Exploiting the limit of BEM solvers in moonpool type floaters

Dimitris Ntouras¹, Dimitris Manolas²,³, George Papadakis¹, Vasilis Riziotis³

¹ School of Naval Architecture & Marine Engineering, National Technical University of Athens (NTUA), GR15780 Athens, Greece
² iWind Renewables Private Company, GR15344 Athens, Greece
³ School of Mechanical Engineering, National Technical University of Athens (NTUA), GR15780 Athens, Greece

Author contact email: ntourasd@fluid.mech.ntua.gr

Abstract. Solvers based on Boundary Element Method (BEM) are fast and they have proven to provide accurate results in a wide range of applications. On the other hand, computational fluid dynamics (CFD) solvers are high-fidelity tools able to account for viscous effects. However, they are computationally demanding. In the present work, the limitations of BEM solvers are exploited considering the case study of a moonpool type floater in which the viscous effects near the sharp edges of the body (vortex shedding) are not negligible. The BEM results are compared with the results from an unsteady Reynolds averaged Navier-Stokes (URANS) CFD solver and experimental data, while viscous corrections of the BEM method are assessed.

1. Introduction

Offshore deployment of Wind Turbines (WT) is becoming more popular as WT rotors increase in size. Alongside the expansion of the offshore wind sector, the need for cost-effective support structures emerges. During the last decade, a number of offshore floating concepts (i.e. spar-buoy, semisubmersible and TLPs) have been proposed, analysed and tested with the aim to qualify the optimum solution for high sea depths. Among those, moonpool type floaters are one of the options that have attracted the attention of WT designers. The moonpool type floaters are defined as floating bodies with an opening in the middle of the structure. The free water surface inside the body is subjected to motion, caused by the interaction of the structure with the waves and the sea currents. The moonpool concept was originally introduced in sailing vessels for safer access to the water and since its first use has been carefully studied. Recently, the moonpool concept has been considered as a foundation option for offshore Wind Turbines that enhances stability of the structure against the wave excitation. The free water surface inside the moonpool is subjected to a vertical motion commonly referred to as piston mode, while additional sloshing modes appear. Near the piston mode resonance, the free surface elevation inside the moonpool is significantly damped due to, mainly, the vortex shedding in the lower entrance of the moonpool. To design this type of floaters and to accurately predict their response, care must be taken to account for the viscous effects of the flow.

The use of the BEM has become an irreplaceable tool in the early stages of the design process, due to its capability to provide fast results and also its reliability in a wide range of applications. It is known that the BEM method, when applied to the moonpool type floaters, overpredicts the amplitude of the piston mode in certain wave frequencies (i.e. near the natural frequency of the piston mode - see [1,2]). In the aforementioned works, artificial damping terms have been introduced in the flow equations to control this behaviour.
In the present work, the application of a BEM solver for designing purposes of moonpool type floaters is assessed in comparison to high fidelity URANS CFD simulations. The case study addressed in the paper is supported by experimental data published in [3]. Moreover, viscous correction modelling in BEM solver is assessed, by introducing viscous damping terms in the free surface dynamic equation (inside the moonpool) and in the equation of motion of the floater.

2. Numerical Framework

The two solvers used in this work are presented in the following sections.

2.1. Boundary Element Method – hFlow solver

hFLOW [4] is a fully nonlinear, potential, two dimensional solver based on BEM. The solver uses linear singularity distributions on planar elements and it is based on the mixed Eulerian-Lagrangian formulation of the free surface equations. Wave generation is implemented along the inflow boundary by imposing the stream function wave solution, while wave absorption at both ends of the domain exclusively relies on the application of absorbing layers. The simulation of free-floating bodies is performed using the so-called iterative method (see [4]). A linear BEM approach is also available by considering the linearized equations on the free surface and the BEM geometry fixed (both free surface and body). To account for viscous effects due to piston mode resonance inside the moonpool, an artificial, nonlinear damping term is introduced in the dynamic free surface equation inside the moonpool [1, 2]. The abovementioned modified equation is,

$$\frac{\partial \phi}{\partial t} + \frac{1}{2} |\nabla \phi|^2 + g \zeta = -\frac{1}{2} \xi \frac{\partial^2 \phi}{\partial t^2}$$

where the damping term appears in the right hand side. In the above equation, $\phi$ denotes the velocity potential, $g$ is the acceleration due to gravity, $\zeta$ and $\xi$ denote the surface elevation and its first time derivative respectively and $\xi$ is the damping coefficient. In the present work $\xi=1.2$, as defined in [1].

Furthermore, in the floating case, a quadratic damping term is added in the equation of motion of the floater. The quadratic term resembles the viscous drag term that appears in Morison’s equation. It was found to consistently capture the quadratic dependence of the viscous damping on the wave height, as opposed to a linear damping term that was initially tested.

2.2. Computation Fluid Dynamics – MaPFlow Solver

The incompressible equations are solved using the artificial compressibility formulation [5], while for the two phase flows an additional equation of the volume of fluid (VOF) [6] is added to the coupled system of equations. The governing equations can be written in the following integral form.

$$\Gamma \int_{\Omega} \frac{\partial \bar{Q}}{\partial t} \, d\Omega + \Gamma \int_{\Omega} \frac{\partial \bar{Q}}{\partial t} \, d\Omega + \int_{\Omega} \left( F_v - \bar{F}_v \right) \, dS = \int_{\Omega} \bar{S}_q \, d\Omega$$

In the above system of equations, the unsteady equations are augmented by the fictitious time derive. For every physical time-step, a pseudo-steady problem is solved until the fictitious time derivatives become zero and convergence is accomplished. The vector $\bar{Q} = [p, \bar{u}, a]^T$ denotes the primitive variables, which contains the pressure, the velocity vector and the volume fraction. The real and the fictitious time are denoted by $t$ and $\tau$ respectively. The inviscid fluxes are denoted by $F_v$, while $\bar{F}_v$ are the viscous fluxes and $\bar{S}_q$ are the source terms. The above system of equations is closed with the artificial compressibility parameter $\beta$, where it is assumed that there is a relation between density and pressure in pseudo-time $\beta = \frac{\partial \rho}{\partial p}$. 

Although the system of equations is casted in primitive form, for unsteady simulations the conservative variables, $\bar{U} = [0, \rho \bar{u}, \bar{a}]^T$, are used to advance the solution in time. For this purpose, the
The governing equations are discretized using the finite volume method. The discretized form of the system (2), for a given control volume $\Omega_j$, is written as

$$\Gamma_e \frac{\partial (\bar{\Omega}_j)}{\partial t} + \Gamma_{ei} \frac{\partial \bar{\Omega}}{\partial r} = -\bar{R}_\alpha,$$

The residual of the equations $\bar{R}_\alpha$ is computed as a sum of the fluxes over the interfaces of the control volume with the addition of the source terms $\bar{S}_d$, which are considered to be constant in $\Omega_j$.

$$\bar{R}_\alpha \simeq \sum_j \left( \bar{F}_j - \bar{F}_j \right) \Delta S_j + \Omega_j \bar{S}_d$$

The inviscid fluxes are evaluated by solving the preconditioned local Riemann problem, between the neighbors of an interface using Roe’s approximate Riemann solver [8], [9]. For the reconstruction of the velocity field a piecewise linear interpolation scheme is used. For the pressure field a similar methodology is followed, but due to the discontinuity of the pressure gradient at the free surface, a density weighted interpolation scheme is adopted [10]. Furthermore, for the reconstruction of the VOF field a high order interface capturing scheme is used [11], in order to reduce the numerical diffusion near the free surface.

For the turbulence modeling the two equations model k-ω SST of Menter is used [12], while a buoyancy term is added to the kinetic energy equation to reduce the overproduction of the turbulent viscosity at the free surface [13]. In order to account for flows with moving objects that perform small amplitude motions MaPFlow implements the technique of the deforming grids [14]. For unsteady simulations, an implicit second order backwards difference scheme is used [15], in combination with a dual time stepping technique in order to facilitate convergence. Finally, for the inversion of the implicit operator, the Gauss-Seidel iterative method is used combined with the Reverse Cuthill-McKee reordering scheme.

The above methodology was implemented as an extension in the compressible solver MaPFlow [16–18]. MaPFlow solves the Unsteady Reynolds Averaged Navier Stokes equations on unstructured grids, in a multiprocessor environment utilizing the MPI protocol and it is an in-house NTUA code.

2.3. Dynamic Analysis
The fluid structure interaction problem consists of solving the rigid-body motion equations in each timestep. The equations of motion are given by the Newton’s second law and have the following form.

$$m \ddot{\eta}_j = F_j; \quad m \ddot{\eta}_3 = F_z; \quad I \ddot{\eta}_4 = M_z,$$

In the above equations, $\eta_2$ is the translation in the x-direction (sway), $\eta_3$ is the translation in the z-direction (heave) and $\eta_4$ the rotation around the y-axis (roll). The excitations of the system consist of a hydrodynamic component, which is evaluated from the solver, as well as a stiffness and a damping
part, which result from the physical parameters of the mooring system (flexibility and damping). In the present work, the modeling described in [14] is adopted.

3. Results
In the first part of this section, the computational setup is presented, as well as a parametric study for the grid and time step resolution of the CFD solver. In the second part, a fixed body analysis is described. The results of the BEM solver are assessed, with and without viscous correction, and they are compared against the results of the CFD analyses. The last subsection addresses the floating body case study and the two solvers are compared against experimental data from [3].

3.1. Computation Setup
The computational setup follows the experiments presented in [3]. Two barges with a rectangular cross section of 20cm and breadth of 56.8cm form a moonpool opening of 10cm. The draft of the model is 9.7cm, the wave tank has constant depth of 1m and two horizontal mooring lines prevent the model from drifting as illustrated in Figure 1. The present study considers two types of waves, defined based on the wave height-to-wavelength steepness ratios: \( H/\lambda = 1/60 \) and 1/30. The wave periods vary from 0.5s to 1.2s. Wave generation is performed in the left side of the computational domain based on the analytical solution of the incoming wave, while damping zones are defined at both ends to avoid reflections from the boundary wall.

Regarding the CFD solver, in order to ensure solution independence to grid and time step parameters, a sensitivity study is conducted. Three different grids are considered M1, M2 and M3 that consist of 90k, 200k and 390k cells, respectively. The left side of the computational mesh is almost uniform (from the wave generator to the left side of the floater), while towards the right side, coarsening of the grid is performed following a geometric rule (from the right side of the body to the damping zone). In addition, in order to accurately capture the viscous effects close to the body (flow separation, vortex formation), the unstructured mesh, in a region close to the body, has a constant characteristic length. Three rules are adopted for the mesh refinement, first the number of cells per wave length in the left side of the domain, second the number of cells per wave height, in the same region, and lastly the characteristic length of the mesh around the body.

In Figure 2 the signal of the wave elevation inside the moonpool is presented for the three meshes. The wave steepness of the incident wave is \( H/L=1/60 \) and the wave period equals to 0.88s. The amplitude of the piston mode in the case of the M3 mesh is smaller and a phase shift can also be observed. The solutions obtained from the other two meshes have a slight difference in amplitude, but it is not considered to be significant. In Figure 3, for the same test case, the time step sensitivity study is presented for the M2 mesh. The larger time step (\( dt=0.002s \)) also appears to have a phase shift compared with the smaller time steps which are in very good agreement. In conclusion, the M2 mesh with time step of 0.001s is adopted for all cases, for both the fixed and the floating body case. In BEM, 75 nodes per wavelength have been used on the free surface boundary and 10 nodes inside the moonpool, while 50 time steps per wave period are considered.

![Figure 1. Schematic representation of the moonpool case study.](image-url)
In CFD simulations, either laminar or turbulent conditions may be considered. They mainly depend on the amplitude of the motion inside the moonpool. For small amplitudes of motion, separation occurs but it is rather limited, while for high motion amplitudes the separation is larger whereas vorticity is shed from the corners of the structure. It is worth noting that in the amplitude regime where the moonpool motion is small, laminar and turbulent simulations yield very similar results. Consequently, we decided to consider the flow fully turbulent also because the model has sharp edges and so separation bubbles will appear at these regions.

3.2. Fixed Body Case Study

Initially, the performance of the BEM solver is assessed when the body is considered to be fixed for both BEM formulations (linear and nonlinear), without applying any viscous correction terms. In Figure 4, the non-dimensional amplitude of the 1st harmonic of the surface elevation inside the moonpool is compared against the CFD results. The natural period of the piston mode is \(~0.88\)s. As
expected, BEM results overpredict the surface elevation inside the moon pool near the critical wave frequencies. In addition, CFD results indicate a nonlinear dependence of the non-dimensional elevation inside the moonpool on the wave steepness.

Figure 5 shows a snapshot of the velocity magnitude, for the case the wave steepness is equal to $H/\lambda = 1/30$ and the wave period 0.88 s (piston mode resonance). It can be seen that flow separation at both edges inside the moonpool is significant. The BEM solver produces erroneous results at the critical frequencies due to the absence of the viscous effects that characterize the flow. For wave frequencies close to the piston mode resonance the motion of the free surface is directly affected by the flow separation at the sharp edges of the body.

In order to regulate the BEM behavior near the resonant frequencies, the damping term that appears in equation (1) is added to the dynamic boundary condition of the free surface. In Figure 6 the results with viscous correction are presented. The amplitude in resonant states has been significantly reduced and it is found to be in good agreement with the CFD results, while the behavior of the BEM solver outside the resonance region remains unchanged.

![Figure 5](image_url)

Figure 5. Contour of velocity magnitude during the upstroke of the piston motion. Flow separation is visible at the edges of the two barges. Case: $H/\lambda = 1/30$ and $T=0.88s$ for the fixed body test case.

![Figure 6](image_url)

Figure 6. Non-dimensional amplitude of the 1st harmonic of the surface elevation inside the moonpool. Comparison between the corrected BEM (with viscous correction $\xi=1.2$ [1,2]) and CFD results for the fixed body. Comparison of the CFD results and modified BEM.
Since the viscous correction term is a non-linear one that depends on the time derivative of the free surface elevation, the linear model produces different results for the two wave steepnesses considered ($H/\lambda=1/30$ and $H/\lambda=1/60$). Furthermore, the linear and non-linear BEM formulations appear to produce almost identical results, indicating that close to resonance the results are driven primarily by the viscous correction term.

Figure 7 presents the amplitude of the 1st harmonic of the vertical hydrodynamic force on the structure as predicted by the two solvers. It is seen that the predictions of the linear BEM solver with viscous correction are similar to those obtained with the CFD solver for both wave heights.

3.3. Floating Body Case Study
In this section, the free-floating case is considered. Similar to the previous section, two wave steepnesses are considered, $H/\lambda=1/60$ and $H/\lambda=1/30$. In two dimensions, the rigid body has three degrees of freedom, two translatory ($\eta_2$: sway, $\eta_3$: heave) and one rotational ($\eta_4$: roll). Results are compared against experimental data from [3].

Figure 8 to Figure 13 present the Response Amplitude Operators (RAO’s) of the floater. Comparisons between the linear BEM, both corrected BEM formulations (linear and nonlinear), CFD and experimental data are shown for $H/\lambda=1/60$ (left plots) and $H/\lambda=1/30$ (right plots).

The BEM solver without viscous correction is able to predict the motions of the floater in frequencies far from resonance. Near the critical frequencies, where higher amplitudes of the motion degrees of freedom are obtained, viscous effects have a significant impact on the response of the floater and the baseline BEM overpredicts the motions of the floater. (i.e. heave motion at $T\approx0.75$ and roll motion at $T\approx0.95$).

By applying the viscous correction model in the dynamic boundary condition of the free surface and including the quadratic damping terms in the equations of motion, the BEM model accurately predicts the amplitudes of heave and roll, but not of the sway motion (see Figure 8 and Figure 9).

Regarding the CFD results, the amplitude of the sway motion shows a good agreement with the experiments. The amplitude of the heave motion is slightly underpredicted at $T=0.7s$ and $0.75s$ and the amplitude of the roll motion at $T=0.9s$ and $0.95s$. This slight underprediction of the amplitudes can be attributed to the assumption of the fully turbulent flow, due to which higher hydrodynamic damping effects may be induced. Despite the abovementioned localized discrepancies, the overall agreement of the CFD results with the measurements is very good.
Figure 8. RAOs in sway direction for H/L=1/60

Figure 9. RAOs in sway direction for H/L=1/30

Figure 10. RAOs in heave direction for H/L=1/60

Figure 11. RAOs in heave direction for H/L=1/30

Figure 12. RAOs in roll direction for H/L=1/60

Figure 13. RAOs in roll direction for H/L=1/30
In Figure 14, the non-dimensional amplitude of the 1\textsuperscript{st} harmonic of the surface elevation inside the moonpool is presented. Similar to the fixed body test case, the surface amplitude in the critical frequencies regime is overestimated by the BEM solver. By applying the viscous corrections, the amplitude is reduced, but remains higher than that predicted by the CFD tool.

4. Conclusions
In the present paper the prediction capabilities of a BEM solver are assessed in a moonpool type floater test case. The potential BEM solver provides accurate predictions in a range of wave frequencies where the viscous effect can be neglected. However, in the critical wave frequencies near resonance, viscous effects primarily due the vortex shedding at the sharp edges of the floater, are dominant and the BEM solver provides conservative results (much higher amplitudes of motion). Application of viscous corrections in the BEM code mitigates the above effect as indicated by the comparisons against the CFD results and experimental data. For the fixed body test case a nonlinear damping in the free surface dynamic equation inside the moonpool is only required. In this case the methodology of [1, 2] provides the theoretical basis for estimating the damping coefficient without any tuning. The same coefficient is considered in the free-floating case but additional quadratic drag terms are added in the equation of motion of the floater to obtain accurate predictions near resonance. In the latter case the parameters of the viscous damping are not known a priori. Tuning needs to be performed in accordance with the results provided by a viscous CFD simulation or appropriate measured datasets.

Acknowledgements
MaPFlow computations were supported by computational time granted from the Greek Research & Technology Network (GRNET) in the National HPC facility - ARIS - under project "WAVEMAP" with ID pr008051. The grids for MaPFlow computations are generated using ANSA meshing software granted from BETA CAE Systems.
References

[1] Faltinsen OM, Timokha AN. On damping of two-dimensional piston-mode sloshing in a rectangular moonpool under forced heave motions. 2015 *J. Fluid Mech.* 772 R1

[2] Tan L, Lu L, Tang GQ, Cheng L, Chen XB. A viscous damping model for piston mode resonance. 2019 *J. Fluid Mech.* 510 pp. 510-533

[3] Fredriksen AG, Kristiansen T, Faltinsen OM. Wave-induced response of a floating two-dimensional body with a moonpool. 2015 *Philos Trans R Soc A Math Phys Eng Sci.* 373.

[4] Manolas D. Hydro-aero-elastic analysis of offshore wind turbines. 2015 PhD Dissertation National Technical University of Athens

[5] Chorin AJ. A numerical method for solving incompressible viscous flow problems. 1967 *J. of Comput. Phys.* pp. 12–26

[6] Hirt CW, Nichols BD. Volume of fluid (VOF) method for the dynamics of free boundaries. 1981 *J. Comput Phys.* 39 pp. 201–225

[7] Kunz RF, Boger DA, Stinebring DR, Chyczewski TS, Lindau JW, Gibeling HJ et al. A preconditioned Navier-Stokes method for two-phase flows with application to cavitation prediction. 2000 *Computer & Fluids* 29 pp. 849-875.

[8] Roe PL. Approximate Riemann solvers, parameter vectors, and difference schemes. 1981 *J. Comput. Phys.* 43 pp. 357–372.

[9] Parameswaran S, Mandal JC. A novel Roe solver for incompressible two-phase flow problems. 2019 *J. Comput. Phys.* 390 pp. 405–424

[10] Queutey P, Visonneau M. An interface capturing method for free-surface hydrodynamic flows 2007 *J. Comput. Fluids* 36 pp.1481–1510.

[11] Darwish M, Moukaled F. Convective Schemes for Capturing Interfaces of Free-Surface Flows on Unstructured Grids 2006. *Numer. Heat Transf. Part B* 49 pp. 19–42.

[12] Menter FR. Two-equation eddy-viscosity turbulence models for engineering applications. 1994 *AIAA J.* 32 8 pp. 1598–1605.

[13] Devolder B, Rauwoens P and Troch P. Application of a buoyancy-modified k-ω SST turbulence model to simulate wave run-up around a monopile subjected to regular waves using OpenFOAM®. 2017 *Coast Eng* 125 pp. 81–94

[14] Zhao Y, Tai J, Ahmed F. Simulation of micro flows with moving boundaries using high-order upwind FV method on unstructured grids 2002 *Comput. Mech.* 28 pp.66-75

[15] Biedron R, Vatsa V and Atkins H. Simulation of Unsteady Flows Using an Unstructured Navier-Stokes Solver on Moving and Stationary Grids. 2005 23rd *AIAA Appl. Aerodyn. Conf.*

[16] Papadakis G, Voutsinas SG. A strongly coupled Eulerian Lagrangian method verified in 2D external compressible flows. 2019 *Computers & Fluids* 195

[17] Diakakis K, Papadakis G and Voutsinas SG. Assessment of transition modeling for high Reynolds flows. *Aerosp Sci Technol* 85 pp.416-428

[18] Papadakis G, Filippas ES, Ntouras D and Belibassakis KA. Effects of viscosity and nonlinearity on 3D flapping-foil thruster for marine applications. 2019 *OCEANS 2019 - Marseille*, Marseille, France pp. 1-10