A Novel Fitting Method for Steady Free Surface Flow

Toon Demeester, Joris Degroote, Jan Vierendeels
Department of Flow, Heat and Combustion Mechanics, Ghent University, Belgium
toon.demeester@ugent.be

1 Introduction

Computational methods to simulate water-air flows around ships can be divided in two categories: surface capturing and surface fitting methods. Capturing methods reconstruct the position of the free surface (FS) to apply the boundary conditions, an example is the volume-of-fluid method. In fitting methods the mesh lies along the free surface and deforms with it, so that its position is always known. The air phase is often neglected, leading to one phase flow with FS boundary conditions (BCs). The dynamic boundary condition (DBC) which requires continuity of the stresses, reduces to a constant pressure condition. The kinetic boundary condition (KBC) requires that the FS is impermeable.

For steady FS flows, two clearly different fitting methodologies can be found in literature. Both consist of an iterative process with two steps that are repeated until convergence is reached: first calculation of the flow field with a fixed FS position and suitable boundary conditions, then update of the FS position. The first methodology uses the DBC in the first step and the KBC in the second (Tzabiras (1997); Muzaferija and Perić (1997)). Using the KBC for updating the surface results in a time-stepping method, which is not efficient for steady cases due to the large number of time steps before transient phenomena disappear. Van Brummelen et al. (2001) describe a steady iterative fitting method which uses a combined boundary condition (KBC + DBC) in the flow solver and the DBC for the surface update. This method is efficient but requires a dedicated coupled solver, making it less flexible.

The goal of this paper is to test a novel fitting method which combines a steady iterative approach with a black box flow solver. This requires that the BCs in the flow solver are easy to implement (only KBC) and that the update procedure is time-independent (only DBC).

2 Theory

2.1 Modal analysis

The KBC is easily enforced in the flow solver by setting the FS boundary as a free slip wall. When the flow field is solved, the pressure $p$ at the FS will not be constant, yielding a pressure error $\tilde{p}$. To satisfy the DBC, the FS must be updated in an attempt to compensate this error $\tilde{p}$. If a relationship can be found between the known pressure error $\tilde{p}$ and the unknown position error $\tilde{\eta}$, such an update procedure can be constructed.

A relatively general relationship is obtained by doing a modal analysis of these errors for the inviscid flow over a flat horizontal bottom as shown in Fig. 1. In this case the solution is known (a horizontal FS) so that a relation between $\tilde{\eta}$ and $\tilde{p}$ can be found by calculating the flow field for a given error mode $\tilde{\eta}$. This derivation was done using potential flow theory, but the details are skipped here for the sake of brevity. The result is a proportional relation between $\tilde{\eta}$ and $\tilde{p}$

$$\tilde{\eta}(x) = K \cdot \tilde{p}(x) \quad \text{with} \quad K = \frac{1}{\rho g \left( \frac{F^2}{\text{Fr}^2} - \frac{kh}{\text{tanh} \lambda} \right)}$$

(1)

Fig. 1: Domain studied in modal analysis
where $K$ is a function of the wavenumber $k = 2\pi/\lambda$ and the Froude number $Fr = U/\sqrt{gh}$. In these formulas, $\rho$ is the density, $g$ the gravitational acceleration, $h$ the undisturbed water depth, $\lambda$ the wavelength and $U$ the average flow velocity. The validity of Eq. (1) was checked with numerical experiments.

Two dimensionless parameters are present in Eq. (1): $Fr$ and $h/\lambda = kh/2\pi$. A visual representation of $K$ as a function of these two parameters is shown in Fig. 2. $h/\lambda$ is the aspect ratio of the domain: small values signify shallow water with respect to the wavelength, large values deep water. The Froude number describes the relative importance of inertial to gravitational forces. At low $Fr$, gravitational forces dominate, so that the flow is governed by the hydrostatic effect: if $\tilde{\eta} > 0 \Rightarrow \tilde{p} < 0 \Rightarrow K < 0$. At high $Fr$, inertial forces dominate, so that the flow is governed by the Bernoulli effect: if $\tilde{\eta} > 0 \Rightarrow v \downarrow \Rightarrow \tilde{p} > 0 \Rightarrow K > 0$. In Fig. 2, these two regions are clearly present and separated by an asymptote. Close to the asymptote, the opposing effects of gravity and inertia on $\tilde{p}$ balance each other out. Along the asymptote the phase velocity of the waves is equal to the flow velocity, giving rise to steady surface gravity waves. These are correct solutions of the linearized free surface problem ($\tilde{p} = 0$, DBC satisfied) and therefore give $K_{th} = \infty$.

2.2 Iterative procedure

The relation in Eq. (1) can now be used to assess the theoretical performance of the iterative procedure mentioned in the introduction: in the first step the flow is calculated for a FS position $\eta$ with a flow solver $F(\eta)$, in the second step the surface position is updated with the procedure $S(p, \eta)$, forming the system

$$\begin{cases}
p = F(\eta) \\
\eta = S(p, \eta)
\end{cases}$$

(2)

The update procedure is now defined as

$$p = F(\eta) \quad \text{and} \quad \eta = S(p, \eta) \xrightarrow{\Delta} \eta - K^* \cdot \tilde{p}$$

(3)

where $K^*$ is calculated as in Eq. (1) for a wavenumber $k^*$ which is chosen further on. With this definition of $S(p, \eta)$, the residual $R(\eta)$ of the system (2) is found as

$$R(\eta) \xrightarrow{\Delta} S(F(\eta), \eta) - \eta = -K^* \cdot \tilde{p}$$

(4)

which gives the Gauss-Seidel (GS) iteration (without underrelaxation) for iteration $n$ as

$$\eta^{n+1} = \eta^n + R(\eta^n) = \eta^n - K^* \cdot \tilde{p}$$

(5)

The new position error $\tilde{\eta}^{n+1}$ is then found by subtracting the correct position from both sides. If $\tilde{\eta}$ is composed of multiple Fourier modes $i$, the amplification $\mu_i$ of each error mode $\tilde{\eta}_i$ is

$$\mu_i = \frac{\tilde{\eta}_i^{n+1}}{\tilde{\eta}_i^n} = 1 - K^* \frac{\tilde{p}_i^n}{\tilde{p}_i^n} = 1 - \frac{K^*}{K_i}$$

(6)
where Eq. (1) is used in the last step to model the flow solver. Only if $|\mu_i| < 1$, error mode $\tilde{\eta}_i$ will damp out during Gauss-Seidel iterations and is stable. If modes are present in the left region of Fig. 2 (low Fr), it is not possible to choose a $K^*$ for which all modes are stable, because $K_i$ changes sign when the asymptote is crossed. However it is possible to make all high wavenumber modes stable by choosing $k^* = k_{\text{max}} = \pi/\Delta x$ (with $\Delta x$ the mesh width) for the calculation of $K^*$, leading to a limited number of unstable low wavenumber modes. The amplification of the highest mode is then 0. As an example, $\mu_i$ is plotted in Fig. 3 for a mesh with 100 and one with 200 harmonics. The base mode corresponds to $h/\lambda = 1/8$, so that quite some unstable modes are produced for low Fr. If the base harmonic has a higher $h/\lambda$ or the Froude number is higher, the number of unstable modes reduces quickly.

For the cases which have one or more unstable modes, GS iterations will diverge because those error modes grow while iterating. In the fluid-structure interaction community, similar iterative processes with a limited number of unstable modes are often stabilized and accelerated using a quasi-Newton coupling algorithm such as IQN-ILS (Interface-Quasi-Newton with Inverse Jacobian from a Least-Squares model, Degroote et al. (2009)). Such an algorithm gives a better prediction of the new FS position $\eta^{n+1}$ based on information from the previous iterations.

In short, the workings of IQN-ILS can be explained as follows. For all previous GS steps, both the residual $r = R(\eta)$ and the result of the GS iteration $\hat{\eta} = S(F(\eta), \eta)$ are stored. The differences of these variables between two consecutive time-steps are denoted respectively as $\Delta r$ and $\Delta \hat{\eta}$. To get from the current position $\eta^n$ to the new position $\eta^{n+1}$, the update $\Delta \eta^n$ must be calculated. From the definition of the residual in Eq. (4) this can be rewritten as

$$\Delta \eta^n = \Delta \hat{\eta}^n - \Delta r^n$$

Ideally, the iterations should converge in the next time-step, i.e. have $r^{n+1} = 0$, so that $\Delta r^n = -\eta^n$ is known. $\Delta r^n$ is now written as a least squares approximation of old $\Delta r$'s. An estimate of $\Delta \hat{\eta}^n$ can then be constructed by taking the same linear combination of old $\Delta \hat{\eta}$'s.

If a mode appears in the residual which has been encountered in one of the previous iterations, the algorithm predicts how the system reacts to this error, and is able to efficiently damp this mode. If the system is linear, the algorithm knows all the unstable modes after a number of iterations and the solution converges. Badly damped stable modes are also recognized by the algorithm and converge faster than would be the case for pure Gauss-Seidel iterations. In this paper, the IQN-ILS algorithm will only be used for this last purpose. Subcritical flows with unstable modes are currently under investigation, but problems regarding the boundary conditions have to be overcome.

3 Numerical experiments

The iterative procedure discussed in the previous section is tested on two academic test cases in 2D. Viscous forces are not taken into account in the calculations. Three different coupling algorithms are
used: Gauss-Seidel, IQN-ILS and a slightly adapted version of the last which is explained later. For the
GS iterations, deviations between numerical experiments and theory are to be expected.

First of all, the actual \( K_i \) for the highest frequencies on a given mesh will certainly be different
from the values predicted with Eq. (1), because these modes are not well discretized. For example, the
highest frequency will look like a triangle wave to the flow instead of a sinusoid. By doing numerical
experiments, the correct \( K_i \) could be determined for these modes, but it is hard to find a generally valid
result as \( K_i \) heavily depends on the mesh resolution perpendicular to the surface (y-direction).

Next, the potential theory derivation from which the formula for \( K \) was derived, is based on the
assumption that \( \tilde{\eta} \) is a small perturbation. As this is not the case in the numerical experiments, this may
also lead to different values of \( K \). In addition the modes may influence each other so that the relation
between \( \tilde{\eta} \) and \( \tilde{p} \) is in fact non-linear.

Finally, the choice of boundary conditions in the flow solver will play a role: the correct boundary
values of the free surface flow are not known so that the implemented values are only an approximation.
A fixed inlet x-velocity is used in combination with a constant pressure at the outlet, instead of periodic
boundary conditions in the potential theory. This constant pressure outlet can be used because gravity
is turned off in the simulations, so that in fact the hydrodynamic pressure \( p + \rho g y \) is calculated in the
flow solver, from which the hydrostatic pressure \( p \) is easily derived. To improve convergence, an inlet
y-velocity based on the slope of the FS is added, so that the inlet flow is nicely aligned.

### 3.1 Supercritical flow with horizontal bottom

The first case is very similar to the one used in the modal analysis in Section 2.1, for which the domain is
shown in Fig. 1 and the converged solution is a horizontal free surface. The initial error \( \tilde{\eta}^0 \) consists only
of the base harmonic. A cosine is used instead of a sine to make the implementation of the boundary
conditions easier.

\[
\tilde{\eta}^0 = \frac{\lambda}{10} (1 - \cos kx)
\]

A hexahedral mesh of 300 \( \times \) 70 cells is used which means there are 150 error modes. The aspect ratio of
the domain is \( h/\lambda = 1/16 \). The flow is chosen to be supercritical (\( Fr = 2 \)) because in that case \( K_i \) does
not change sign and GS iterations are stable. For the surface update, \( K^* = K(k_{max}) \) is chosen, as was
motivated in Section 2.2.

The residuals of \( \tilde{p} \) are calculated by taking the root mean square (rms) of \( \tilde{p} \). It can be seen in Fig. 4a
that they converge very slowly for GS iterations. The initial error \( \tilde{\eta}^0 \) is also the harmonic with the worst
amplification, in theory \( \mu_1 = 0.985 \) for \( \rho g K_1 = 0.29 \).

As no other modes are present (nor appear), the iterations should converge with the same rate as predicted.
The numerical result is of the same order: during 100 coupling iterations, \( \rho g K_{1,\text{num}} \) is on average around 0.40,
but the value is getting closer to the predicted one while the error decreases.

When the IQN-ILS coupling algorithm is used, convergence is much faster as shown in Fig. 4a.
When the evolution of \( \tilde{p} \) is monitored, it is observed that high frequency modes start to appear after a
few iterations, unlike what is seen during GS iterations. These sharp peaks in $\tilde{p}$ first appear at both the inlet and outlet due to the boundary conditions, gradually polluting more of the domain. This behavior is shown in an early stage (coupling iteration 11) in Fig. 5 (top). As $r = -K^* \tilde{p}$, these peaks immediately appear in the modes $\Delta r$ which are used in the IQN-ILS algorithm for the least squares approximation of the residual $r^n$. These modes are shown in Fig. 6 (left) for iteration 11. As $\tilde{p}$ is getting more polluted with high frequencies, so are the modes $\Delta r$. This is unwanted behavior and likely deteriorates the performance of IQN-ILS.

In an attempt to get rid of the high frequency noise in $\tilde{p}$ and in the modes $\Delta r$, the IQN-ILS algorithm is altered. The error $\tilde{p}$ is split into lower and higher frequency parts. The lower frequency part of $\tilde{p}$ is used to construct an IQN-ILS update $\Delta \eta$. The higher frequency part is used for a simple GS update, but in theory these modes should converge fast enough due to their low $\mu_i$. The total update is the sum of these two parts. A 5-point moving average filter\(^1\) is applied twice to split $\tilde{p}$ in lower and higher frequency parts. In terms of filtering performance this is a poor choice, but it was chosen only as proof of concept. This procedure will be referred to as IQN-ILS-low.

During the first coupling iterations, IQN-ILS-low behaves the same as IQN-ILS. When high frequencies appear in $\tilde{p}$ for IQN-ILS, IQN-ILS-low starts to converge faster as shown by the residuals in Fig. 4a. Fig. 5 shows that $\tilde{p}$ is less prone to develop high frequency peaks, Fig. 6 shows that the modes $\Delta r$ at that point are much smoother. When the residual reaches values of the order $10^{-5}$, convergence stops because it is limited by the convergence criteria in the flow solver. At this point, the consecutive values of $r$ are so close to each other that $\Delta r$ consists of numerical noise. The presence of such modes causes the quasi-Newton algorithm to give oscillating pressure residuals at the end.

The number of coupling iterations given in Fig. 4a does not give an accurate view of the total computational cost as the flow calculation becomes cheaper when the coupling iterations are converging. As the flow solver iterations are the dominant time-cost in the iteration process, the pressure residual is given as a function of the total number of flow solver iterations in Fig. 4a. Quasi-Newton coupling is clearly very advantageous and IQN-ILS-low is even faster than the original algorithm. The total number of iterations is quite high because the convergence criteria in the flow solver had to be very strict to get such a deep convergence of the coupling algorithm (> 8 orders of magnitude).

### 3.2 Supercritical flow with bottom topography

The second case that is studied is an academic case for which experimental data was collected by Cahouet (1984), but this data is not available for public download. It consists of the supercritical flow ($Fr = 2.05$) over a bottom topography described by the equation

$$y = \frac{27}{4} \frac{H}{L^3} x(x-L)^2 \quad \text{with} \quad H = 0.2, L = 2$$

A graphical representation of the domain is given in Fig. 7, with the flow from left to right. Both the initial solution $\eta^0$ and the solution $\eta^{60}$ after 60 iterations with IQN-ILS-low are shown. Tzabiras (1997) studied the same geometry for multiple Froude numbers to test a surface fitting method. Van Brummelen et al. (2001) demonstrated the steady-iterative approach with this geometry for subcritical flow.

---

\(^1\)k-point moving average: the new value is the average of the point itself and its $(k - 1)$ closest neighbors

---

Fig. 7: Initial and final solution for supercritical flow over bottom topography
The same three coupling algorithms are tested on this case: GS, IQN-ILS and IQN-ILS-low, with the same settings as used for the case with horizontal bottom. Fig. 8a and Fig. 8b show the pressure residuals: the results are very similar to those for the horizontal bottom. Both quasi-Newton methods converge to the limit set by the convergence criteria in the flow solver, but IQN-ILS-low is considerably faster.

4 Conclusion

The use of IQN-ILS and IQN-ILS-low in combination with a simple update method, gives promising results for solving steady free surface flows. Both methods converge well for supercritical 2D flows. There is much room for improvement: better filters for IQN-ILS-low must be designed, inlet and outlet boundary conditions must be improved to decrease appearance of high error frequencies, a better choice of $K^*$ might be found by studying discrete triangle waves and a good method must be found to remove old inaccurate information from the quasi-Newton algorithm. Furthermore, an increase in number of coupling iterations as the mesh is refined is currently observed, which should be avoided.

Currently the bottom topograhpy with subcritical flow (Fr = 0.43) is being investigated, but there are still convergence problems. The steady surface gravity wave mode ($\mu_i = 1$) is not correctly predicted because of incompatibility of the boundary conditions, the main problem being that the constant pressure outlet is not a good assumption anymore. To get realistic results, viscosity and turbulence modelling will also be added in the future, although this is not expected to be much of a problem as there is no immediate link with the free surface update.

References


