



Flanders
State of
the Art

19_040_1
FHR reports

Literature survey on Boundary Element Methods for the Efficient Simulation of Ship Propellers in CFD Codes

DEPARTMENT
**MOBILITY &
PUBLIC
WORKS**

www.flandershydraulicsresearch.be

Literature survey on Boundary Element Methods for the Efficient Simulation of Ship Propellers in CFD Codes

Van Hoydonck, W.; Eloot, K.; Mostaert, F.

Legal notice

Flanders Hydraulics Research is of the opinion that the information and positions in this report are substantiated by the available data and knowledge at the time of writing.

The positions taken in this report are those of Flanders Hydraulics Research and do not reflect necessarily the opinion of the Government of Flanders or any of its institutions.

Flanders Hydraulics Research nor any person or company acting on behalf of Flanders Hydraulics Research is responsible for any loss or damage arising from the use of the information in this report.

Copyright and citation

© The Government of Flanders, Department of Mobility and Public Works, Flanders Hydraulics Research, 2020

D/2020/3241/182

This publication should be cited as follows:

Van Hoydonck, W.; Eloit, K.; Mostaert, F. (2020). Literature survey on Boundary Element Methods for the Efficient Simulation of Ship Propellers in CFD Codes. Version 2.0. FHR Reports, 19_040_1. Flanders Hydraulics Research: Antwerp

Reproduction of and reference to this publication is authorised provided the source is acknowledged correctly.

Document identification

Customer:	Flanders Hydraulics Research	Ref.:	WL2020R19_040_1
Keywords (3-5):	Boundary Element Method, potential code, ship propeller, literature survey, isogeometric analysis		
Knowledge domains:	Harbours and Waterways > Manoeuvring behaviour > Open water > Numerical calculations		
Text (p.):	24	Appendices (p.):	3
Confidentiality:	No	☒ Available online	
Author(s):	Van Hoydonck, W.		

Control

	Name	Signature
Revisor(s):	Eloit, K.	Getekend door: Katrien Eloit (Signature) Getekend op: 2020-07-08 08:00:17 +01:00 Reden: Ik keur dit document goed 
Project leader:	Van Hoydonck, W.	Getekend door: Wim Van Hoydonck (Signat) Getekend op: 2020-07-07 15:30:57 +01:00 Reden: Ik keur dit document goed 

Approval

Head of division:	Mostaert, F.	Getekend door: Frank Mostaert (Signature) Getekend op: 2020-07-07 15:32:18 +01:00 Reden: Ik keur dit document goed 
-------------------	--------------	--



Abstract

The objective of this report is to give an overview of the Boundary Element Method (BEM) for the efficient simulation of ship propellers in Reynolds-Averaged Navier Stokes (RANS) Computational Fluid Dynamics (CFD) simulations. In the first chapter, first the background of the BEM is given. This is followed with a short discussion about methods to determine the correct shape of the vortical propeller wake. Methods to adapt the BEM to include an estimate of viscous effects on the blades are also discussed. After this, the Isogeometric Boundary Element Method (IgBEM) is discussed. The first chapter finishes with an overview of the literature related to coupling BEM with CFD codes. In the second chapter, an overview of existing commercial, research and open-source BEM software is given. This report does not pretend to give a complete overview of all literature available on the subject of panel methods and software that implements the methods, but does try to list some of the more relevant references. In the third chapter, a summary is given and a discussion is held about possibilities for future research.

Contents

Abstract.....	III
Nomenclature	VII
1 Boundary element method background	1
1.1 Panel methods	1
1.1.1 Statement of the problem	1
1.1.2 Implementation choices	2
1.1.3 Boundary conditions	2
1.1.4 Discretisation of the blade geometry	3
1.1.5 Discretisation of the singularity distribution	4
1.2 Shape of the wake	4
1.2.1 Initial wake shape (trim) computation	4
1.2.2 Time stepping approaches.....	5
1.2.3 Vortex particle wake methods	5
1.3 Boundary layer methods	6
1.4 Panel methods and Isogeometric Analysis	6
1.5 Interfacing BEM and CFD	7
1.5.1 Coupling procedure: literature overview	7
1.5.2 FINE/Marine.....	8
2 Overview of existing boundary element software	9
2.1 Commercial codes	9
2.1.1 ProCAL	9
2.1.2 VSAERO/USAERO	9
2.1.3 CMARC.....	10
2.1.4 FlightStream.....	12
2.2 Research codes	13
2.2.1 PMARC	13
2.2.2 panMARE	13
2.3 Open source codes	13
2.3.1 PANAIR	13
2.3.2 Apame	14
3 Summary and discussion	16
References	18
A1 Descriptions of Commercial Software	A1
A1.1 VSAERO	A1
A1.2 USAERO.....	A1
A1.3 OMNI3D	A2
A1.4 SPIN(w)	A3

List of Figures

Figure 1	Panel arrangement for propeller P4119: conventional grid and grid with <i>hydrodynamic tip</i>	3
Figure 2	USAERO pressure distribution on a propeller computation in non-uniform inflow	10
Figure 3	Pressure distribution on the P-51 Mustang computed with CMARC.	11
Figure 4	Streamlines and pressure distribution around an aerofoil with extended flap.....	11
Figure 5	F16-A surface discretisation and vorticity distribution with wake filaments.	12
Figure 6	Surface pressure visualisations of computations executed with VSAERO.....	A1
Figure 7	Unsteady simulation of the release of payload from the wings of a jet fighter.	A2
Figure 8	Visualisation of the pressure on the fuselage of a jet trainer with OMNI3D.	A3
Figure 9	Wake preprocessing using SPINw: configuration of wake geometry of a jet engine aircraft.	A3

Nomenclature

Abbreviations

BEM	Boundary Element Method
BIE	Boundary Integral Equation
CAD	Computer Aided Design
CCCM	Constant Circulation Contour Method
CFD	Computational Fluid Dynamics
CRS	Cooperative Research Ships
CVC	Constant Vorticity Contour
DWT	Digital Wind Tunnel
FEA	Finite Element Analysis
FHR	Flanders Hydraulics Research
FLAG	Flow Adapted Grid
FMM	Fast Multipole Method
FV	Finite Volume
GPL	GNU General Public License
IgA	Isogeometric Analysis
IgBEM	Isogeometric Boundary Element Method
MOL	Method of Lines
NURBS	Non-Uniform Rational B-Spline
ODE	Ordinary Differential Equation
PANAIR	Panel Aerodynamics
PDAS	Public Domain Aeronautical Software
PMARC	Panel Method Ames Research Center
ProCAL	Propeller Calculation
PSW	Personal Simulation Works
RANS	Reynolds-Averaged Navier Stokes
VPM	Vortex Particle Method

1 Boundary element method background

1.1 Panel methods

This section focuses solely on potential panel methods, without placing them in a wider context of other approximations of the Navier-Stokes equations. More information on other approximations of these equations can be found in Bertin and Cummings (2009), Peerlings (2018) and Wendt (2009) and the references therein.

1.1.1 Statement of the problem

The type of problem that can be solved with panel methods is one where the majority of the flow is inviscid, irrotational and incompressible. For incompressible flow, the volume of a fluid element is constant as it moves along a streamline,

$$\nabla \mathbf{V} = 0. \quad (1)$$

For irrotational flow, the fluid element does not rotate as it moves along a streamline. For this flow type, the velocity can be expressed as the gradient of a scalar *velocity potential* function ϕ ,

$$\mathbf{V} = \nabla \phi. \quad (2)$$

The combination of Eqs. 1 and 2 gives Laplace's equation,

$$\nabla \cdot \nabla \phi = \nabla^2 \phi = 0. \quad (3)$$

Viscous and rotational flow is allowed, but it is assumed to be confined to thin surfaces (such as boundary layers and wakes behind a lifting body).

With an appropriate set of boundary conditions, a boundary value problem can be constructed on the surface boundary S . Due to the lack of viscosity, the (no-slip) tangential velocity on a solid body cannot be prescribed, only the normal velocity can be enforced. The wake surface S_W has zero thickness, and the normal velocity jump and pressure jump across it are zero, while a jump in the potential is allowed (Kerwin *et al.*, 1987). For a lifting surface, the potential jump across the wake surface equals the circulation on the body and is constant in the streamwise direction. At the trailing edge, a Kutta condition is required to fix the circulation to ensure that the flow leaves the trailing edge smoothly (Kerwin *et al.*, 1987; Wendt, 2009). On the outer surface, the perturbation velocity due to the presence of the (lifting) body vanishes. These boundary conditions are discussed in more detail in § 1.1.3.

By assuming a fictitious fluid inside the body with an associated perturbation velocity potential ϕ' the integral equation can be obtained (Kerwin *et al.*, 1987; Maskew, 1987),

$$4\pi\phi(p) = \iint_{S_B} \left[(\phi(q) - \phi'(p)) \frac{\partial}{\partial n_q} \frac{1}{R(p;q)} - \left(\frac{\partial \phi(q)}{\partial n_q} - \frac{\partial \phi'(q)}{\partial n_q} \right) \frac{1}{R(p;q)} \right] dS \\ + \iint_{S_W} \Delta \phi(q) \frac{\partial}{\partial n_q} \frac{1}{R(p;q)} dS, \quad (4)$$

where ϕ and ϕ' are the perturbation potentials in the real and fictitious domains, $p(x, y, z)$ is a field point where the potential is computed, $q(x, y, z)$ is location of the singularity and $R(p; q)$ is the distance between the two points.

Eq. 4 computes the value of $\phi(p)$ in terms of ϕ and $\partial\phi/\partial n$ on the surface boundaries. The difference between the external and internal potential is defined as a *doublet* μ while the difference between the normal derivatives of the external and internal potential is a *source* σ .

By choosing an appropriate value for the fictitious potential ϕ' inside the body, different kinds of potential methods can be formulated with different sets of singularities. If $\phi' = 0$, the perturbation potential method is obtained (Kerwin *et al.*, 1987; Maskew, 1987). The total potential method is obtained if the fictitious internal potential is set to the negative of the inflow velocity potential $\nabla\phi_\infty = \mathbf{U}_\infty$, hence $\phi' = -\phi_\infty$.

By taking the normal derivative w.r.t. the field point p of Eq. 4, an alternative integral equation can be constructed: with appropriate choices of the internal velocity potential, velocity field formulations can be constructed (Kerwin *et al.*, 1987) but these will not be discussed here.

1.1.2 Implementation choices

Different choices must be made when implementing a panel method:

- the type of singularity used to describe lifting and non-lifting objects and wakes (sources, dipoles, vortices, and combinations thereof);
- the order of the singularity used: constant, linear, quadratic, and/or higher;
- *direct* methods (where the velocity is solved for directly) and *indirect* methods (where the potential is solved for).

Note that the above distinction between *direct* and *indirect* methods is found in literature (Erickson, 1990), although recently, Bertram (2012) and Gaschler (2017) switched the description for the direct and indirect formulation.

The direct method (where the velocity is solved for directly) was first introduced by Hess and Smith (1964) while the indirect method was introduced approximately a decade later by Morino (1974).

The different steps required with all panel methods are:

- discretisation of the geometry (both solid bodies and wake);
- discretisation of the singularity distribution;
- boundary conditions satisfied at the collocation points of the panels;
- a Kutta condition imposed to obtain a correct flow field.

Not all reported panel method implementations seem to perform (or converge) equally well for the simulation of marine propellers (Lee, 1987). Convergence characteristics of multiple implementations of panel methods have been tested for two-dimensional test cases (Vaz *et al.*, 2000). The conclusion was that the Morino formulation (based on the disturbance potential) – which is often used for propeller panel implementations – has convergence issues with foils with a cusped trailing edge.

1.1.3 Boundary conditions

The external Neumann boundary condition must be satisfied at the solid surface S (Maskew, 1987),

$$\mathbf{n} \cdot \nabla\phi = -V_N - \mathbf{n} \cdot \mathbf{V}_S, \quad (5)$$

where V_N is the resultant normal velocity component relative to the surface, which is zero for a solid surface with no transpiration. $\mathbf{V}_S$ is the local velocity of the surface (due to body rotation, for example). For steady problems in a body-fixed coordinate system, $\mathbf{V}_S$ is zero and Eq. 5 reduces to

$$\mathbf{n} \cdot \nabla\phi = -V_N. \quad (6)$$

Infinitely far away from the body, the disturbance in the flow caused by the presence of the body should vanish,

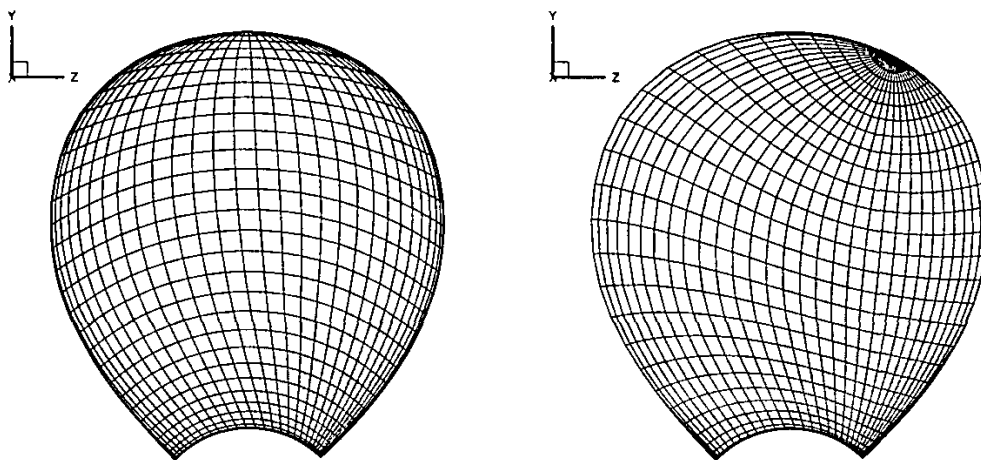
$$\lim_{r \rightarrow \infty} \nabla \phi = 0. \quad (7)$$

The wake is a zero-thickness surface that carries all shed and trailed vorticity with the local velocity away from the body. Two boundary conditions are applied to the wake. The first one states that the wake is carried away with the local velocity and the second one states that the pressure on either side of the wake is equal (force-free condition). To uniquely define the circulation around the lifting surface, the Kutta condition must be imposed at the trailing edge. The most widely used Kutta condition is the one introduced by Morino (Morino, 1974), which states that dipole strength of the first wake panel equals the difference between the dipole strength of the two panels adjacent to the trailing edge (Morino, 1974). This method works well for two-dimensional cases and cases without a significant cross-flow component at the trailing edge, but for cases with a significant cross-flow at the trailing edge, the method contains a fundamental error (Lee, 1987). By requiring that the potential across the wake equals the total potential of the upper and lower panels at the trailing edge, the circulation converges to the correct analytical value for two-dimensional cases. For three-dimensional cases with significant cross flow, this modified Kutta condition is not enough. By explicitly requiring that the pressure at the last panels are equal, an iterative pressure Kutta condition is obtained (Kerwin *et al.*, 1987; Lee, 1987). This iterative pressure Kutta condition has been used in more recent (maritime) panel implementations as well (Baltazar, 2008; Gaschler, 2017; Vaz, 2005).

1.1.4 Discretisation of the blade geometry

It has been shown that the panel discretisation near the tip of a blade and the associated (and a-priori determined) location (and end) of the trailing edge has an impact on convergence of the pressure distribution near the tip (Baltazar, 2008; Baltazar *et al.*, 2005; Pyo, 1995). The *hydrodynamic tip*¹ grid (right-hand side of Fig. 1) as compared to the usual propeller blade discretisation (left-hand side of Fig. 1) based on the blade geometric characteristics (rake, skew, chord length, pitch and blade profile as a function of radial distance) (Baltazar and Falção de Campos, 2006; Hoshino, 1989; Jennings, 2010) has been shown to improve convergence characteristics of the iterative scheme to determine the pressure Kutta condition (Baltazar, 2008; Pyo, 1995).

Figure 1 – Blade panel arrangement for propeller P4119: conventional grid (left) and new panel arrangement with *hydrodynamic tip* (right) (from Baltazar *et al.* (2005)).



¹The *hydrodynamic tip* is the location on the blade where the wake ends which does not necessarily coincide with the geometric tip (Baltazar, 2008; Baltazar *et al.*, 2005).

Likewise, the use of a Flow Adapted Grid (FLAG), where the blade grid is adapted such that the computational tip corresponds to the location where the tip vortex detaches from the blade, has been shown to improve the convergence characteristics of the panel method applied to marine propellers (Pyo, 1995).

This discussion shows that discretizing the blade geometry of a marine propeller is not as straightforward as the discretisation of an aircraft wing, where the wing tip is clearly defined.

1.1.5 Discretisation of the singularity distribution

Discretisation of the blade and wake geometry in a finite number of panels converts Eq. 4 into a system of linear equations in the unknown doublet (and source) values on the panels. The order of the singularity distribution over each panel has an influence on the convergence characteristics of the method and the difficulty of the actual implementation. The simplest (and nowadays most-widely used) methods use constant distributions of doublets and sources per panel (Ashby *et al.*, 1991; Baltazar, 2008; Bosschers, 2007; Filkovic, 2008; Gaschler, 2017; Hundemer, 2013; Jennings, 2010; Kerwin *et al.*, 1987; Lee, 1987; Martin, 2015; Maskew, 1987; Pyo, 1995; Takinaci *et al.*, 2003; Tarafder *et al.*, 2010; Vaz, 2005; Willis *et al.*, 2006; Yari and Ghassemi, 2016; Yiran *et al.*, 2018). Linear and higher-order potential distributions have been used in the past (Baltazar, 2005; de Koning Gans, 1994; Hoeijmakers, 1989; Magnus and Epton, 1981; Pease Moore, 2013; Willis, 2006) to (amongst others) reduce or eliminate issues with leakage of constant strength implementations, and to get more accurate solutions at coarser discretisations. The issue of flow leakage due to the non-flat nature of propeller panel discretisations has been solved by assuming that quadrilateral panels are hyperboloidal in shape: the corner points are connected by straight line segments and the influence coefficients are obtained by a geometric argument avoiding direct integration (Newman, 1986). In general, higher-order implementations require more computations per panel, but need less panels to get a similar level of accuracy as low-order methods (Kerwin *et al.*, 1987).

1.2 Shape of the wake

During a time simulation in which the boundary conditions vary (such as the angular velocity, the angle of attack of the propeller or the velocity field in which the propeller operates), the loads on the propeller blades will change and as a consequence, the shed and trailed vorticity in the wake will vary in time. In turn, this change in vorticity in the wake affects the shape of the wake: a higher blade loading will result in stronger trailed (tip) vortices that will have a larger pitch. The tip vortex strength affects the inflow (or induced velocity) in the wake plane and thus the angle of attack of the propeller blades. In general, the mechanism of feedback between the different parts of the system means that a solution needs to be obtained in an iterative fashion, even for a steady (harmonic) condition where the boundary conditions are fixed in time. Determining the shape of the wake can be split in two parts:

- determination of wake shape for fixed boundary conditions (*trim* computation²);
- evolution in time of wake shape for fixed or variable boundary conditions.

1.2.1 Initial wake shape (trim) computation

An overview of the literature related to the computation of a harmonic wake shape under fixed boundary conditions is given in Van Hoydonck (2013). Normally, the problem is written as a set of partial differential equations

²Note that the term *trim* computation is taken from the aerospace sciences where it refers to the general process of determining states and controls of a dynamic system given a set of boundary conditions (Van Hoydonck, 2013), whereas in naval architecture, *trim* of a vessel is only related to the amount of pitch of a vessel in a certain sailing condition.)

that are discretised using a finite difference scheme and these are solved with a relaxation scheme (Chopra *et al.*, 1997). The process of determining the correct partial differential equation and the subsequent discretisation using a finite difference approach can be left alone by looking at the problem from a perspective of the streamlines that emanate from the propeller blades and that carry the vorticity downstream. By reversing the equation to compute the derivative along a Non-Uniform Rational B-Spline (NURBS) curve at a fixed parametric point, an interpolation algorithm follows that can reconstruct a streamline given its starting point and derivative data along the curve (Van Hoydonck, 2013). This method has been shown to be robust (independent of the initial wake shape) and to work well for cases where the slipstream of the propeller is blocked by a solid surface.

1.2.2 Time stepping approaches

Similar to the trim problem described in the previous section, the time stepping problem can be described by a partial differential equation that is discretised using some finite difference approximation. Another approach is the use of the Method of Lines (MOL) for the solution of partial differential equations by reducing the number of dependent variables to only one, usually replacing the spatial variable with algebraic approximations, see Hidalgo (2014) and the references therein. The PDE problem is thus reduced to a system of Ordinary Differential Equation (ODE) that can be solved with standard ODE solvers.

1.2.3 Vortex particle wake methods

One of the biggest disadvantages of the standard engineering dipole-based wake models as used in the majority of panel methods (Baltazar, 2008; Bosschers, 2007; de Koning Gans, 1994; Gaschler, 2017; Hundemer, 2013; Magnus and Epton, 1981; Vaz, 2005; Willemsen, 2013), is that these wake models inherently assume that the flow is *ordered* (Hoeijmakers, 1989). The individual Lagrangian marker points (i.e. the corner points of a doublet panel, or the marker points of a vortex filament section) are transported with the local velocity which may mean that as time progresses, the panel becomes severely distorted and the assumption of ordered flow may no longer hold. Most of the codes do not contain features to adaptively refine panels as they become larger or severely distorted because this means that the linear systems that are solved during BEM simulations change size, which in turn can negatively affect the computing time to advance the system in time. Apart from engineering corrections to account for viscous diffusion by changing the core sizes as time progresses, these methods have no means to account for viscous diffusion in a first principles way. The only method compatible with panel methods that can account for viscous diffusion in the wake and that does not assume that the flow is ordered is the Vortex Particle Method (VPM) (Winckelmans, 1989) where the part of the fluid domain with non-zero vorticity (i.e., the shed and trailed vortical wake behind a propeller) is discretised with vortex particles that are periodically redistributed to ensure that the vortex particles overlap (Winckelmans *et al.*, 2005; Winckelmans and Leonard, 1993). Viscous vortex particle methods have been used in combination with a Fast Multipole Method (FMM)-accelerated arbitrarily high-order panel method (Pease Moore, 2013), a multipole-accelerated unsteady panel method with quadratic curved panels (Willis, 2006; Willis *et al.*, 2006), and a low order panel method (Martin, 2015). For engineering applications where panel methods are used with geometries such as ducted propellers (Kerwin *et al.*, 1987), standard structured doublet lattice wake models may require a lot of tuning and effort to improve the predictions in the gap between the propeller tip and shroud (Baltazar and Falcao de Campos, 2009; Baltazar *et al.*, 2018; Willemsen, 2013).

1.3 Boundary layer methods

Boundary layer methods can use the pressure distribution from the panel code to compute the displacement thickness of the viscous layer around the discretised body. This displacement thickness is then used in the panel code by either modifying the body shape by adding the incremental thickness to the discretised body shape or, by adjusting the source strengths of the individual panels such that each panel ejects or sucks enough fluid to cause the resulting flow field to be approximately displaced by the displacement thickness that was computed in the previous iteration (Drela, 1989; Erickson, 1990; Maskew, 1987).

Simpler methods use the tangential velocity distribution along a streamline and semi-empirical methods (Moran, 1984) to compute the growth of the boundary layer along the aerofoil (Gillebaart, 2018; Gillebaart and De Breuker, 2016).

1.4 Panel methods and Isogeometric Analysis

Isogeometric Analysis (IgA) has been introduced as a means to bridge the gap that exists between Finite Element Analysis (FEA) and Computer Aided Design (CAD). By utilising the NURBS basis functions used in CAD as a basis for the discretisation of both the geometry and the solution fields in the FEA, the time-consuming and non-trivial task of converting the input geometry to a form that is suitable for the FEA can be reduced significantly (Hughes *et al.*, 2005). The use of higher-order elements and exact geometry representation (which means that refinement of the discretisation can be executed without communicating with the original CAD model) in IgA have shown to be advantageous for the accuracy of solutions (Cottrell *et al.*, 2009). However, due to the three-dimensional nature of FEA, one often needs to construct (by hand) a trivariate spline mapping of the geometry, a task that is not always trivial for real-world geometry (Marussig *et al.*, 2015).

In recent years, the concept of IgA has been applied to BEM problems (Beer *et al.*, 2020; Chen *et al.*, 2019; Dölz *et al.*, 2018; Marussig *et al.*, 2015; Politis *et al.*, 2014). Three-dimensional BEM problems only require a two-dimensional discretisation of the boundary surfaces, which means that IgA can be applied in this field more easily to real-world problems as many CAD applications (such as Rhino) only model the bounding surfaces of the geometry. But here also, the need to use trimmed surfaces in geometry poses a challenge for the application of IgBEM for which researchers only recently have found a solution (Beer *et al.*, 2015).

In Politis *et al.* (2014), problem formulations are based on a Boundary Integral Equation (BIE) for the velocity potential combined with a Kutta condition for lifting surfaces. The resulting IgBEM method utilises the geometric NURBS representation also for representing the velocity potential. Compared to a low-order conventional panel method, the IgBEM method was shown to require significantly fewer degrees of freedom to reach similar or better levels of accuracy. The general features of BEM formulations also apply to IgBEM implementations: the resulting linear system of equations is dense and the evaluation of the field functions is still an N-body problem that scales with the square of the number of elements. For this reason, FMM-type enhancements have been implemented in IgBEM formulations as well (Chen *et al.*, 2019; Dölz *et al.*, 2018; Marussig *et al.*, 2015).

In (aero)nautical engineering, IgBEM has been used recently for the simulation and shape optimization of lifting flows around hydrofoils (Kostas *et al.*, 2016; Politis *et al.*, 2014), wave scattering problems governed by the Helmholtz equation (Chen *et al.*, 2019), wave propagation and wave-structure interaction (Heredia, 2016), wave loading on ships (Kaklis *et al.*, 2017), wave resistance of ships (Wang *et al.*, 2017) the open-water marine propeller flow problem (Chouliaras *et al.*, 2017), ship hull shape optimization (Kostas *et al.*, 2015), low-fidelity aeroelastic analysis and optimization of morphing airfoils (Gillebaart and De Breuker, 2016) and aerostructural optimization of composite aircraft wings (Gillebaart, 2018).

The specific issues that have been discussed previously (such as the location and extend of the trailing edge line where the wake forms, and the shape of the wake) also apply to IgBEM implementations. Although the IgBEM

promises a more accurate solution with less variables, the user should make sure that the correct problem is solved accurately (i.e., the wake release point near the propeller blade tip should be located at its correct position).

1.5 Interfacing BEM and CFD

When a propeller BEM code is interfaced with a Finite Volume (FV) CFD code to include the effects of a propeller without the disadvantages of completely resolving the viscous flow around the rotating propeller (such as increased computing times), information must be exchanged between the BEM code and the CFD code. The BEM code requires a uniform or non-uniform flow field in order to solve for the non-penetrating boundary conditions at the panel control points. With this flow field, the linear systems of equations are solved and the distribution of pressure on the propeller is determined. This pressure distribution (which can be integrated to give the resultant forces and moments on the propeller) is passed to the CFD code where they are added as body forces to the governing equations.

In the coupling scheme (Hally, 2015), the propeller influence on the RANS solution is introduced via a force field that mimics the propeller action, similar to how an actuator disk model is added in a RANS computation. The influence of the hull on the propeller is introduced by supplying an inflow wake computed from the RANS solution from which the propeller induction calculated with the BEM has been subtracted. This procedure is repeated until the effective propeller wake no longer changes.

1.5.1 Coupling procedure: literature overview

The coupling procedure between the RANS flow solver *FresCo+* and the potential panel method *panMare* is presented in Wöckner *et al.* (2011). Coupling is based on the transfer of inviscid propeller-induced body forces to the viscous solver and the transfer of the effective wake velocities to the inviscid solver. Details of the spatial mapping techniques required to exchange data between two grids of significantly different nature are given as are details on the temporal transfer of information. The viscous velocity distribution is extracted from the RANS solver 0.1 D and 0.2 D upstream of the propeller. The inviscid pressure values on the individual panels of the propeller are distributed to adjacent fluid cell centers and added to the RANS solver as body forces. For the coupling procedure, the inviscid solver needs the velocity field from the viscous solver, without the influence of the induced velocities due to the action of the propeller in the inviscid method. The paper explains the details of two coupling modi: an explicit and an implicit procedure. The method is also extended to free-surface flows, where the location of the free surface is accounted for in the inviscid panel method with an inviscid wall boundary. Instead of transferring the actual free surface as determined by the viscous solver to the panel code, Fourier transforms are calculated along two-dimensional longitudinal sections and these Fourier coefficients are transferred to the inviscid solver. The latter one uses the Fourier coefficients to reconstruct the free surface at arbitrary locations.

Rijkema *et al.* (2013) explains the procedure that is used to couple Propeller Calculation (ProCAL) with Re-Fresco. First, a steady RANS simulation is executed and the wake field at the propeller plane is evaluated. This field is used for the first BEM computation, where the loading distribution on the propeller blades is determined. The non-uniform nature of the wake field, means that an unsteady BEM computation is required. The average forces on the camber plane are computed at different positions of the blades, a three-dimensional time-averaged force distribution is constructed. This time-averaged BEM propeller loading distribution is used in the RANS computation as a body force field. The force field is interpolated to the RANS grid. To correct for deficiencies in the interpolation procedure, the body force field can be scaled to ensure equal thrust in the RANS computation as determined by the BEM. With the propeller body forces included in the RANS computation, the total wake field can be computed. The effective wake of the propeller is obtained by subtracting the

propeller induced velocities from the total wake field. The averaged propeller induced velocities are determined in the BEM computation. This procedure is repeated with the new effective wake field until convergence (change in thrust and effective wake velocities) is reached.

Hally (2015) details a procedure where ProCAL is coupled with OpenFOAM. The paper focuses on the correct representation of the propeller blade blockage in the RANS computations by adding source mass terms to the equations of motion. Failure to include the propeller blade blockage in the RANS computation results in an increase of the axial velocity in the effective wake which increases the advance ratio and reduces the predicted thrust and torque. In a subsequent publication, Hally details the procedure to interpolate the velocity field that is used in the exchange of information between the ProCAL and RANS procedures (Hally, 2017). The method uses multiple planes normal to the rotation axis of the propeller which means that axial velocity variations are allowed. This contrasts with simpler methods where the velocity is sampled at a single plane (Anon., 2020; Hally, 2012).

1.5.2 FINE/Marine

FINE/Marine has a predefined interface for adding a Boundary Element Method to CFD computations through a dynamic Fortran library. The coupling can be made directly in the dynamic library or this can be kept separately. In the latter case, the communication is handled through system calls and text files.

The interface consists of two parts where information is exchanged between the two programs:

- a velocity field ahead of the propeller is passed to the BEM software for use as boundary condition by the panel code;
- a body force distribution (pressure) at the propeller blade location used in the CFD code to compute the propeller forces.

According to the manual of FINE/Marine (Anon., 2020), only the thrust and torque forces are supported through this interface, similar to the functionality of the available actuator disk model. Once the propeller code works and has been validated for simple cases, this assumption could be tested in conditions where a significant lateral flow is added. Then it can be verified if the restriction present in FINE/Marine is justified.

The AC&E company in the UK that provides support for the panel methods of AM Inc. in Europe also uses the CFD software of Numeca³. Two-way coupling, where information is exchanged between both programs has not been used, only a one way coupling has been used. In this method, a non-uniform velocity field directly upstream of the propeller or in a volume around the propeller is extracted from the CFD computation and used as input for the panel code.

³<http://www.ace1.co.uk>

2 Overview of existing boundary element software

This chapter discusses existing boundary element software that solve the Laplace equation in an infinite domain (suitable for low-speed aero and hydrodynamics), other potential methods (for things such as sound propagation) are not discussed. In addition, the interface between the BEM and CFD codes is discussed as well.

2.1 Commercial codes

2.1.1 ProCAL

In 2003, the Cooperative Research Ships (CRS) organisation initiated the development of a panel code for the analysis and simulation of marine propellers (Bosschers, 2007). A working group was formed consisting of interested parties such as shipyards, a propulsor manufacturer, classification societies, navies and ship research institutes. Within this working group, a panel method (named ProCAL and implemented by MARIN) and a graphical user interface with grid generation capabilities have been developed. The code is available for members of the CRS organisation, which has a yearly participation fee of €25 000 (Toxopeus, 2015). One of the current ongoing projects is the adaptation the PROCAL program for the simulation of ducted propellers (Moulijn *et al.*, 2019)⁴. In the recent past, the PROPDEV working group⁵ aimed at developing a tool based on ProCAL for the design of propulsion including the interaction with the hull and appendages. At TU Delft, ProCAL has recently been used in research to simulate flexible marine propellers in an efficient way (Maljaars *et al.*, 2018). Other participants of the CRS have coupled ProCAL with OpenFOAM (Hally, 2017, 2015).

2.1.2 VSAERO/USAERO

VSAERO(Maskew, 1988, 1987) is a low-order panel method developed and commercialised by Analytical Methods, Inc. since 1980s. It can be used to compute non-linear aerodynamic characteristics of arbitrary three-dimensional configurations in subsonic flow. It contains an iterative wake shape calculation procedure and the method is coupled with an integral boundary layer solver which means that the effects of viscosity can be accounted for. Geometries are discretised using quadrilateral panels on which piecewise constant source and doublet singularities are distributed. Panel source values are determined by the external Neumann boundary condition that controls the normal component of the local resultant flow while the doublet values are solved after imposing the internal Dirichlet boundary condition of zero perturbation potential at the underside of the panel centres. The original code was written in standard FORTRAN IV and the first version of the code allowed for the solution of problems with up to 1000 panels. This version was used as a base for the Panel Method Ames Research Center (PMARC) panel method. As part of this literature survey, a request for information regarding the current capabilities of VSAERO and USAERO was send to AMI Aero, LCC (the successor of Analytical Methods, Inc.). Fig. 2 shows the pressure distribution on a propeller in a non-uniform flow field computed with USAERO (Nathman, 2020). Other than that this case required less than 10 minutes on a Linux machine 5 years ago, no information was provided about it. Although VSAERO and USAERO are capable of computing the potential flow around marine propellers, it seems not much time has been dedicated to thoroughly validate the

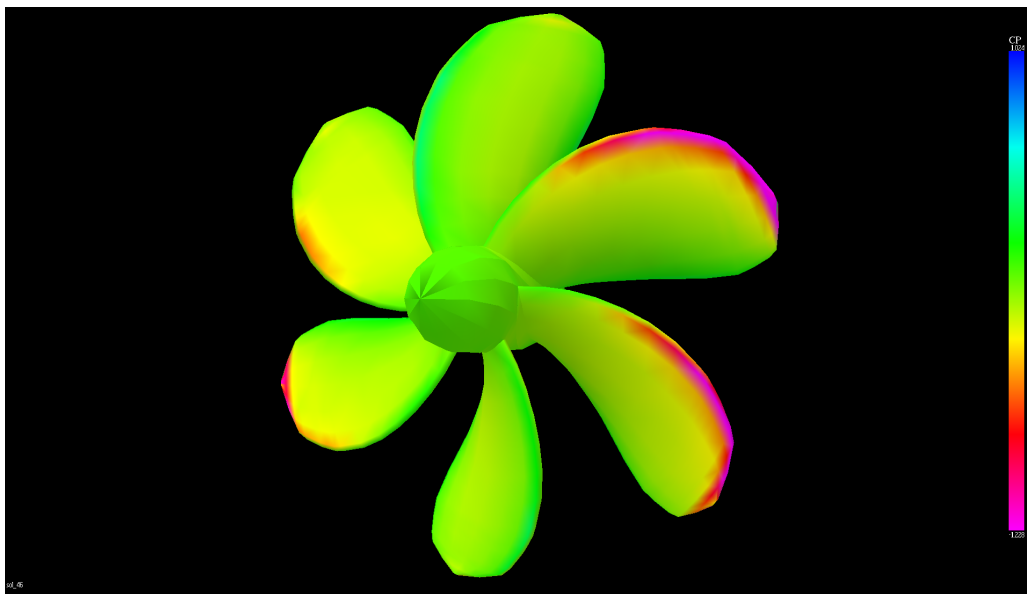
⁴<http://www.crships.org/web/Ongoing-Projects/Ducted-Propellers.htm>

⁵<http://www.crships.org/web/Recent-Projects/Propeller-Design-and-Development.htm>

software for this type of problem as has been done more recently with dedicated marine software (Baltazar, 2008; Moulijn *et al.*, 2019; Vaz, 2005).

A disadvantage of VSAERO and USAERO is that separate programs are used for unsteady and steady computations, while ideally, this should be possible with a single program. AC&E in the UK⁶ provides support for VSAERO and USAERO to customers in Europe. Upon inquiry, the programs can be licensed on an annual or permanent basis. Typically, a single user floating license price for either VSAERO or USAERO is €10 500 including SPINW for the wake definition. For postprocessing the output of VSAERO and USAERO, AM Inc. has developed the interactive 3D data visualisation tool OMNI3D. This has an additional license cost of €4500 per year. These prices include annual support and maintenance service. Descriptions of the four programs mentioned in this section that can be found on the website of aero attack⁷ are given in Appendix A1.

Figure 2 – USAERO pressure distribution on a propeller computation in non-uniform inflow (Nathman, 2020).



2.1.3 CMARC

CMARC is an enhanced version of NASA's PMARC-12, which in turn is a descendant of an early version of VSAERO. It is part of AeroLogic's⁸ Digital Wind Tunnel (DWT) that can be used to determine longitudinal and lateral static and dynamic stability derivatives, neutral point and elevator deflections required for trim. The software suite contains a lofting program (Loftsman/P) for designing the outer surface of aeroplanes and boats and a postprocessing program (Postmarc) for visualisation of the solution. The lofting program can output the lofted shapes in IGES and DXF format as well as creating input files for CMARC and VSAERO. The website contains analysis examples of general aviation type aircraft and boat hulls generated with the software suite, but lacks examples of propeller analyses. The software is written in ANSI-C and executables are only provided for Windows. The yearly licensing costs of the software components are listed on their website and these are repeated in Table 2.

An example of the panelling and pressure distribution on the fuselage and wings of a P-51 Mustang are shown in Fig. 3, while in Fig. 4, the two-dimensional flow around an aerofoil with extended flap is shown. This example seems to include the effects of viscous interaction in the boundary layer, because the pressure around the aerofoil does not seem to decrease in the direction normal to the surface as the distance to the aerofoil is

⁶<http://www.acel.co.uk>

⁷<http://www.aeroattack.com>

⁸<http://www.aerologic.com>

increased.

Table 2 – License cost for the Aerologic software

component	price (US dollar)
Personal Simulation Works (PSW) with DWT	3750
PSW with CMARC only	3125
CMARC ANSI-C source code (unix compatible)	625
Loftsman/P	1250
CMARC	625
DWT (includes CMARC)	1250
Postmarc	1250

Figure 3 – Pressure distribution on the P-51 Mustang computed with CMARC. Retrieved from <http://www.aerologic.com/example/p51.html> © AeroLogic.

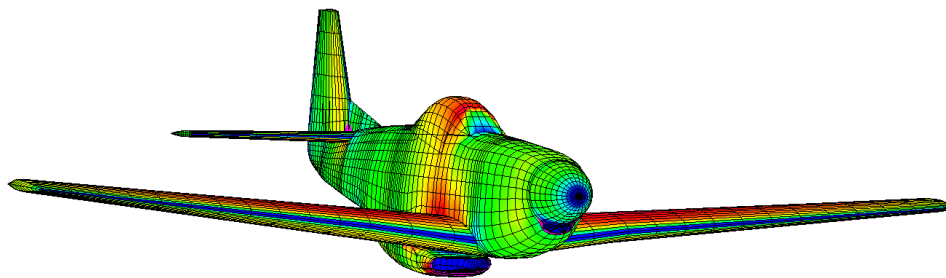
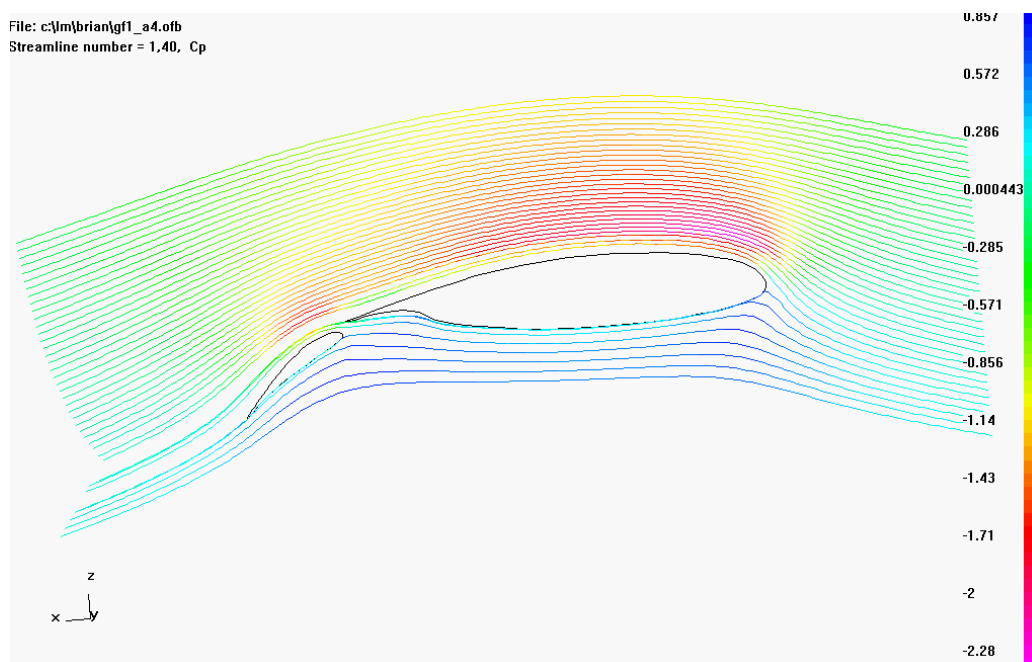


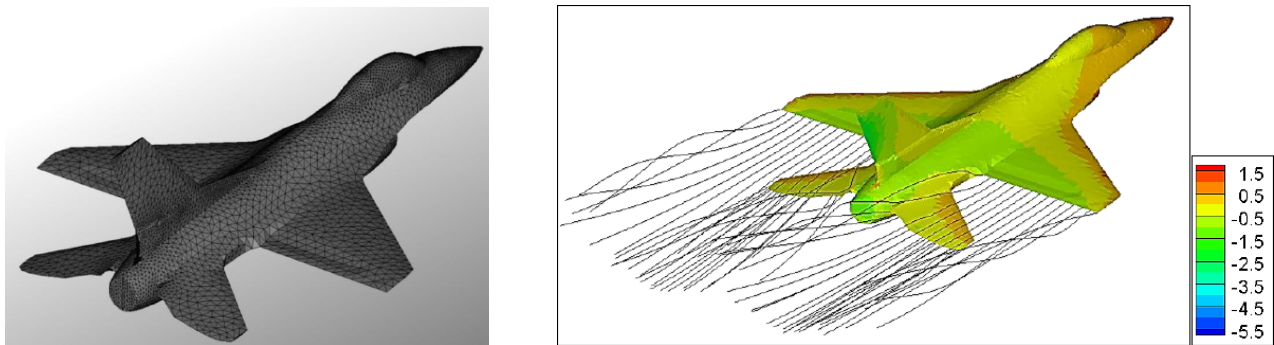
Figure 4 – Streamlines and pressure distribution around an aerofoil with extended flap. Retrieved from <http://www.aerologic.com/example/gaw.html>, © AeroLogic.



2.1.4 FlightStream

FlightStream® is a surface vorticity flow solver developed and marketed by Research in Flight⁹ to allow users to develop optimized designs for compressible and incompressible subsonic vehicles as well as transonic vehicles. Applications range from modeling UAVs, high-altitude-aerostats, high subsonic transport and military aircraft, marine propellers and under-water vehicles, wind turbines, propellers, store separation and high-lift components. An example of the surface discretisation and vorticity distribution around an F16-A is shown in Fig. 5.

Figure 5 – F16-A surface discretisation (left) and vorticity distribution (right) with wake filaments (Ahuja, 2013).



The surface vorticity solver uses an unstructured triangular surface discretisation in combination with an unstructured wake strand model (Ahuja, 2013). The wake filaments originate at nodes that are flagged by the software as trailing edges and the strength of the vorticity trailed at these points is a summation of the strength of the edges that are connected to these points. A point that is not touched upon in the thesis of Ahuja is (unsteady) time simulation. By the nature of the wake implementation, only steady-state simulations can be executed as vorticity is only trailed in the wake, and not shed. This is a shortcoming that makes the software unsuitable for performing propeller simulations with unsteady boundary conditions. The Constant Vorticity Contour (CVC) wake model has elegantly solved the issue of combining shed and trailed vorticity in a vortex filament wake model almost 30 years ago by placing the vortex filaments along contours of constant sheet strength for the sheet of vorticity trailed from each lifting surface (Quackenbush *et al.*, 1991), so there is certainly a possibility to implement unsteady time simulations in vorticity panel methods that utilize vortex filament wakes. Recently, this type of wake model was applied to time simulations of wind turbine wakes (Su and Bliss, 2019), where it is called the Constant Circulation Contour Method (CCCM).

For use in a marine context, where the trailed wake is (highly) curved, the use of straight line segments to approximate a curved vortex filament (as is done by Ahuja) is far from optimal because (a) many segments (and hence longer computing times) are required to get a good approximation¹⁰ and (b) without corrections for the local curvature at a point on a filament where the velocity is computed, fundamental errors are made (Bliss *et al.*, 1987). Hence, in a marine context where vorticity shed from a propeller blade has high curvature, the use of longer curved vortex filaments as compared to shorter straight line segments may prove to be advantageous for both the accuracy of the results and the reduced computing times.

⁹www.researchinflight.com

¹⁰This argument also holds for (low-order) panel methods where a doublet lattice is used to represent the wake due to the equivalence of a doublet panel and a vortex ring.

2.2 Research codes

2.2.1 PMARC

PMARC is a low-order potential panel method developed by NASA Ames Research Center at the end of the 1980s (Ashby *et al.*, 1988, 1991) to predict flow fields around complex three-dimensional geometries. Experience with panel methods such as Panel Aerodynamics (PANAIR) and VSAERO had shown that low-order methods can provide nearly the same accuracies as higher order methods for many cases. The computing times are significantly shorter for a low-order implementation and at the same time, the implementation of a low-order method is significantly easier. Furthermore, adjacent panels do not have to match exactly in a low-order method. The code is based on versions of VSAERO delivered to NASA Ames in the first half of the 1980s. The PMARC low-order panel method is available through the NASA software website¹¹ for U.S. Government-authorized use only¹².

2.2.2 panMARE

panMARE is a potential panel method developed at the Technical University of Hamburg at the Institute for Fluid Dynamics and Ship Theory¹³. The potential code has been developed with German government funding and is not available for general use outside Germany (Abdel-Maksoud, 2015). The implementation of the basic method for marine propellers is discussed in Hundemer (2013) (in German) while the addition of a cavitation model is detailed in Gaschler (2017).

2.3 Open source codes

2.3.1 PANAIR

PANAIR is an advanced method for the solution of linear integral equations of inviscid subsonic and supersonic potential flow in three dimensions (Magnus and Epton, 1981). The method uses source and doublet panels as boundary surfaces. Complex geometries are split in multiple networks on which panels are laid out in a structured manner. Higher order (i.e. curved) panel geometries are supported over which linearly varying source or quadratically varying doublet singularities are employed. All influence coefficients are computed in closed form. The program has been extensively validated and shows a marked insensitivity to the size, shape and arrangement of panels. For complex configurations, the method has advantages as fewer panels are required than standard first-order methods. Originally developed for NASA Ames by Boeing in the 1980s, the original Fortran program has been released through the Public Domain Aeronautical Software (PDAS) website¹⁴. This website also contains a preprocessor (PANIN¹⁵) to format the input files for PANAIR.

Evaluation of the code

The source code is contained in a single file with more than 69×10^3 lines of fixed-format Fortran 77. The theory manual (Magnus and Epton, 1981) mentions that the program is capable of superimposing an incremental

¹¹<https://software.nasa.gov/software/ARC-14407-1>

¹²The research executed at Flanders Hydraulics Research (FHR) does not fall under U.S. Government-authorized use.

¹³<https://www.tuhh.de/panmare/home.html>

¹⁴<http://www.pdas.com/panair.html>

¹⁵<http://www.pdas.com/panin.html>

velocity on the freestream to simulate effects such as rotational motion or a propeller slipstream. The placement of the wake (or its alignment with the local velocity) is not discussed in the theory manual. However, the user manual (Sidwell *et al.*, 1990) mentions that PANAIR does not calculate the shape of a wake unless indirectly as part of a design problem. The user must provide the program with an approximate shape for the wake in the form of panelled surfaces, and the quality of the solution depends on the quality of the wake shape approximation.

The above evaluation means that although rotating bodies such as propellers could be simulated, the wake shape shed by the propeller is part of the input that must be provided by the user: the program cannot align the wake with the local velocity. Adapting the PANAIR code to implement this feature may provide a challenge given the size and dialect of the PANAIR source code.

2.3.2 Apame

Aircraft Panel Method (APAME)¹⁶ is a fairly standard low-order panel method for the analysis of lifting flow around three-dimensional bodies in subsonic flow conditions. It is a redesign (in Fortran) of the software package (written in Matlab) developed by the author for his thesis (Filkovic, 2008). The software is released as open source (GNU General Public License (GPL)) and contains a solver module and a graphical user interface. The solver supports unstructured meshes consisting of a mix of triangular and quadrilateral panels. It can compute lift and induced drag over bodies in attached flow conditions. As-is, the software cannot handle rotating bodies such as propellers. In addition, the wake model is rigid, i.e., it is not aligned with the local flow direction. From running some of the examples, the solvers seems to work fine, generating lift when a body has an angle of attack. However, validation of code is required to verify that the potential solver has been implemented correctly.

Evaluation of the code

This short evaluation will focus on the issues in the current code and the efforts required to adapt it for computing the flow around marine propellers.

The code does not contain functionality to make the object under investigation rotate around an axis which is required for propellers simulations. The lack of this functionality is however not envisioned as a big issue, implementing this should be fairly straightforward.

More work will be required to implement functionality for aligning the wake with the local flow. In addition, it appears to miss functionality to construct a reasonable prescribed wake as well: all wake geometry panels are defined directly in the input files. Implementing this is more work, but fortunately the author of this report has some experience in this field (Van Hoydonck, 2013).

The source code is implemented in single precision: the keyword `double precision` is absent from the source code, and the parametric real facility of modern Fortran is not used. The source code makes calls to (single precision) external functions from LAPACK for solving the linear systems that are typical for potential flow solvers, which is also an indication of the single precision nature of the program.

The `GOTO` statement (which is considered harmful by some (Dijkstra, 1968)) appears 105 times in the source code, especially in the module where input files are parsed. The readability of the code can be improved because old-style logical evaluations are used (such as `.ge.` and `.le.`) instead of the easier readable and more mathematical forms `>=` and `<=`. Mathematical constants (such as π) are not defined in a central location in the source code and then *used* wherever necessary. Instead, the constants are defined at the location where they are needed in the code.

¹⁶See <http://www.3dpanelmethod.com>.

Furthermore, some derived constants, such as $1/(4\pi)$ are defined without referring to the definition of π itself. This makes it difficult and error-prone to alter the source code if the value of π should be changed.

No build system is in place, only the source code is provided along with executables. It should be easy to add configuration files for the CMake build system.

3 Summary and discussion

This report gives an overview of past and present research in the area of panel methods applied to the problem of simulating marine propellers and the vortical wake they produce. The cited literature is far from complete and only the very basics of potential panel methods has been given, but it should be enough as a starting point for future research. Apart from this literature review, an overview is given of existing software: commercial, open-source and research codes for the simulation of potential flow around marine propellers are discussed.

Topics discussed include models for the wake and how to align them with the local flow, grid generation on the blades of the propeller and its effect on the overall solution convergence and accuracy, methods to estimate viscous effects in panel methods, coupling of potential panel codes with CFD software for increased efficiency of CFD computations and the recent research of IgBEM, where IgA is combined with the BEM in a FEA. With a good grasp of the underlying physics (related to the wake and where and how it leaves the propeller blades) and high-quality CAD models of the propellers, IgBEM promises to give significantly more accurate results with less work in the future.

Given the high annual costs of commercial panel codes (yearly, VSAERO and USAERO combined cost €21 000, proCAL is even more expensive at €25 000), it seems more beneficial to either start from scratch or use an existing (open-source) panel code such as APAME and adapt it to the needs of nautical research at FHR. Most panel codes these days that are developed specifically for marine applications, solve for the velocity potential using a distribution of constant sources and doublets on a discretized version of the propeller geometry and wake. Some of the biggest disadvantages of these codes are summarised here and are discussed in more detail afterwards:

- the discretisation of the propeller geometry and wake is completely structured and uses quadrilateral panels;
- the wake consists of a lattice of doublets;

A structured discretisation is easy to implement: neighbours of a panel can be found by index based on the number of panels in each row and column, but a structured discretisation is more difficult to adapt to the flow efficiently. Also, a structured discretisation may result in extraordinary points (such as the tip region). Implementing a data structure for an unstructured panel discretization is more involved than for a structured discretisation, but once implemented, provides the advantage that local refinements in the discretisation are possible, keeping the number of panels to a minimum. This is important because the underlying problem of computing the influence coefficients at every panel scales with the square of the number of panels.

The use of (flat) quadrilateral panels can be seen as an advantage, because fewer panels are required for a quadrilateral discretisation with the same edge length as a triangular discretisation. However, for implementations where quadrilateral panels must be flat, gaps may be present between panels in areas with high curvature. Triangular panels do not have this issue. In principle, an edge-based vorticity solver can also use quadrilateral panels without problems because the vorticity is only defined on the edges of the faces. This means that for quadrilateral panels that are not flat, there will be no gaps between adjacent panels on a (doubly) curved surface.

Using a lattice of doublets for the wake discretisation means that in both the spanwise and streamwise directions, flow is assumed to be ordered. This means for example that the interaction between the propeller wake and a rudder is difficult to take into account in a first principles way: the interaction of the propeller wake lattice with the rudder means that the lattice might have to split to pass the rudder on either side, or that certain doublet panels of the wake would stretch or distort so much that they have to be subdivided. In summary, the ordered flow assumption might be too strict for applications other than open-water tests. The

number of panels in the spanwise direction on a blade and on its associated wake are generally kept the same, which means that the number of panels can quickly get very large. By using a buffer layer between the blade trailing edge and the actual (filament) wake, it is possible to use a lower number of spanwise wake elements than the number of spanwise blade elements. However, it seems this is not used in marine research codes. A vortex filament wake model only assumes ordered flow in the streamwise direction which makes interaction with (lifting) surfaces easier to implement. This type of wake¹⁷ is not directly compatible with standard panel methods that solve for the velocity potential, but it is compatible with surface vorticity panel methods. This type of panel method is also compatible with viscous vortex particle methods for cases where the streamwise ordering of the wake as implied by vortex filament wake models is too strict as well.

Marine CFD computations where the turning propeller is taken into account directly generally require a lot of computational resources. Replacing the turning propeller with a potential panel simulation can reduce the computational burden significantly by relaxing the time step size requirements and reducing the grid size in the region of the propeller. This way, it will be possible to execute this type of CFD computations in a reasonable amount of time.

To get there, a choice has to be made between the development (or adaptation) of a low-order *run-of-the-mill* panel code with source and doublet singularities, a low-order vorticity-based panel method in combination with a vortex filament wake method and a method based on IgBEM¹⁸. Of these three, the last option is judged to require most effort to implement a working version. The reward would be high, because the method promises high quality results with excellent convergence characteristics. One of the requirements for this is that the surface geometry description must be of high quality as well, which sadly is not always the case in the nautical sciences. The development effort for the other two options are expected to be lower, with a slight advantage for a vorticity-based method due to the use of only one singularity as opposed to the use of two singularities for potential-based panel methods. Pure vorticity-based methods are relatively new, which means that there are more opportunities for innovation as compared to potential-based panel methods that have been around for more than 50 years. A vorticity-based method would also set FHR apart from other maritime research institutes that have developed in-house panel codes for the simulation of propellers. Although open for discussion, the author has a preference for a vorticity-based solver with a vortex filament wake model.

Adapting and extending an existing potential code such as APAME all by oneself is a task that should not be underestimated, especially given that this is not the main research task of the author. For this reason, a number of subtasks can be defined that could be executed by students as master thesis assignments once a basic working version has been developed:

- implementation of a free wake module (either a vortex filament-based or doublet lattice based) with efficient and robust wake relaxation and time stepping algorithms;
- coupling of the panel method with CFD solvers such as OpenFOAM and FINE/Marine;
- implementation of a method to adaptively refine the grid based on the distribution of field variables;
- comparison of the performance of the direct (velocity-based) and indirect (potential-based) panel method;
- implementation of (or coupling with) a viscous vortex particle method including acceleration methods such as the FMM to reduce the computational burden;
- implementation of a method to estimate viscous effects in the boundary layer around the propeller geometry;
- extending the applicability of the method to situations where wake-piercing objects (such as rudders) are present.

¹⁷Possibly implemented using curved vortex filaments laid out along contours of constant vorticity in the wake.

¹⁸Note that the option of using a commercial code was already discarded.

References

Abdel-Maksoud, M. (2015). Private communication

Ahuja, V. (2013). Aerodynamic Loads over Arbitrary Bodies by Method of Integrated Circulation. (PhD Thesis). Auburn University: Auburn, Alabama

Anon. (2020). Theory Guide FINE/Marine 8.2

Ashby, D.; Dudley, M.; Iguchi, S. (1988). Development and Validation of an Advanced Low Order Panel Method. *report*, NASA-TM-101024. NASA Ames Research Center: Moffett Field, California

Ashby, D.; Dudley, M.; Iguchi, S.; Browne, L.; Katz, J. (1991). Potential flow theory and operation guide for the panel code PMARC. *report*, NASA-TM-102851. NASA Ames Research Center: Moffett Field, California

Baltazar, J. (2008). On the Modelling of the Potential Flow About Wings and Marine Propellers Using a Boundary Element Method. (PhD Thesis). University of Lisbon: Lisbon, Portugal

Baltazar, J.; Falcao de Campos, J. (2009). On the Modelling of the Flow in Ducted Propellers With a Panel Method. *in: Proceedings of the First International Symposium on Marine Propulsors S:P09*. Trondheim, Norway. p. 9

Baltazar, J.; Falção de Campos, J. (2006). Propeller Unsteady Potential Panel Method. *Report*, IST-MARETEC - TR - J583 - 14. University of Lisbon: Lisbon, Portugal

Baltazar, J.; J.A.C., F.; Bosschers, J. (2005). A study on the modeling of marine propeller tip flows using BEM. *in: Proceedings of the Congreso de Métodos Numéricos en Ingeniería*. Granada, 4th–7th July 2005. p. 17

Baltazar, J.; Rijpkema, D.; J.A.C., F.; Bosschers, J. (2018). Prediction of the Open-Water Performance of Ducted Propellers with a Panel Method. *Journal of Marine Science and Engineering* 6 (27) (1): 17. DOI: 10 . 3390 / jmse6010027

Baltazar, J. (2005). A Higher order potential based BEM for three-dimensional flows. *report*, POSI-SFRH/BD/14334/2003. University of Lisbon: Lisbon, Portugal

Beer, G.; Marussig, B.; Duenser, C. (2020). The Isogeometric Boundary Element Method. Vol. 90. Lecture Notes in Applied and Computational Mechanics. Springer International Publishing. ISBN: 978-3-030-23338-9

Beer, G.; Marussig, B.; Zechner, J. (2015). A simple approach to the numerical simulation with trimmed CAD surfaces. *Computer Methods in Applied Mechanics and Engineering* 285. DOI: 10 . 1016 / j . cma . 2014 . 12 . 010

Bertin, J.; Cummings, R. (Eds.). (2009). Aerodynamics for Engineers. 5th ed. Pearson Education: London, UK. ISBN: 978-0-132-272681

Bertram, V. (2012). Practical Ship Hydrodynamics. 2nd ed. Butterworth-Heinemann: Oxford, UK. ISBN: 978-0-08-097150-6

Bliss, D.; Teske, M.; Quackenbush, T. (1987). A New Methodology for Free Wake Analysis Using Curved Vortex Filaments. *report*, CR-3958. NASA Ames Research Center: Moffett Field, CA

Bosschers, J. (2007). PROCAL beats the panel! online

Chen, L.; Zhao, W.; Liu, C.; Chen, H.; Marburg, S. (2019). Isogeometric Fast Multipole Boundary Element Method Based on Burton-Miller Formulation for 3D Acoustic Problems. *Archives of Acoustics* 44 (3). DOI: 1024425/aoa.2019.129263

Chopra, I.; Celi, R.; Leishman, J.; Vizzini, A.; Baeder, J.; Werely, N. (1997). Center for Rotorcraft Education and Research: Progress Report on Army Research Office Contract No. DAAL03-88-C-0002 and DAAHO4-94-G-0074 "Rotorcraft Center of Excellence" Final Report. *Report*, DAAHO4-94-G-0074. University of Maryland, Department of Aerospace Engineering: College Park, Maryland

Chouliaras, S.; Kaklas, P.; Ginnis, A.-A.; Kostas, K.; Politis, C. (2017). An IGA-BEM Method for the Open-Water Marine Propeller Flow Problem. *in: Proceedings of the 4th International Conference on Isogeometric Analysis*. Pavia, Italy, 11th–13th September 2017. p. 12

Cottrell, J. A.; Hughes, T.; Bazilevs, Y. (2009). Isogeometric analysis: toward integration of CAD and FEA. 1st ed. John Wiley & Sons Ltd: Chichester, UK. ISBN: 978-0-470-74873-2

de Koning Gans, H. (1994). Numerical Time Dependent Sheet Cavitation Simulations using a Higher Order Panel Method. (PhD Thesis). Delft University of Technology: Delft, the Netherlands

Dijkstra, E. W. (1968). Go To Statement Considered Harmful. *Communications of the ACM* 11 (3)

Dölz, J.; Harbrecht, H.; Kurz, S.; S., S.; Wolf, F. (2018). A Fast Isogeometric BEM for the Three Dimensional Laplace- and Helmholtz Problems. *Computer Methods in Applied Mechanics and Engineering* 330. DOI: 10.1016/j.cma.2017.10.020

Drela, M. (1989). XFOIL: An Analysis and Design System for Low Reynolds Number Airfoils. *in: Mueller, T. (Ed.). Low Reynolds Number Aerodynamics*. Vol. 54. Lecture Notes in Engineering. Springer-Verlag. ISBN: 978-3-540-51884-6. p. 14. DOI: 10.1007/978-3-642-84010-4

Erickson, L. (1990). Panel Methods—An Introduction. *report*, TP-2995. NASA Ames Research Center: Moffett Field, California

Filkovic, D. (2008). Graduate Work. (MSc Thesis). University of Zagreb - Faculty of Mechanical Engineering and Naval Architecture: Zagreb, Croatia

Gaschler, M. (2017). Numerical Modelling and Simulation of Cavitating Marine Propeller Flows. (PhD Thesis). Hamburg University of Technology: Hamburg, Germany

Gillebaart, E. (2018). Towards Geometrically Consistent Aerostructural Optimisation of Composite Aircraft Wings. (PhD Thesis). Delft University of Technology: Delft, the Netherlands. ISBN: 978-94-028-1081-3

Gillebaart, E.; De Breuker, R. (2016). Low-Fidelity 2D Isogeometric Aeroelastic Analysis and Optimization Method with Application to a Morphing Airfoil. *Computer Methods in Applied Mechanics and Engineering* 305. DOI: 10.1016/j.cma.2016.03.014

Hally, D. (2017). A RANS-BEM coupling procedure for calculating the effective wakes of ships and submarines. *in: Bensow, R. (Ed.). Proceedings of the 5th International Symposium on Marine Propulsion SMP'17*. Espoo, Finland. p. 9

- Hally, D.** (2015). Propeller Analysis Using RANS/BEM Coupling Accounting for Blade Blockage. *in: Proceedings of the 4th International Symposium on Marine Propulsors SMP'15*. Austin, Texas, USA. p. 9
- Hally, D.** (2012). Using RANS to Calculate the Flow Past Marine Propellers. *in: Proceedings of the Annual Conference of the CFD Society of Canada*. Canmore, Canada. p. 7
- Heredia, J. M.** (2016). Towards the application of the isogeometric boundary element analysis to fluid mechanics: non-linear gravity waves and dynamics of deformable capsules in shear flows. (PhD Thesis). Universitat Rovira i Virgili: Tarragona, Spain
- Hess, J.; Smith, A. M. O.** (1964). Calculation of nonlifting potential flow about arbitrary three-dimensional bodies. *Journal of Ship Research* 8: 22–44
- Hidalgo, D.** (2014). Assessment of a State-Space Free Wake Model for rotorcraft flight mechanics applications. (MSc Thesis). Delft University of Technology: Delft, the Netherlands
- Hoeijmakers, H.** (1989). Computational Aerodynamics of Ordered Vortex Flows. (PhD Thesis). Delft University of Technology: Delft, Netherlands
- Hoshino, T.** (1989). Hydrodynamic Analysis of Propellers in Steady Flow Using a Surface Panel Method. *Journal of The Society of Naval Architects of Japan* 1989 (165). DOI: 10.2534/jjasnaoe1968.1989.55
- Hughes, T.; Cottrell, J.; Bazilevs, Y.** (2005). Isogeometric analysis: CAD, finite elements, NURBS, exact geometry, and mesh refinement. *Computer Methods in Applied Mechanics and Engineering* 194 (39-41). DOI: 10.1016/j.cma.2004.10.008/sym11020196
- Hundemer, J.** (2013). Development of a procedure for the calculation of the unsteady potential theory propeller flow. (PhD Thesis). Hamburg University of Technology: Hamburg, Germany
- Jennings, s.** (2010). Creation, Verification and Validation of a Panel Code for the Analysis of Ship Propellers in a Steady, Uniform Wake. (MSc Thesis). University of New Orleans: New Orleans
- Kaklis, P.; Politis, C.; Belibassakis, K.; Ginnis, A.; Kostas, K.; Gerostathis, T.** (2017). Encyclopedia of Computational Mechanics. *in: Stein, E.; de Borst, R. et al. (Eds.). 2nd ed. Vol. 5. John Wiley & Sons, Ltd. ISBN: 978-1-119-00379-3. Chap. Chapter XX Boundary-Element Methods and Wave Loading on Ships. DOI: 10.1002/9781119176817.ecm2115*
- Kerwin, J.; Kinnas, S.; Lee, J.-T.; Shih, W.-Z.** (1987). A Surface Panel Method for the Hydrodynamic Analysis of Ducted Propellers. *The society of naval architects and marine engineers*: 22
- Kostas, K.; Ginnis, A.; Politis, C.; Kaklis, P.** (2016). Shape-optimization of 2D hydrofoils using an Isogeometric BEM solver. *Computer-Aided Design* 82. DOI: 10.1016/j.cad.2016.07.002
- Kostas, K.; Ginnis, A.; Politis, C.; Kaklis, P.** (2015). Ship-hull shape optimization with a T-spline based BEM-isogeometric solver. *Computer Methods in Applied Mechanics and Engineering* 284. DOI: 10.1016/j.cma.2014.10.030
- Lee, J.-T.** (1987). A Potential Based Panel Method for the Analysis of Marine Propellers in Steady Flow. *Report*, 87-13. Massachusetts Institute of Technology: Cambridge, Massachusetts

- Magnus, A.; Epton, M.** (1981). PAN AIR – A computer program for predicting subsonic or supersonic linear potential flows about arbitrary configurations using a higher order panel method. Volume I: Theory document (Version 3.0). *report*, NASA-CR-3251. NASA Ames Research Center: Moffett Field, California
- Maljaars, P.; Kaminski, M.; den Besten, H.** (2018). Boundary Element Modelling Aspects for the Hydro-Elastic Analysis of Flexible Marine Propellers. *Journal of Marine Science and Engineering* 6 (67): 32. DOI: 10.3390/jmse6020067
- Martin, E.** (2015). Assessment of Panel and Vortex Particle Methods for the Modelling of Stationary Propeller Wake Wash. (MSc Thesis). Memorial University of Newfoundland: St. John's, Newfoundland
- Marussig, B.; Zechner, J.; Beer, G.; Fries, T.-P.** (2015). Fast Isogeometric Boundary Element Method based on Independent Field Approximation. *Computer Methods in Applied Mechanics and Engineering* 284. DOI: 10.1016/j.cma.2014.09.035
- Maskew, B.** (1988). Program VSAERO - A Computer Program for Calculating the Non-linear Aerodynamic Characteristics of Arbitrary Configurations - USER'S MANUAL. *report*, NASA-CR-166476. NASA Ames Research Center: Moffett Field, California
- Maskew, B.** (1987). Program VSAERO Theory Document - A Computer Program for Calculating the Non-linear Aerodynamic Characteristics of Arbitrary Configurations. *report*, NASA-CR-4023. NASA Ames Research Center: Moffett Field, California
- Moran, J.** (1984). An introduction to theoretical and computational aerodynamics. 1st ed. John Wiley & Sons, Inc.: New York. ISBN: 0-471-87491-4
- Morino, L.** (1974). A General Theory of Unsteady Compressible Potential Aerodynamics. *report*, CR-2464. NASA: Washington, D.C.
- Moulijn, J.; Bosschers, J.; Tornros, S.; Schreiber, C.; Grassi, D.; Goedbloed, I.; Bijlard, M.** (2019). Validation studies of a boundary element method for ducted propellers. *in: Proceedings of the Sixth International Symposium on Marine Propulsors*. Rome, Italy. Institute of Marine Engineering (CNR-INM). p. 11
- Nathman, J.** (2020). Private communication
- Newman, J.** (1986). Distribution of sources and normal dipoles over a quadrilateral panel. *Journal of Engineering Mathematics* 20 (2). DOI: 10.1007/BF00042771
- Pease Moore, J.** (2013). An Arbitrarily High-Order, Unstructured, Free-Wake Panel Solver. (MSc Thesis). Massachusetts Institute of Technology: Cambridge, Massachusetts
- Peerlings, B.** (2018). A review of aerodynamic flow models, solution methods and solvers – and their applicability to aircraft conceptual design. *Literature study report*. Delft University of Technology: Delft, Netherlands
- Politis, C.; Paggiannopoulos, A.; Belibassakis, K.; Kaklis, P.; Kostas, K.; Ginnis, A.; Gerostathis, T.** (2014). An isogeometric BEM for exterior potential-flow problems around lifting bodies. *in: Onate, E.; Oliver, J. et al. (Eds.). Proceedings of the 11th World Congress on Computational Mechanics (WCCM XI), 5th European Conference on Computational Mechanics (ECCM V), 5th European Conference on Computational Fluid Dynamics (ECFD VI)*. p. 12
- Pyo, S.** (1995). Numerical Modeling of Propeller Tip Flows with Wake Sheet Roll-up in Three Dimensions. (PhD Thesis). Massachusetts Institute of Technology: Massachusetts, MA

Quackenbush, T.; Wachspress, D.; Boschitch, A. (1991). Computation of Rotor Aerodynamic Loads with a Constant Vorticity Contour Free Wake Model. *in: Proceedings of the AIAA 9th Applied Aerodynamics Meeting*. Baltimore, MD, 23rd–25th September 1991

Rijpkema, D.; Starke, B.; Bosschers, J. (2013). Numerical simulation of propeller-hull interaction and determination of the effective wake field using a hybrid RANS-BEM approach. *in: Proceedings of the 3rd International Symposium on Marine Propulsors SMP'13*. Launceston, Tasmania, Australia. p. 9

Sidwell, K.; Baruah, P.; Bussoletti, J.; Medan, R.; Conner, R.; Purdon, D. (1990). PAN AIR – A computer program for predicting subsonic or supersonic linear potential flows about arbitrary configurations using a higher order panel method. Volume II: User's Manual (Version 3.0). *report*, NASA-CR-3252. NASA Ames Research Center: Moffett Field, California

Su, K.; Bliss, D. (2019). A Novel Hybrid Free-Wake Model for Wind Turbine Performance and Wake Evolution. *Renewable Energy* 131. DOI: 10.1016/j.renene.2018.07.108

Takinaci, A.; Altar, M.; Korkut, E. (2003). A practical surface panel method to predict velocity distribution around a three-dimensional hydrofoil including boundary layer effects. *Ocean Engineering* 30 (2). DOI: 10.1016/S0029-8018(02)00015-X

Tarafder, M.; Saha, G.; Mehedi, S. (2010). Analysis of potential flow around 3-dimensional hydrofoils by combined source and dipole based panel method. *Journal of Marine Science and Technology* 18 (3)

Toxopeus, S. (2015). Private communication

Van Hoydonck, W. R. M. (2013). Computation of Rotorcraft Wake Geometry using NURBS. (PhD Thesis). Delft University of Technology: Delft, the Netherlands. ISBN: 9789461917447. DOI: <https://doi.org/10.4233/uuid:1d00c8c8-6a69-4fd9-a631-0aedfa063ac3>

Vaz, G. (2005). Modelling of Sheet Cavitation on Hydrofoils and Marine Propellers using Boundary Element Methods. (PhD Thesis). University of Lisbon: Lisbon, Portugal

Vaz, G.; Eca, L.; Falcao de Campos, J. (2000). Boundary Elements XXII. *in: Brebbia, C.; Power, H. (Eds.). Advances in Boundary Elements*. WITpress: Southampton, UK. ISBN: 978-1853128240. Chap. A comparison of velocity and potential based boundary element methods for the analysis of steady 2D flow around foils

Wang, X.; Chouliaras, S.; Kaklis, P.; Ginnis, A.-I.; Politis, C. (2017). Wave-Resistance Computation via CFD and IGA-BEM Solvers: A Comparative Study. *in: Proceedings of the 27th International Ocean and Polar Engineering Conference*. San Fransisco, CA. International Society of Offshore and Polar Engineering. p. 7

Wendt, J. (Ed.). (2009). Computational Fluid Dynamics - An Introduction. 3rd ed. Springer-Verlag: Berlin Heidelberg. ISBN: 978-3-540-85055-7

Willemsen, J. (2013). Improving Potential Flow Predictions for Ducted Propellers. (MSc Thesis). University of Twente: Twente, the Netherlands

Willis, D. (2006). An Unsteady, Accelerated, High Order Panel Method with Vortex Particle Wakes. (PhD Thesis). Massachusetts Institute of Technology: Cambridge, Massachusetts

Willis, D.; Peraire, J.; White, J. (2006). A combined pFFT-multipole tree code, unsteady panel method with vortex particle wakes. *International Journal for Numerical Methods in Fluids* 53 (8). DOI: 10.1002/flid.1240

- Winckelmans, G.** (1989). Topics in Vortex Methods for the Computation of Three- and Two-Dimensional Incompressible Unsteady Flows. (PhD Thesis). California Institute of Technology: Pasadena, CA
- Winckelmans, G.; Cogle, R.; Dufresne, L.; Capart, R.** (2005). Vortex methods and their application to trailing wake vortex simulations. *Comptes Rendus Physique* 6 (4-5)
- Winckelmans, G.; Leonard, A.** (1993). Contributions to Vortex Particle Methods and their application to trailing wake vortex simulations. *Journal of Computational Physics* 192 (2)
- Wöckner, K.; Greve, M.; Scharf, M.; Rung, T.; Abdel-Maksoud, M.** (2011). Unsteady Viscous/Inviscid Coupling Approaches for Propeller Flow Simulations. *in: Proceedings of the Second International Symposium on Marine Propulsors smp'11*. Hamburg, Germany. p. 8
- Yari, E.; Ghassemi, H.** (2016). Hydrodynamic analysis of the surface-piercing propeller in unsteady open water conditions using boundary element method. *International Journal of Naval Architecture and Ocean Engineering* 8 (1). DOI: 10.1016/j.ijnaoe.2015.09.002
- Yiran, S.; Kim, S.; Kinnas, S.** (2018). Prediction of Propeller-Induced Hull Pressure Fluctuations via a Potential-Based Method: Study of the Effects of Different Wake Alignment Methods and of the Rudder. *Journal of Marine Science and Engineering* 6 (52) (2): 14. DOI: 10.3390/jmse6020052

A1 Descriptions of Commercial Software

Due to a lack of description of the different software packages on the website of the developer (AM Inc.), the descriptions below were taken from the website of aero attack systems¹⁹.

A1.1 VSAERO

VSAERO couples integral methods of potential and boundary layer flows for low runtimes – full configurations in as little as five minutes. Compatible with structured, unstructured and hybrid surface meshes, VSAERO can also be expanded to models with 50,000 or more surface panels for extremely complex models. Besides standard surface calculations (Fig. 6), flowfield properties are computed for off-body velocity surveys and on/off-body streamlines.

Figure 6 – Surface pressure visualisations of computations executed with VSAERO. Retrieved from <https://www.aeroattack.com/vsaero>, © Analytical Methods.



A1.2 USAERO

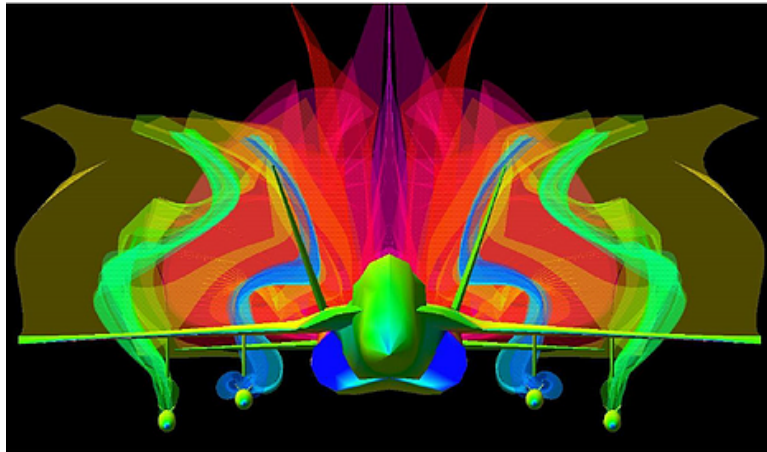
USAERO enables analysis of the aerodynamic effect of relative motion aerodynamic simulations. For example it can analyze multi-store weapons release from a military aircraft.

USAERO is one of the best tools for solving transient calculations. A unique coupling of potential flow and boundary layer methods with a time-stepping procedure for arbitrary motions assures timely and cost effective assessments of unsteady surface pressures and loads. As integral potential flow and boundary layer methods are the basis of this CFD software, the aerodynamics solution is only required on the boundary surfaces for each time step and requires only surface meshing. Furthermore, while components or bodies may involve relative motions no regridding is required with this solution scheme.

USAERO supports practical engineering solutions for such problems as maneuvering aircraft, formation flying, aircraft stores carriage (see Fig. 7 for an example of this), gust response, rotor/body interactions, and train passing and tunnel entry. Special application modules, FPI and FSP, couple with USAERO to provide, respectively, flight-path integration calculations with six degrees of freedom and nonlinear free-surface simulations.

¹⁹<http://www.aeroattack.com> – Design and Development of Military Aircraft and Access Control Systems.

Figure 7 – Unsteady simulation of the release of payload from the wings of a jet fighter. Retrieved from <http://www.aeroattack.com/usaero>, © Analytical Methods.



A1.3 OMNI3D

OMNI3D is an interactive, 3D data visualization tool designed for efficient post-processing of the CFD simulation results of AMI software.

CFD Data Input

- Accepts data in the VSAERO plot file format, a fully general CFD output format
- Reads both formatted (ASCII) and unformatted (binary) data files

2D Displays

- Plots any solution parameter on arbitrary 2D cuts
- Plots solution parameters vs. time
- Displays streamlines and boundary layer parameters
- Computes 2D cuts relative to specific reference frames
- Calculates frame loads in a particular axis system

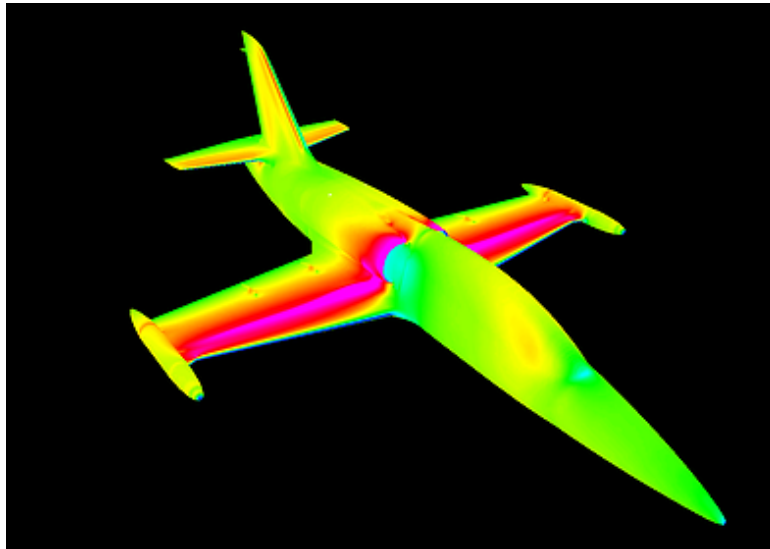
3D Displays

- Displays body and wake geometry using high quality shading or wireframe
- Uses sophisticated color mapping to show computed fluid flow and measured experimental data on the geometry
- Displays wakes as wireframe, ribbons, cross-sections, or transparent surface
- Uses color to show wake strength along wake lines
- Displays contours of surface pressure, flow velocity, Mach number, or any other property
- Provides real-time, 3D view manipulation for interactive data investigation
- Animation capability for displaying time dependent results with changing surface geometry and several different reference frames
- Displays on-body and off-body streamlines, using color to show streamline and boundary layer properties
- Mouse-based techniques for probing geometry and solution data

Comparison with Experimental Data

- Accepts experimental data from a separate input file
- Displays experimental data defined at specific locations on the geometry
- Displays experimental data on 2D cuts
- Provides specially-designed visual tools for comparing experimental data with CFD results

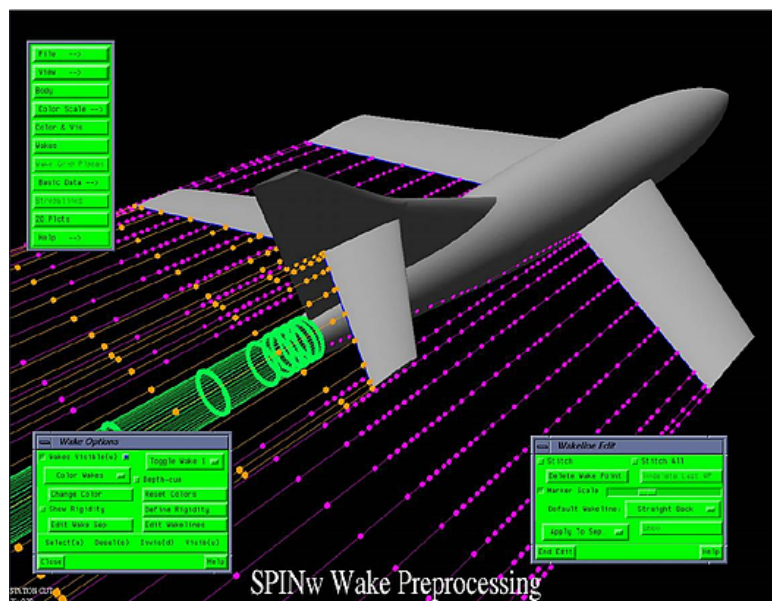
Figure 8 – Visualisation of the pressure on the fuselage of a jet trainer with OMNI3D. Retrieved from <https://www.aeroattack.com/omni3d>, © Analytical Methods.



A1.4 SPIN(w)

SPIN(w) is a wake and data preprocessor for VSAERO. SPIN(w) allows users to generate the wakes graphically and provides an interface for VSAERO data input.

Figure 9 – Wake preprocessing using SPINw: configuration of wake geometry of a jet engine aircraft. Retrieved from <https://www.aeroattack.com/spinw>, © Analytical Methods.



DEPARTMENT **MOBILITY & PUBLIC WORKS**
Flanders hydraulics Research

Berchemlei 115, 2140 Antwerp

T +32 (0)3 224 60 35

F +32 (0)3 224 60 36

waterbouwkundiglabo@vlaanderen.be

www.flandershydraulicsresearch.be