(Abstract submitted to the 25th IWWWFB, China, 2010)

Meshfree simulation of free surface flow and fluid-structure interaction *

Ole Lindberg, Harry B. Bingham and Allan Engsig-Karup
Technical University of Denmark,
2800 Kgs. Lyngby, Denmark
olli@mek.dtu.dk, hbb@mek.dtu.dk and apek@imm.dtu.dk

At this year's workshop we will present one of our newest initiatives, meshfree modelling of free surface flow and fluid-structured interaction. The motivation for this work is to develop enhanced methods and models for simulation of violent wave-structure interaction. The focus for this work is on ocean energy devices, e.g. wind turbine foundations and wave energy devices, i.e. both fixed and floating structures.

At the 22nd, 23rd and 24th International Workshop on Water Waves and Floating Bodies we presented results using a higher-order finite difference (FD) based solution of the exact Laplace problem for nonlinear water waves and their interaction with fixed structures, see Bingham et al. [2007], Engsig-Karup et al. [2008] and Engsig-Karup and Bingham [2009]. The model is also described in Bingham and Zhang [2007] and Engsig-Karup et al. [2009]. These publications establish the convergence, accuracy and stability properties of the model and demonstrate the that the model is robust and efficient.

The goal of the current project is to develop an accurate meshfree model for violent fluid-structure interaction and couple it with the existing finite difference model mentioned above. The higher order finite difference model is used to accurately simulate smooth ocean waves over a large area of open sea or in a coastal region. The meshfree model is used to simulate the, potentially violent, interactions between ocean waves and fixed or floating structures and the flow in the near field of these structures. The expected result is a robust, accurate, flexible and efficient solver for nonlinear wave-structure interaction.

The meshfree model under development is based on an arbitrary Lagrangian-Eulerian (ALE) formulation. The model solves the incompressible and inviscid Euler equations by means of a weakly compressible flow approximation using isotropic and isothermal equations. The compressible Euler equations are non-linear conservation laws which can develop steep gradients in the solution, i.e. shocks, even from smooth initial conditions, see Toro [1999], hence careful attention must be payed to the approximation of the spatial derivatives. To prevent spurious oscillations in the vicinity of large gradients, the derivatives in are approximated by the moving least squares method (MLS), see Zienkiewicz et al. [2006], in combination with a monotone flux calculation scheme. The monotone flux calculation scheme approximates the solution to the Riemann problem between the particles. This solution method gives a smooth and accurate pressure field which is especially important for wave-structure interaction problems. robust and accurate calculation of the pressure field is still a research topic in the meshfree community, see Ferrari et al. [2009]. The above described solution procedure has a number of attractive features: It is consistent; the order of accuracy of the scheme is flexible; the choice of the approximate Riemann solver gives some control over numerical diffusion; it is free of undetermined parameters; and it is possible to impose boundary conditions directly. Finally, it is well suited to a direct coupling with a finite difference method.

We present the ALE isotropic and isothermal Euler equations for a compressible fluid. The extensive and conserved properties volume v, mass m and momentum $m\mathbf{u}$ are moved by a regular transport field

*We wish to thank Statkraft, Norway for funding this project under their Ocean Energy Research Program, and the Danish Center for Scientific Computing for supercomputing resources.

$\mathbf{u}_0$ according to the conservation laws

$$\frac{\mathrm{d}v}{\mathrm{d}t} - v\nabla \cdot \mathbf{u}_0 = 0, \tag{1}$$

$$\frac{\mathrm{d}m}{\mathrm{d}t} + v\nabla \cdot (\rho\mathbf{u} - \rho\mathbf{u}_0) = 0, \tag{2}$$

$$\frac{\mathrm{d}m\mathbf{u}}{\mathrm{d}t} + v\nabla \cdot (\rho\mathbf{u} \otimes \mathbf{u} + p\mathbf{I} - \rho\mathbf{u} \otimes \mathbf{u}_0) = m\mathbf{g}, \tag{3}$$

which is the conservation laws of space, mass and momentum respectively. In the equations ρ is the density, $\mathbf{u}$ is the flow velocity, p is the pressure, $\mathbf{g}$ is the gravitational acceleration and $\mathbf{I}$ is the identity matrix of size $d \times d$, where $d = \dim(\Omega)$ is the dimension of the physical space.

The Eulerian equations are obtained by setting the transport velocity to zero $\mathbf{u}_0 = \mathbf{0}$ and the Lagrangian equations are obtained by setting the transport field to the flow velocity $\mathbf{u}_0 = \mathbf{u}$.

The extensive conserved properties are centered at a position $\mathbf{x}$ and moves with the kinematic equation

$$\frac{\mathrm{d}\mathbf{x}}{\mathrm{d}t} = \mathbf{u}_0. \tag{4}$$

The pressure is related to the density through the isothermal equation of state

$$p = c_s^2(\rho - \rho_0), \tag{5}$$

where c_s is the speed of sound, which is constant though out the fluid volume due to the isothermal approximation, and ρ_0 is the reference density.

The equations are solved in a spatial domain Ω with the boundary Γ, to which we apply physical boundary conditions inflow, outflow, solid wall and free surface. The kinematic and dynamic boundary conditions at the solid wall are

$$(\mathbf{u} - \mathbf{u}_w) \cdot \mathbf{n} = 0 \quad \wedge \quad \frac{\partial p}{\partial n} = \rho\mathbf{n} \cdot \mathbf{g}, \quad \mathbf{x} \in \Gamma_w, \tag{6}$$

where $\mathbf{u}_w$ is the velocity of the wall, $\mathbf{n}$ is the outward normal vector to the wall and $n = \mathbf{x} \cdot \mathbf{n}$ is the coordinate of the normal direction. At the free surface the kinematic and dynamic boundary conditions are

$$(\mathbf{u} - \mathbf{u}_{fs}) \cdot \mathbf{n} = 0 \quad \wedge \quad p = p_{atm} \approx 0, \quad \mathbf{x} \in \Gamma_{fs}. \tag{7}$$

where $\mathbf{u}_{fs}$ is the velocity at the free surface. The dynamic boundary condition states that the pressure is equal the atmospheric pressure at the free surface.

The applied numerical method for approximation the divergence is very similar to the one presented by Shu et al. [2005]. The main difference is that they use the radial basis functions (RBF) for the meshfree derivative approximation, where we apply the MLS method to construct the basis function. This is because the MLS method is believed to be more robust and has better consistency properties that the RBF method as presented in Shu et al. [2005].

The physical domain is discretized in a point-based representation, where each point represents a physical volume and mass. All the flow variables are collocated at these points. The numerical approximation is localized such that each point only communicates with other points within a given radius. The support of a point i is an index set $N_i = \{j \in [1, N] \mid |\mathbf{x}_i - \mathbf{x}_j| < r\}$, where N is the total number of points. The MLS derivative approximation of a function $f(\mathbf{x})$ at the point $\mathbf{x}_i$ has the representation

$$\nabla f(\mathbf{x}_i) \approx \sum_{j \in N_i} \mathbf{w}_{i,j} f(\mathbf{x}_j) \tag{8}$$

where the summation is carried out over all the points in the support of point $\mathbf{x}_i$ and the vectors with weights $\mathbf{w}_{i,j}$ are calculated using the MLS method.

To formulate the Riemann problem between the calculation points, it is practical to formulate the ALE conservation laws (1) to (3) in the general form

$$\frac{\mathrm{d}(v\mathbf{q})}{\mathrm{d}t} + v\nabla \cdot (\mathbf{F}(\mathbf{q}) - \mathbf{q} \otimes \mathbf{u}_0) = v\mathbf{b}(\mathbf{q}), \tag{9}$$

where $\mathbf{q}$ is the vector of intensive conserved variables, $\mathbf{F}(\mathbf{q})$ is the flux tensor and $\mathbf{b}(\mathbf{q})$ is the source vector. The approximate Riemann solver is applied to the divergence of the flux tensor $\nabla \cdot \mathbf{F}(\mathbf{q})$. A new divergence approximation is calculated using a flux approximation at the midpoint between the points $\mathbf{x}_i$ and $\mathbf{x}_j$. The weights for the new divergence approximation is

$$\boldsymbol{\alpha}_{i,j} = \frac{1}{|\mathbf{w}_{i,j}|}\mathbf{w}_{i,j} \quad \wedge \quad \beta_{i,j} = |\mathbf{w}_{i,j}|, \tag{10}$$

where the first is a unit direction vector and the latter is a scaling factor. The direction vector is used to calculate a the directional flux between at two calculation points

$$\mathbf{f}(\mathbf{q}_i) = \boldsymbol{\alpha}_{i,j} \cdot \mathbf{F}(\mathbf{q}_i) \quad \wedge \quad \mathbf{f}(\mathbf{q}_j) = \boldsymbol{\alpha}_{i,j} \cdot \mathbf{F}(\mathbf{q}_j). \tag{11}$$

Now a flux scheme is applied to approximate the flux at the midpoint $\mathbf{f}(\mathbf{q}_i, \mathbf{q}_j)$ and the divergence approximation is

$$\nabla \cdot \mathbf{F}(\mathbf{q}_i) \approx 2 \sum_{j \in N_i} \beta_{i,j} \mathbf{f}(\mathbf{q}_i, \mathbf{q}_j). \tag{12}$$

The simplest choice of flux scheme is the the central difference scheme

$$\mathbf{f}(\mathbf{q}_i, \mathbf{q}_j) = \frac{1}{2}(\mathbf{f}(\mathbf{q}_i) + \mathbf{f}(\mathbf{q}_j)), \tag{13}$$

but it's not a monotone scheme and it can create oscillations in the solution around steep gradients. Another simple scheme is the Rusanov scheme

$$\mathbf{f} = \frac{1}{2}(\mathbf{f}(\mathbf{q}_i) + \mathbf{f}(\mathbf{q}_j)) - \frac{1}{2}|\hat{\mathbf{A}}|(\mathbf{q}_j - \mathbf{q}_i), \tag{14}$$

where $|\hat{\mathbf{A}}|$ is a matrix with the maximum local wave speed on the diagonal. It is a monotone scheme and hence it's a possible choice when solving compressible Euler equations. Other, more accurate possibilities, are the Roe or MUSCL schemes and many other schemes can be found in e.g. Toro [1999].

In the meshfree community, the still water hydrostatic test case is very important and perhaps surprisingly difficult. In fact, very few meshfree methods can solve this seemingly trivial problem satisfactorily. We consider a reference density of $\rho_0 = 1000kg/m^3$ and the gravitational acceleration is $g = -9.81m/s^2$. The simulation time is $t_{max} = 10s$. A fourth order Runge-Kutta method is used for the time stepping with $\Delta t = C\frac{\Delta x}{c_s}$, $C = 0.9$ and Δx set to the initial uniform point spacing. A first order polynomial basis has been used in the MLS basis functions and the Rusanov flux has been used for the approximate Riemann solver.

Figure 1a gives a qualitative impression that the hydrostatic pressure is calculated satisfactory and 1b show the relative maximum difference between the simulated density and the hydrostatic density. It is seen that there is a linear relationship between the sound speed and the density difference. At the maximum sound speed $c_s = 25m/s$ the relative difference from the hydrostatic density is less than 0.1%, which is satisfactory. We note however, that it may be necessary to use a higher sound speed to simulate dynamic problems satisfactorily.

The method outlined above is a solid basis for further development, but is still in the intial stages of developement. At the workshop we will show how the method performs on standing waves and shoaling tests. We believe it is very important to demonstrate good performance on these very fundamental validation cases, before the method is applied to more advanced simulations. One issue that needs attention is the minimization of numerical diffusion from the approximate Riemann solver, because it has a damping effect on the wave motion. We will also present cases with fluid structure interaction, both for fixed and floating structures. The meshfree models are interesting, because they are attractive for simulation of very violent and strongly Lagrangian flows. The meshfree nature is also suited for violent fluid structure interaction. At the workshop we will demonstrate the quality of the meshfree models on these types of flows.

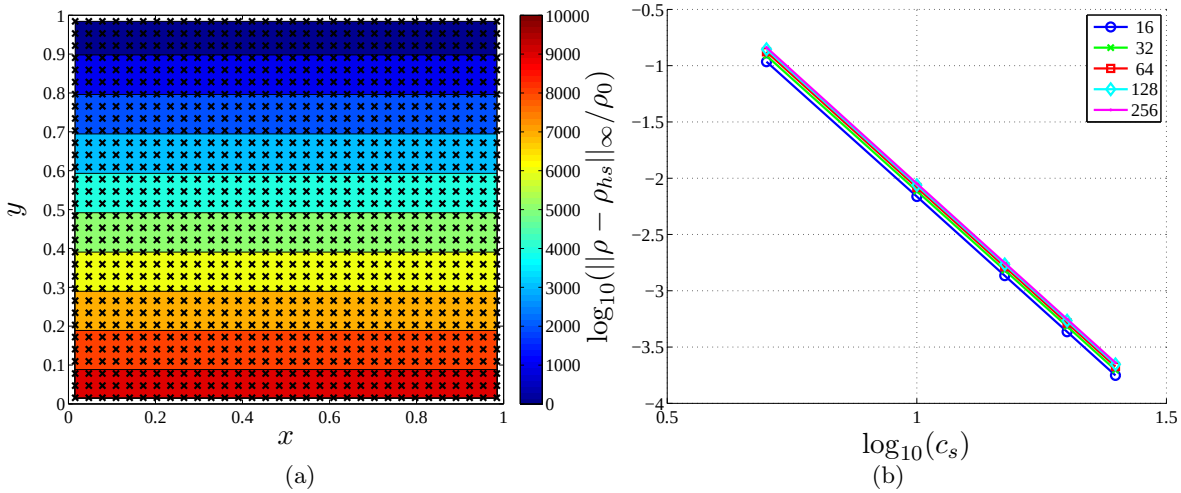

Figure 1: The hydrostatic validation case. (a): The contour levels show the calculated pressure at $t = 1s$ and the crosses show the calculation points. The sound speed is $c_s = 25m/s$. (b): The maximum relative difference of the density ρ from the hydrostatic density ρ_{hs} as a function of the sound speed c_s. The results are for $N_z = 16, 32, 64, 128, 256$ points in the vertical direction.

H. B. Bingham and H. Zhang. On the accuracy of finite-difference solutions for nonlinear water waves. *Journal of Engineering Mathematics*, 58(1-4):211–228, 2007.

H. B. Bingham, A. P. Engsig-Karup, and O. Lindberg. A high-order finite difference method for nonlinear wave-structure interaction. In *22nd International Workshop on Water Waves and Floating Bodies*, 2007.

A. P. Engsig-Karup and H. B. Bingham. Boundary-fitted solutions for 3d nonlinear water wave-structure interaction. In *24th International Workshop on Water Waves and Floating Bodies*, 2009.

A. P. Engsig-Karup, H. B. Bingham, and O. Lindberg. Multigrid preconditioning for efficient solution of the 3d laplace problem for wave-body interaction. In *23rd International Workshop on Water Waves and Floating Bodies*, 2008.

A. P. Engsig-Karup, H. B. Bingham, and O. Lindberg. An efficient flexible-order model for 3d nonlinear water waves. *Journal of Computational Physics*, 228:2100–2118, 2009.

A. Ferrari, M. Dumbser, E. F. Toro, and A. Armanini. A new 3D parallel SPH scheme for free surface flows. *Computers & Fluids*, 38(6):1203–1217, 2009.

C. Shu, H. Ding, H. Q. Chen, and T. G. Wang. An upwind local RBF-DQ method for simulation of inviscid compressible flows. *Computer Methods in Applied Mechanics and Engineering*, 194(18-20): 2001–2017, 2005.

E. F. Toro. *Riemann Solvers and Numerical Methods for Fluid Dynamics, A Practical Introduction*. Springer, 2 edition, 1999.

O. C. Zienkiewicz, R. L. Taylor, and J. Z. Zhu. *Finite Element Method, Its Basis and Fundamentals*. McGraw-Hill, 6 edition, 2006.