6th European Conference on Computational Mechanics (ECCM 6)
7th European Conference on Computational Fluid Dynamics (ECFD 7)
1115 June 2018, Glasgow, UK

NEW TECHNIQUES FOR SOLVING THE STEADY FREE SURFACE FLOW PROBLEM

TOON DEMEESTER[1], E. HARALD VAN BRUMMELEN[2], JORIS DEGROOTE[1,3], JAN VIERENDEELS[1,3]

[1] Department of Flow, Heat and Combustion Mechanics, Ghent University
Sint-Pietersnieuwstraat 41, 9000 Ghent, Belgium
toon.demeester@ugent.be

[2] Department of Multiscale Engineering Fluid Dynamics, Eindhoven University of Technology
PO Box 513-5600 MB Eindhoven, the Netherlands
e.h.v.brummelen@tue.nl

[3] Flanders Make, Belgium

Key words: Free Surface Flow, Fitting Method, Stability Analysis, Quasi-Newton, Multigrid

Abstract. Steady free surface (FS) flows can be solved numerically with capturing or fitting methods, the latter being the subject of this paper. Most fitting methods are (pseudo-)transient and thus quite slow for steady flows; the so-called *steady iterative method* is much faster, but requires a dedicated solver because of the complex FS boundary conditions. The goal is to develop a (currently 2D) fitting method which is fast and can be used with a black box flow solver. Results from a perturbation analysis are used in combination with the IQN-ILS algorithm to construct such a method, applicable to supercritical flows. To tackle this method's scaling problem when the mesh is refined, an extension is proposed which uses a multigrid technique for the surface update. The flow over an object is simulated with the original and multigrid enhanced methods for three meshes. The multigrid method clearly outperforms the original one and is even mesh independent during part of its convergence.

1 INTRODUCTION

Steady free surface (FS) flows of water and air are often encountered in maritime and hydraulic applications, such as flow around ships or in confluences. The behavior of these flows can be simulated numerically (CFD) using either capturing or fitting methods. Capturing methods use a marker to transport the FS through the mesh. Fitting methods deform the mesh along with the FS. Usually they neglect the air phase so that the FS becomes a domain boundary, where the dynamic and kinematic boundary conditions (DBC and KBC) have to be fulfilled. The DBC commonly assumes atmospheric pressure

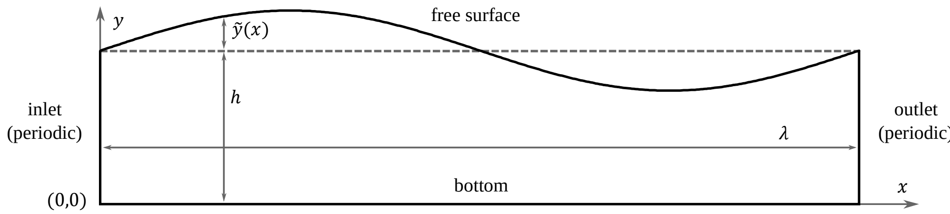

Figure 1: Domain studied in modal analysis.

at the FS boundary. The KBC requires that the FS is impermeable, i.e. zero mass flow rate through the FS.

Fitting methods are iterative and consist of two steps: solution of the flow and update of the FS position. Convergence of the iterations is reached when both the DBC and KBC are fulfilled. Most fitting methods use the DBC in the flow solver and the KBC for the surface update, which leads to a (pseudo-)time-stepping scheme [1]. As a consequence, these methods are inefficient for solving steady flow problems. The so-called *steady iterative method* [2] contains no time derivative and is therefore a truly steady method and much faster. This is achieved by using a combination of the two FS conditions in the flow solver. On the downside, this combined boundary condition requires a dedicated flow solver.

The goal of this research is to develop a fast steady method which can be used with a general purpose black box flow solver. In this paper such a method for 2D supercritical flows is presented and tested.

2 SOLVING SUPERCRITICAL STEADY FREE SURFACE FLOW

To ascertain compatibility with a black box solver, only the KBC will be used in the flow solver. This is done by treating the FS as a free slip wall. The DBC will be used to calculate the surface update, as it contains no time-dependent terms. The consequences of this choice are first studied with a modal analysis. The results are then used to construct an update method. Scaling of this method with respect to the mesh resolution is subsequently improved using a multigrid-like technique.

2.1 Modal analysis

In the first iteration of a fitting method, an initial estimate is made of the free surface position $y(x)$. Most likely the position error $\tilde{y}(x)$ will not be zero, so that the flow solver gives a non-zero pressure error $\tilde{p}(x)$. The position of the FS at the intersection with the inlet is assumed to be correct, which means that it has $\tilde{p} = 0$ by definition, giving a reference point for $\tilde{p}$. Based on $\tilde{p}$ the FS must now be displaced in an attempt to satisfy both boundary conditions. For this purpose a relationship between $\tilde{p}$ and $\tilde{y}$ is required.

Such a relationship can be obtained by doing a modal analysis of the position error $\tilde{y}$ for 2D inviscid flow over a horizontal bottom, as depicted in Fig. 1. Sinusoidal perturbations are denoted as $\tilde{y}_k$ and $\tilde{p}_k$ with $k = 2\pi/\lambda$ the perturbation's wavenumber. This flow has a flat FS as solution, which means a relationship between $\tilde{y}_k$ and $\tilde{p}_k$ can be derived by

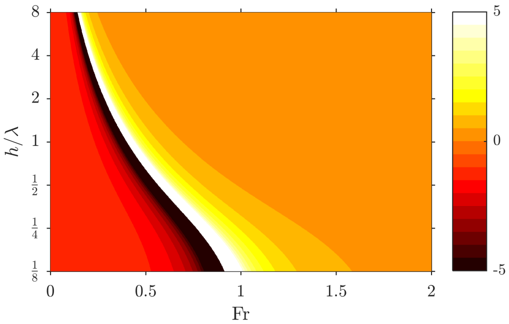

Figure 2: The proportional factor $\rho g K$.

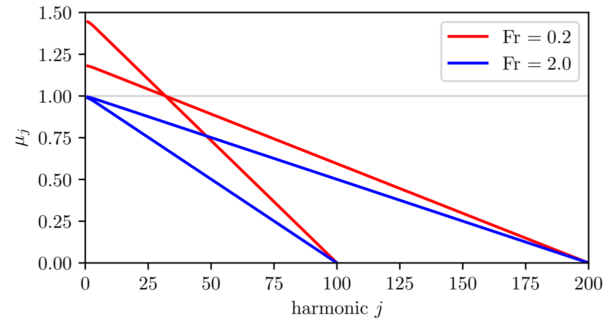

Figure 3: Amplification μ_j for 100 and 200 harmonics.

calculating the flow field with potential flow theory [3]. This leads to the linear relation

$$\tilde{y}_k(x) = K \cdot \tilde{p}_k(x) \qquad \text{with} \qquad K = \frac{1}{\rho g \left(\mathrm{Fr}^2 \frac{kh}{\tanh kh} - 1\right)} \tag{1}$$

where the factor K is a function of the perturbation's wavenumber k and the flow's Froude number $\mathrm{Fr} = U/\sqrt{gh}$. ρ is the density, g the gravitational acceleration, h the undisturbed water depth and U the average flow velocity.

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. The ratio h/λ 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 are dominant so that the flow is governed by the hydrostatic effect. At high Fr, inertial forces dominate the flow through the Bernoulli effect. In Fig. 2, these two regions are clearly present and separated by an asymptote. Close to the asymptote, the opposing effects on $\tilde{p}$ of gravity and inertia 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 and KBC satisfied) and therefore give $K \to \infty$.

The phenomenon of these steady surface gravity waves is problematic for cases where this mode can be present in the error $\tilde{y}$, namely subcritical flows ($\mathrm{Fr} \leq 1$). As this mode gives no response in $\tilde{p}$, there are infinitely many solutions y which fulfill both KBC and DBC. Additional conditions have to be added to get a unique solution, e.g. requiring that the FS is flat close to the inlet. The currently proposed method does not yet include these additional conditions, so for now it can only be used for supercritical flows ($\mathrm{Fr} > 1$).

2.2 Quasi-Newton iterative method

The DBC requires that the pressure p at the FS is equal to the atmospheric pressure, or analogously that

$$\tilde{p}(x) = 0. \tag{2}$$

The solution to this system can be found with Newton's method, if it converges. Iteration n –denoted with a superscript– has the form

$$\tilde{p}^n + \frac{\partial \tilde{p}}{\partial y}\Delta y^n = 0 \qquad \text{with} \qquad \Delta y^n = y^{n+1} - y^n \tag{3}$$

and $\partial\tilde{p}/\partial y$ the Jacobian of the system. This gives a way to calculate the new position y^{n+1}, which is evaluated in the flow solver to get the next error $\tilde{p}^{n+1}$. Because the flow solver is a black box, its Jacobian is unknown and must be approximated, so that the iteration is in fact a quasi-Newton method. Alternatively the Jacobian's inverse $\partial y/\partial\tilde{p}$ can be approximated, so that

$$y^{n+1} = y^n - \widehat{\frac{\partial y}{\partial \tilde{p}}}\,\tilde{p}^n. \tag{4}$$

A simple approximation is

$$\widehat{\frac{\partial y}{\partial \tilde{p}}} = K^* I \tag{5}$$

with I the identity matrix and K^* from Eq. (1) using k^*, the highest wavenumber on the given mesh. The stability of Newton iterations with this simple model can be investigated by looking at the amplification factor μ_j of a Fourier-mode $\tilde{y}_j$ present in the error:

$$\mu_j = \frac{\tilde{y}_j^{n+1}}{\tilde{y}_j^n} = \frac{\tilde{y}_j^n - K^*\,\tilde{p}_j^n}{\tilde{y}_j^n} = 1 - \frac{K^*}{K_j}. \tag{6}$$

Eq. (1) is used in the last step. For supercritical flows, K_j and K^* are both positive and $K^* \leq K_j$ by definition, so that $0 \leq \mu < 1$ and all modes are stable. For subcritical flows, μ_j can be bigger than one for some low wavenumber modes, which are then unstable. Fig. 3 confirms these properties. It shows μ_j for uniform meshes with 100 and 200 harmonics, for a subcritical flow (Fr = 2.0) and a supercritical one (Fr = 0.2). The base mode has $h/\lambda = 1/8$.

The approximation of the Jacobian may be improved with known input-output pairs $(y, \tilde{p})$ of the black box flow solver. Differences between consecutive iterations are defined as

$$\Delta y^n = y^{n+1} - y^n \qquad \text{and} \qquad \Delta\tilde{p}^n = \tilde{p}^{n+1} - \tilde{p}^n \tag{7}$$

and can be collected in matrices

$$W = \left[\Delta y^{n-1} \cdots \Delta y^0\right] \qquad \text{and} \qquad V = \left[\Delta\tilde{p}^{n-1} \cdots \Delta\tilde{p}^0\right]. \tag{8}$$

Using these matrices, $\partial y/\partial\tilde{p}$ can be approximated with the IQN-ILS[1] technique [4]. This technique is used for partitioned fluid-structure interaction (FSI) problems with black box solvers. It stabilizes the coupling iterations by correcting unstable modes ($|\mu_j| > 1$), and

[1] Interface Quasi-Newton with Inverse Jacobian from a Least-Squares model

it accelerates convergence of badly damped modes ($|\mu_j|$ close to 1). It approximates the (inverse) Jacobian as

$$\widehat{\frac{\partial y}{\partial \tilde{p}}} = WR^{-1}Q^T \qquad \text{with} \qquad V = QR. \tag{9}$$

After every iteration, the V and W matrices grow and the approximation improves. It can be seen that this Jacobian only affects the part of $\tilde{p}^n \in \text{span}(V)$. For the remaining part of $\tilde{p}^n$ the simple approximation from Eq. (5) can be used. Splitting up $\tilde{p}^n$ using complementary orthogonal projectors QQ^T and $I - QQ^T$, the Jacobian can be approximated as

$$\widehat{\frac{\partial y}{\partial \tilde{p}}} = WR^{-1}Q^T + K^* \left(I - QQ^T\right). \tag{10}$$

In the remainder of this paper, this FS update method will be referred to as IQN.

3 MULTIGRID-LIKE TECHNIQUE

3.1 The scaling problem

The amplification factor μ_j for supercritical flow is approximately linear, with μ_j close to 1 for the lowest frequency and $\mu_j = 0$ for the highest one (see Fig. 3). This means that the fraction of modes which are badly damped is independent of the mesh resolution. In other words: when the mesh is refined, the number of difficult modes increases at the same rate as the number of unknowns. The number of coupling iterations will increase at the same rate as only one mode is added to the IQN-ILS model of the Jacobian during each iteration. This means the current IQN method has bad scaling behavior for this problem. By contrast, the number of coupling iterations is typically independent of the mesh refinement when applied to normal FSI problems [4].

The fact that low frequency errors are badly damped and high frequency errors disappear quickly brings to mind the use of a multigrid method, except that the problem now concerns coupling iterations of a black-box flow solver. It would be impractical to change the mesh used in the flow solver every iteration, so the multigrid principle should only be used in the surface update.

A multigrid-like approach for the IQN method from Section 2.2 is now sketched. In each iteration a grid is chosen for the surface update, based on the modal content of the pressure error $\tilde{p}$. The inputs for the IQN method (namely $\tilde{p}$, y and the matrices V and W) are restricted to this grid. The highest possible frequency on this grid will be (much) lower than on the original mesh, so that a higher value K^* can be chosen. The higher modes on this grid damp out easily, the lower ones are effectively dealt with by the IQN-ILS algorithm. The update Δy is then interpolated back to the original mesh and passed to the flow solver.

3.2 Multigrid enhanced IQN

Implementing a multigrid enhanced IQN method (MG-IQN) requires procedures to choose the different free surface grids and for the restriction/interpolation of variables to

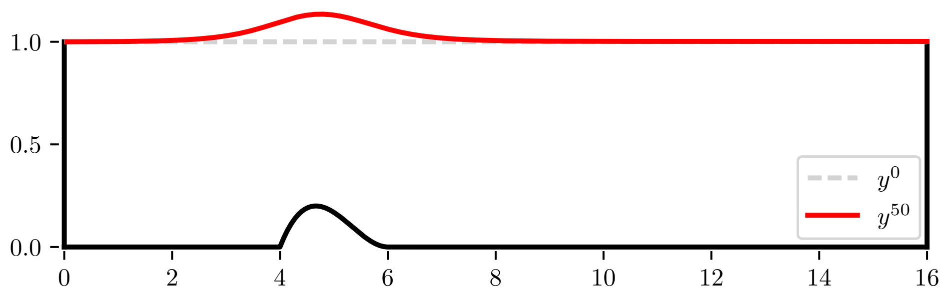

Figure 4: Initial position y^0 and final solution y^{50} for supercritical flow over bottom topography.

coarser grids.

Assuming that the computational cost of the surface update is negligible with respect to that of the flow solver, the coarse grid iterations are not cheaper as opposed to classical multigrid methods as only the free surface grid is coarsened and not the grid in the fluid domain. For that reason, it was decided not to make use of the common V-cycles or W-cycles for switching between grids. Instead, in each iteration that grid should be chosen which will reduce the residual the most (obviously, this is no triviality). A method is proposed which is likely not optimal, but nevertheless gave much faster convergence than V-cycles. The method chooses a grid from a list of allowed grids (see Section 4) based on $\tilde{p}$. On each grid, $\tilde{p}$ is restricted, then interpolated back and compared to the original $\tilde{p}$. If the difference is larger than a given threshold, the grid is removed from the list of choices, to make sure that a fine enough grid is used when high frequencies are dominant. The coarsest remaining grid is then chosen.

The restriction and interpolation steps are both done with cubic spline interpolation. In the restriction operation the signal is filtered first, so that high frequencies (which cannot be represented on the coarse grid) have no influence on the spline interpolation. The implemented filter is a combination of Gaussian windows, which gives a sharp frequency cut-off without overshoot. Gaussian windows were used because the frequency response of an infinite continuous window is known and easily approximated by a finite discrete window. The cut-off frequency (signal's amplitude reduced to 10%) is chosen to coincide with the smallest wavelength on the coarse grid.

4 NUMERICAL EXPERIMENTS

The IQN and MG-IQN methods are compared for the academic test case of 2D flow over an obstacle as shown in Fig. 4. Experimental work was done by Cahouet [5], and the data was used in several papers for evaluating surface fitting methods [1, 2]. The shape of the obstacle is described by

$$y_{obst}(x) = \frac{27}{4}\frac{H}{L^3}(x-x_0)(x-x_0-L)^2\ ,\ 0 \leq x-x_0 \leq L \tag{11}$$

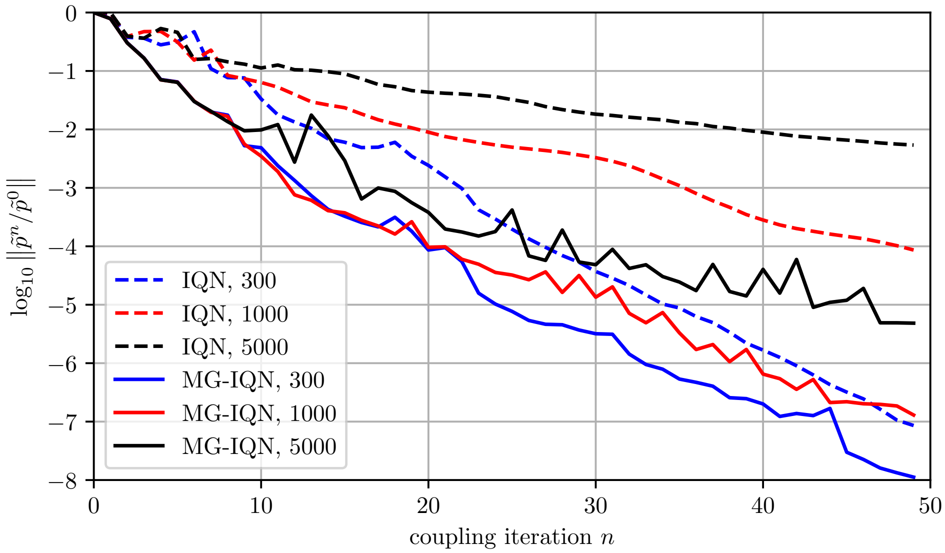

Figure 5: Convergence of IQN and MG-IQN on three meshes. The number in the legend refers to te number of intervals in the FS grid.

with $x_0 = 4$ the starting position, $H = 0.2$ the height and $L = 2$ the length of the obstacle. The Froude number is 2.05 and determines the inlet velocity. Boundary conditions are a uniform velocity inlet, a hydrostatic pressure outlet and a free-slip wall for the FS. To check the scaling behavior of the surface update methods, three uniform rectangular meshes are used with respectively 300, 1000 and 5000 cells in the x-direction and 100 cells in the y-direction. The results shown further on were obtained with ANSYS Fluent, but the model works just as well with OpenFOAM[2]. To assess convergence, only the pressure error at the FS is monitored. To speed up calculations, viscous forces are not taken into account. The initial position y^0 is a horizontal FS.

As the flow solver mesh is uniform and rectangular, it is straightforward to define the coarser FS grids used in MG-IQN: these are also uniform and defined by their number of cells in x-direction. The coarsest grid has 40 cells, the finest one is the original mesh. The grids in between change with approximately a factor 2, so that the meshes with 300, 1000 and 5000 FS cells have respectively 4, 6 and 8 grid-levels[3].

Fig. 5 summarizes the simulation results. The dashed lines show the convergence of the normalized pressure error for the three meshes with IQN, the continuous lines with MG-IQN.

The scaling behavior can clearly be seen for IQN: the convergence rate decreases fast when the mesh is refined. For the coarsest mesh, IQN and MG-IQN converge at the same rate, the only advantage of MG-IQN is that convergence is faster in the first few

[2]Switching between Fluent and OpenFOAM is done by changing a single line in the code. Because of data transfer issues, the surface update in OpenFOAM does not work yet in parallel. Therefore, Fluent was used for this paper.

[3]In practice, the first level (original mesh) is never used, but a correction is done at the end of every iteration for the highest frequencies using Eq. (5).

iterations, as $\tilde{p}$ contains mainly low modes then. Later on the error modes are spread out more uniformly over the frequency domain and this advantage disappears. When the mesh is refined, MG-IQN performs much better than IQN: the number of coupling iterations is mesh independent for the first part of the simulation. When $\tilde{p}$ becomes very small convergence slows down for the finest mesh. The more jagged convergence behavior of MG-IQN is probably due to the constant grid switching and its effect on the IQN-ILS Jacobian approximation.

5 CONCLUSIONS

A straightforward FS fitting method –dubbed IQN– has been derived for 2D supercritical flows, based on a modal analysis and the IQN-ILS method for approximating the Jacobian of a black box flow solver. The IQN method inherently has bad scaling behavior when the mesh is refined for this problem, while the number of iterations is typically independent of the grid refinement when applied to FSI problems. As a solution MG-IQN is proposed, a multigrid-like extension of IQN. Numerical simulations show that this idea definitely has merit, although its exact implementation may not yet be optimal and subject to change in the future.

REFERENCES

[1] G. Tzabiras, "A numerical investigation of 2D, steady free surface flows," *International Journal for Numerical Methods in Fluids*, vol. 25, no. 5, pp. 567–598, 1997.

[2] E. Van Brummelen, H. Raven, and B. Koren, "Efficient numerical solution of steady free-surface Navier–Stokes flow," *Journal of Computational Physics*, vol. 174, no. 1, pp. 120–137, 2001.

[3] T. Demeester, J. Degroote, and J. Vierendeels, "Stability analysis of a partitioned iterative method for steady free surface flow," *Journal of Computational Physics*, 2017.

[4] J. Degroote, K.-J. Bathe, and J. Vierendeels, "Performance of a new partitioned procedure versus a monolithic procedure in fluid–structure interaction," *Computers & Structures*, vol. 87, no. 11, pp. 793–801, 2009.

[5] J. Cahouet, *Etude numérique et experimentale du probleme bidimensionnel de la résistance de vagues non-linéaire.* Ecole nationale supérieure de techniques avançées, 1984.