VISCOUS WAVE INTERACTION WITH STRUCTURES USING ADAPTING QUADTREE GRIDS AND CARTESIAN CUT CELLS

Deborah M. Greaves*

* Department of Architecture and Civil Engineering,
  University of Bath,
  Claverton Down,
  Bath BA2 7AY,
  U.K.
  e-mail: d.m.greaves@bath.ac.uk

Key words: Quadtree grids, grid adaptation, cut cells, volume of fluid

Abstract. This work involves the application of adaptive hierarchical grids to free surface Navier-Stokes simulation of viscous waves in a stationary tank and their interaction with a cylinder. Adapting quadtree grids are combined with a volume of fluid (VoF) approach, in which a high resolution interface capturing scheme is used for advection of the interface. The Navier-Stokes equations are discretised using finite volumes with collocated primitive variables and solved using the PISO (Pressure Implicit with Splitting of Operators) algorithm. The cylinder is included by using the technique of Cartesian cut cells. Grids adapt to follow the movement of the free surface and to provide refinement in areas of high vorticity. The method is used to simulate small amplitude viscous waves in a unit square tank, separated flow past a cylinder and preliminary results are presented for the fluid structure interaction occurring when a cylinder moves into a tank of still water and for viscous waves in a tank containing a submerged or surface piercing cylinder. Results agree reasonably well with available analytical, experimental and numerical data.

1 INTRODUCTION

The aim of this work is to develop a numerical technique for simulating viscous wave interaction with a structure. Many applications of wave interaction with a floating structure can be suitably approximated using potential flow theory, which assumes that the fluid is incompressible and inviscid. Potential flow theory is a powerful tool\(^1,2\), but usually limited to non-breaking wave simulations. In some situations, such as the response of a floating wave energy device, flow separation, turbulence and wave breaking all make significant contributions to the fluid loading. In these cases, the fluid viscosity must be accounted for, which usually means solving the full Navier-Stokes equations or RANS equations if the flow is turbulent. To model wave breaking a two-fluid approach may be taken, in which the fluid flow equations are solved both in air and water so that complex free surface motions can be modelled including wave overturning and break up into spray.

An alternative to the two fluids model is to use a particle method such as smoothed particle hydrodynamics (SPH)\(^3,4\) which has been applied to sloshing and green water problems, but is problematic in ensuring incompressibility of the fluid and also requires a very large number of particles, which can become expensive. Of the two-fluid methods, known as interface capturing techniques, level set and volume of fluid (VoF) are most popular. The level set method has produced some impressive results, but mass conservation may not necessarily be enforced. In the volume of fluid method, mass conservation is ensured as the continuity equation is used to advect the fluid interface in the calculated flow field. However, the
method traditionally suffers from smearing of the interface over many cells and the requirement for a large number of calculation cells and very small time steps.

In this work these traditional drawbacks of the VoF are tackled by using adapting quadtree grids\(^5\). These are combined with a volume of fluid (VoF) approach, in which the CICSAM high resolution interface capturing scheme, derived by Ubbink\(^6\), is used for advection of the interface. The Navier-Stokes equations are discretised using finite volumes with collocated primitive variables and solved using Issa’s\(^7\) PISO (Pressure Implicit with Splitting of Operators) algorithm. Grids adapt to follow the movement of the free surface and to provide refinement in areas of high vorticity for the separated flow cases. This results in a significant reduction in the grid size and CPU time required for a given accuracy of simulation and is found to be effective in simulating separated flow past a cylinder as well as small amplitude viscous waves in a tank and interaction with a cylinder. Full details of the method and its application to interfacial flows, dam collapse, wave simulation in a tank and interaction with a submerged cylinder are described in previous publications\(^8,9,10\). Here, the main points of the scheme are described and results for some new applications are presented.

2 SOLUTION OF THE NAVIER-STOKES EQUATIONS

The Governing equations in primitive form for a two-dimensional incompressible flow are the mass conservation equation and the Navier-Stokes momentum conservation equations

\[
\frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} = 0 \tag{1}
\]

\[
\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \nu \nabla^2 u + \frac{\partial}{\partial x} \left( v \frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial y} \left( v \frac{\partial u}{\partial y} \right), \tag{2}
\]

\[
\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -g - \frac{1}{\rho} \frac{\partial p}{\partial y} + \nu \nabla^2 v + \frac{\partial}{\partial x} \left( v \frac{\partial v}{\partial x} \right) + \frac{\partial}{\partial y} \left( v \frac{\partial v}{\partial y} \right), \tag{3}
\]

where \(x\) and \(y\) define an orthogonal Cartesian co-ordinate system, \(u\) and \(v\) are the corresponding velocity components, \(t\) is time, \(p\) is pressure, \(\rho\) is the fluid density, \(g\) is the gravitational acceleration and \(\nu\) is the fluid kinematic viscosity. For situations where the fluid viscosity is variable, such as the multi-fluid (air and water) flow simulations considered here, extra diffusion terms appear in the momentum equations: the last two terms in (2) and (3). According to Ferziger and Peric\(^11\), these terms are small compared to the other diffusion term and so can be treated explicitly and included in the source term.

The governing equations are discretised on quadtree grids using finite volumes with collocated primitive variables \((u, v\) and \(p\) are stored together at cell centres). The discrete expression for the \(u\)-velocity is obtained by integrating the \(u\)-momentum equation over the control volumes and multiplying by the cell area, \(V_p = \delta x \delta y\), to give

\[
a_p u_p = \sum a_{ab} u_{ab} + u_p^0 \rho \frac{V_p}{\delta t} - \frac{\partial p}{\partial x} V_p = H(u) - \frac{\partial p}{\partial x} V_p, \tag{4}
\]

where the control volume (cell) in question is given the subscript \(P\) and the sum of the neighbour contributions are denoted, \(nb\). The superscript \(0\) indicates that the term is from the previous time step, \(\delta x\), \(\delta y\) and \(\delta t\) are the horizontal and vertical dimensions of the cell and the time step for the calculation and \(H(u) = \sum a_{ab} u_{ab} + u_p^0 \rho \frac{V_p}{\delta t}\). The coefficients, denoted \(a\),
combine the momentum fluxes for convection and diffusion transport. The discrete equation for the $v$ component of velocity is similar. The continuity equation is discretised to give

$$S_e u_e - S_e u_w + S_n v_n - S_s v_s = 0,$$

where the lower case subscripts denote face values of the velocity and $S_i$ is the area of face $i$.

In order to solve for the velocity and pressure field, it is necessary to couple the momentum and continuity equations. The PISO\textsuperscript{7} algorithm was developed initially for non-iterative computation of unsteady compressible flows, but has since been successfully adapted for steady and unsteady incompressible flows. In this scheme, the discretised $u$-momentum equation (4) is interpolated to cell faces using Rhie and Chow\textsuperscript{12} treatment to prevent checkerboard errors. The face velocity expressions are then directly substituted into the discretised continuity equation (5) to give an equation for pressure.

$$a_j p_j = a_E p_E + a_W p_W + a_N p_N + a_S p_S + b.$$

Here, $a_j$ are coefficients for node $j$ and the source term is

$$b = -S_e \left( \frac{H(u)}{a_p} \right)_e - S_w \left( \frac{H(u)}{a_p} \right)_w + S_n \left( \frac{H(u)}{a_p} \right)_n - S_s \left( \frac{H(u)}{a_p} \right)_s.$$

The algorithm for solution of the equations using the PISO scheme is:

1. solve momentum equations (2, 3) using guessed pressure field
2. solve pressure equation (7)
3. calculate volumetric fluxes for use in momentum equation coefficients
4. correct velocity

The loop from (2) to (4) is repeated iteratively until a prescribed tolerance is achieved before proceeding to the next time step (1).

3 THE VOLUME OF FLUID METHOD, VOF

In the VoF method, we consider two immiscible incompressible fluids. The location of the two fluids is specified using a volume fraction function, $C$, with $C = 1$ inside one fluid and $C = 0$ in the other. The interface between the two fluids is contained in cells for which $0 < C < 1$. The continuity equation applies,

$$\nabla \cdot \mathbf{u} = 0$$

and the volume conservation of the first fluid can be expressed as

$$\frac{\partial C}{\partial t} + \nabla \cdot (\mathbf{u} C) = 0.$$

The original VoF scheme of Hirt and Nichols\textsuperscript{13} has a fluxing scheme that uses either upwinding or a downwinding donor–acceptor cell approach depending on the local orientation of the interface. The advantage of the upwind scheme is that it is stable, but it is diffusive and may spread the interface over many cells. The downwind scheme is unstable, but sharpens the interface and so is advantageous in interface tracking. Various VoF fluxing methods have been developed, most of which aim for a balance between the stability advantages of the upwind scheme and the front sharpening advantages of the downwind scheme\textsuperscript{8}.

3.1 CICSAM VoF
Ubbink’s\textsuperscript{6} compressive differencing scheme for discretisation of the volume fraction equation is named CICSAM (Compressive Interface Capturing Scheme for Arbitrary Meshes). In this scheme, the cell face values of $C$ are determined from a combination of the Convection Boundedness Criteria (CBC) value given by

$$
\tilde{C}_{fCBC} = \begin{cases} 
\min \left\{ 1, \frac{\tilde{C}_D}{\sigma_D} \right\} & \text{when } 0 \leq \tilde{C}_D \leq 1 \\
\tilde{C}_D & \text{when } \tilde{C}_D < 0, \tilde{C}_D > 1
\end{cases},
$$

and the Ultimate Quickest (UQ) value given by

$$
\tilde{C}_{fUQ} = \begin{cases} 
\min \left\{ \frac{8\sigma_D \tilde{C}_D + (1 - \sigma_D)(6\tilde{C}_D + 3)}{8}, \tilde{C}_{fCBC} \right\} & \text{when } 0 \leq \tilde{C}_D \leq 1 \\
\tilde{C}_D & \text{when } \tilde{C}_D < 0, \tilde{C}_D > 1
\end{cases}.
$$

Here $\sigma_D$ is the cell Courant number, calculated by summing the fluxes over each cell face. $\tilde{C}_D$ is the normalised variable for the donor cell, calculated from

$$
\tilde{C}_D = \frac{C_D - C_U}{C_A - C_U},
$$

where subscript $U$ indicates the upwind cell, $A$ the acceptor and $D$ the donor cell. These are determined depending on the velocity at a given face. The weighting factor used to combine the CBC and UQ contribution at a given face takes into account the orientation of the interface and the direction of motion. This method is compared with others and demonstrated for various interfacial flow calculations in earlier work\textsuperscript{8}.

3 QUADTREE GRIDS

Quadtree grids are quick to generate by recursive subdivision of a unit square about a set of seeding points. The seeding points are defined to lie around the cylinder boundary for the separated flow cases considered here, and along the initial position of the air-fluid interface for the wave simulations. The grids have a tree structure that enables them to be stored efficiently in a data tree. A reference numbering system can be derived comprised of integers. These reference numbers may be manipulated to obtain information relating to the position of a cell within the grid, such as division level, centre coordinates and grid neighbourhood. Quadtree grids can be readily adapted by the addition or removal of cells from the tree, whilst maintaining the tree structure. Quadtree grid generation, grid reference numbering schemes, grid data retrieval and neighbour finding routines are described in detail in previous publications\textsuperscript{5,14,15}.

3.1 Grid Adaptation

Two types of grid adaptation are used in this work. For separated flow cases, the adaptation indicator is vorticity. The flow vorticity in each cell is calculated and if it lies above a prescribed threshold, then the cell is divided into four. If, on the other hand, a cell and its three siblings all have vorticity less than the minimum, then the four cells are removed and replaced with their parent. For free surface wave cases, the adaptation is driven by proximity to the interface between fluids. Remeshing of the grid operates by dividing a cell into four if it lies on the interface; derefinement takes place by removing four sibling cells and replacing them with their parent if each of the four sibling cells lies away from the interface.
Velocity and pressure variables are interpolated onto new cells using bi-linear interpolation from the neighbours of the divided cell. Alternatively, when four sibling cells are removed and replaced with their parent, the variables assigned to the parent are the average of the four sibling values.

4 CARTESIAN CUT CELLS

Quadtree grids are made up of square cells and so a stepped approximation to a smooth curved boundary, such as a cylinder, is inevitable. This drawback was mentioned by Greaves and may be eliminated by using the Cartesian Cut Cell technique. In this technique, the smooth shape of the body is cut out of the grid, leaving cut cells around the body boundary, as in Figure 1. The technique is described for a uniform Cartesian grid by Yang et al. It was also used by Wang to simulate flow past a cylinder with a uniform grid and fractional step method.

The cut cell process usually generates some very small cut cells around the boundary and these may require very small time steps for stability. Small cut cells can be seen around the cylinder in Figure 1. The problem of very small cut cells may be overcome by merging cut cells that have fluid volume less than a specified minimum. If the specified minimum is 0.5 of the finest quadtree cell, the merged grid is shown in Figure 2. As can be seen, merging may result in cut cells having more than one face neighbour. Thus hanging nodes are created and these need to be carefully identified and treated using the same approach, but with appropriate cell volume and centroid, as those occurring naturally within the quadtree grid.

Special interpolations are used to calculate gradients and variables at faces for the finite volume scheme at these cut cells, following recommendations made by Qian et al. Boundary conditions are applied at the fictional mirror node R, shown in Figure 3. If no-slip wall boundary conditions are used, the pressure and velocity are

\[ p_R = p_P - \rho g \left( n_y |RP| \right) \]  
\[ u_R = u_P - 2(u_P \cdot n)n + 2(u_b \cdot n)n \]

where \( \rho p \) is the density at cell \( P \), \( g \) is the acceleration due to gravity for wave cases, \( |RP| \) is the distance between nodes \( R \) and \( P \), \( u_b \) is the velocity of the cylinder and \( n \) is the unit normal vector to the cut face.
5 RESULTS

5.1 Separated flow past a cylinder

The adaptive quadtree cut cell method is first demonstrated for fluid flow past a cylinder at
Reynolds number, $Re = \frac{\rho u_in d}{\mu} = 100$. Here, $d$ is the cylinder diameter and $u_{in}$ is the inlet
velocity. No slip boundary conditions are used on the cylinder, free boundary conditions are
applied on the right-hand, top and bottom walls and a steady unidirectional inlet velocity, $u = u_{in}$, $v = 0$ is applied on the left-hand wall. The grid is initially refined around the cylinder
boundary only, and as the vortex shedding flow develops, it adapts to areas of high vorticity.
The vorticity limits for grid adaptation are found by trial and error and for this case, cells are
divided if the vorticity exceeds $\omega = 3.0$ and derefinement occurs if the vorticity in four sibling
cells is less than $\omega = 1.0$. The quadtree grid has maximum division level of 9 at the cylinder
and in the adapted region as it develops and a minimum background division level of 5. It is
adapted every 20 time steps throughout the simulation and the non-dimensional time step size,
$\Delta t = \Delta t_d/u_{in} = 0.005$.

Figure 4 shows the streamlines and Figure 5 the velocity vectors once vortex shedding has
established. Figure 6 shows the time history of lift and drag force coefficients, calculated by
integrating the pressure and viscous forces around the cylinder surface. The Strouhal number
is predicted to be, $St = 0.129$, the mean drag coefficient to be $c_{Dave} = 1.38$ and the rms lift
coefficient to be, $c_{Lrms} = 0.17$, which agree reasonably with experimental and numerical data
given by Zhou and Graham$^{19}$ who recorded values of $St = 0.152 – 0.174$, $c_{Dave} = 1.29 – 1.82$
and $c_{Lrms} = 0.14 – 0.34$. 
5.2 Sloshing

5.2.1 Potential flow

This case was also calculated by Ubbink\textsuperscript{6} on a uniform grid. A rectangular tank of length, $b = 2h$, where $h$ is the mean water depth is half filled with liquid with air above it. The liquid has initial asymmetric surface elevation profile, $\eta = a \cos(\pi x / b)$, where $x$ is measured along the length of the tank and $a = 0.02$ is the wave amplitude. In each of the two-fluid test cases presented in this work the density ratio is 1000, where the upper fluid has density equal to 1 and the lower fluid density equal to 1000. The sloshing wave is initially calculated in a potential flow, in which the fluid viscosity, $\nu = 0$. Initial conditions for velocity are zero, slip boundary conditions are applied on walls and the calculation time step $dt = 0.0001$s. The theoretical solution for this case is that the wave should continue sloshing indefinitely with the same amplitude and wave period given by Raad \textit{et al}\textsuperscript{20}:
\[ P = 2\pi \sqrt{gk \tanh(kh)}, \tag{15} \]

Where \( k \) is the wavenumber and \( h \) is the mean water depth. The wave elevation is recorded at the left hand end of the tank and plotted in Figure 7 against non-dimensional time, \( \tau = t\sqrt{g/h} \), together with the theoretical result. The wave period is predicted well and the amplitude agrees reasonably with the theoretical solution over the first nine wave periods calculated.

![Figure 7 Wave elevation history at the LH wall of the tank](image)

**5.2.2 Viscous fluid**

In this case, the fluid is given the same initial elevation, time step and conditions as in 5.2.1 above, but the simulated fluid is viscous. The Reynolds number is calculated from \( Re = \frac{\nu gh}{h} \), in which \( \nu \) is the kinematic viscosity. Here, \( Re = 200 \) waves are simulated using both a uniform Cartesian grid and an adapting quadtree grid with refinement in a band surrounding the free surface. The air-liquid interface and adapted quadtree grid after one wave period are plotted in Figure 8. In this case, the maximum division level is 7 and the minimum division level is 5 and it will be referred to here as a 7 x 5 quadtree grid. The band around the interface is 10 cells wide. In Figure 9, the wave elevation history at the left hand end of the tank is plotted for both the uniform and adapting quadtree grids. There is very little difference between the two results, although some small differences in the peak amplitude are evident. Application of this method to small amplitude symmetric sloshing waves in fluids of different viscosity was found to agree well with analytical solutions in previous work\(^9\). The method was also found to be effective for predicting the large scale wave overturning and break up into spray occurring after collapse of a water column and its subsequent interaction with an obstacle\(^9\). The use of adapting quadtree grids was shown to lead to considerable savings both in CPU and grid size\(^9,10\).
Combining the Cartesian cut cell method with the VoF method, preliminary results are shown here for a cylinder entering a liquid, with an initially still free surface, in a tank. The tank is width $b$ and filled with liquid to depth $0.51b$. A cylinder of diameter, $d = 0.1b$, starts from a position at the centre of the tank and at $0.065b$ above the liquid surface and proceeds vertically downwards with velocity $0.2\text{m/s}$. At each time step, the new position of the cylinder is calculated and the new cut cell mesh determined. Results showing the progress of the cylinder into the tank are given in Figures 10 – 13. The velocity vectors, cylinder and air-fluid interface are plotted at time, $t = 0.164$, $0.296$, $0.478$ and $0.545$. No-slip boundary conditions are applied at the cylinder and the free surface is seen to be drawn down as the cylinder enters the water and to splash up as it submerges.
5.4 Wave cylinder interaction

Preliminary results for viscous wave interactions with a submerged cylinder in the tank are presented here. The length of the tank, \( b = 2h \), where \( h \) is the mean water depth. The wave has symmetric initial surface elevation profile, \( \eta = a \cos\left(\frac{2\pi x}{b}\right) \). A submerged cylinder of diameter \( 0.2h \) is positioned at the horizontal centre of the unit square tank. The fluid Reynolds number, \( Re = h \sqrt{gh}/\nu = 200 \). A refinement band of 10 cells is maintained around the cylinder boundary as well as the interface and the grid adapts dynamically at each time step. Two cases are considered. In the first, the wave amplitude, \( a = 0.05 \) and the cylinder is positioned at depth \( 0.2h \) below the mean water level. The air-liquid interface and velocity vectors calculated on an adapting 7 x 5 quadtree grid at \( t = 0 \) and at the first peak are shown in Figures 14 and 15. The wave is clearly distorted over the cylinder and appears to be breaking. In the second case, the cylinder is located at the mean water level and the wave amplitude, \( a = 0.01 \), and the adapted quadtree grid and interface are plotted in Figures 16 and 17. Here the free surface oscillates up and down the curved sides of the surface piercing cylinder. These preliminary fluid structure interaction cases demonstrate the power of the method and its potential for simulating extreme wave interactions with offshore structures. However, further work is required in order to confirm the results shown here and to validate properly the method for these applications. An important part of this will be to source appropriate experimental data.
6 CONCLUSIONS

A method has been presented combining adapting quadtree grids with a PISO finite volume solution procedure for the Navier-Stokes equations together with a high resolution volume of fluid scheme and the Cartesian cut cell technique for curved boundary modelling. Results have been shown for each part of the new method: cut cells combined with adapting quadtree grids for separated flow simulation; high resolution combined with adapting quadtree grids for viscous wave simulation; the combination of each method to simulate a cylinder moving through a fluid with a free surface in the water entry problem and waves interacting with a submerged and surface piercing cylinder. These investigations are developmental steps of the code towards the ultimate aim of simulating violent wave interactions with a floating wave energy device. The proposed methodology has been demonstrated to give reasonably accurate results, although a finer grid would improve the solution in some cases, and time and grid size savings have been made through the use of adapting quadtree grids. However, further investigation of the water entry and wave-cylinder interaction problems is necessary to properly validate the code for these cases. The method shows good potential for simulating viscous wave interaction with floating structures and each of the techniques utilised may be readily extended to three dimensions.

7 ACKNOWLEDGEMENTS

The author is very grateful to the Royal Society for supporting this work through a Royal Society University Research Fellowship.

8 REFERENCES