Numerical Method for the Analysis of Cavitating Waterjet Propulsion Systems

Shu-Hao Chang  
Ocean Engineering Group, Department of Civil,  
Architectural and Environmental Engineering  
The University of Texas at Austin, Austin, TX  
78712, USA  
shuchang@utexas.edu

Spyros A. Kinnas  
Ocean Engineering Group, Department of Civil,  
Architectural and Environmental Engineering  
The University of Texas at Austin, Austin, TX  
78712, USA  
kinnas@mail.utexas.edu

ABSTRACT
A previously developed numerical panel method has been refined, and extended to predict the hydrodynamic performance of a rotor inside a water-jet pump. The governing equations and boundary conditions on the water-jet rotor are formulated and solved by distributing constant dipoles and sources on blades, hub and shroud surfaces, and constant dipoles on the shedding wake sheets behind the trailing edge of the rotor. The algorithm of finding the correct discrete cavity shape on every section of the blade is initiated with a guessed cavity plan-form, and the cavity extent and thickness are determined iteratively until both the dynamic and kinematic boundary conditions are satisfied within a set criterion on the cavity surface. The present method is validated against the experimental data in the case of an actual water-jet pump (ONR AxWJ-2). The predicted numerical results such as thrust and torque coefficients as well as cavity patterns are compared with those from the measurements and observation.

INTRODUCTION
The use of water-jet propulsions has gradually increased on commercial and navy applications recently. Due to the absence of appendages under the waterline, the vessels with water-jet propulsion systems can reduce the resistance and are ideal for the shallow water maneuvering. Moreover, the possibility of cavity occurrence and the debris damages to the blade can be lowered inside water-jet pumps. However, cavitation is still an inevitable problem. The thrust breakdown caused by super cavitation is the foremost hydrodynamic issue accompanied by noise, vibration, and erosion when a water-jet is operating in off design circumstances.

At present, numerical methods for analyzing the cavitating performance and assisting the design of water-jet propulsors are still limited. A more detailed description and literature review of cavitating water-jets can be found in International Towing Tank Conference [ITTC, 2008]. RANS solvers have become more and more popular and played important roles for simulating flows inside water-jets. Chun et al. [2002] and Brewton et al. [2006] used RANS solver on the rotor and stator interaction by considering the former in an unsteady sense and the latter in a circumferentially averaged sense. Lindau et al. [2009; 2011] also applied a RANS solver with homogeneous-multiphase modeling and turbulent simulation capabilities, and by using 3-D powering iteration methodology, they simulated water-jet flows over a wide range of flow coefficients and into the cavitation driven breakdown. An intermediate approach which combines potential flow solver and RANS tool was applied to the prediction and design of water-jet components. Taylor et al. [1998] and Kerwin et al. [2006] applied a vortex lattice method (VLM) coupled with either a RANS solver or an Euler equation solver to take effects of the hull and other appendages into account, and to analyze the global flow through the water-jet pump. Kinnas et al. [2007a; 2010] applied a panel method to predict performances of a water-jet. Sun [2008], Sun and Kinnas et al. [2006; 2008] and Kinnas et al. [2007b] used a viscous and inviscid interactive approach successfully by coupling a panel method with a boundary layer solver to simulate the viscous flow around single, ducted and water-jet propulsion systems, including boundary layer effects on the cavities over the blades.

In this paper, a numerical scheme based on a panel method by Fine and Kinnas [1993] and Kinnas and Fine [1993] is improved and refined fairly to predict cavitating performance of water-jets. The numerical results are validated against the experimental data from a series of measurements by Chesnakas et al. [2009] on ONR-AxW12 pump at Naval Surface Warfare Center at Carderock Division (NSWCCD).

FORMULATION
A panel method PROPCAV [PROPeller CAVitation, Kinnas and Fine (1992)] based on the potential flow theory is utilized to analyze 3-D unsteady flow around cavitating propulsors.

For a water-jet, assuming subject to a uniform inflow $U_{in}$ at the inlet of the pump and is defined in a ship fixed coordinate system $(X_s, Y_s, Z_s)$. An impeller rotates with a constant angular
velocity vector \(\mathbf{v}_i\). Thus, the inflow velocity can be defined as \(\mathbf{v}_\text{in} = \mathbf{U}_\text{in} - \omega \times \mathbf{X}\) in a rotating coordinate system or as \(\mathbf{v}_\text{in} = \mathbf{U}_\text{in} - \alpha \times \mathbf{X}\) in a ship fixed coordinate system where \(\mathbf{X}\) indicates a propeller fixed coordinate system. The flow is assumed to be inviscid, incompressible and irrotational, and thus the velocity field can be described as following:

\[
\mathbf{q}(x,y,z) = \mathbf{V}_\text{a}(x,y,z) + \nabla \phi(x,y,z)
\]

(1)

where \(\mathbf{q}\) is the total velocity. For the rotor only problem, the perturbation potential \(\phi(x,y,z)\) at any point located either on the wetted surfaces of rotor blades (\(S_R\)) or on the hub and casing surfaces (\(S_{HC}\)) or on the cavitating surfaces of the rotor (\(S_{RC}\)) must satisfy Green’s third identity as follows:

\[
2\pi \phi = \int_{S_R} \left[ \phi \frac{\partial G(p;q)}{\partial n_q} - G(p;q) \frac{\partial \phi}{\partial n_p} \right] ds + \int_{S_{HC}} \Delta \phi_{nq} \frac{\partial G(p;q)}{\partial n_q} ds + \int_{S_{RC}} \left[ \phi \frac{\partial G(p;q)}{\partial n_q} - G(p;q) \frac{\partial \phi}{\partial n_p} \right] ds
\]

(2)

where the \(p\) and \(q\) correspond to the variable points and the field points, respectively. \(G(p;q) = 1/R(p,q)\) is the Green function and \(R(p;q)\) is the distance between the field point \(p\) and the variable point \(q\). \(n_q\) indicates the normal direction pointing into the flow field. \(\Delta \phi_{nq}\) is the potential jump across the trailing wake sheets shedding from the rotor blade trailing edge.

**Figure 1**: A water-jet rotor only problem subject to a general inflow and coordinate systems.

In order to solve the rotor only problem, the following boundary conditions must be satisfied:

(i) The flow is tangent to the wetted rotor blades, hub and casing surfaces.

\[
\frac{\partial \phi}{\partial n} = -\mathbf{V}_\text{a}(x,y,z) \cdot \hat{n}
\]

(3)

(ii) The Morino’s [Morino and Kuo, 1974] steady Kutta condition is applied to ensure the fluid velocities are finite at the trailing edge of the blade. An iterative pressure Kutta (IPK) condition [Kinnas and Hsin, 1992] is required to force a zero pressure jump between the pressure and suction sides at the blade trailing edge.

(iii) The dynamic boundary condition on the blade cavity surface requires that the pressure on the cavity surface is constant and equal to the cavitating pressure, \(p_c\). By applying Bernoulli’s equation in the propeller fixed coordinate system in terms of the cavitation number, the total velocity \(\mathbf{q}\) on the cavity surface can be expressed as following:

\[
|\mathbf{q}| = \sqrt{\frac{2}{\gamma} \left( -2g\rho \frac{\partial \phi}{\partial t} - \frac{\partial^2 \phi}{\partial t^2} \right)}
\]

(4)

where \(r\) is the distance from the axis of the rotation. \(g\) and \(\gamma\) are the gravitational constant and the vertical distance from the horizontal plane through the axis of rotation. \(n\) and \(D\) are the rotating frequency and the propeller diameter, respectively. The cavitation number \(\sigma_c\) is defined as following:

\[
\sigma_c = \frac{p_c - p_i}{0.5 \rho \omega^2 D^2}
\]

(5)

where \(p_i\) is the pressure far upstream on the shaft axis and \(\rho\) is the fluid density.

(iv) The kinematic boundary condition on cavity is the requirement that the substantial derivative vanishes on the cavity surface.

\[
\frac{\partial q}{\partial t} + \mathbf{q} \cdot \nabla h = 0
\]

(6)

where \(\mathbf{n}\) is the coordinate normal to the blade surface and \(h\) is the cavity thickness normal to the blade.

(v) The cavity detachment location is determined iteratively to satisfy the smooth detachment conditions which described in Young [2002].

(vi) The cavity closure condition implies that the cavity needs to be closed at the end of the cavity. Initiating from a guessed cavity plan-form, the cavity shape may not be closed if the pressure coefficient on the cavity plan-form is not the same as a given cavitation number. In the present method, the Newton-Raphson iterative scheme is adopted to search correct cavity extents which satisfy the cavity closure condition at the given cavitation number.

The solutions, \(\phi\) on the wetted surfaces and \(\frac{\partial \phi}{\partial n}\) on cavity surface, of a boundary value problem for a cavitating water-jet rotor is obtained by solving eqn. (2) with the boundary conditions described above. Similar descriptions of boundary conditions can be referred to Lee and Kinnas [2006] or Stefano et al. [2009], and for more details of governing equations and boundary conditions of solving inviscid wetted and cavitating water-jet flows can be found in Sun [2008]. It should be noted that the water-jet inlet panels are removed and the perturbation potentials are assumed to be zero at the inlet to obtain a unique solution of an internal boundary value problem.

**RESULTS AND DISCUSSION**

The present method is applied to do a series of numerical calculations and is validated against the experimental data of an actual water-jet pump (ONR AxWJ-2). It should be noted that the effects of the stator on the rotor have been found to be small (Kinnas et al. [2010]) and have not been included in this work.
ONR Axial Flow Waterjet 2 Pump

ONR AxWJ-2 water-jet pump was designed by NSWCCD. The experimental data were obtained from the report of Chesnakas et al. [2009]. The geometry of the tested pump has an inlet diameter of 0.3048 m, and nozzle has an outlet diameter of 0.2134 m. The design pump has a 6-bladed rotor and an 8-bladed stator as shown in Fig. 2. Rotor tip gap size is 0.0508 cm which is 2.3% of the blade radius. For the numerical simulations by the present scheme, however, the tip gap is sealed. The design advance ratio \(J_s\) is 1.19 and the rotational frequency is 2000 rpm for the cavitating condition and 1400 rpm for the non-cavitating condition respectively.

\[ J_s = \frac{Q}{nD^3} \]  
\[ N = \frac{P_r - P_0}{\rho n D^2} \left( \frac{V_f}{n D} \right) = \frac{P_r - P_0}{\rho n D^2} + \frac{1}{2} \frac{V_f^2}{n D^2} = \frac{1}{2} C_v + \frac{1}{2} J_s \]

where \(P_r\) is the measured pressure at station 3 in the pump. We assume the station 3 (in Fig. 2) is far upstream, thus, the pressure \(P_0\) equals to \(P_r\) and velocity \(V_f\) equals to \(U_{in}\).

The comparisons of cavitation coverage on the rotor blade are as shown in Fig. 3 with those from observations. The prediction of the cavity patterns seems to agree well with those from experiments. At highest cavitation coefficient condition \((N^* = 4.044)\), the present method is able to predict small sheet cavities starting from the leading edge of the blade. For lower cavitation coefficients, the present method tends to predict slightly larger extent of cavity patterns than those from the experiments.

The convergence of cavitating circulation distributions of the rotor is shown in Fig. 4. The number of circumferential panels between two blades on the hub and shroud is kept the same as 20 elements, and changing the number of panels on the blade does not affect the blade circulations significantly. The circulation \((\Gamma)\) is defined as following:

\[ \Gamma = \Delta \phi / 2 \pi R \sqrt{V_f^2 + (0.7 \pi n D)^2} \]
The sensitivity of the cavity patterns with number of panels on the rotor is investigated at $J_s=0.988$ ($Q^*=0.706$) and $\pi_n=1.130$ ($N^*=1.046$) as shown in Fig. 5. Except for the slightly differences in the case of 60x20 on the blade, the present method predicts very similar results regardless of the number of panels on the rotor.

**Thrust Breakdown**

The experimental values and the numerical computations of the normalized thrust and torque are compared at two flow rate restriction conditions (by the flow control nozzle in Fig. 2): 100% nozzle ($Q_s^*=0.83$) and 90% nozzle ($Q_s^*=0.774$). Due to large amounts of cavitation, the flow rate through the pump is also reduced. Therefore, in the thrust breakdown plots shown here, all values of thrust and torque are normalized by the non-cavitating values of those quantities at that particular flow rate. The normalization for each point on the plot also changes as the flow rate changes. The $J_s$ and $\pi_n$ used in the calculations by the present method keep changing with the corresponding $Q$ and $N$. The numerical results are obtained by using 60x20 panels on the rotor and 20 elements in the circumferential direction between two blades. The comparison of pressure distributions between cavitating and fully-wetted solutions at $r/R=0.988$ for $Q^*=0.830$ and $N^*=0.993$ is shown in Fig. 6.

Since there is super cavitation on the back (suction) side of this section as shown in Fig. 7, the pressure coefficient remains the same as the corresponding cavity number ($\pi_n=0.724$). The experimental measurements and the numerical computations of the normalized thrust and torque at $Q^*=0.83$ are compared as shown in Fig. 8 and Fig. 9. The comparison of pressure distributions between cavitating and fully-wetted solutions at $r/R=0.988$ for $Q^*=0.774$ and $N^*=0.901$ is shown in Fig. 10. Due to the super cavitation on the back (suction) side of the section as shown in Fig. 11, the pressure coefficient remains the same as the corresponding cavity number ($\pi_n=0.749$). The comparison of the normalized thrust and torque at $Q^*=0.774$ are as shown in Fig. 12 and Fig. 13. Some discrepancies occurred when cavitation coefficients become lower, and the present method seems to under-predict the performance. Generally, the thrust breakdown phenomenon can be simulated by the present numerical scheme, and the trends of the prediction follows well with those from the experiments.
Figure 7: Cavity patterns on the rotor blade (at $Q^*=0.830$ and $N^*=0.993$).

Figure 8: Comparison of the predicted rotor normalized thrust (using non-cavitating thrust at $Q^*=0.83$) with experimental data for various flow coefficients ($N^*$) from Chesnakas et al. (2009).

Figure 9: Comparison of the predicted rotor normalized torque (using non-cavitating torque at $Q^*=0.83$) with experimental data for various flow coefficients ($N^*$) from Chesnakas et al. (2009).

Figure 10: Pressure distributions on the rotor at the section of $r/R=0.988$. Comparison between fully-wetted and cavitating solutions (at $Q^*=0.774$ and $N^*=0.901$).

Figure 11: Cavity patterns on the rotor blade (at $Q^*=0.774$ and $N^*=0.901$).

Figure 12: Comparison of the predicted rotor normalized thrust (using non-cavitating thrust at $Q^*=0.774$) with experimental data for various flow coefficients ($N^*$) from Chesnakas et al. (2009).
Figure 13: Comparison of the predicted rotor normalized torque (using non-cavitating torque at \( Q^* = 0.774 \)) with experimental data for various flow coefficients (\( N^* \)) from Chesnakas et al. (2009).

**CONCLUSIONS**

A refined numerical panel method has been extended to predict the hydrodynamic performance of a rotor inside a water-jet pump subject to a uniform inflow. Brief theorems and algorithms for solving the steady potential flow problem inside the water-jet have been presented in this paper. The application of the present method in the rotor only case of ONR AxWJ-2 pump has satisfactory correlations with the experimental measurements. The predicted cavity patterns have slightly larger extent than those from the observations. The predicted thrust and torque breakdown of the rotor due to lower values of cavitation coefficients agree well with those measured.

In the future, the authors plan to include the effects of stator and the effect of viscosity to the cavitation on the rotor and stator blades by extending the methods of Sun and Kinnas [2006, 2008].

**ACKNOWLEDGMENTS**

Support for this research was provided by the U.S. Office of Naval Research (Contract N00014-07-1-0616) and Phases VI of the “Consortium on Cavitation Performance of High Speed Propulsors” with the following current members: American Bureau of Shipping, Kawasaki Heavy Industry Ltd., Rolls-Royce Marine AB, Rolls-Royce Marine AS, SSPA AB, Andritz Hydro GmbH, Wärtsilä Netherlands B.V., Wärtsilä Norway AS, Wärtsilä Lips Defese S.A.S., and Wärtsilä CME Zhenjiang Propeller Co. Ltd.

**NOMENCLATURE**

\( C_p \): pressure coefficient, \( C_p = P - P_0 / 0.5 \rho u^2 D^2 \)

\( D \): propeller diameter

\( J_3 \): advance ratio, \( J_3 = \rho_0 / n D \)

\( n \): propeller rotation frequency (rev/s)

\( N^* \): cavitation coefficient, \( N^* = (\rho_3 - \rho_1) / \rho u^2 D^2 \)

\( P_0 \): pressure far upstream

\( P_3 \): total pressure at station 3

\( P_4 \): vapor pressure

\( Q \): volumetric flow rate

\( Q^* \): flow coefficient, \( Q^* = Q / \rho u D^3 \)

\( R \): propeller radius

\( u_R \): reference velocity, \( u_R = \sqrt{V_S^2 + (0.7 \pi n D)^2} \)

\( V_S \): ship speed

\( \sigma_n \): cavitation number, \( \sigma_n = (P_3 - P_0) / 0.5 \rho u^2 D^2 \)

\( \phi \): perturbation potential

**REFERENCES**


