

See discussions, stats, and author profiles for this publication at: <https://www.researchgate.net/publication/304398831>

# Computational Fluid Dynamics (CFD) Models

Chapter · December 2016

DOI: 10.1016/B978-0-12-803210-7.00006-2

CITATIONS

9

READS

2,577

3 authors, including:



[Ling Qian](#)

Manchester Metropolitan University

83 PUBLICATIONS 956 CITATIONS

[SEE PROFILE](#)



[D.M. Causon](#)

Manchester Metropolitan University

145 PUBLICATIONS 3,764 CITATIONS

[SEE PROFILE](#)

Some of the authors of this publication are also working on these related projects:



Extreme Loading on FOWT under Complex Environmental Conditions [View project](#)



VOWS - Violent Overtopping by Waves at Seawalls [View project](#)

## Chapter 7 CFD models - Clive Mingham, Ling Qian and Derek Causon

### 7.1 Introduction and fundamental principles

As computers have become ever more powerful the discipline of computational fluid dynamics (CFD) has become an increasingly viable numerical approach for simulating the dynamic behaviour of wave energy converters (WECs). In principle CFD can be used to study the design of a particular WEC, conduct parametric studies to optimise its performance and investigate wave loadings to characterise its survivability in extreme seas. Given enough compute power CFD can also simulate the performance of arrays of WECs.

In CFD the underlying physical laws describing fluid flow (e.g. conservation of mass, momentum etc.) are expressed mathematically as a system of partial differential equations (PDEs). Classically, these equations are the well known Navier-Stokes equations together with the Continuity equation. To complete the mathematical model, initial conditions are given together with the internal and external boundary conditions needed to define a WEC geometry, bathymetry and the wave field. In general, the describing system of PDEs cannot be solved analytically so approximate solutions are obtained via numerical algorithms (called solvers) implemented on digital computers. These solutions are the values of the relevant dependent variables (e.g. pressure, velocity etc.) at discrete spatial points in the computational domain and at discrete times as the simulation progresses. In principle, solutions can be found to any specified degree of accuracy at specified points in space and time depending upon the formal accuracy of the underlying numerical algorithm or discretisation scheme.

CFD methods can be separated broadly into two categories: Eulerian and Lagrangian. In the Eulerian approach the computational domain is discretised by a finite set of points called a grid (or mesh) and the approximate solution is computed at these grid points. These approximations can be carried out in several different ways depending on the underlying mathematical theory. Two popular approaches are the Finite Difference Method (FDM) and the Finite Volume Method (FVM). The FDM is based on Taylor theory to approximate partial derivatives at the grid points. An introduction to the FDM can be found in Causon and Mingham (2010). The FVM uses the Gauss Divergence Theorem to express spatial partial derivatives as surface integrals. In the FVM the spatial grid is viewed as a set of cells and the approximate numerical solutions are usually but not necessarily computed at the centroid of each cell. An introduction to the FVM can be found in Causon et al. (2011). The majority of Eulerian CFD methods applied to WEC simulation are based on the FVM because of its geometric flexibility and physically consistent treatment of the flow across cell boundaries.

In the Lagrangian approach the computational region is discretised by a set of particles which move at the local flow velocity and approximate solutions are computed at the position of each particle at each discrete time. Amongst several Lagrangian methods the Smooth Particle Hydrodynamics (SPH) method in its various forms is becoming popular although it is not yet a mature technology. The interested reader may find the SPHERIC website a useful SPH resource and this can be found at, [https://wiki.manchester.ac.uk/spheric/index.php/SPHERIC\\_Home\\_Page](https://wiki.manchester.ac.uk/spheric/index.php/SPHERIC_Home_Page).

As the authors' expertise is primarily in the FVM we will concentrate mainly on this Eulerian CFD method as there are many recent examples of the FVM applied to the study of WECs.

The CFD approach has several potential advantages over the physical modelling of WECs in wave tanks: CFD models are relatively cheap to set up, the WEC geometry and wave conditions can be changed easily, CFD models do not normally suffer from scaling problems and data can be obtained at any points of interest in the computational domain. To be fair it should also be noted that CFD models have some disadvantages compared to physical tank models: run times may be slower than for the execution of a tank test, and, if an inappropriate CFD model is used, the model may not capture all of the relevant physics correctly and the approximating algorithms may therefore introduce large errors in the numerical solutions.

The use of a CFD code in the form of a so-called numerical wave tank (NWT) to simulate a WEC is a non-trivial task. Even with a pre-written code significant expertise is generally needed to produce useful results and there are certain fundamental questions to be answered before running a CFD simulation. The most important question to ask is, 'does the CFD model capture all of the relevant

physics of the situation to be simulated?', i.e. do the underlying equations describe all the important physical processes that will occur? No CFD model can capture every aspect of the physics and decisions will have to be made as to what physics to model and what to ignore. Before running the required simulations the CFD model must be *validated* by comparing numerical results to those from a physical model, usually via a wave tank or field data. Validation must be done for tests involving the physical processes that are to be ultimately simulated. This is not an easy task since it is usually difficult to ensure that physical and computational boundary conditions match exactly so some degree of difference in the compared results must be accepted even if the CFD model does capture all of the relevant physics. A second important question is, 'have the numerical algorithms been coded correctly?', i.e. is the code doing what it is supposed to do? This is the issue of *verification*. Ideally all branches of the code must be rigorously tested for coding accuracy via small test problems with known output data and this is inevitably time consuming as there are often thousands of lines of code; but, of course, this is impossible if the user of the CFD package does not have access to the source code. In such cases, the best that can be done is to compare the results to other independent CFD codes over a range of situations and to use known analytical solutions for simpler test cases. It should be noted that numerical algorithms, even if correctly coded, contain intrinsic errors due to the numerical approximations to the derivatives referred to earlier so it is important to extend the verification process to check that the algorithms do not produce excessive numerical dissipation or diffusion. Only after a CFD code has been validated and verified should it be used for WEC simulations. A final fundamental step in obtaining reliable simulation results addresses the issue of convergence. Errors are introduced by the spatial discretisation so it is desirable to demonstrate that results are independent of the computational grid. Any simulation should be repeated on finer grids until corresponding results on successive grids are within some specified tolerance. In this way a mesh converged solution is obtained.

Looking to the future, given the continuing advances being made in fluid dynamics models, numerical methods and computer architecture, the authors believe that eventually CFD will become the standard approach for WEC engineers.

## 7.2 CFD models

The WEC engineer has a large number of CFD models from which to choose. Some of these models are freely available, including source code, while others are commercial 'black box' packages available often under license. The authors have many years experience of developing in-house CFD models for fluid/structure interaction and in recent years have used both commercial packages and in-house codes for WEC simulation. Without exception, significant expertise is needed both to select the appropriate model and then to set up and run a simulation. Generally speaking, commercial packages come with user support and are well documented and are easier to use than the open source CFD models so may be the best choice for the WEC engineer who wants results quickly. However, some commercial packages do not use the most up to date solver algorithms giving rise to unnecessary numerical errors: it has been the authors' experience that some commercial packages that purport to be able to simulate fully 3D WEC dynamics do not necessarily simulate a simple wave field correctly (although promotional literature may include enticing images of devices in waves!). All models have their limitations and the choice of model should be informed by the important physics in the WEC simulation. The following is a non-exhaustive list of some commercial CFD packages that have been used for WEC simulation: ANSYS Fluent, CFX, FLOW-3D and Star-CD/CCM+. Some of the free open-source or in-house CFD codes that have been used to model WECs are: AMAZON, Code-Saturne, ComFLOW and OpenFoam (more on this package later). A comparison of several commercial and non-commercial CFD codes, including SPH models, applied to the Pelamis and Manchester Bobber WECs, is presented in Westphalen et al. (2009).

Rather than detail the attributes of the various competing CFD codes the authors will make some general observations on what features are desirable in a code and outline the features of their own AMAZON suite of in-house codes and present some of their results.

Most WECs operate in two fluids, namely air and water. Furthermore waves are likely to *break* in shallow water and/or storm conditions. WEC simulations have been carried out using *single* fluid CFD solvers where the numerical solution is computed only in the water component and the air/water

interface (i.e. the free surface) is found via a surface boundary condition. This approach breaks down or gives the wrong solutions when water heights become multi-valued under wave breaking. More useful are *two-fluid* solvers where the underlying fluid equations are solved simultaneously in both air and water components. This is a numerically challenging task because of the large discontinuity in density across the air/water interface (water being a thousand times denser than air). The Volume of Fluid (VoF) method (see e.g. Ubbink, 1997) is one way of obtaining the air/water interface. Essentially at each time step the volume fraction,  $F$ , of water in each computational cell ( $F = 0$  means that the cell contains only air,  $F = 1$  means that the cell contains only water) is calculated and regarded as a transported flow variable which satisfies a local advection equation. The  $F$  values are found at the new time level by solving the advection equation and the new interface is reconstructed from local  $F$  values. The VoF method can handle breaking waves although reconstruction of the interface becomes complicated in 3D.

A quite different method, used by the authors, is to capture the air/water interface *automatically* as the numerical solution proceeds in a manner similar to shock capturing in aerodynamics (Qian et al., 2006). In this approach the fluid density,  $\rho$ , (which for an incompressible model is constant in each fluid) is treated as a variable and solved for throughout the computational region along with the other flow variables like pressure and velocity. A modern Riemann based approach is used together with appropriate flux limiters to suppress oscillations caused by large spatial gradients. For visualisation purposes the air/water interface is determined by contour values where  $\rho = 500 \text{ kg m}^{-3}$ . This method handles wave breaking naturally but care must be taken to use an appropriate high resolution method to avoid unnecessary numerical errors. In the authors' incompressible finite volume AMAZON-SC code the air/water interface is sharply defined being spread over a couple of computational cells due to unavoidable numerical diffusion. The inviscid form of the AMAZON-SC model and the basic underlying equations are,

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}_1}{\partial x} + \frac{\partial \mathbf{F}_2}{\partial y} + \frac{\partial \mathbf{F}_3}{\partial z} = \mathbf{B} \quad (7.1a)$$

where  $\mathbf{U}$  is a column matrix of conserved variables,  $\mathbf{F}_i$  are column matrices containing flux terms and  $\mathbf{B}$  is a column matrix of source terms.  $\mathbf{U}$ ,  $\mathbf{F}_i$  and  $\mathbf{B}$  are given by,

$$\mathbf{U} = [\rho, \rho u, \rho v, \rho w, p]^T \quad (7.1b)$$

$$\mathbf{F}_1 = [\rho u, \rho u^2 + p, \rho uv, \rho uw, u]^T \quad (7.1c)$$

$$\mathbf{F}_2 = [\rho v, \rho uv, \rho v^2 + p, \rho vw, v]^T \quad (7.1d)$$

$$\mathbf{F}_3 = [\rho w, \rho uw, \rho vw, \rho w^2 + p, w]^T \quad (7.1e)$$

$$\mathbf{B} = [0, 0, -\rho g, 0, 0]^T \quad (7.1f)$$

and  $p = p(x, y, z)$  is pressure,  $\rho = \rho(x, y, z)$  is density and  $u$ ,  $v$ , and  $w$  are the components of the flow velocity in  $x$ ,  $y$  and  $z$  directions respectively. Note that these equations are actually re-written in integral form for the finite volume approach used in AMAZON-SC but we present them here in differential form for ease of comparison with the related compressible two-phase model in section 7.3. Viscous effects including suitable turbulence models, where these are available, can be included easily via additional source terms.

Details of the incompressible surface capturing method applied to a WEC simulation can be found in Qian et al., (2005). AMAZON-SC simulations of the LIMPET oscillating water column (OWC) (Figure 7.1) and a prototype of the Oyster oscillating WEC (Figure 7.2) are given below.

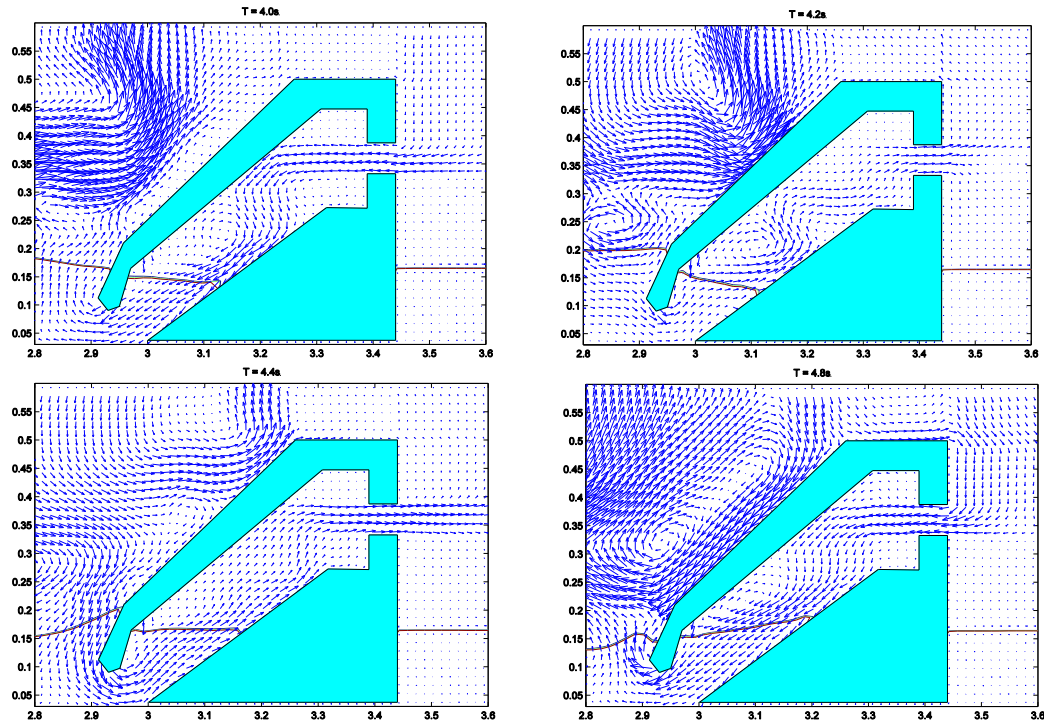


Figure 7.1: AMAZON-SC simulation of the LIMPET WEC

In Figure 7.1, the free surface patterns and velocity vectors in both water and air around a scaled LIMPET WEC model are shown at four typical instants during one period of the flow development. Regular waves with a wave length of 1.5m were generated by a moving paddle located at the left hand side of the numerical wave tank. The boundary surface contours of the device itself are represented by the Cartesian cut cell method which represents these accurately (see below). As the wave front interacts with the device it drives the water column inside the chamber to move up and down with the same period as the waves, but with a slightly different phase. A jet of air is clearly seen to be alternately driven out of the chamber and sucked into it due to the motion of the oscillating water column.

The prototype of the Oyster WEC, which has one or more vanes with a rotational axis spanning a recess in the shoreline or a caisson in the near-shore zone and responds to the pre-dominant and amplified horizontal fluid motion in shallow and intermediate depth waves, has also been simulated under regular waves. The angular velocity of the vane is derived from the motion of the waves, i.e. by calculating the torque from the pressure exerted on its surface. Several snapshots showing the wave profiles and velocity vectors around the device are presented in Figure 7.2. This test case clearly demonstrates the potential of CFD to deal with complex wave/paddle interactions and real fluid flow problems such as the Oyster WEC.

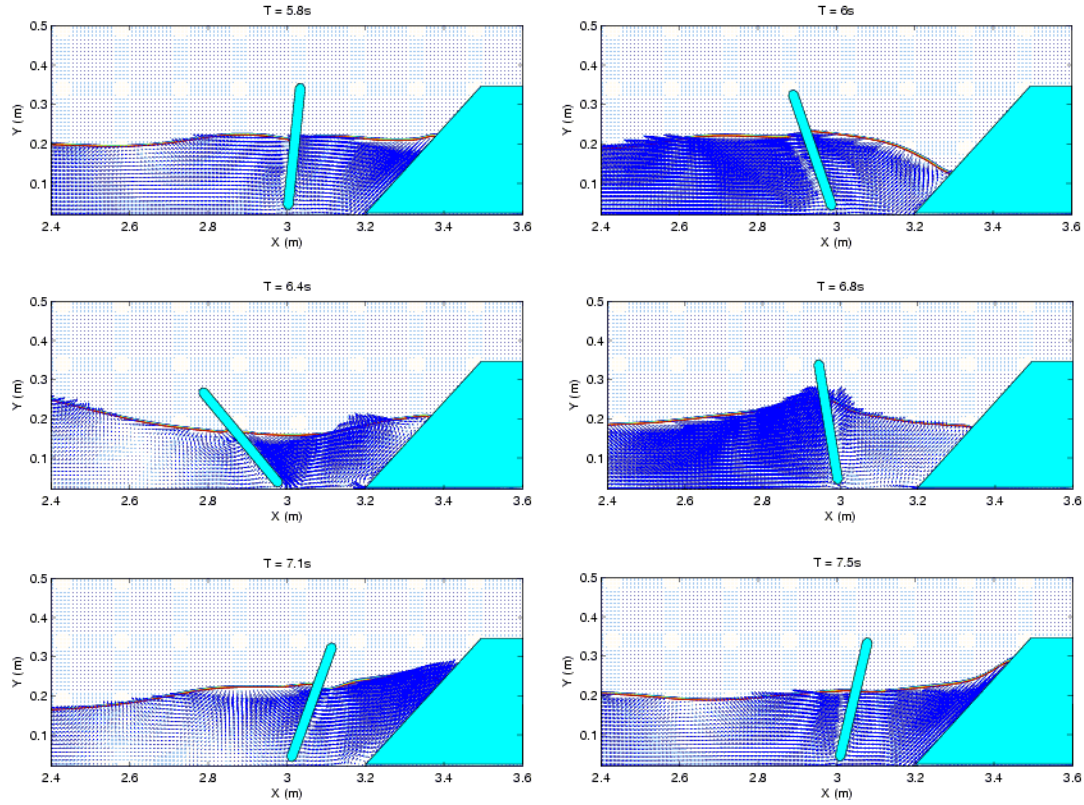
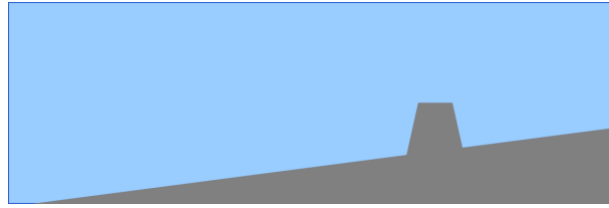


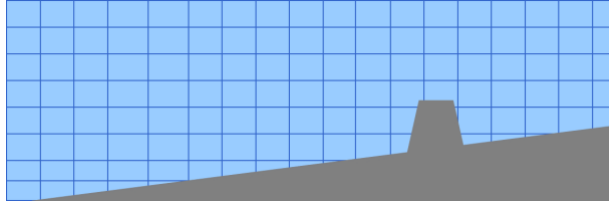
Figure 7.2: AMAZON-SC simulation a prototype Oyster WEC

A second important feature of a CFD NWT code for simulating WECs is the ability to deal with the motion of the device correctly. Typical WEC movement involves 6 DoF and the motion may be extreme in storm conditions. Grid generation has a key role to play here. One gridding approach is to stretch and compress the grid locally around the WEC as it moves in order to ensure that the grid remains body fitted at all times. This approach is efficient for small amplitude oscillatory motion of the WEC but breaks down or needs frequent remeshing in violent wave conditions when the device motion becomes large and grid cells become highly skewed. Another approach is to embed the WEC in its own interior body fitted sub grid which moves with the WEC without changing its topology within a fixed external grid. At each time step a procedure is used to interpolate the data at the intersection of the sub grid and the fixed external grid. Such a gridding strategy can, in principle, cope with arbitrary WEC motion although there remains the problem of generating a suitable body fitted internal grid and the data interpolation at the interior and exterior grid boundaries must be done with care.

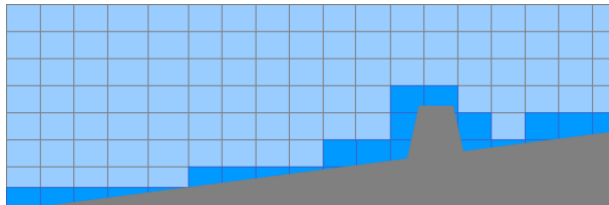
The authors have used another gridding approach within their AMAZON suite of codes. This approach is the Cartesian cut cell method (Causon et. al., 2001). In 2D the computational domain is overlaid with a simple Cartesian grid. The intersections of the grid with any irregular external and internal (i.e. the WEC) solid boundaries are computed and the affected grid cells are *cut* so as to produce a body fitted grid in which cut cells have 5 or more sides whilst the majority of cells remain rectangular Cartesian. Figure 7.3 shows the basic idea of the cut cell method.



Input vertices of solid boundary (and domain)



Overlay background stationary Cartesian grid



Identify Cut Cells and compute intersection points and other geometrical information

Figure 7.3: Cut cell generation in 2D

Moving boundaries are treated by simply recomputing cell cuts as the body moves through the background regular Cartesian grid for as long as the motion continues so that the computational grid remains boundary fitted at all times. This process is computationally efficient and, once the solid boundaries in the computational domain are specified, grid generation is automatic and no special user expertise is needed. The Cartesian cut cell method extends naturally to 3D {Ref}. AMAZON simulations for a Bobber type WEC undergoing free fall motion are presented in Figure 7.4 based on the work reported in Hu et al. (2011).

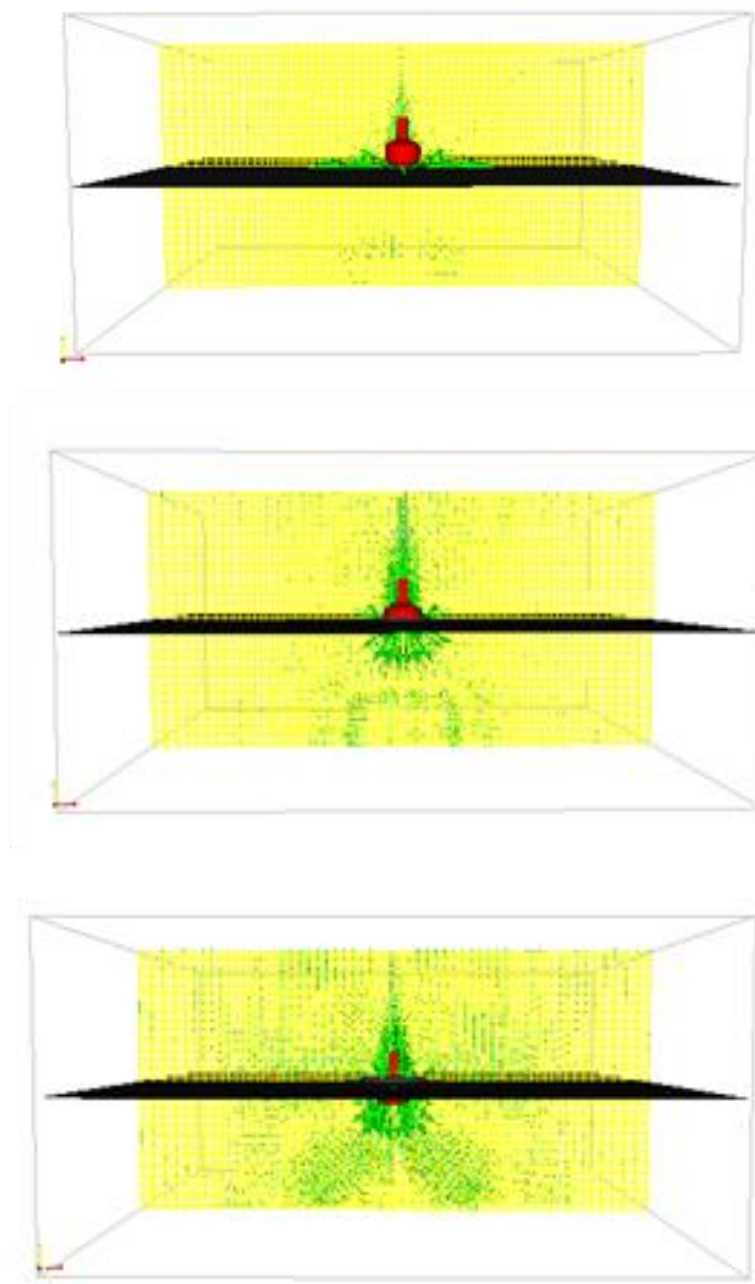


Figure 7.4: AMAZON-SC3D Bobber type WEC simulation

### 7.3 Compressible two-phase fluid models

Historically most CFD simulations of WECs have been based on *incompressible* models for the air and water components and this is true of the AMAZON results given previously (note that AMAZON-SC is a two-*fluid* model and each fluid is treated as incompressible and cannot undergo phase change). The assumption of incompressibility is often valid in many cases as water is difficult to compress and the air above the free surface does not affect the WEC dynamics appreciably. However, compressibility effects may be significant when considering WEC survivability in storm conditions. Violent wave interaction with WECs produce a variety of complex physical phenomena. Air bubbles may be entrained into the surrounding water and air pockets may be trapped against the WEC during wave interaction. Aerated water behaves very differently to pure water due to its increased compressibility. The sound speed in aerated water drops quickly (from  $1500 \text{ ms}^{-1}$  for pure



water to less than 100 ms<sup>-1</sup> at only 5% aeration) with aeration level leading to the possibility of transonic shock wave and rarefaction phenomena even at the relatively low particle velocities occurring at wave impact. Large fluctuations of pressure within trapped air pockets can also occur. Pressures may drop so low due to rapid local air pocket expansion that the surrounding water evaporates (and then recondenses once the air pressure rises again). Clearly incompressible codes cannot capture these physical processes, nor even those that treat the air component as compressible and the water component as incompressible. It should also be noted that physical tank tests are difficult to perform with aerated water due to difficulties in generating and controlling aeration levels and measuring particle velocities and pressures in a bubbly environment. Such physical models are also often intrinsically unscalable due to the different scaling laws that apply for air and water which is a good reason for using CFD where this problem does not generally occur.

It may be thought that, since the cushioning effect of compressible air and aerated water will reduce violent wave loadings on WECs, incompressible codes would still be useful for survivability calculations as they would tend to overestimate loadings especially for single fluid based solvers. However there are two problems with this view. Firstly because the incompressible assumption, especially for the water phase, does not capture the correct physics they can produce spurious results and significant *underestimates* of impact pressures. This has been demonstrated conclusively by comparing the impact pressures generated by a range of incompressible and compressible codes for the benchmark test case of the free drop of a water column and its impact onto a 2D tank floor (Figure 7.5, Ma et al. 2014). Secondly the complex physics produced by compressible fluids could potentially give rise to *increased* impact pressure in cases where reflected shocks combine or focus. The associated physics is not well understood and is a subject of current research.

Clearly, more sophisticated CFD models are required and the authors have extended their incompressible in-house code AMAZON-SC CFD model to AMAZON-CW in which both water and air components are treated as compressible fluids and the water can change phase in response to large local pressure variations (Ma et al. 2014). The basic equations extended from those of (7.1) are,

$$\frac{\partial U}{\partial t} + \frac{\partial F_1}{\partial x} + \frac{\partial F_2}{\partial y} + \frac{\partial F_3}{\partial z} = B \quad (7.2a)$$

$$U = [\alpha_1 \rho_1, \alpha_2 \rho_2, \rho u, \rho v, \rho w, \rho e]^T \quad (7.2b)$$

$$F_1 = [\alpha_1 \rho_1 u, \alpha_2 \rho_2 u, \rho u^2 + p, \rho u v, \rho u w, \rho h u]^T \quad (7.2c)$$

$$F_2 = [\alpha_1 \rho_1 v, \alpha_2 \rho_2 v, \rho v^2 + p, \rho v w, \rho h v]^T \quad (7.2d)$$

$$F_3 = [\alpha_1 \rho_1 w, \alpha_2 \rho_2 w, \rho u w, \rho v w, \rho w^2 + p, \rho h w]^T \quad (7.2e)$$

$$B = [\alpha \quad \alpha \quad \alpha \quad -\rho g \quad \alpha \quad -\rho g]^T \quad (7.2f)$$

where the variables are defined as in equations (7.1) and  $e$  is energy,  $\rho_i$  is the density of fluid  $i$  ( $i = 1$  indicates air,  $i = 2$  indicates water),  $\alpha_i$  is the volume fraction of fluid  $i$  in a cell and  $h$  is the enthalpy given by,

$$h = (\rho e + p) / \rho \quad (7.2g)$$

The underlying flow model treats the dispersed water wave as a compressible mixture of air and water with homogeneous material properties. The corresponding mathematical equations are based on a multiphase flow model which builds on the conservation laws of mass, momentum and energy as well as the gas-phase volume fraction advection equation. A high-order finite volume scheme based on

monotone upstream-centred schemes for conservation law (MUSCL) reconstruction is used to discretize the integral form of the governing equations. The numerical flux across a mesh cell face is estimated by means of the HLLC approximate Riemann solver. A third-order total variation diminishing Runge–Kutta scheme is adopted to obtain a time-accurate solution. The present model provides an effective way to deal with the compressibility of air and water–air mixtures. Several test cases have been calculated using the present approach, including a gravity-induced liquid piston, free drop of a water column in a closed tank, water–air shock tubes, slamming of a flat plate into still pure and aerated water and a plunging wave impact at a vertical wall. The obtained results agree well with experiments, exact solutions and other numerical computations (Ma et. al. 2014). Furthermore, the results illustrate through compressible simulations that during a violent wave impact a vertical wall may be subject to both positive and negative loading as shown in Fig. 7.6. This demonstrates the potential of the current method to tackle more general violent wave–air–structure interaction problems including the simulation of extreme wave loads on WEC devices.

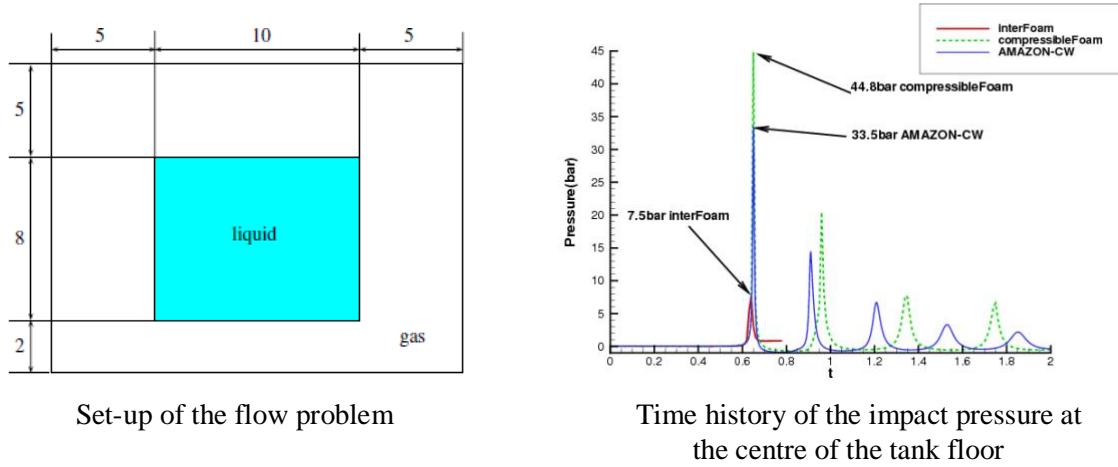


Figure 7.5: Comparison of impact pressure from incompressible and compressible CFD codes

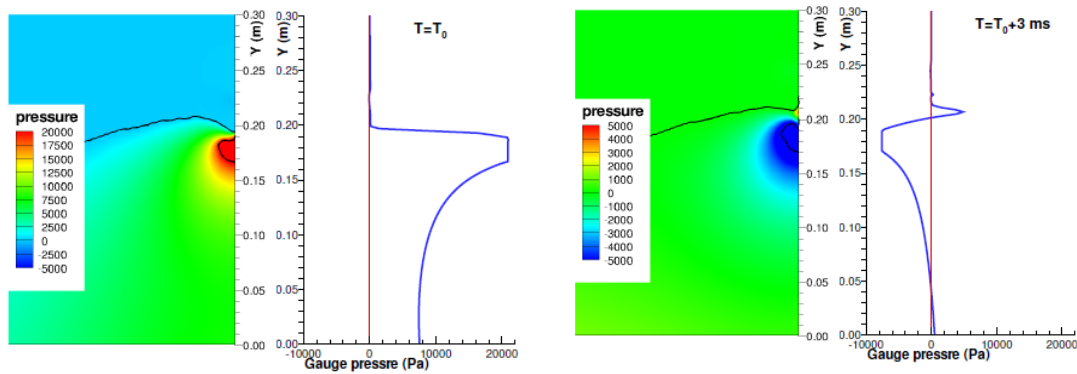


Figure 7.6: Compressible CFD model simulation of violent impact loads on a vertical wall

#### 7.4 Limitations

In general WEC dynamics involves complex physical processes which present severe challenges for CFD modelling. It is only by honestly acknowledging the limitations of CFD models and solvers that progress can be made in WEC simulation. One practical limitation of CFD at the present time is compute speed. Ideally WEC engineers would like to be able to run parametric studies with 3D models on desk top computers and obtain useful results in minutes or perhaps a few hours at most.

This would allow WEC designs to be modified and their performance to be optimised over a range of wave climates. However at the present time 3D simulations typically take hours, days (or even weeks!) depending on the case being studied. The authors' AMAZON-SC incompressible code, although based on modern solver technology, uses time steps of the order of microseconds (for numerical stability) and is therefore impractical for simulations of more than a few seconds of real time. Compressible models require similar resources. The grid generation procedure can be another limitation on compute speed and solution accuracy. A fine grid is needed around the WEC to render its geometry accurately and at the free surface to capture the air/water interface. Coarser grid cells can be used away from these regions. A balance must be achieved between solution accuracy from a fine grid and compute speed from a coarse grid. However this is further complicated by the dynamics of the WEC. If a locally body fitted grid moves in response to large amplitude WEC motion then cells may become stretched so much that the simulation breaks down. If an overlapping block grid approach is used then the necessary interpolation of data between small and large cells could produce unacceptable errors in the solution. Another limitation arises from the numerics. Approximate solutions to the CFD model are obtained from algorithms derived from discretisations of the underlying continuous equations. To a greater or lesser degree these algorithms give rise to intrinsic and unavoidable numerical dissipation and/or dispersion which degrade solutions particularly over long simulations. We have observed linear waves reduce greatly in amplitude as they cross a numerical wave tank due to numerical dissipation (and this was from an expensive commercial CFD package!). Yet another limitation arises when considering peak impact pressures. Pressure peaks occur over short time scales and seem to be very sensitive to cell size. It is important to use small time steps to capture or sample the pressure peak accurately but this can give excessively long run times. We have also found that it is necessary to use a fine mesh to obtain a mesh converged solution when considering peak pressures and this is illustrated in Figure 7.6 which shows time series of pressure for a water column falling onto a tank floor using the AMAZON-CW code on successively finer grids. This topic is of great current interest to the CFD wave/structure interaction (WSI) community and a special session at the International Society of Offshore and Polar Engineers (ISOPE) meetings has invited participants to submit their solutions to specified impact problems such as that shown in Fig. 7.7 with a view to making independent blind test code comparisons {Ref}.

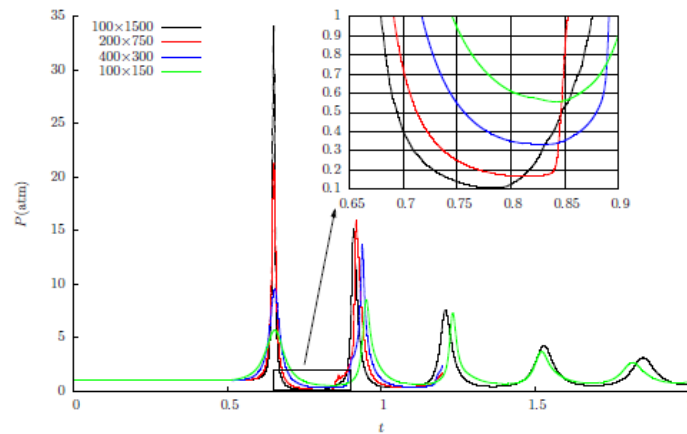


Figure 7.7: AMAZON-CW solutions for peak pressure for different grid sizes for a water column falling onto the tank floor

The previous stated limitations are essentially based on the flow solver. More fundamental limitations come from the model itself which necessarily contains assumptions and approximations. It is assumed that microscopic flow details can be ignored such as the dynamics of a single air bubble but, as we have shown, the phenomenon of air entrainment can be an important feature of the physics. It is clearly not possible to model the production and collapse of each microscopic air bubble over the size of domain needed for WEC simulation, even if bubble physics was completely understood, so some

form of integral fluid mixture model is used in the belief that enough of the essential physics will be captured and the material properties of the aerated fluid will be faithfully replicated. In some WEC simulations there will be a need to include turbulence but there are limits to the accuracy and applicability of turbulence models and there are many competing models of turbulence which itself is not completely understood and is highly problem dependent. Then there is the water. WECs operate in sea water but numerical models (and wave tank tests) currently use fresh water. Sea water can have quite different properties to fresh water depending on its temperature and salt content and this is a topic that does not seem to have been studied extensively. Finally, most numerical models assume that WECs are completely rigid when in fact there may deform appreciably in response to a large wave so these models may need to be extended to include scenarios where structures are flexible and detailed calculations of extreme loads are required.

## 7.5 Future developments

Significant advances have been made in CFD during the last few decades which have also seen massive improvements in computer hardware including massively-parallel implementations. This has led to the development of a large number of independent CFD codes for the fluid/structure interaction application written in different languages and running on different systems. These codes possess elements which, if extracted and put together into a single code, could improve current WEC simulation. However research codes are usually written in a piecemeal way, are poorly documented and are difficult for third parties to adapt to their particular needs. There is also the problem that some researchers have been reluctant to make their codes available to the wider community (even though they were funded from public research grants) and it is a very difficult and time consuming undertaking to re-write code from scratch using the available published literature. We believe that the current situation is an impediment to progress in CFD modelling of wave/structure interaction in general and WEC simulation in particular.

The authors at Manchester Metropolitan University (MMU) (along with colleagues at Plymouth University (PU) and STFC The Rutherford Appleton Laboratory (RAL)) are seeking to address these problems via a grant from the UK Engineering and Physical Sciences Research Council's (EPSRC) 'Software for the Future Project' (EPSRC Ref: EP/K038303/1). A decision was taken to develop open source code for generic wave/structure interaction (WSI) problems based around the OpenFoam software. OpenFoam (<http://www.openfoam.com>) is a freely available open source package which is becoming widely used in CFD and facilitates the development of custom-built open source CFD NWT software for the present WSI application. OpenFoam contains a large number of models, advanced gridding routines and parallelization options. Furthermore the source code is provided and the package is well structured, documented and comes with tutorial examples. Our approach has been to take a fixed version of OpenFoam as our starting point and embed it into a dedicated software repository maintained under the auspices of the Software Engineering Support Project (SESP) by the Software Engineering Group at STFC The Rutherford Appleton Laboratory. In this way, as software is developed it is tested, documented, structured and maintained to professional software engineering standards. Modules can be checked from the main branch of the repository, developed within the code developer's branch and checked back into the main repository subject to stringent code verification tests being passed. By adhering to the OpenFoam structure it should be possible for the CFD community to make rapid advances by contributing to and making use of the latest innovations without having to constantly replicate code. We have thus also developed OpenFoam modules to simulate the Oyster WEC (2D and 3D simulation in Figure 7.8 and Figure 7.9 respectively). Work is currently being undertaken with partners to produce a more general OpenFOAM framework for coupling different flow solvers such as full potential codes, CFD NWT codes and structural dynamics models to produce an efficient hybrid NWT code for wave structure interaction (WSI-Foam). Such a code will be freely available on the understanding that any developments are put back into the repository for the benefit of the CFD community. This aim is further supported by the UK Research Councils through the establishment of a recent EPSRC Collaborative Computational Project for Wave/Structure Interaction (CCP-WSI). This multi-partner project (EPSRC Ref: EP/M022382/1), which starts in July 2015, will provide the WSI community with a code development infrastructure to

produce a national numerical wave tank for use by all interested parties including the WEC community.

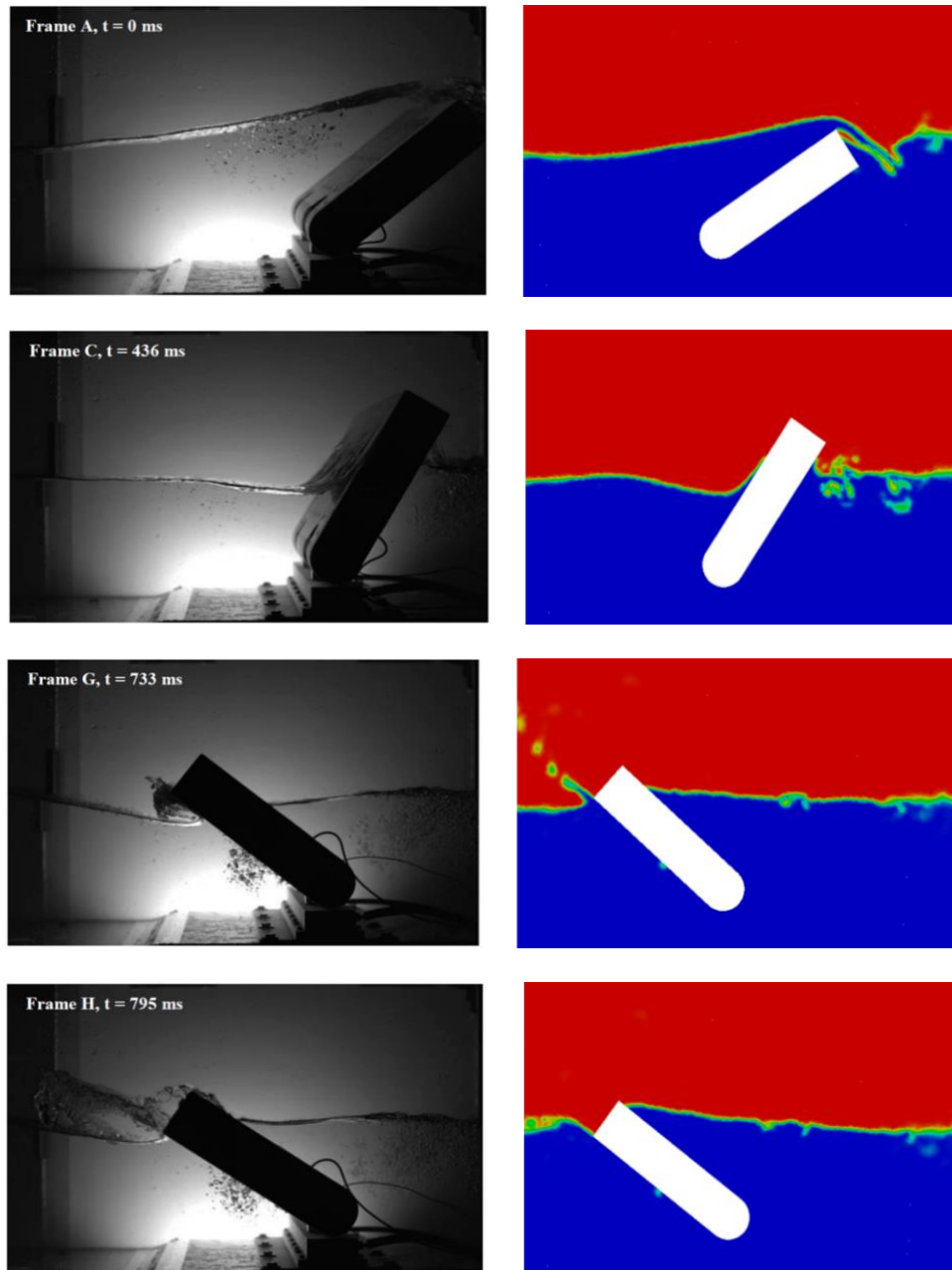
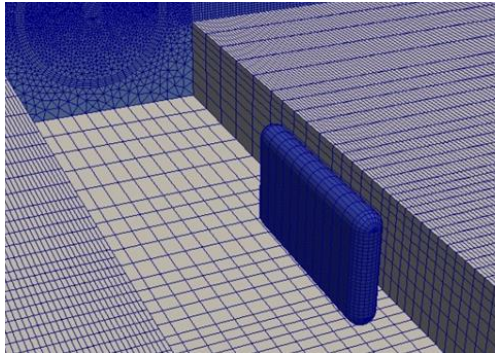
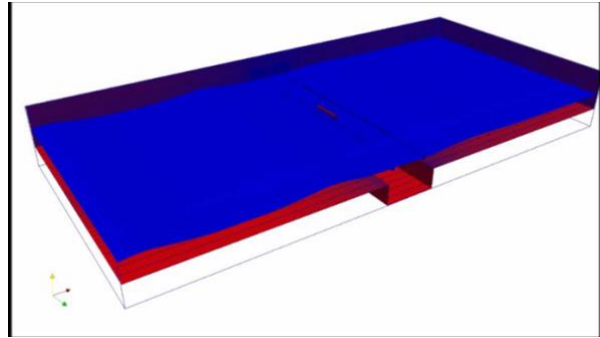


Figure 7.8: OpenFoam simulation of the Oyster WEC using AMI gridding and the waves2Foam solver (2D), Left: experiments, Right: numerical simulation



Surface representation the flap  
and the local mesh



The Oyster WEC under waves

Figure 7.9: OpenFoam simulation of the Oyster WEC  
using AMI gridding and the waves2Foam solver (3D)

## References

DM Causon and CG Mingham, (2010), 'Introductory Finite Difference Methods for PDEs', ISBN: 978-87-7681-642-1.

DM Causon, CG Mingham and L Qian, (2011), 'Introductory Finite Volume Methods for PDEs', ISBN: 978-87-7681-882-1.

J Westphalen, DM Greaves, CJK Williams, PH Taylor, DM Causon, CG Mingham, ZZ Hu, PK Stansby, BD Rogers, P Omidvar, (2009), 'Extreme wave loading on offshore wave energy devices using CFD: a hierarchical team approach', Proceedings 8th European Wave and Tidal Energy Conference (EWTEC), Uppsala, Sweden. 501-508.

O Ubbink (1997), 'Numerical prediction of two fluid systems with sharp interfaces', PhD Thesis, Imperial College of Science, Technology and Medicine, London.

L Qian, C Mingham, D Causon, D Ingram, M Folley, T Whittaker, (2005), 'Numerical simulation of wave power devices using a two-fluid free surface solver', Modern Physics Letters B 19 (28n29), 1479-1482.

L Qian, DM Causon, CG Mingham and DM Ingram (2006), 'A free-surface capturing method for two fluid flows with moving bodies', Proceedings of the Royal Society: A, 462 (2065), 21-42.

DM Causon, DM Ingram, CG Mingham, (2001), 'A Cartesian cut cell method for shallow water flows with moving boundaries', Advances in Water resources 24 (8), 899-911.

ZZ Hu, DM Causon, CG Mingham, L Qian, (2011), 'Numerical simulation of floating bodies in extreme free surface waves', Natural Hazards and Earth System Science 11 (2), 519-527.

ZH Ma, DM Causon, L Qian, Mingham CG, Gu HB and Martinez Ferrer P (2014). 'A compressible multiphase flow model for violent aerated wave impact problems'. Proceedings of the Royal Society A, Vol. 470, 20140542.