

# Performance Assessment of the Waterjet Propulsion System through a Combined Analytical and Numerical Approach

Parviz Ghadimi\*, Roya Shademani, Mahdi Yousefi Fard

Department of Marine Technology, Amirkabir University of Technology, Tehran, Iran  
\*Corresponding author: pghadimi@aut.ac.ir

Received December 13, 2012; Revised April 21, 2013; Accepted April 24, 2013

**Abstract** The application of waterjets is rapidly growing and they are increasingly being chosen for propulsion in high-speed crafts. Waterjet as a propulsion system of a vessel is also favorable when it comes to maneuvering, appendage drag, draft and fuel consumption at high speeds. Furthermore, waterjet system has recently gained more credibility for its acceptable efficiency because of the advent of more efficient and large pumps. This type of propulsion system consists of many components working together harmoniously, thus establishing a complex system. A significant problem facing designers when predicting performance of the waterjet is the interaction between the hull and the waterjet. This paper describes the powering performance of a vessel equipped with a waterjet system. The interaction between the hull and waterjet is studied in order to predict the powering characteristics. The work starts with an introduction of the waterjet and a review of its current status in design and analysis. Subsequently, hydrodynamic properties of its components are computed and interactions among them are analyzed. Finally, numerical computation is performed for acquiring pressure distribution by a two dimensional computer code in the suction area of the waterjet inlet to predict the possible occurrence of cavitation for the inlet duct.

**Keywords:** waterjet, inlet flow, numerical analysis, cavitation, propulsion system

## 1. Introduction

In 16<sup>th</sup> century, Toogood and Hays for the first time proposed waterjet propulsion system, as reported by J.S. Carlton [1]. At that period of time, waterjet propulsions were used in high-speed pleasure craft and work boats. However, in recent years, this system has been considered for large high-speed crafts. Accordingly, many huge waterjet units have been used in wide range of ships such as passenger and naval crafts.

The waterjet propulsion is a complex system. On the contrary, the screw propellers are simpler, lighter and more efficient than waterjet system. However, the arrival of more efficient pumps, the necessity for timely delivering the critical commercial cargoes, and the required maneuverability for particular vessels have made the usage of waterjets more attractive.

It is normal to divide this type of propulsion system into a hull and a waterjet. It has been demonstrated that waterjet-hull interaction can affect the overall efficiency more than 20% [2]. Usually, waterjet system is broken down into subsystems and an explicit modular approach is applied to analyze them. In order to assess the interaction between the hull and the jet, a parametric method is used.

In 1980, an early contribution related to the jet-hull interaction is attributed to Etter et al. [3]. A complete review of the existing relations for waterjet-hull performance is presented by Allison [4]. Van Terwisga [2]

proposed a parametric propulsion prediction method for the waterjet driven craft. Several numerical methods were devoted to study the flow behavior in every part of waterjet system by many researchers whose works were submitted to the ITT Conference [5].

A complete design method that utilized the numerical scheme for the analysis of system parts has been used by CCDOTT<sup>1</sup> [6]. N.W.H. Bulten [7] used CFD methods to design and analyze the whole jet system. Also, a flush-type waterjet propulsion unit was designed with different inner diameter impellers by Moon et al. [8]. In a related work, analysis of a waterjet axial pump was performed numerically [9]. They investigated the performance of the axial-flow-type waterjet based on the variation of the impeller tip clearance.

The main distinction between the procedure implemented here and the previous methods is that the present method applies the empirical, analytical and numerical methods simultaneously to reach a conceptual design of the waterjet system. By numerical analysis, inlet design parameters that describe model geometry have been specified. Moreover, a simple 2-dimensional inlet model has been used here instead of a complex 3-D geometry. By this method, we have an acceptable propulsion system in a short time. Accordingly, a complete jet system design code has been developed that combines the empirical, analytical and numerical methods.

<sup>1</sup> Center for the Commercial Deployment of Transportation Technologies

By considering the ship geometry limitations and hydrodynamic properties, this software proposes suitable jet conceptual design parameters. Consequently, by using simple 2-D domain for inlet duct and computed pressure distributions for different inlet angles, an optimum inlet has been proposed which leads to minimum losses and maximum efficiency.

## 2. Basic Theory of System

Waterjet propulsion system has three main components: Inlet, Pump and Nozzle. Figure 1 shows typical waterjet arrangement while Figure 2 demonstrates the idealized profile of the jet system. Sea water enters the system with the velocity  $V_{in}$  and leaves it with a different velocity  $V_{jet}$ . The mass flow rate of the water through the waterjet is given by

$$\dot{m} = \rho A_{jet} V_{jet} \quad (1)$$

where  $A_{jet}$  and  $\rho$  are the area of nozzle outlet and density of water, respectively. The thrust produced by the system is equal to the rate of change of momentum:

$$T = \rho A_{jet} V_{jet} (V_{jet} - V_{in}) \quad (2)$$

Hence, the effective propulsion power delivered by the system is given by

$$P_E = TV_S = \dot{m} V_S (V_{jet} - V_{in}) \quad (3)$$

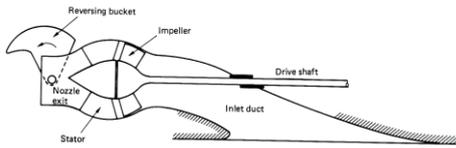


Figure 1. Typical waterjet general arrangement [1]

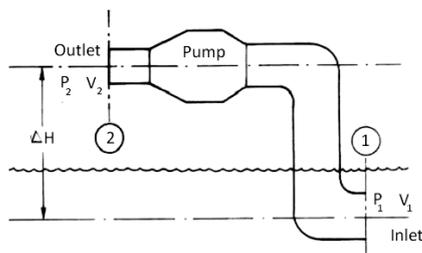


Figure 2. Idealized waterjet arrangement [1]

In Figure 2, the energy equation between points 1 and 2 can be written as

$$\frac{P_1}{\rho g} + \frac{V_1^2}{2g} + H_P = \frac{P_2}{\rho g} + \frac{V_2^2}{2g} + \Delta H + h_{loss} \quad (4)$$

where

$H_P$ : Pump head

$\Delta H$ : Vertical distance between inlet and outlet

$h_{loss}$ : Total head loss.

The jet efficiency is defined as

$$\eta_{jet} = \frac{P_E}{P_E + Losses} \quad (5)$$

This indicates that the jet efficiency is equal to the ratio of useful power to the total power absorbed by the system.

Because of the effect of hull and geometry of the inlet duct, the inlet velocity is different from ship velocity. Hence, the inlet velocity ratio is defined as follows:

$$IVR = \frac{V_{in}}{V_S} \quad (6)$$

The inlet has an optimal inlet velocity ratio at which jet has minimum losses at that point. This optimum point has been found by studying a range of velocity ratio and computing related system efficiency. Head losses in the duct inlet and outlet are functions of squared velocity [1]:

$$h_{Inlet} = \xi V_{in}^2 / 2g \quad (7)$$

$$h_{Outlet} = \psi V_{jet}^2 / 2g \quad (8)$$

where  $\xi$  and  $\psi$  are empirical coefficients. These parameters can be assumed as fixed values depending on the inlet and outlet geometries [1].

Power input to the system (received by the pump) is given by

$$P_{pump} = \dot{m} \left[ \frac{1}{2} (V_{jet}^2 - V_{in}^2) + g (h_j + h_{loss}) \right] \quad (9)$$

where  $h_j$  is the vertical distance between the outlet nozzle and the water line. By separating system losses and applying loss coefficients, we obtain

$$P_{pump} = \frac{\dot{m}}{2} [V_{jet}^2 (1 + \psi) - (1 - \xi)(1 - \omega)^2 V_S^2 + 2gh_j] \quad (10)$$

By using equation (10) and assuming the velocity ratio as  $\mu = V_S / V_{jet}$ , equation (5) can be written as

$$\eta_j = \frac{2\mu(1 - (1 - \omega)\mu)}{1 + \psi - (1 - \xi)(1 - \omega)^2 \mu^2 + \frac{2gh_j}{V_S^2} \mu^2} \quad (11)$$

Jet efficiency can be calculated from equation (11). Other design parameters have been found based on the velocity ratios. Figure 3 shows that jet outlet diameter increases at high velocity ratio, while the maximum value of jet efficiency is observed to occur at high velocity ratio. Thus, to achieve high efficiency, a large jet diameter should be selected. However, the aft geometry of ship makes limitations for jet diameter. Therefore, a specified jet diameter is selected and other design parameters are assessed based on this main restriction. On the other hand, the optimum velocity ratio is found by setting the derivative of equation (11) with respect to  $\mu$  equal to zero to obtain:

$$\mu_{opt} = \frac{1 \pm \sqrt{1 - \gamma}}{(1 - \omega)\gamma} \quad (12)$$

where  $\gamma$  is given by

$$\gamma = \frac{(1-\omega)^2(1-\xi) - \frac{2gh_j}{V_s^2}}{(1+\psi)(1-\omega)^2} \quad (13)$$

By using the pump and hull efficiencies, the total efficiency is defined by the following equation:

$$\eta_{OA} = \eta_{Pump} \cdot \eta_{Hull} \cdot \eta_{jet} \quad (14)$$

$$\eta_{Hull} = \frac{1-t}{1-\omega} \quad (15)$$

where  $\eta_{Hull}$ ,  $\omega$  and  $t$  are the hull efficiency, wake and thrust deduction factors, respectively. Wake factor can be estimated by the boundary layer theory which will be explained in the next section.

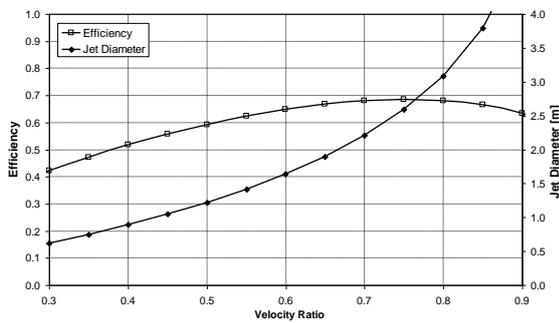


Figure 3. Efficiency and jet diameter as a function of velocity ratio

Figure 4. shows how the pump head is inversely proportional to the mass flow rate of the jet system. Therefore, for a specified velocity ratio, a pump should be selected as such that its head and mass flow rate correspond to those values obtained from the plots in Figure 4 at that designated velocity ratio. On the other hand, the engine and propulsion system spaces have dimensional limitations, since mixed flow pumps is a better option than other types like axial pumps.

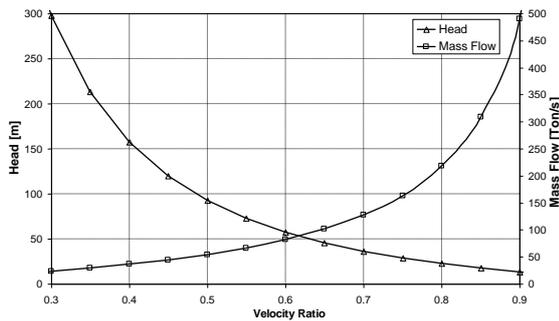


Figure 4. System head and flow rate as a function of velocity ratio

### 3. Assessment of Wake Parameter

The waterjet inlet provides flow from beneath the ship to the waterjet pump. The waterjet inlet is located near the stern area of the ship, and as a result inflow to the flush waterjet inlets will include a significant amount of hull boundary layer flow. This means that for a flush waterjet inlet, the flow entering the waterjet inlet has a momentum velocity,  $U_m$ , which is less than the ship speed,  $V_S$ , due

to the inclusion of hull boundary layer flow. The amount of the hull boundary layer flow which is included in the inlet flow is important as it can impact the size, performance, and propulsive efficiency of the waterjet pump and must be taken into consideration during the design phase of a waterjet pump.

Prediction of the inlet momentum velocity requires a thorough understanding of the boundary layer velocity profile which is instrumental for estimation of the boundary layer thickness,  $\delta$ .

Svenson et al. [10] presented data for hull boundary layer thickness for high Reynolds number applications, as:

$$\delta = 0.27 \times L_x \times Re_x^{-1/6} \quad (16)$$

where

$$Re_x = \frac{U_\infty \times L_x}{\nu} \quad (17)$$

Accordingly, velocity at any point in the boundary layer region is given by:

$$\frac{U}{U_\infty} = \left( \frac{y}{\delta} \right)^{1/n} \quad (18)$$

where

$$n = \log_{10} Re_x \quad (19)$$

A standard method of expressing the impact of the inlet momentum velocity is the inlet wake fraction or factor given as:

$$\omega = \left( 1 - \frac{U_m}{U_\infty} \right) \quad (20)$$

which can be conversely written as:

$$U_m = (1-\omega) \times U_\infty \quad (21)$$

where  $U_\infty$ , is the uniform speed at far stream (in this case is considered to be the ship-speed  $V_S$ ) and  $U_m$  is the velocity inside the boundary layer (but here, it is the flow speed at the inlet due to the boundary layer). Figure 5 shows the computed values of wake parameter and the boundary layer thickness as functions of ship velocity. It is quite obvious that these two significant parameters decline as ship velocity increases. However, the value of wake parameter is about 0.9 and this parameter could be considered as a constant quantity.

### 4. Validation of Design Parameters

As mentioned before, main results of the current code in the form of mass flow rate, pump head and power have been compared with the available results of design software and model test [6]. A range of data made available by the current code was compared with that provided by the CCDOTT software [6]. Furthermore, comparison was made between the results of the current code with the CCDOTT experiment for the design point shown in figures 6, 7, and 8. 'WJ' sign in comparison

figures shows the results of the present method. Figure 6 shows the comparison of the mass flow rate for the experimental design point, the current method and fast ferry computed results which was offered by CCDOTT [6]. Also, figures 7 and 8 present the same comparisons for system head and power, respectively. These figures demonstrate that the existing errors between the results of the current code and the experimental data are generally lower than 5%. Based on the presented validations, it can be concluded that the off-design results are quite reliable. The off-design results become particularly important when the high-speed craft tends to reach a maximum speed over the main hump of the speed-resistance curve, because the power which is produced by the engine in off-design condition is sometimes lower than the required value

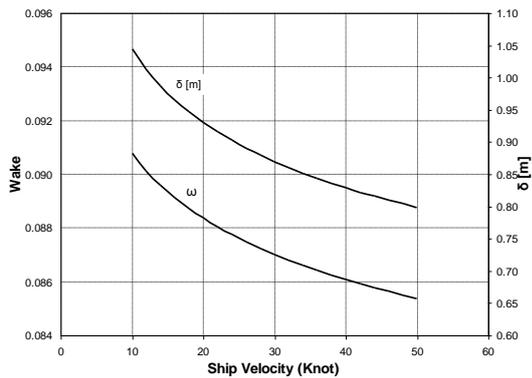


Figure 5. Boundary layer thickness and wake factor as a function of ship velocity

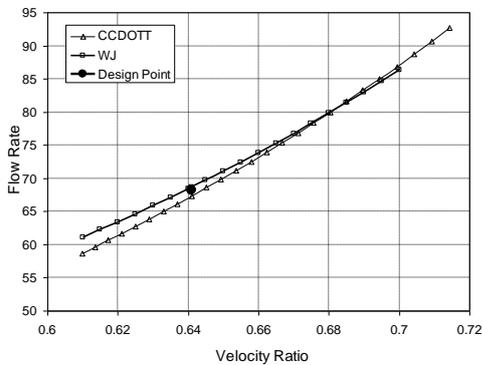


Figure 6. Comparison of flow rate results

### 5. Analysis of the Inlet Design Parameters

The current code is capable to compute the inlet area and pump diameter from momentum equation and geometry limits. Based on these computed results, empirical theory was used to predict the system losses and basic geometrical parameters. It was found that elliptical inlets have more advantages than the rectangular inlets. Experiments show that elliptical inlet geometry with aspect ratio of 1.3 is the best inlet form for flush type jet [2]. Figure 9 shows the parameterized geometry of the inlet profile. Obviously, a fair inlet profile has many advantages, but sometimes transom geometry and aft form of ship impose limitations. To survey the inlet profile shape and to analyze the flow over this domain, some geometrical parameters have been defined. Cavitation and

separation could occur at two points at different inlet velocity ratios [11]. Figures 10 and 11 show the positions of the cavitation and separation phenomena at low and high inlet velocity ratios, respectively. The probability of cavitation occurrence was shown by the current work to be a function of the inlet angle, ramp radius among other design parameters. In the meantime, Inlet angle was found to be the most influential parameter which affects the cavitation occurrence and as such, the best inlet angle was determined. Based on this observation, it was decided that controlling of the cavitation occurrence be done through variation of the inlet angle while keeping other parameters constant.

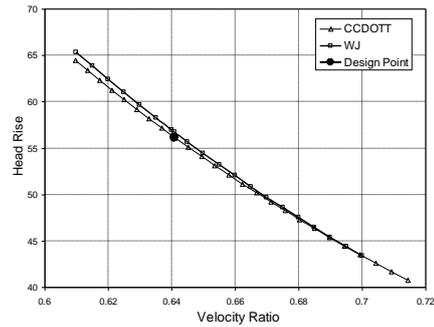


Figure 7. Comparison of system head results

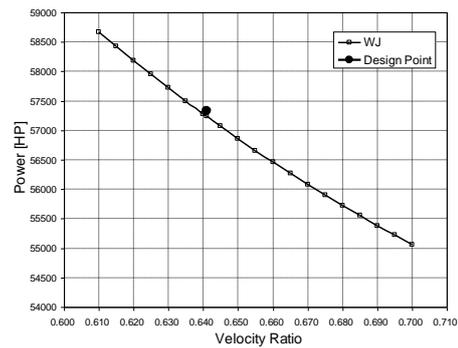


Figure 8. Comparison of system power results

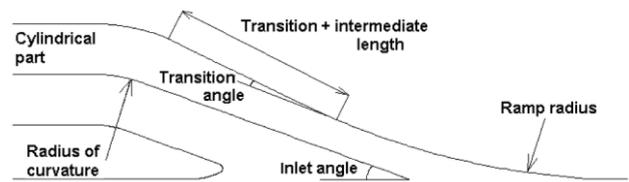


Figure 9. Inlet design parameters

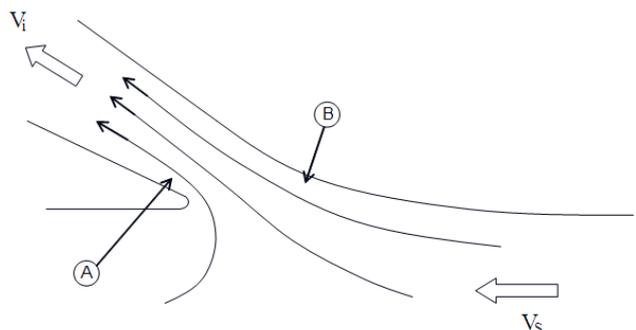


Figure 10. Cavitation (point A) and separation (point B) at low Inlet velocity ratio (IVR)

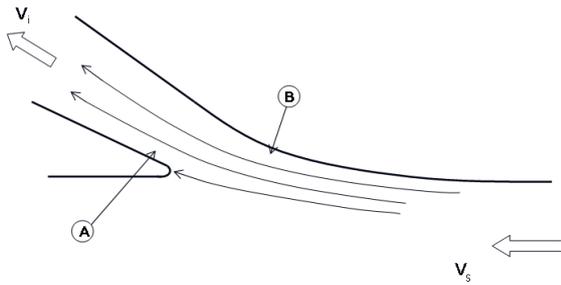


Figure 11. Separation (point A) and cavitation (point B) at high Inlet velocity ratio (IVR)

## 6. Numerical Analysis of the Inlet Duct

When designing a high-speed waterjet inlet, it is difficult to achieve an optimum design that is both efficient and has low drag. This is because cavitation occurs in the water that is unfavorable to both drag and efficiency.

Cavitation generally occurs when the local pressure on a body moving in a fluid drops to or below the vapor pressure of the fluid. When reaching the vapor pressure, small “bubbles” or cavities are produced. These cavities will collapse when they reach a higher-pressure region and cause a small “water hammer” to form. This phenomenon is called cavitation. Cavitation can produce the negative effects of noise, vibration, and erosion or damage to the inlet, and therefore, must be avoided in order to safeguard the efficiency and the drag.

RANS<sup>2</sup> code has been used to find pressure distribution around the inlet duct. Two-dimensional geometry of waterjet inlet duct has been modeled. Additionally, a rectangular region is constructed around the hull to represent the boundaries of the ocean. Since the ocean is not actually bounded, those boundaries will have to either be placed adequately far away from the ship hull and defined as a wall, or placed closer to the hull to reduce the size of the problem and defined as an opening.

Once the modeling was accomplished, an inlet duct was analyzed for four different inlet angles ranging from 26 to 34 degrees, and minimum pressure at the given inlet angles were compared against the numerical findings of CCDOTT group [6]. This comparison in Figure 12 shows a good agreement between the validation base and numerical calculations. Selection of the inlet angle was done based on two different design considerations; suitable length of propulsion system and avoidance of cavitation. Finding the optimum inlet angle requires a trial and error progress. Accordingly, a large angle was selected and possibility of cavitation was examined, needless to say that assuming a proper safety factor is very important in this progress.

Cavitation number is defined as:

$$\sigma = \frac{p - p_v}{\frac{1}{2} \rho U_\infty^2} \quad (22)$$

When “ $\sigma < 0$ ”, cavitation starts. This implies that high cavitation numbers give less risk for cavitation. Figure 13 shows contours of cavitation numbers in XY plane while

Figure 14 presents the cavitation number at the centerline of the system for two different inlet angles. These figures could furthermore indicate how and where the cavitation could occur. Figure 14 shows how the system will operate at a design speed range without cavitation at the inlet ramp when the inlet angle becomes 26 degrees.

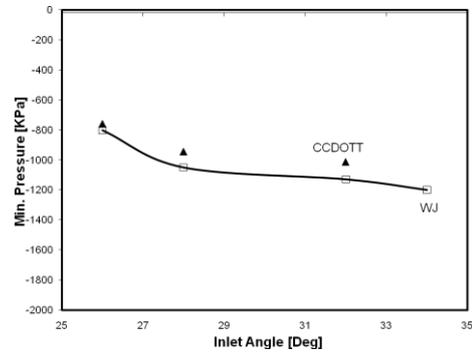


Figure 12. Comparison of minimum pressure for different inlet angles

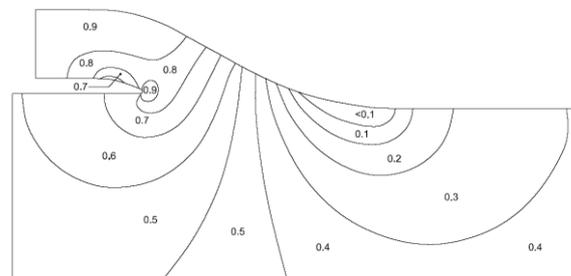


Figure 13. Cavitation number around inlet duct (Inlet angle 26 °)

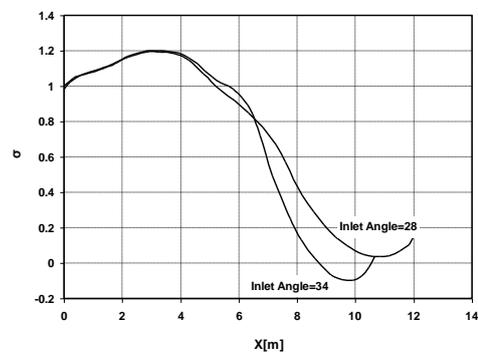


Figure 14. Cavitation number on top inlet wall at two inlet angle (see Figure 9)

3-D analysis of inlet duct which was done by CCDOTT shows that the minimum pressure occurs at the centerline. On the other hand, the results of the current study have good conformity with those of the mentioned 3D analysis. Thus, Figure 14 could be considered as a reliable source for designing the inlet duct.

## 7. Conclusion

Simultaneous accomplishment of the conceptual and basic design as well as the analysis of waterjet propulsion system in a single process is a cumbersome task. Many analytic and empirical methods have been proposed to evaluate the influential parameters of this system. Numerical methods are used for the analysis of the system

<sup>2</sup> Reynolds Averaged Navier Stokes

performance, albeit not in a design process. In the current work, a particular method has been presented which covers all stages of the design and the analysis of waterjet propulsion system which has also been validated by reliable results. Two different codes have been developed, one for the analytical assessment of the main waterjet propulsion system parameters and the other of numerical investigation of the inlet duct impacts.

In the current scheme, a practical method is used to predict the powering characteristics of systems. Prediction of the performance of the waterjet system at the design point starts with determination of the required thrust, jet diameter, shaft horsepower, and RPM which is done by a developed computer program. During the detailed hydrodynamic optimization study, the RPM, radial blade loading distribution, and other parameters of the waterjet were varied to arrive at the optimum design for this power level.

Computed values of flow rate, head rise, and power versus the velocity ratio were compared against earlier reported experimental and numerical results. These comparisons demonstrated good agreements indicating that the separate handling of the waterjet and hull designing process appears to be quite possible. The adopted approach leads to a set of parametric relations that describes the interaction between the hull and the waterjet system. Because of this modular approach, the results can simply be refined during the design process of the vessel.

Empirical relations have been used for preliminary estimation of the required power for the vessel. From this calculated thrust and the assessed internal losses in the waterjet system, the required pump power can be found.

Possible cavitation at the inlet duct leads to erosion or vibrations which must be avoided. In this paper, cavitation characteristics at the inlet were found by looking at the pressure distribution of the water around the inlet opening. Accordingly, the possibility of cavitation was investigated and controlled by close scrutiny of the cavitation number. Along the same line, optimum inlet geometry was found based on the observation of pressure distributions at different inlet angles. This whole process was achieved by a 2-D numerical code written for the flow investigation at the inlet of the waterjet system.

Numerical findings of the current 2-D code compared with the earlier results of the 3-D modeling show an excellent match between the pressure distributions. As a result, one may conclude that a lengthy process of 3-D computation can indeed be avoided and that similar results can easily be achieved by a 2-D analysis which is a less laborious process and saves much computational time.

Main geometrical parameters for the design of waterjet propulsion system have been determined by the present method. By utilizing these important parameters, a designer is able to predict the waterjet performance on a ship before arriving at the basic design stage. Needless to say that there are other considerations that must be taken into account which include necessary number of jets, position of the propulsion system, and the effect of the system on compartmental arrangement of the ship.

## Nomenclature

$\dot{m}$ :	Mass flow rate
$\rho$ :	Density
$A_{jet}$ :	Area of nozzle outlet
$V_{jet}$ :	Jet velocity
$T$ :	System thrust
$V_{in}$ :	Inlet velocity
$P_E$ :	Effective power
$V_S$ :	Ship speed
$H_P$ :	Pump head
$h_{loss}$ :	Total head losses
$\eta_{jet}$ :	Jet efficiency
$IVR$ :	Inlet Velocity Ratio
$h_{inlet}$ :	Inlet head losses
$h_{outlet}$ :	Outlet head losses
$\xi$ :	Inlet losses coefficient
$\psi$ :	Outlet losses coefficient
$P_{pump}$ :	Pump power
$h_j$ :	jet height from waterline
$\omega$ :	Wake factor
$\mu$ :	Velocity ratio
$\eta_{pump}$ :	Pump efficiency
$\eta_{hull}$ :	Hull efficiency
$t$ :	Trust reduction factor
$\delta$ :	Boundary layer thickness
$L_x$ :	Distance from bow
$Re_x$ :	Reynolds number
$\nu$ :	Kinematical viscosity
$y$ :	Vertical distance from hull
$U_\infty$ :	Uniform flow velocity (ship speed)
$U_m$ :	Flow speed at the inlet due to boundary layer
$\sigma$ :	Cavitation number
$p$ :	Fluid pressure
$p_v$ :	Fluid vapor pressure

## References

- [1] Carlton.J.S. (John S.), "Marine Propeller and Propulsion", Oxford, Butterworth-Heinemann, 1994.T.J.C. Van Terwisga, "A Parametric Propulsion Prediction Method For Waterjet Driven Craft", Fast'97 Conference, paper No. 151, 1997.
- [2] T.J.C. Van Terwisga, "A Parametric Propulsion Prediction Method For Waterjet Driven Craft", Fast'97 Conference, paper No. 151, 1997.
- [3] Etter, R.J., Krishnamoorthy, V. and Sherer, J.O., "Model Testing of waterjet Propelled Draft", Proceedings of the 19<sup>th</sup> ATTC, 1980.
- [4] Allison, J.L., "Marine Waterjet Propulsion", SNAME Annual meeting, New York, 1993.
- [5] ITTC, "The Specialist Committee on Waterjets", 22th ITTC, International Towing Tank Conference, 1998.
- [6] Stanley Wheatley, "Development of a High-speed Sealift Waterjet Propulsion System", Final Report, Center for the Commercial Deployment of Transportation Technologies California State University, Long Beach Foundation, September 30, 2003.
- [7] Bulten, N.W.H., "Numerical Analysis of Waterjet Propulsion System", PhD Thesis, Technical University of Eindhoven, 2006.
- [8] Moon-Chan Kim, Ho-Hwan Chun, "Experimental Investigation into the performance of the Axial-Flow-Type Waterjet according to the Variation of Impeller Tip Clearance", Ocean Engineering 34, pp.275-283, 2007.
- [9] Hong Gao, Wanlai Lim, Zhaohui Du, "Numerical Flow and Performance Analysis of a Waterjet Axial Flow Pump", Ocean Engineering 35, pp.1604-1614, 2008.
- [10] Svensson, R., et al., "Trial Results Including Wake Measurements from the World's Largest Waterjet Installation", RINA International Conference on Waterjet Propulsion, Amsterdam, 22-23 October 1998.
- [11] Donald M. MacPherson, "A Universal Parametric Model for Waterjet performance", FAST '99, Aug 1999.