Kevin Raleigh

Development of Numerical Code to Model Submerged 2D Hydrofoils

ABSTRACT

Hydrofoils are used on a wide range of marine applications such as high speed ferries, high speed crafts and sailing boats. This thesis focuses on the determination of the pressure profile on fully submerged 4-digit NACA hydrofoils in an ideal fluid. The approximation of the fluid as inviscid, irrotational and incompressible allows for the use of a potential flow based model. Potential flow based models or Boundary Element Methods (BEM) are widely used in early ship design phase because of their computational speed that allows for a comparatively fast evaluation of a number of designs, as BEM reduces the fluid domain to the boundary of the submerged body. The submerged hydrofoil is discretized into flat panels where each panel has a Kelvin source at the control point. The use of Kelvin sources will ensure the free-surface condition is fulfilled. The basic hydrofoil resistance estimations are shown to be consistent with the findings from proven computational methods used to determine the pressure on submerged hydrofoils. The thesis also details, some of the various steps to be taken as future developments with this program or in similar programs of its kind.

INTRODUCTION

Over time, computational modelling has become increasingly important since they allow engineers to design and optimize systems faster and cheaper than experiments. Experimental modelling requires physical models, which present additional expense and time, as each iteration must be manufactured and tested independently of its predecessors. Computational and mathematical models allow for a large number of tests conditions to be evaluated, with virtually no added expense. The additional time is only limited to the processing power available.

In a marine context, conventional testing methods of ships and submerged bodies resistance involve machining of a scale model and testing it in a towing tank or fluid flow facility. Such tests are time consuming and to make an alteration in the design of a vessel once testing has been done often requires the use of an entirely new to scale model. Computational fluid dynamics programs allow for variations in design to be tested without the need for time consuming alterations. Rather, new geometries can be introduced as quickly as they can be made in a modelling software.

This paper presents a MATLAB code which allows a user to specify a submerged body in order to find the pressure distribution along its length.

BACKGROUND

MODELLING APPROACH

A 2D potential flow model is developed in the complex plane in order to determine the pressure distribution on submerged hydrofoils. Potential flow allows simplicity in calculation, which in turn allows for economy of computational time. Since hydrofoils are prismatic they can accurately be modelled in 2D, neglecting wing tip effects. Using a potential flow based model to evaluate submerged geometries has been proven successful many times for several decades as shown by Giesing et al (1967) and Hess et al (1967). In a potential flow model, the submerged body is represented by a number of sources, vortexes or doublets, here sources are placed as shown by the stars in Figure 1.

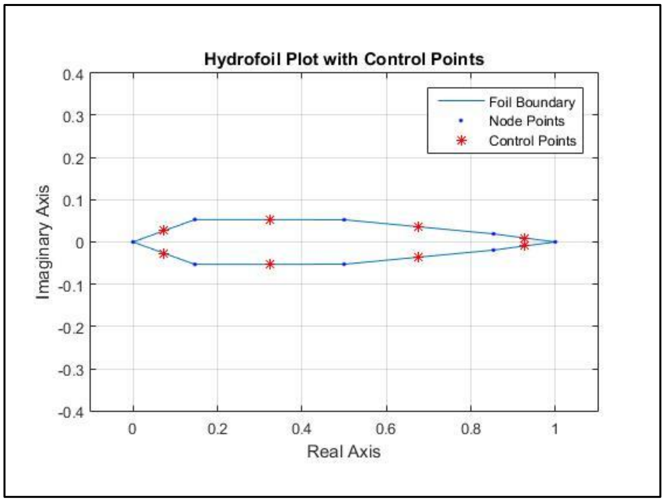

Figure 1 Representation of a NACA 0012 hydrofoil using 8 panels

By modelling, the body as a series of sources, with a tip vortex, all the influences of the singularities can be added together in order to find the total effect from the fluid flow on the body. This technique of demonstrating a geometry computationally as a series of sources and vortexes follows the procedure described by Fürth (2011) and Chen (2012).

Calculations of sources, and vortex are done at discrete points along the outer edge of the hydrofoil. Because of this, the geometry of the hydrofoil is modelled as a finite number of straight panels representing the curved geometry, called a Boundary Element Method, BEM

The geometry of the hydrofoil and the effects around it are modelled in the complex plane. The complex plane allows for a better utilization of vectors. Thus each control point "z" is described as:

$$z = x + iy \qquad (1)$$

One source is located at each control point and the flow speed is determined at each control point. A single vortex is placed at the trailing edge in order to fulfill the Kutta condition which states that at the trailing edge of the hydrofoil there is a stagnation point due to the upper and lower velocities. Using Kelvin sources is especially beneficial in the complex plane. Kelvin sources rely on meshing the object and calculating pressure relative to its geometry as opposed to Rankine sources which require meshing of the free surface as well. Thus, the system is modelled such that the undisturbed surface is always at a height of zero, the complex conjugate of each source matches the mirror image of the source above the free surface. No flow can cross the surface, as there is a streamline at the free surface, thus the surface is a barrier between the fluid above it and the fluid below it.

GOVERNING EQUATIONS

Fluid flow is governed by the Navier-Stokes equations, however the equations are complex causing numerical models solving the full Navier- Stokes equation to be computationally time consuming.

In order to reduce the Navier Stokes equations a potential is defined satisfying the Laplace equation so that:

$$\nabla^2 \phi = 0 \qquad (2)$$

This reduction comes at a cost, the modelled fluid will now be:

- Irrotational
- Inviscid
- Incompressible

This is known as an ideal fluid. Assuming the fluid to be irrotational satisfies the Laplace equation. By assuming the flow is inviscid, all shear forces in the fluid are neglected. It is through this that the Navier Stokes equation is reduced to the Euler equations. When discussing incompressible flow, a compressible fluid can still be modelled with some accuracy, however, it allows for the assumption of no divergence in the flow, thus further simplifying the governing equations. The physical limitations of potential flow are that systems with boundary layers cannot be modelled. However, by layering sources, a boundary layer can be modelled to imitate a real-life system. The potential is a field variable valid throughout the fluid domain. The flow speed is:

$$(u, v) = \nabla\phi \tag{3}$$

DISCRETIZATION

In order to model the geometry of the hydrofoil, a number of panels has to be used. Each panel is defined as the line between two nodes on the outer edge of the foil section. A control point, z_c, is specified as the midpoint along the line defined by:

$$z_C = \frac{z_B - z_A}{2} \tag{4}$$

Where A represents the first, and B represents the second node traversed around the geometry counterclockwise.

HYDROFOILS USED

In order to test the versatility of the program, both symmetrical and cambered hydrofoils are evaluated. Both hydrofoils are NACA four-digit foils whose geometry is defined by:

$$\frac{dy_c}{dx} = \begin{cases} \frac{2m}{p^2}\left(p - \frac{x}{c}\right) & 0 \le x \le pc \\ \frac{2m}{(1-p)^2}\left(p - \frac{x}{c}\right) & pc \le x \le c \end{cases} \tag{5}$$

Where m is the maximum camber, p is the location of maximum camber, c is the chord length, and x is the position along the chord from 0 to c. This equation is used for both symmetrical and asymmetrical foil numbers and shapes. Here a NACA 0012 symmetrical foil and a NACA 4412 cambered foil are used. These hydrofoils are chosen because a maximum thickness of 12% of the cord length is common for hydrofoils leading to a large amount of research on foils of this shape (Bal, 1998). The NACA 4412 hydrofoil has the same maximum thickness and a moderate camber to display the adaptability of the code.

FREE SURFACE MODELLING

After modelling the hydrofoil, the free surface was modelled for the cases of operations near the free surface. The location of the free surface, η, was determined using the dynamic free surface boundary condition.

$$\eta = -\frac{U\phi_x}{g} \qquad (4)$$

Here, g is the gravity acceleration. The surface profile is seen in Figure 2.

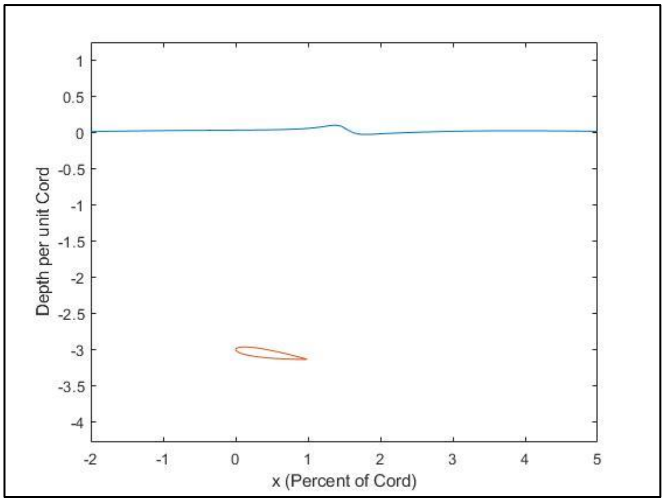

Figure 2 Illustration of Free Surface in Relation to Hydrofoil

RESULTS

CONVERGENCE OF PANELS

In order to verify the precision of the results acquired from the program, convergence study is conducted on the number of panels required to model the sources that the hydrofoil is to be represented with. In order to make an efficient code taking full advantage of the computational speed associated with potential flow based models the number of panels used depends not only on the precision of the values that were calculated, but also on the amount of time taken for the calculations to be done. To calculate the percent error of each number of panels, each value of pressure coefficient is compared to a pressure value determined using twice as many panels in an iterative process using:

$$Error = 100 * \left| \frac{C_{p_{j+1}} - C_{p_j}}{C_{p_{j+1}}} \right| \qquad (6$$

Through this process, the error amount converged as the number of panels increases.

Table 1 displays the percent error for each number of panels.

Table 1 Error Convergence

Number of Panels	Error	CPU Time (s)
8	29.9%	0.65
16	8.3%	1.02
32	2.4%	12.47
64	1.46%	20.31
78	**0.97%**	**26.91**
128	0.08%	64.55

As seen in Table 1, the optimal number of panels is 78 as this value allows a percent error of less than 0.1% while maintaining a computational time of less than one minute on a Lenovo Y500 Laptop while the code ran non-concurrently.

In addition to investigating the appropriate number of panels, the program was evaluated to show that even at a low number of panels, the results matched similar data. At 20 panels, the results for the pressure profile were compared to those of the PANEL method which similarly discretizes the hydrofoil into flat panels, as shown in Figure 3, developed by Cummings (2015).

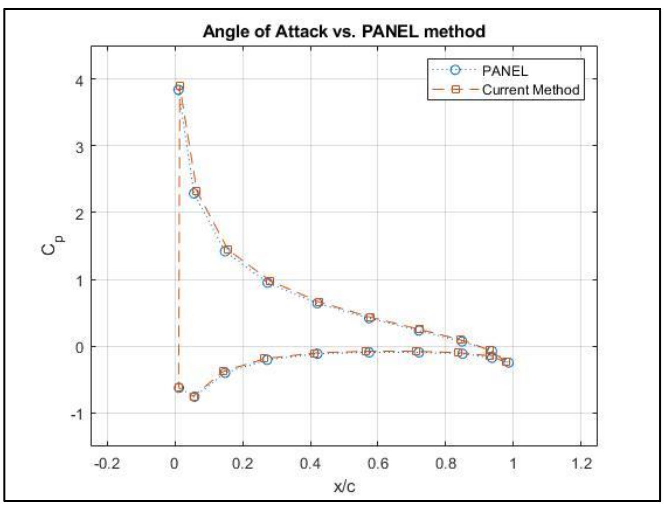

Figure 3 Low panel number comparison between the Current Method and the PANEL method for a NACA 4412 hydrofoil at an 8 degree angle of attack

DETERMINATION OF SIGNIFICANT DEPTH

The XFoil program to which the results were compared assumes a hydrofoil in deep water with no surface effect. Due to this, it is important to discover when the depth, *d* of the hydrofoil is enough to be considered deeply submerged. This is done by comparing the results of the hydrofoil as the depth normalized by cord length below the surface increased as shown in Figure 4.

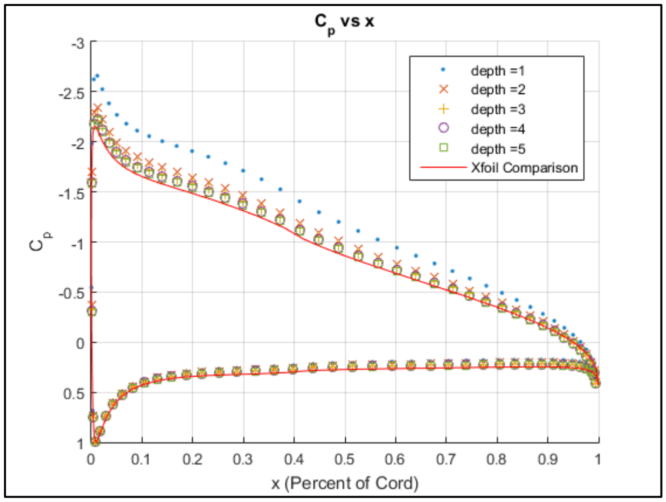

Figure 4 Deep water study of a 4412 NACA hydrofoil at a 6 degree angle of attack

From these results it is concluded that the results at a depth of 6 d/c is considered deep water compared to the XFoil program. This is consistent with the findings of Bal. (1998)

COMPARISON DATA

Using 78 panels to discretize the hydrofoil, the acquired data from the MATLAB code is validated using a secondary source. This test is done using XFoil hydrofoil design software. Xfoil makes use of potential flow as well to make its calculations. The pressure on a fully submerged hydrofoil determined by the present method and by XFoil is shown in Figure 5.

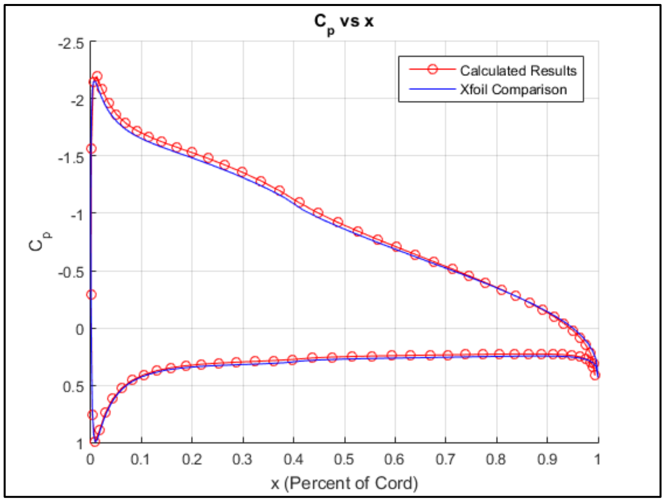

Figure 5 The pressure on a 4412 NACA hydrofoil at a 6 degree angle of attack

The results shown here display a close match between the values obtained through the MATLAB code and those from the XFoil program.

After determining agreement between current results and the reference, proper angle of attack orientation was determined by comparing test data to angle of attack data from the PANEL method which similarly uses a discrete element method to evaluate a potential flow program as shown in Figure 6.

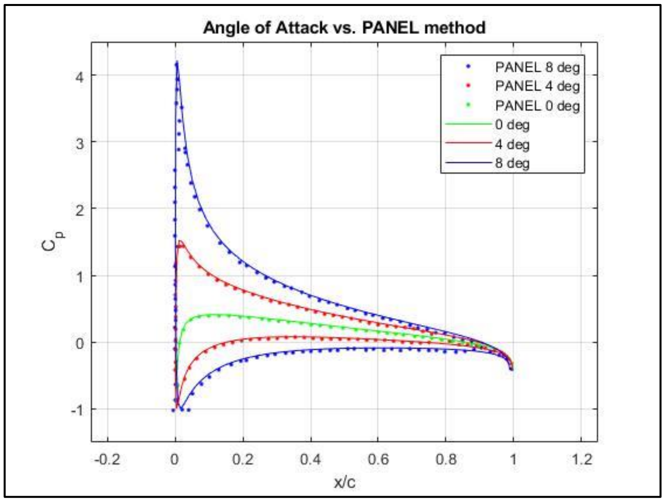

Figure 6 Comparison of Test Data to PANEL method data for NACA 4412 foil with varying Angles of Attack

CONCLUSIONS

In this experiment a code is developed that accurately modelled proven results. This code was shown to work not only for symmetrical, but for asymmetrical hydrofoils with an angle of attack.

The presented method showed good accuracy when compared to an existing numerical method. This means that the program can be used as a timely way to test geometries alongside other methods.

Moving forward with this analysis, the wave profile behind the hydrofoil should be compared to existing data to ensure it is accurately modelling the trailing wave. In addition, new applications of the data should be investigated. Future development of a 3D model is another possibility as demonstrated in Kinnas et al (1993).

REFERENCES

Bal, Sakir. "A Potential Based Panel Method for 2-D Hydrofoils." Ocean Engineering, vol. 26, no. 4, 1998, pp. 343–361.

Chen, Zhi-Min. "A vortex based panel method for potential flow simulation around a hydrofoil." *Journal of fluids and Structures* 28 (2012): 378-391.

Cummings, Russell M., et al. Applied Computational Aerodynamics: a Modern Engineering Approach. Cambridge University Press, 2015.

Fürth, Mirjam. *Determining the Pressure Distribution on Submerged 2D Bodies Using Dissipative Potential Flow*. Master diss, KTH, Royal Institute of Technology, (2011).

Giesing, Joseph P., and A. M. O. Smith. "Potential flow about two-dimensional hydrofoils." *Journal of Fluid Mechanics* 28.01 (1967): 113-129.

Hess, John L., and A_M O. Smith. "Calculation of potential flow about arbitrary bodies." *Progress in Aerospace Sciences* 8 (1967): 1-138.

Kinnas, Spyros A., and Neal E. Fine. "A numerical nonlinear analysis of the flow around two-and three-dimensional partially cavitating hydrofoils." *Journal of Fluid Mechanics* 254 (1993): 151-181.

Raleigh, Kevin. “NASEC 2016.” Potential flow based code for hydrofoil pressure profiles, 2016.

Raleigh, Kevin. “NASEC 2017.” Determining the Flow around 2D Bodies Using a Potential Flow Based Model, 2017.

Raleigh, Kevin. “SNAME SMC 2017.” Potential Flow-Based Boundary Element Method for Submerged Rigid 2D Bodies, 2017.