A NOVEL SPH FORMULATION FOR 2-PHASE FLOWS

M. Landrini
INSEAN, The Italian Ship Model Basin
Via di Vallerano 139, 00128, Roma, Italy
Email: maulan@waves.insean.it

A. Colagrossi
INSEAN, The Italian Ship Model Basin
Via di Vallerano 139, 00128, Roma, Italy
Email: andreac@rios13.insean.it

M. P. Tulin
Ocean Engineering Laboratory, UCSB
6740 Cortona Dr., Goleta 93117 (CA)
Email: mpt@engineering.ucsb.edu

INTRODUCTION

In previous Workshops, Fontaine *et al.* (2000), Landrini *et al.* (2001a), a gridless Lagrangian method to simulate breaking and fragmentation in free-surface flows has been developed starting from the SPH method, introduced by Monaghan and co-authors (see *e.g.* Monaghan (1988)).

The resulting algorithm features several improvements in the treatment of boundary conditions, and in stability and efficiency properties. Verification and validation of the code have been performed by comparison with other numerical solutions and experimental data (see *e.g.* Colicchio *et al.* (2002)).

The method has been successfully applied to study breaking waves in shallow water and around ships by Tulin and Landrini (2000). In particular, the structure of the breaking wave pattern around slender ships has been highlighted by combining a 2D+t approximation to the steady flow with the SPH method (Landrini *et al.* 2001a, 2001b). In both cases, cyclical splash up has been observed, with formation of vortical structures and cavities entrapping air, as confirmed by experimental observations.

The long-term evolution of such structures can be influenced by the entrapped air, evolving into a bubbly mixture with peculiar physical properties. The fate of such aerated regions is relevant to oxygenation processes of oceans, diffusion of pollutants, as well as to radar and acoustic signatures of ships. Finally, impact loads are largely affected by air-cushioning effects, as those observed in sloshing flows Faltinsen (2001). In this paper we describe our more recent activity aimed to extend the SPH formulation to multi-phase flows.

SPH-based approaches to multi-phase flows have been already proposed in the literature. The formulation presented in Monaghan (1996), Monaghan *et al.* (1999) is suitable for density ratios of order $O(0.5)$ between the two fluids. Unphysical surface tension effects of numerical origin affect this approach Cummins (1999). Moreover, for smaller values of the density ratios, the method is highly unstable and not applicable. For a collection of particles highly dispersed in a fluid, Monaghan and Kocharyan (1995), Monaghan (1997) adopted correction terms in the fluid-flow equations to take into account the presence of the suspended particles.

Here, we like to consider flows where the two phases are dynamically interacting through a sharp interface, and the two fields have to be described through the fluid-flow equations, without any further approximation. On this ground, we developed a new formulation, described in the next section, which overcomes all the mentioned drawbacks. Preliminary results for a prototype problem are presented in the last section.

FORMULATION

Basic details of the SPH method The essential features of the SPH method are i) the Lagrangian character, allowing self-adaptability to large fluid-domain deformations, and ii) the meshless character removing the burden of building a mesh in a computational domain of complex geometry.

The fluid is divided into a collection of N particles interacting each other through evolution equations of the general form:

$$
\begin{aligned}
\frac{d\rho_i}{dt} &= -\rho_i \sum_j \mathcal{M}_{ij}(\rho_i, \vec{u}_i) \\
\frac{d\vec{u}_i}{dt} &= -\frac{1}{\rho_i} \sum_j \mathcal{F}_{ij}(\rho_i, \vec{u}_i, p_i) + \vec{f}_i \quad . \\
\frac{d\vec{x}_i}{dt} &= \vec{u}_i
\end{aligned}
\tag{1}
$$

The interaction terms $\mathcal{M}_{ij}$, $\mathcal{F}_{ij}$ follow from the manipulation of the equations of mass and momentum balance for an inviscid fluid, respectively, and depend on density ρ_j, velocity $\vec{u}_j$ and pressure p_j of the particles. The last equation in (1) simply represents the Lagrangian evolution of the i-th particle.

In the basic implementation, the interaction terms $\vec{\mathcal{F}}_{ij}$ model the pressure interactions and contain the pressure p_k which here is determined by the value of the density ρ_k through an equation of state of the form

$$
p(\rho) = B\left[\left(\frac{\rho}{\rho_0}\right)^{\gamma} - 1\right] \quad . \tag{2}
$$

The parameters B, ρ_0, γ are chosen to have maximum density oscillations of order $O(1\%)$ of a reference value ρ_0. In practice, this is accomplished by choosing the sound speed $c_s = dp/d\rho$ ten times or more larger than the highest fluid velocity expected in the physical problem. We note that the use of the actual speed of sound in water would imply a time-steps too small for any practical use.

Upon considering a weakly compressible fluid, we can avoid the solution of the Poisson equation for the pressure and the method does not require the solution of an algebraic problem. As a consequence, the memory occupation is proportional to the number of particles, and the efficiency is rather high. Moreover, the particles can be arbitrarily scattered over the fluid domain leading to a completely grid-free method.

The interaction terms can be computed independently of each other. Therefore, the method is explicit and can be easily implemented on parallel computers. The resulting algorithm is rather robust, even for large free-surface fragmentation and folding, efficient, and relatively easy-to-code at least in its most naive implementation. Modeling of no-slip body boundary conditions and of turbulent flows are less obvious. Finally, the stability of the method requires some subtleties. Some of these issues are discussed by Colagrossi *et al.* (2001).

Multi-phase version of the SPH method We collect here, the main differences between the Monaghan formulation and the new formulation here introduced. The discrete SPH equations are obtained by using discrete approximation to the interpolation integral

$$\langle u(\vec{x}_P)\rangle = \int_\Omega u(\vec{x}^\star)\, W(\vec{x}_P - \vec{x}^\star; h)\, dV^\star . \qquad (3)$$

In particular, Monaghan adopted the following approximation to the field and its gradient

$$\langle \vec{u}_i\rangle \simeq \sum_j \vec{u}_j W_{ji}\, dV_j\,, \quad \langle \nabla\vec{u}_i\rangle \simeq \sum_j \vec{u}_j \otimes \nabla W_{ji}\, dV_j\,. \qquad (4)$$

Here and in the following discrete approximations, the kernel function W_{ji} is evaluated at the points $\vec{x}_i$, $\vec{x}_j$, and the gradient operator ∇ is taken with respect to the variable $\vec{x}_i$. We note that $dV_j = m_j/\rho_j$, that is each particle carries a constant mass m_j during the evolution. By using the identities:

$$div(\vec{u}) = \frac{div(\rho\vec{u}) - \nabla\rho\cdot\vec{u}}{\rho} \quad \nabla\mathcal{A} = \frac{\nabla(\rho\mathcal{A}) - \mathcal{A}\nabla\rho}{\rho}\,, \qquad (5)$$

we find the expressions for the divergence and gradient operators:

$$\begin{aligned} div(\vec{u}_i) &= \sum_j (\vec{u}_j - \vec{u}_i)\cdot\nabla W_{ji}\,\frac{m_j}{\rho_i} \\ \nabla\mathcal{A}_i &= \sum_j (\mathcal{A}_j - \mathcal{A}_i)\,\nabla W_{ji}\,\frac{m_j}{\rho_i} \end{aligned} \quad . \qquad (6)$$

Monaghan discretized the pressure gradient by using the identity:

$$\sigma := \frac{1}{\rho} \qquad \nabla p = \frac{\nabla(\sigma p) - p\nabla\sigma}{\sigma} = \left[\nabla\left(\frac{p}{\rho}\right) + p\frac{\nabla\rho}{\rho^2}\right]\rho \qquad (7)$$

and equations (4), to get

$$\nabla p_i = \sum_j \left(\frac{p_j}{\rho_j^2} + \frac{p_i}{\rho_i^2}\right)\nabla W_j(\vec{x}_i)\, m_j\,, \qquad (8)$$

which allows for a formulation locally conservative.

As we will show, with the above discretization formulae applied to each of the fluids forming the multi-phase system, the resulting method exhibits some drawbacks, as illustrated in the following. Here, we just note that these are related to the sharp change of the density across the interface. Therefore, we propose a different formulation based on the discrete approximations

$$\begin{aligned} div(\vec{u}_i) &\simeq \sum_j (\vec{u}_j - \vec{u}_i)\cdot\nabla W_{ji}\, dV_j \\ \langle\nabla\vec{u}(\vec{x}_i)\rangle &\simeq \sum_j (\vec{u}_j - \vec{u}_i)\otimes\nabla W_{ji}\, dV_j \end{aligned} \quad . \qquad (9)$$

The divergence and gradient operators now read:

$$\begin{aligned} div(\vec{u}_i) &= \sum_j (\vec{u}_j - \vec{u}_i)\cdot\nabla W_{ji}\,\frac{m_j}{\rho_j} \\ \nabla\mathcal{A}_i &= \sum_j (\mathcal{A}_j - \mathcal{A}_i)\,\nabla W_{ji}\,\frac{m_j}{\rho_j} \end{aligned} \quad , \qquad (10)$$

which avoid the use of the gradient of the density. The main difference between (10) and Monaghan's (6) is the use of m_j/ρ_j instead of m_j/ρ_i, which becomes crucial for small density ratios. With the same motivations, the pressure gradient is now computed by

$$\nabla p_i = \sum_j (p_j + p_i)\nabla W_{ji}\, dV_j\,, \qquad (11)$$

which is still locally conservative.

A second distinctive feature of the present implementation is related to the Monaghan's velocity correction, the so called *XSPH* formulation. This correction takes into account neighbors velocity through a mean velocity evaluated within the particle support, i.e.

$$\vec{u}_i = \vec{u}_i + \frac{\varepsilon}{2}\sum_j \frac{m_j}{\bar{\rho}_{ij}}(\vec{u}_j - \vec{u}_i)W_{ji} \qquad \bar{\rho}_{ij} := \frac{\rho_i + \rho_j}{2}\,.$$

For particles i close to the two-fluids interface the mean density $\bar{\rho}_{ij}$ is wrongly evaluated and the *XSPH* correction leads to wrong results. In our implementation, when considering one medium, the XSPH correction is computed without considering influence of the other media possibly present.

A SAMPLE CASE

As a test case, we consider an initially circular bubble of lighter fluid Y underneath the interface, separating the two phases X and Y, respectively (*cf.* Fig. 1). A no-penetration boundary condition is enforced on the outer boundary. In the computations, the symmetry has been enforced explicitly and only half of the fluid domain will be shown. We also note that for such problem, surface tension may be physically relevant and the present results are mainly meant to show the capabilities of the present formulation. Surface tension can be introduced as shown by Morris (2000), or approximated by modifying the equation of state as suggested by Nugent (2000). The latter approach is much simpler and also provide a simple mechanism to control the numerical fragmentation of the interface. This issue will be further discussed at the Workshop.

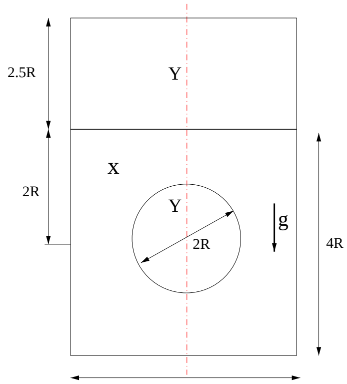

Figure 1. Rise of a gas bubble through water. Sketch of the problem and adopted nomenclature.

Figure 2 (left) shows the result obtained by the the Monaghan model for the density ratio $\rho_Y/\rho_X = 0.5$. The interface shows large oscillations and the whole evolution (not reported) is affected by an unphysical numerical surface tension (as discussed by Cummins (1999)). In the right plot of the same figure, we report the domain configuration obtained by the new formulation. Apparently, the interface is smoother and the following evolution is confirmed by reference solutions obtained by a Navier-Stokes code with Level Set to capture the interface (solid line in the right plot).

For a smaller density ratio, Fig. 3 left, $\rho_Y/\rho_X = 0.1$, a strong instability appears soon which prevents the computation to proceed, as can be argued by the highly irregular velocity field in proximity of the bubble. The present improved formulation, right plot, does not exhibit such behavior and the computation can be arbitrarily prolonged in time.

As a last result, we show the more stringent case $\rho_Y/\rho_X = 0.001$, Fig. 4. Time increases from left to right and from top to bottom ($t\sqrt{g/R}$ =1.897, 2.530, 3.162, 3.795, 4.427, 5.060, 5.692, 6.325). The present solution is compared with a viscous solution obtained by using a level-set technique to capture the interface between the two fluids. In the heavier fluid, the SPH particles are colored according to their initial vertical coordinate, giving a simple and effective representation of their motion.

As time increases, the bubble deforms and rises, pushing up the interface and forming a central hump. The thickness of the bubble near the line of symmetry gradually diminishes. Eventually, the bubble splits in two parts, and two counter-rotating vortices are created, inducing an upward motion of water from the bottom of the tank. The separation distance between the two structures slightly increases, while the central hump moves downward under the restoring action of gravity. In the SPH simulation, the air bubble breaks and some air escapes upwards. The agreement between the two solutions is reasonable, at least at the beginning of the evolution. Later stages are characterized by more pronounced differences, though the solutions remain qualitatively similar.

Using the new SPH model, we like to investigate the the influence of the air on the breaking and post-breaking evolution of water waves. New results on this issue will be presented at the Workshop.

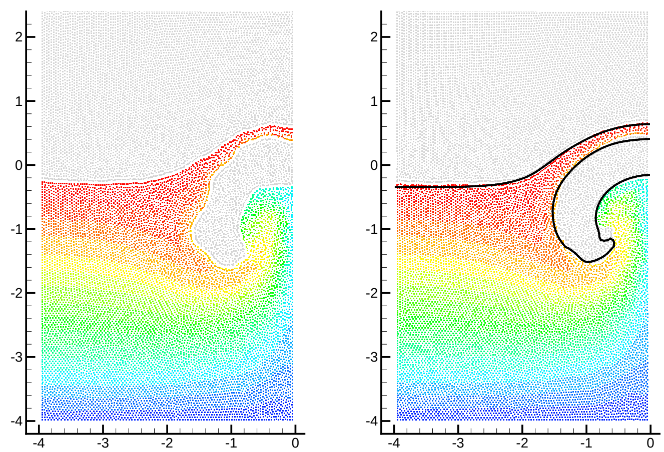

Figure 2. Left: standard SPH solution; right: improved model. $\rho_Y/\rho_X = 0.5$, $t\sqrt{(g/R)} = 4.427$. The solid lines in the right plot represent a Navier-Stokes solution based on the Level-Set algorithm for capturing the interface.

ACKNOWLEDGEMENTS

This work has been supported as part of a program for the simulation of ship breaking waves by the Ship Hydrodynamics Program of ONR, Dr. Pat Purtell Program Manager. INSEAN research activity is supported by the Italian *Ministero per le Infrastrutture ed i Trasporti* through INSEAN Research Program 2000-2002.

REFERENCES

A. Colagrossi, M. Landrini, M. P. Tulin (2001), *A Lagrangian Meshless Method for Free-surface Flows* Fourth Numerical Towing Tank Symposium, Hamburg.

G. Colicchio, A. Colagrossi, M. Greco, M. Landrini (2002), *Free-surface Flow After a Dam break: A Comparative Study.* Ship Tech. Res.

A. Colagrossi, M. Landrini, M.P. Tulin (2000), *Near Shore Bore Propagation and Splashing Processes: Gridless Simulations*, Monterey (CA), OEL Tech. Report 00-224.

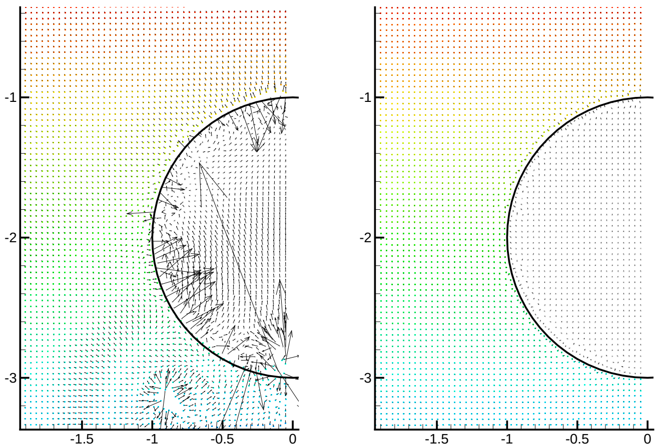

Figure 3. Growth of instabilities at the bubble interface. Left: standard SPH solution; right: improved model. $\rho_Y/\rho_X = 0.1$, $t\sqrt{(g/R)} = 0.127$.

S.J. Cummins and M.Rudman (1999), *An SPH Projection Method*, J. Comp. Phys. **152**, pp. 584-607.

O.M. Faltinsen (2001), *Slamming with Application to Planing Vessels, Green Water Loading and Sloshing*, Hydrodynamics in Ship and Ocean Engineering, Ed. M. Kashiwagi, pp. 27-58.

E. Fontaine, M. Landrini, M.P. Tulin (2000), *On modeling the post-breaking phase: splashing*. Proc. of 15th Int. Work. Water Waves Float. Bodies, Eds. T. Miloh and G. Zilman.

J.J. Monaghan (1988), *An Introduction to SPH*, Comp. Phys. Comm. 48, pp. 89-96.

J.J. Monaghan (1996), *Gravity currents and solitary waves*, Physica D **98**, 523.

J.J. Monaghan, A. Kocharyan (1995), *SPH simulation of multi-phase flow*, Comp. Phys. Comm. **87**, pp. 225-235.

J.J. Monaghan (1997), *Implicit SPH Drag and Dusty Gas Dynamics*, J. Comp. Phys. **138**, pp. 801-820.

J.J. Monaghan, R.A.F. Cas, A.M. Kos, M. Hallworth (1999), *Gravity currents descending a ramp in a stratified tank*, J. Fluid Mech. **379**, pp. 39-69.

J.P. Morris (2000), *Simulating Surface Tension with Smoothed Particle Hydrodynamics*, Int. J. Numer. Methods Fluids, **33**, pp. 333-.

S. Nugent, H.A. Posch (2000), *Liquid drops and surface tension with smoothed particle applied mechanics.* Physical Review E, **62** (4), pp.4968-4975, Part A

M. Landrini, A. Colagrossi, M. P. Tulin (2001a), *Breaking Bow and Stern Waves: Numerical Simulations*, Proc. of 16th Int. Work. Water Waves Float. Bodies, Hiroshima (Japan).

M. Landrini, A. Colagrossi, M. P. Tulin (2001b), *Numerical Studies of Breaking Bow Waves Compared to Experimental Observations* 4th Numerical Towing Tank Symposium, Hamburg (Germany).

M. P. Tulin, M. Landrini (2000), *Breaking waves in the ocean and around ships* Proc. of 23rd ONR Symp. on Naval Hydrodynamics, Val de Reuil, France.

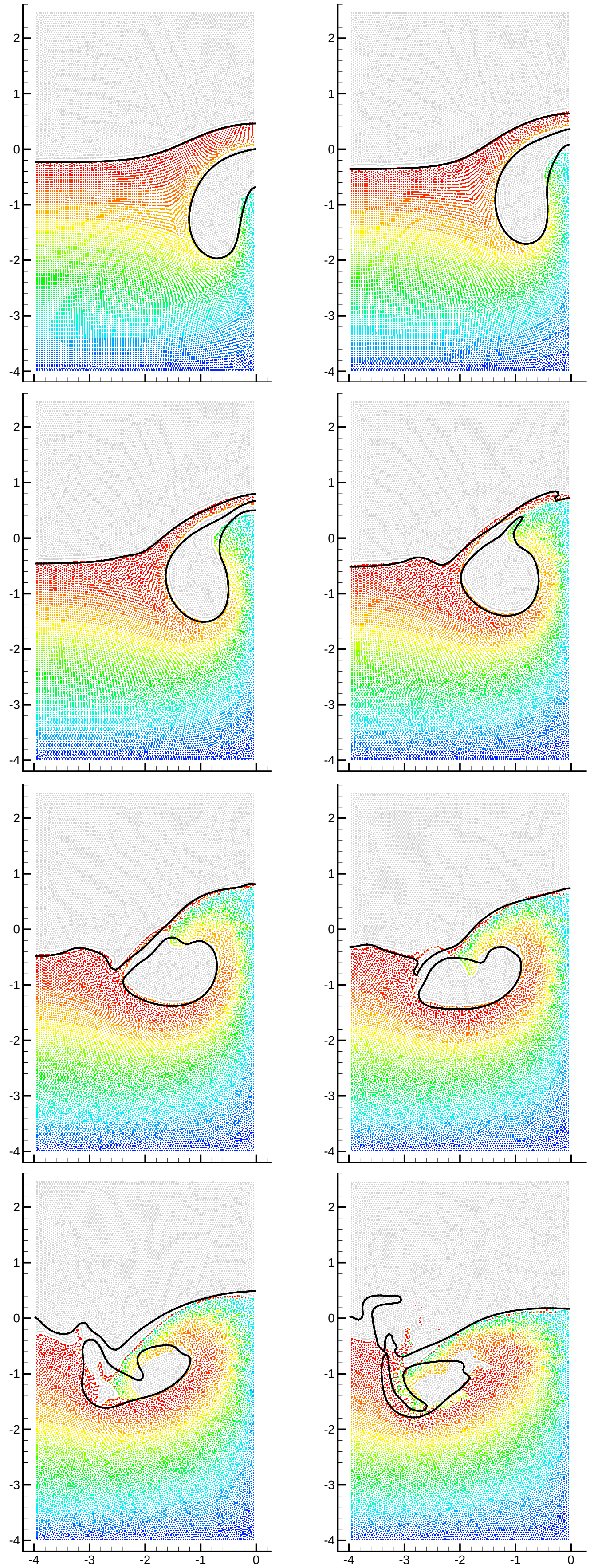

Figure 4. Evolution of an air bubble ($\rho_Y/\rho_X = 0.001$). The SPH solution is compared with a Navier-Stokes solution by Level-Set technique for interface capturing (solid lines).

Discussion Sheet

Abstract Title :	A Novel SPH formulation for 2-phase flows		
(Or) Proceedings Paper No. :	25	***Page :***	095
First Author :	Landrini, M (Colagrossi, A presented)		
Discusser :	Xiao Bo Chen		

Questions / Comments :

You mentioned in the case of the rise of a gas bubble through water, that there exists "Numerical" surface tension. How is it generated? Does it play the same role as the real surface tension which you neglected in the mathematical model?

Author's Reply :
(If Available)

Author did not respond.

Discussion Sheet

Abstract Title :	A novel SPH formulation for two-phase flows		
(Or) Proceedings Paper No. :	25	***Page :***	95
First Author :	M. Landrini, A. Colagrossi and M.P. Tulin		
Discusser :	John Grue		

Questions / Comments :

Could you include the effect of viscosity in your formulation, and what would be the effect of the value of the Reynolds number?

Author's Reply :
(If Available)

So far, we have never applied the SPH technique to the solution of the Navier-Stokes equations.

In the literature, successful applications for incompressible low Reynolds-number flows have been presented by Takeda et al. (1994) and Morris et al. (1997).

SPH approaches to compressible turbulent flows have been reported Pope (1995), Welton and Pope (1997), Welton (1998).

Finally, by noticing the similarity between the finite interaction-radius in SPH and the sub grid-scale concept of LES, Bicknell (1991) suggested the possibility of developing Large Eddy Simulations based on SPH.

Meshless Eulerian methods for the Navier-Stokes equations have been discussed e.g. by Wagner and Liu (2000), Zhang et al. (2002).

G.V. Bicknell,
The Equations of Motions of Particles in Smoothed Particle Hydrodynamics
SIAM J. Sci. Stat. Comput., 12, 5, 1198-1206 (1991)

S.B. Pope,
Particle Method for Turbolent Flows: Integration of Stochastic Model Equations,
J. Comp. Phys., 117, 332--349 (1995)

W.C. Welton, S.B. Pope,
PDF Model Calculations of Compressible Turbolence Flows Using SPH,
J. Comp. Phys., 134, 150--168 (1997)

W.C. Welton,
Two-Dimensional PDF/SPH Simulations of Compressible Turbolent Flows,
J. Comp. Phys., 139, 410--443 (1998)

G.J. Wagner, W.K. Liu,
Turbolence simulation and multiple scale subgrid models
Comp. Mech., 25, 117--136 (2000)

L.T. Zhang, G.J. Wagner, W.K. Liu,
A Parallelized Meshfree Method with Boundary Enrichment for Large--Scale CFD,
J. Comp. Phys., 176, 483--506 (2002)