The Use of a Meshless CFD Method in Modelling Progressive Flooding and Damaged Stability of Ships

Dag Skaar, *SSRC, Univ. of Glasgow and Strathclyde, UK*

Dracos Vassalos, *SSRC, Univ. of Glasgow and Strathclyde, UK*

Andrzej Jasionowski, *SSRC, Univ. of Glasgow and Strathclyde, UK*

ABSTRACT

The paper demonstrates the applicability of meshless CFD techniques to model complex free-surface problems for marine applications. As an illustration the Smoothed Particle Hydrodynamics method (SPH) has been used to model the progressive flooding of a damaged ship section. A brief description of the numerical method used is given followed by a numerical example. The test case used is a simple qualitative case, but nevertheless a case that clearly demonstrates the power of meshless techniques when it comes to modelling free-surface flow phenomena with moving geometries.

Keywords: *SPH, meshless, flooding*

1. INTRODUCTION

The numerical modelling of the flooding process of a damaged ship is a complicated task. The state-of-the-art of suitable numerical tools today consists of potential theory methods and Reynolds Averaged Navier-Stokes Equations (RANSE) based CFD methods. Potential theory methods are computationally efficient but lack the ability to model violent flows, and it is particularly difficult to accurately model water ingress and egress and the subsequent sloshing inside flooded compartments. Because of this it has long been desirable to use CFD tools to model the flooding process although this is rarely done due to the considerable computational effort involved. CFD methods such as the Finite Volume Method (FVM) with a Volume of Fluid (VOF) treatment of fluid interfaces are capable of modelling violent free surface effects such as overturning waves. Sloshing and flooding can therefore be modelled by this approach but there is an issue with numerical diffusion of density, and thus smeared interfaces, arising from the water-air mixing in the grid cells at the free surface. Also, due to its mesh based nature it is difficult to allow for moving geometries in cases where body forces cannot be used to substitute the movement of the geometry. A computational method that has the capabilities of a VOF method without being constrained by a mesh seems, therefore, very appealing.

While such methods exist in many shapes and forms one of the most well-known and popular meshless methods is Smoothed Particle Hydrodynamics (SPH). SPH was originally presented in the late 1970's for astrophysical purposes. It is a fully Lagrangian method where any fluid or solid medium is represented by a number of discrete particles in space. From a mathematical point of view these particles are simply computational nodes, but since they carry their own mass along with all other field values they are commonly seen as discrete chunks of mass that have their field values interpolated smoothly from their surrounding neighbours. The particles are unconstrained and free to move wherever the force field takes them. Deformation and fragmentation of fluids or solids is therefore easily dealt with. Because of its high degree of flexibility it has been successfully applied in fields such as solid dynamics, computer graphics and visualisation, geophysical flows and biomedical engineering to name a few. It is only during the last few years however that it has caught interest within naval architecture and ocean engineering. That there now is more effort being put into research of SPH for incompressible free-surface hydrodynamics is mainly attributed to the ease of which it can simulate violent nonlinear free-surface flows, and the realisation that SPH in many ways may have its main strengths where the mesh based methods have their main weaknesses.

In this paper the aim is to demonstrate some of the capabilities of SPH to model these types of problems, illustrated by simulating the progressive flooding of a simple two-dimensional damaged ship section.

2. SMOOTHED PARTICLE HYDRODYNAMICS

SPH has its basis in interpolation theory, its fundamental principle being that any function at a particular point in space can be represented by weighted interpolation from the function value at neighbouring points. An integral approximation of the function $f(x)$ can then be expressed as

$$f(x) = \int_V f(x')W(|x-x'|,h)dx' \tag{1}$$

In the above expression $f(x')$ denotes the function value at location x' while $W(|x-x'|,h)$ is the weight function, commonly referred to as the kernel for short. h denotes the smoothing length, which is a measure of the weight functions' reach. The smoothing length is representative of a typical length scale in SPH and is proportional to the cell size of a mesh-based method. The smoothing length need not be constant but may change in time and space.

For the expression given in (1) to hold, the kernel is required to satisfy at least two conditions:

$$\int_V W(|x-x'|,h)\partial x' = 1 \tag{2a}$$

$$\lim_{h\to 0} W(|x-x'|,h) \to \delta(|x-x'|) \tag{2b}$$

These two requirements state that the integral of the kernel must be unity and that as the smoothing length goes to zero the kernel approaches the Dirac delta function. In addition, it is preferred that the kernel has compact support to limit the size of its support domain.

To find the derivative of the function $f(x)$, equation (1) can be differentiated directly:

$$\nabla f(x) = \int_S \nabla f(x')W(|x-x'|,h)ds - \int_V f(x')\nabla W(|x-x'|,h)dx' \tag{3}$$

Given that we assume the kernel has compact support, the surface integral (the first term on the right hand side) vanishes in an unbounded domain. This leaves only the volumetric part, which means that the derivative of any function can be found simply by differentiating the kernel analytically.

In discrete form, the function value of $f(x)$ and its gradient at a location a are given by

$$f(x_a) = \sum_{b=1}^{n} V_b f(x_b)W(|x_a - x_b|,h_a) \tag{4}$$

$$\nabla f(x_a) = \sum_{b=1}^{n} V_b f(x_b)\nabla W(|x_a - x_b|,h_a) \tag{5}$$

This are the particle approximations of the integral approximations given in (1) and (3). The summations in these cases are over all particles located inside the kernels' support

domain (figure 1). V_b denotes the tributary volume of particle *b*, taken as particle mass divided by particle density. The particle mass is usually kept constant which ensures exact mass conservation. $f(x_b)$ denotes the function value at particle *b*.

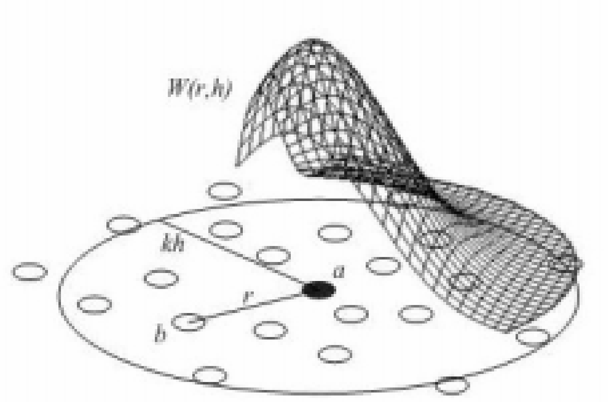

Figure 1. Illustration of the principle behind SPH. The shape of the kernel function centred at particle *a* is illustrated by the mesh (with a cut-out for clarity) while the large circle represent the limit of its support domain.

For standard SPH the kernel function is symmetric with an anti-symmetric first derivative, and typically takes the form:

$$W(r,h) = \frac{\alpha}{h^d}\left|F(s)\right|,\ s = \frac{r}{h} \tag{6}$$

In the above expression *d* is the number of spatial dimensions and $F(s)$ is a scalar function, in most cases Gaussian or spline-based. α is a normalisation constant to ensure that the condition (2a) is satisfied regardless of dimension. The most commonly used weight function, which is also the one used in this paper, is the cubic spline given by:

$$F(s) = \begin{cases} 1 - \frac{3}{2}s^2 + \frac{3}{4}s^3, & 0 \le s \le 1 \\ \frac{1}{4}(2-s)^3, & 1 \le s < 2 \end{cases} \tag{7}$$

Using this function in (6), α takes the value $10/(7\pi)$ in two dimensions.

For the remainder of this paper, the following shorthand notation will be used:

$$W(\left|x_a - x_b\right|, h_a) = W_{ab} \tag{8a}$$

$$\nabla W(\left|x_a - x_b\right|, h_a) = \nabla W_{ab,a} \tag{8b}$$

2.1 Numerical Model

The governing equations are the conservation equations for fluid dynamics, i.e. the mass and momentum conservation equations:

$$\frac{d\rho}{dt} = -\rho \nabla \bullet \mathbf{v} \tag{9}$$

$$\frac{d\mathbf{v}}{dt} = -\frac{1}{\rho}\nabla \boldsymbol{\sigma} + \mathbf{F}_{ext} \tag{10}$$

The only difference from the standard set of equations is that the momentum equation does not contain the convective term since convection is calculated directly by moving the particles. In this paper fluid is modelled as inviscid and only the isotropic part of the stress tensor is used.

<u>Mass Conservation Equation.</u> To discretise the mass conservation equation it is possible to use (5) directly, but it is more accurate to use a symmetrical expression (Monaghan, 1988). An SPH version of (9) can then be taken as

$$\frac{d\rho_a}{dt} = -\rho_a \sum_{b=1}^{n} V_b (\mathbf{v}_b - \mathbf{v}_a) \bullet \nabla W_{ab,a} \tag{11}$$

The term inside the summation is the SPH equivalent of the velocity divergence. The use of a symmetrical version like (11) as opposed to using (5) directly has the advantage that the derivative of a constant function is reproduced exactly. The drawback is that while mass is intrinsically conserved, the consistency between mass, density and volume is not.

This can be mitigated by periodically reinitialising the density field using a higher order interpolation (Colagrossi et. Al., 2003).

Momentum Equation. A similar procedure to that of the velocity divergence could be applied to derive the pressure gradient, but this would lead to an expression that would not conserve linear and angular momentum exactly. In this paper the approach outlined by Vila (1999) has been used, leading to the following form of the pressure gradient term:

$$\nabla P_a = \sum_{b=1}^{n} V_b (P_a \nabla W_{ab,a} - P_b \nabla W_{ba,b}) \qquad (12)$$

Because of the symmetry properties of the kernel it is straight forward to show that the use of this expression in the momentum equation guarantees a local conservation of linear and angular momentum in the absence of external forces.

For numerical stability reasons it is also necessary to employ a small amount of artificial viscosity. In this paper a standard formulation following Monaghan (2003) is used:

$$\mu_{ab} = \begin{cases} h_{av} \dfrac{\mathbf{v}_{ab} \bullet \mathbf{r}_{ab}}{\mathbf{r}_{ab}^{\ 2} + \varepsilon h_{av}}, & \mathbf{v}_{ab} \bullet \mathbf{r}_{ab} < 0 \\ 0, & \mathbf{v}_{ab} \bullet \mathbf{r}_{ab} \geq 0 \end{cases} \qquad (13)$$

$$\mathbf{F}_{visc} = -\alpha \sum_{b=1}^{n} m_b \mu_{ab} \frac{c_{s,av}}{\rho_{av}} \nabla W_{av} \qquad (14)$$

In the above expressions $\mathbf{v}_{ab} = \mathbf{v}_a - \mathbf{v}_b$ and $\mathbf{r}_{ab} = \mathbf{r}_a - \mathbf{r}_b$, and the subscript *av* refers to averaged quantities taken as $f_{av} = 1/2(f_a + f_b)$. The factor $\varepsilon << 1$ is included to guard against a singular expression. The parameter α is typically taken between 0.001 and 0.03 for incompressible flows; in this study a value of 0.01 was used. The use of this artificial viscosity in the momentum equation provides the necessary damping to the system while preserving the momentum conservation properties. It should be noted that this type of artificial viscosity is inherently one dimensional in nature and is therefore too dissipative for shear flows. If the flow is shear dominated a switch should be introduced to reduce the damping in shear (see e.g. Balsara, 1995).

Using (12) and (14) the momentum equation takes the form:

$$\frac{d\mathbf{v}_a}{dt} = -\frac{1}{\rho_a} \nabla P_a + \mathbf{F}_{visc} + \mathbf{F}_{ext} \qquad (15)$$

Equation of State. To close the system of equations a direct link between density and pressure is provided by an equation of state:

$$P_a = \frac{c_s^2 \rho_0}{\gamma} \left[\left(\frac{\rho_a}{\rho_0}\right)^{\gamma} - 1 \right] \qquad (16)$$

For the given equation of state, P_a denotes the pressure at particle *a*, ρ_0 is the reference density of the fluid, c_s is the sound speed and γ is a stiffness constant taken as 7 for water. The use of this equation of state avoids the solution of a Poisson equation of pressure and renders the algorithm fully explicit. The price that has to be paid is that there are always density fluctuations present in the flow field and there is a dependency on sound speed for the maximum allowable time step. To use the real speed of sound would result in time steps far too small for any practical use and therefore an artificial speed of sound is used. This artificial speed of sound must be large enough to ensure that the maximum density variations are sufficiently small to realistically represent incompressible flow, while at the same time be small enough to allow for practical time steps. Since it can be assumed that the density fluctuations are proportional to the Mach number squared (Monaghan, 1994), a speed of sound that is at least 10 times larger than the maximum speed of the bulk flow will result in density variations of 1% or less. When using the Courant (CFL) condition as time step control this result in CFL numbers that are 10 times larger than for a completely incompressible formulation, but as the number of particles increases this is more than offset against the reduction in computational effort per time step. The approach outlined above leads to a weakly compressible formulation where water is modelled as an artificial fluid slightly more compressible than it really is.

Time Integration. The mass conservation equation and momentum equation can be integrated in time by most standard integration schemes. In this paper a second order predictor-corrector scheme (Monaghan, 1994) has been used, but Leap Frog integrators as well as 3rd and 4th order Runge Kutta methods are also common. To ensure numerical stability the following CFL condition has been used to limit the time step:

$$dt = \min\left(\frac{0.25h_{\min}}{c_{s,a} + v_a}\right) \qquad (17)$$

The minimum is taken as the minimum over all fluid particles.

Modelling Geometry. The implementation of wall boundaries in SPH is a well-known difficulty and still a major challenge. There are a variety of methods that can be used, such as repulsive boundary forces (Monaghan, 1994, 2003), ghost particles (Libersky et. Al., 1993) and dummy particles (Morris et. Al., 1997). In this paper a modified version of the repulsive boundary force of Monaghan (1994) has been used to model solid bodies. Boundary particles are placed along the boundary at an approximately equal distance from each other, exerting a short-range boundary force normal to the boundary. This distance dependent force is governed by a Lennard-Jones potential and enters the momentum equation directly as an external body force. While this approach is not as accurate as the use of ghost particles on planar shapes, it allows arbitrarily shaped wall boundaries to be modelled with ease.

3. TEST CASE

To demonstrate the ability of SPH to model progressive flooding of a damaged ship, a simple two dimensional section of a Ro-Ro ship has been chosen as test case (figure 2). The beam of the section is 24.0 metres, with a draft of 6.5 metres. The ship section has a damage opening on the side at the car deck level. A vertical opening consistent with the presence of a staircase allows for progressive flooding to take place from the car deck and all the way down to the auxiliary engine room.

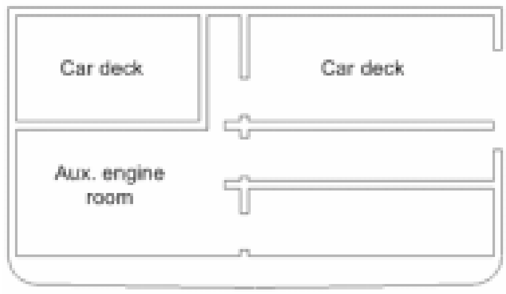

Figure 2. Cross section of the roro ship used as a test case.

In this case to simplify the problem, the ship is subjected simultaneously to prescribed heave and roll motion. The imposed motion is harmonic with untypical heave and roll period of 5.0 and 8 seconds respectively. The amplitude for heave is taken to be 0.5 metres and for roll 9.0 degrees. A train of beamwise incoming regular waves of 1.0 metre amplitude and with a period of 5.0 seconds is generated by a flap type wave maker. In the opposite end of the domain a damping zone was established to reduce the wave reflection from the domain boundary. The damping increased linearly from the beginning of the zone towards the domain boundary. As the ship is forced to move in heave and roll a progressive flooding process will take place. This approach is somewhat simplified, but removing the ship dynamics results in test cases that are well suited for early stage verification due to the reduction in the number of uncertainties present. While no validation is taking place in this case, a set of larger three-dimensional cases will be validated against experiments in the near future as part of ongoing research.

3.1 Initial Setup

The simulation was conducted using a total of 90848 particles. The width of the domain was set to 5 times the beam and the depth to 5 times the draft of the ship (figure 3). The ship geometry and the domain boundaries were modelled as solid walls.

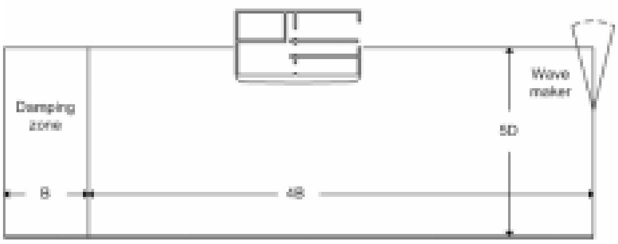

Figure 3. Computational domain.

The initial fluid particle setup was created by a standard grid generator. An in-house pre-processor was used to convert the mesh file into a particle setup by placing a particle in the geometric centre of each cell and subsequently assigning it the volume from that cell. Each particle was then given a smoothing length defined by

$$h_a = 1.5 V_a^{1/d} \tag{18}$$

where V_a is the particle volume and d denotes spatial dimensions. Each particle therefore had an individual smoothing length based on its size and this smoothing length was kept constant during the course of the simulation.

3.2 Results

When the ship starts moving from its initial upright position water enters the damaged area. The incoming wave is small and therefore very little water enters at car deck level. Most of the water enters in the engine control room and progresses into the auxiliary engine room. Since the ship motion is prescribed, sinkage resulting from the flood water is not captured and the sloshing inside the ship can not affect the ship dynamics. The sequence of snap shots below (figures 4a – 4f) show the flooding process in 30 seconds intervals.

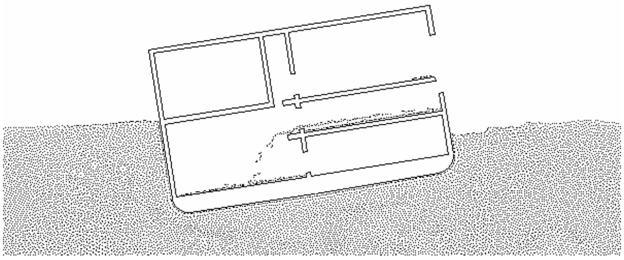
Figure 4a. Flooding after 30 seconds.

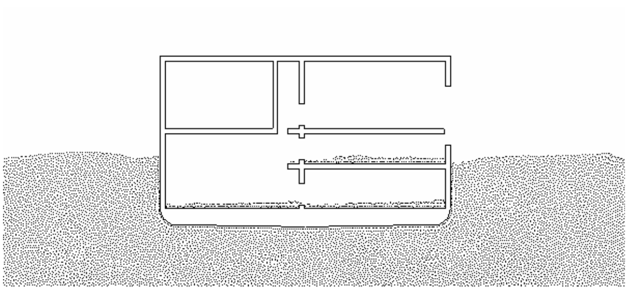
Figure 4b. Flooding after 60 seconds.

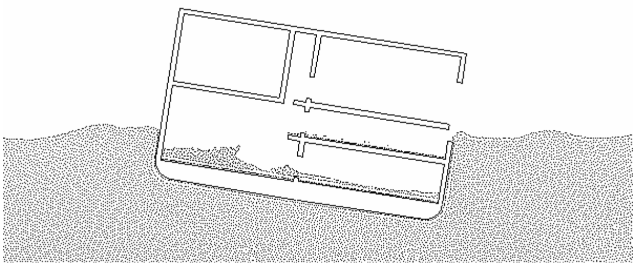
Figure 4c. Flooding after 90 seconds.

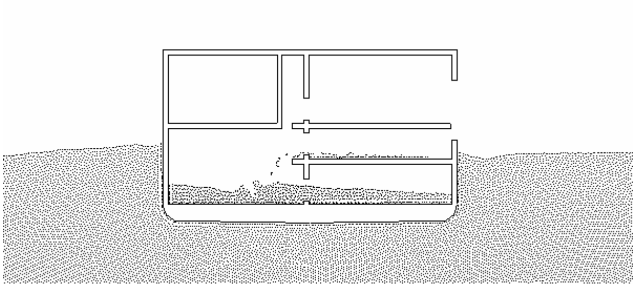
Figure 4d. Flooding after 120 seconds.

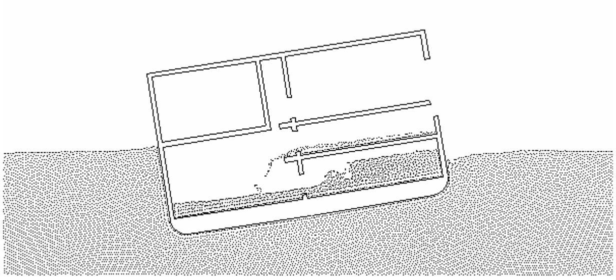

Figure 4e. Flooding after 150 seconds.

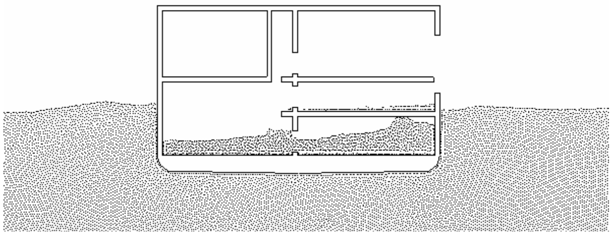

Figure 4f. Flooding after 180 seconds.

This simple example shows that SPH can handle moving geometries with ease, with water being free to flow in and out of the damaged area.

4. DISCUSSION

In the approach adopted in this paper some simplifications were made. While it is fully possible to include ship dynamics, this was not done in this case. A range of similar cases, but in three dimensions, will be run to validate the numerical code before it is extended to include the full ship dynamics. This allows for a validation of forces and moments on the hull as well as free-surface elevation internally and externally without the added uncertainty from the ship dynamics. When satisfactory results are achieved, the full ship dynamics will be incorporated. The inclusion of ship dynamics raises new issues that have to be resolved, in particular related to wave generation, wave reflection on boundaries and solid body dynamics. Accurate wave generation in SPH needs more study to resolve issues concerning wave reflection from domain boundaries as well as issues related to numerical dissipation. The wave reflection problem is significant for simulations that run over several minutes. To reduce the influence from reflected waves, numerical beaches are being developed to dampen waves approaching the domain boundaries. The issue of numerical dissipation is more complicated, but it is believed that an improved interpolation technique can contribute to reduce the numerical damping due to a smoother and more accurate flow field. SPH is a reproducing particle method in the sense that it reproduces existing functions by weighted interpolation. Errors in the interpolation can therefore be readily demonstrated by using SPH to reproduce simple analytic functions. These errors give rise to some well-known practical problems such as tensile instability, sensitivity to particle disorder and lack of angular momentum conservation when the full stress tensor is used. The effect it may possibly have on numerical dissipation is currently not known. It should be pointed out that in general SPH has been shown to be remarkably accurate despite its known difficulties, but resolving the interpolation issues would contribute to further enhancement of the general confidence in the methods' abilities.

Concerning the implementation of boundary conditions, this is still an open issue. In particular, wall boundaries are proving difficult to implement in a mathematically consistent way. The problem is that short of solving large linear systems of equations there are no proper and consistent implementations of wall boundaries for arbitrary geometrical shapes. The approach adopted in this paper is performing well but it should still be seen as a temporary solution that needs to be improved. As for most CFD methods, transmissive (non-reflective) boundaries are also difficult to deal with and as mentioned previously, the common solution is presently the use of damping zones within the domain to dampen waves that approach the domain boundary. Nevertheless, with the gaining popularity of SPH for free-surface hydrodynamics applications it is expected that progress will be seen in this issues in a relative short term.

5. CONCLUSION

A qualitative simulation of the progressive flooding of a two-dimensional ship section has been presented, illustrating the capability of SPH to model this process. While some simplifications of the problem were made in that the ship was subjected to forced motion, the ability to capture the flooding process with the subsequent internal sloshing has been demonstrated.

6. REFERENCES

Balsara, D.S., 1995, “von Neumann Stability Analysis of Smoothed Particle Hydrodynamics – Suggestions for Optimal Algorithms“, Journal of Computational Physics, Vol. 121, pp. 357 – 372

Colagrossi, A. and Landrini, M. 2003, “Numerical Simulation of Interfacial Flows by Smoothed Particle Hydrodynamics”, Journal of Computational Physics, Vol. 191, pp. 448 – 475

Libersky, L.D., Petschek, A.G., Carney, T.C., Hipp, J.R., Allahdadi, F.A. 1993, “High Strain Lagrangian Hydrodynamics”, Journal of Computational Physics, Vol. 109, pp. 67 – 75

Monaghan, J.J. 1994, “Simulating free surface flows with SPH”, Journal of Computational Physics, Vol. 110, pp. 399 – 406

Monaghan, J.J., Kos, A., and Issa, N. 2003, “Fluid Motion Generated by Impact”, Journal of Waterway, Port, Coastal and Ocean Engineering, Vol. 11/12, pp. 250 – 259

Morris, J.P., Fox, P.J., Zhu, Y. 1997, “Modeling Low Reynolds Number Incompressible Flows Using SPH”, Journal of Computational Physics, Vol. 136, pp. 214 – 226

Vila, J.P. 1999, “On Particle Weighted Methods and Smoothed Particle Hydrodynamics”, Mathematical Models and Methods in Applied Sciences”, Vol. 9, pp. 161 – 210