TWO ATTEMPTS OF TURBULENCE MODELLING IN SMOOTHED PARTICLE HYDRODYNAMICS

D. VIOLEAU

Research Engineer, Laboratoire National d'Hydraulique et Environnement
EDF R&D, 6 quai Watier 78400 Chatou, France - E-mail : damien.violeau@edf.fr

S. PICCON

Université de Paris VI, 4 place Jussieu 75005 Paris, France

J.-P. CHABARD

Deputy Director, Modelling and Information Technologies Department
EDF R&D, 1 avenue du Général De Gaulle 92141 Clamart cedex, France
E-mail : jean-paul.chabard@edf.fr

Smoothed Particle Hydrodynamics (SPH) is a lagrangian numerical technique which proved its ability to reproduce a large variety of flows. However, the turbulent effects are often neglected when considering SPH simulations, and to date it appears that no specific turbulence model is available for SPH. Two models are presented here, one based on an eddy viscosity assumption, the other one on a Langevin process. Both are tested in the case of a Poiseuille flow in a pipe. Velocity profiles are in good agreement with theory ; particularly the log-law is well reproduced near the walls.

1 The Smoothed Particle Hydrodynamics numerical method

Smoothed Particle Hydrodynamics (SPH) is a fully lagrangian numerical method which is today used to model fluid dynamics (Monaghan 1994). SPH is a particle method, which does not require a grid to calculate spatial derivatives. The equations of motion and continuity are expressed in terms of ordinary differential equations where the body forces become classical forces between particles. SPH is robust and can predict complicated flows. The main features of SPH are presented in this section ; for more information about the basis of this method, we refer to Monaghan (1992).

1.1 The SPH governing equations

In the SPH formalism, each particle a (which may be understood as a small volume of fluid), located at $\mathbf{r}_a$, has a constant mass m_a and carries a density ρ_a, a pressure p_a and a velocity vector $\mathbf{u}_a$. The classical equation of motion for each particle a is then

$$\frac{d\mathbf{u}_a}{dt} = -\sum_b m_b \left(\frac{p_a}{\rho_a^2} + \frac{p_b}{\rho_b^2} + \Pi_{ab} \right) \nabla_a W_h\left(r_{ab}\right) + \mathbf{F}_a \tag{1}$$

in which the summation runs over all the particles b inside the domain. The first two terms correspond to the pressure gradient, $\mathbf{F}_a$ is a body force (e.g. gravity), and Π_{ab} is a viscous term depending on the particles relative positions and velocities :

$$\Pi_{ab} = -\frac{16\nu}{\rho_a + \rho_b} \frac{(\mathbf{u}_a - \mathbf{u}_b)\cdot(\mathbf{r}_a - \mathbf{r}_b)}{r_{ab}^2 + \eta^2} \tag{2}$$

where ν is the kinematic viscosity of the fluid, and η is a small parameter chosen to keep the denominator nonzero. In equation (1), W_h is an interpolating kernel depending on the particle distance r_{ab}, and ∇_a denotes the gradient with respect to the co-ordinates of particle a. Usually, the kernel has a compact support proportional to a « smoothing lenght » h. Therefore, the contribution of a particle b to the momentum of particle a vanishes if r_{ab} is large enough ; hence a interacts with a finite number of particles. The momentum equation (1) can be considered as an approximation of the Navier-Stokes equation, and is used to calculate the particle velocities with a first order explicit scheme. A suitable SPH form of the continuity equation is given by

$$\frac{d\rho_a}{dt} = \sum_b m_b (\mathbf{u}_a - \mathbf{u}_b)\cdot \nabla_a W(r_{ab}) \tag{3}$$

This equation is used to evaluate the particle densities at each time step. Since SPH works in a compressible formalism, an equation of state is required to deduce the pressure of the particles from their densities. For water, the following form is considered :

$$p_a = \frac{\rho_0 c_0^2}{\gamma}\left[\left(\frac{\rho_a}{\rho_0}\right)^\gamma - 1\right] \tag{4}$$

where ρ_0 is a reference density, c_0 the speed of sound and γ = 7. When particles are locally collapsing, the density increases with respect to the continuity equation. Then because of the high value of c_0, a small density fluctuation yields a large pressure gradient, which acts in the momentum equation to keep the particle distance almost constant.

The authors have developed a code based on SPH. This code was validated on various test cases, including a schematic dam break and gravity wave propagation in a flume (Violeau et al. 2000), and was used in the present work.

1.2 Example : laminar Poiseuille flow in a pipe

The ability of SPH to predict flow characteristics under laminar conditions can be examined with a Poiseuille flow in a pipe. Here 1600 particles are located in a square of size $2e = L = 0.1$ m (see figure 1), the upper and lower boundaries (parallel to the x-axis) being rigid walls. The flow is longitudinally periodic (i. e. each particle which leaves the domain through a vertical boundary immediately goes back through the opposite one). At the initial time step, the velocities are set to zero, and the particles start moving under the effect of a constant horizontal force F, playing the role of a pressure gradient (gravity is not considered here). The velocity of the boundary particles (i. e. located on the walls) is set to zero as a boundary condition, while the other particles accelerate until they reach a steady parabolic profile of mean velocity $U = Fe^2 / 3\nu$. Hereafter, the Reynolds number is defined by $\mathrm{Re} = eU / \nu$.

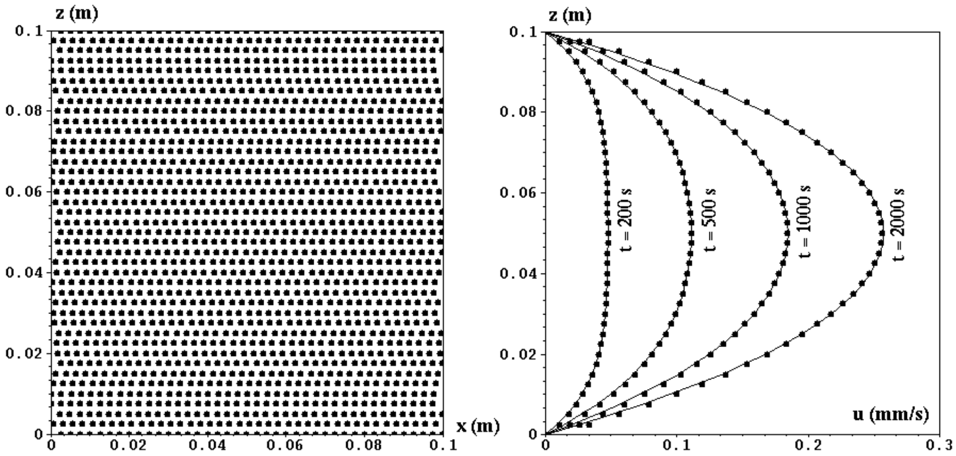

Fig. 1 - Laminar Poiseuille flow in a pipe with SPH. Left part : location of the 1600 particles at t = 2000 s. Right part : computed velocity profiles (solid circles) at different times for Re = 10, compared with theory (solid lines).

Figure 1 represents the velocity profiles obtained with SPH at different times for Re = 10, showing a good agreement with theory. However, to obtain this result it was necessary to introduce « mirror » particles outside the walls, as defined by Morris et al. (1997). These fictitious particles are needed to make a correct prediction of the wall shear stress. The previous result indicates that the SPH form of diffusion given by equation (2) is suitable to reproduce the viscous effects at low Reynolds numbers.

2 An eddy viscosity model for SPH

2.1 Description of the model

To date, it seems that no specific turbulence model has ever been designed for SPH. However, at high Reynolds numbers, it is common to describe the turbulent effects through a first order closure model. The eddy viscosity assumption, for example, consists of a relation between the Reynolds stress tensor and the mean velocity gradients. The momentum equation is then analogous to the Navier-Stokes equation, but the eddy viscosity ν_T is considered in place of ν, and the velocities are Reynolds-averaged. In the SPH formalism, the diffusion term occurring in (1) could then be written as

$$\tilde{\Pi}_{ab} = -8 \frac{\nu_{T,a} + \nu_{T,b}}{\rho_a + \rho_b} \frac{\left(\langle \mathbf{u} \rangle_a - \langle \mathbf{u} \rangle_b\right) \cdot \left(\mathbf{r}_a - \mathbf{r}_b\right)}{r_{ab}^2 + \eta^2} \quad (5)$$

where the brackets refer to the Reynolds average. The eddy viscosity can be calculated from a mixing length model, by

$$\nu_T = L_m^2 \sqrt{2 S_{ij} S_{ij}} \tag{6}$$

where S_{ij} are the components of the rate of strain tensor. Since they depend on the mean velocity gradients, we need an SPH form for them. Starting from the basic principles of SPH (Monaghan 1992), it is easy to see that the following form is correct :

$$\frac{d\langle u_i \rangle_a}{dx_j} = -\frac{1}{\rho_a} \sum_b m_b \left(\langle u_i \rangle_a - \langle u_i \rangle_b \right) \frac{\partial W}{\partial x_{j,a}} (r_{ab}) \tag{7}$$

where the indices i and j denote the components, while a and b refer to the particles.

2.2 Application to a turbulent Poiseuille flow

The previous considerations are applied to a high Reynolds Poiseuille flow. In the lower part of the pipe, the mixing length attached to a particle is defined as $L_m = \kappa \min(z\ ;\ 0.2e)$, where $\kappa = 0.41$ is the Karman constant. In the upper part, similar values are prescribed. For boundary conditions, a wall function approach is used : the boundary particles are not considered to be located on the walls but at a small distance $\delta = 50\nu / u_*$, where u_* is the shear velocity. According to the log-law, the velocity and eddy viscosity of these particles a are then prescribed as

$$\begin{aligned} \frac{u_a}{u_*} &= \frac{1}{\kappa} \log \frac{\delta u_*}{\nu} + 5.2 \\ \nu_{T,a} &= \kappa u_* \delta \end{aligned} \tag{8}$$

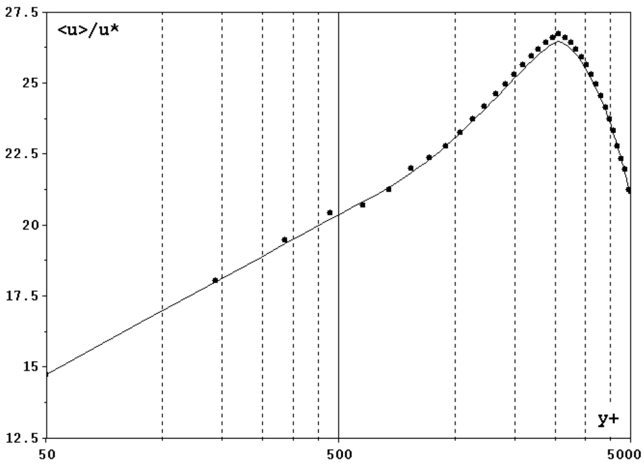

Fig. 2 - Turbulent Poiseuille flow in a pipe with SPH, using the eddy viscosity model. Computed mean velocity profiles after 1.0 s (solid circles) for Re = 64,000, compared with theory (solid line). The horizontal axis represents the dimensionless distance to the lower wall in log-scale.

Note that u_* can be deduced from the horizontal force *F*, by an energy balance : $u_*^2 = Fe$. At the initial time step, we prescribe a parabolic profile having the expected mean velocity. Figure 2 shows the velocity profiles along the dimensionless distance to the lower wall ($y^+ = yu_* / \nu$ y) for a Reynolds number of 64,000, after convergence. The velocity profile shows a very slight discrepancy with theory, but as expected, a log-law develops near the walls. This seams to indicate that the model is appropriate for turbulent mixing problems, or when considering spatially-varying viscosity. However, the applications are preferably restricted to shear flows.

3 Langevin models

3.1 Theoretical aspects

Among the various existing turbulence models, the stochastic approach seem to be particularly relevant for lagrangian modelling. The main idea is to prescribe the particle velocity as a random process, with properties fulfilling the turbulence theoretical hypotheses (Pope 1994). The heart of this approach is the Generalized Langevin Model (GLM), which consists of writing the particle acceleration as

$$d\mathbf{u} = -\frac{1}{\rho}\nabla\langle p\rangle + \mathbf{G}\big(\mathbf{u} - \langle\mathbf{u}\rangle\big)dt + \sqrt{C_0 \varepsilon dt}\,\xi \qquad (9)$$

Here C_0 is a constant, ε is the turbulent dissipation rate and ξ a random vector statistically non correlated with velocities, i.e. satisfying for all *i* and *j*

$$\begin{aligned} \langle \xi_i \xi_j \rangle &= \delta_{ij} \\ \langle \xi_i u_j \rangle &= 0 \end{aligned} \qquad (10)$$

It should be emphasised that the velocities considered in (9) are not averaged. The term $d\mathbf{u}$ in the left-hand side refer to a substantial derivative. Starting from equation (9), it is easy to deduce the behaviour of the average velocity co-ordinates (Minier 1998) :

$$\frac{D\langle u_i \rangle}{dt} = -\frac{1}{\rho}\frac{\partial \langle p \rangle}{\partial x_i} - \frac{\partial \langle u'_i\, u'_j \rangle}{\partial x_j} \qquad (11)$$

where the left-hand side denotes the substantial derivative following the mean motion, and the prime refers to the fluctuations, i. e. $u'_i = u_i - \langle u_i \rangle$. We see that the GLM naturally yields the Reynolds equation. Many other advantages of this model are discussed by Pope (1994), but the most interesting feature for our work is that different closure models can be invoked by specifying the tensor $\mathbf{G}$. Here, we define the tensor's components G_{ij} as

$$G_{ij} = -\frac{1}{2} C_1 \frac{\varepsilon}{k} \delta_{ij} + C_2 \frac{\partial \langle u_i \rangle}{\partial x_j} \tag{12}$$

where k is the turbulent kinetic energy, ε the dissipation rate, and C_1 and C_2 are constants. As shown by Durbin & Speziale (1993), this leads to the LRR-IP model (Launder et al. 1975), appropriate for shear flows :

$$\frac{D\langle u'_i u'_j \rangle}{dt} = -C_1 \frac{\varepsilon}{k}\left(\langle u'_i u'_j \rangle - \frac{2}{3} k\delta_{ij} \right) - C_2 \left(P_{ij} - \frac{2}{3} P\delta_{ij} \right) + P_{ij} - \frac{2}{3}\varepsilon\delta_{ij} \tag{13}$$

where P_{ij} are the components of the production tensor, and P is the turbulent production (see Pope 1994). The model constants C_1 and C_2 are generally set to 1.8 and 0.6 respectively. Strict consistency requires the following value of C_0 (Pope 2000) :

$$C_0 = \frac{2}{3}\left(-1 + C_1 + C_2 \frac{P}{\varepsilon} \right) \tag{14}$$

3.2 *A Generalized Langevin Model for SPH*

Modelling turbulence through a GLM in SPH consists of defining the velocities accordingly to (9) and (12), while the pressure gradient keeps the form indicated in (1) :

$$\frac{d\mathbf{u}_a}{dt} = -\sum_b m_b \left(\frac{\langle p \rangle_a}{\rho_a^2} + \frac{\langle p \rangle_b}{\rho_b^2} \right) \nabla_a W_h(r_{ab}) - \frac{1}{2} C_1 \frac{\varepsilon_a}{k_a} \mathbf{u}'_a + C_2 \nabla \langle \mathbf{u} \rangle_a \cdot \mathbf{u}'_a + \sqrt{\frac{C_0 \varepsilon_a}{\delta t}} \xi_a \tag{15}$$

It is important to notice that equation (13) is not solved, but is virtually included in (15), as well as the Reynolds equation (11), which is a remarkable point.

The components of the mean velocity gradient $\nabla \langle \mathbf{u} \rangle_a$ can be evaluated from (7). An SPH form of the Reynolds average is defined by using a kernel estimation (Pope 2000) :

$$\langle \mathbf{u} \rangle_a = \sum_b \frac{m_b}{\rho_b} \mathbf{u}_b W_h(r_{ab}) \tag{16}$$

This operator can also be used to compute the mean pressure $\langle p \rangle_a$. Here, the local values of k and ε are estimated from a mixing length model, specifying

$$\begin{aligned} \varepsilon_a &= 2\nu_{T,a} \left(S_{ij} S_{ij} \right)_a \\ k_a &= \varepsilon_a \nu_{T,a} / C_\mu \end{aligned} \tag{17}$$

with $C_\mu = 0.09$, as usual. In (17), the S_{ij} are defined using the averaged velocities. Note that estimating the dissipation rate through the true velocity gradients would be a mistake, since the fluctuations of random velocities do not reproduce the small eddies. On the other hand, it could be possible to calculate k with a small error, by averaging the $u'_i u'_i$ resulting from the GLM process (15) at the previous time step, because the large eddies carry the main part of this energy. However, the method given by (17) was chosen here for simplicity.

3.3 Validation on the case of the turbulent Poiseuille flow

Under the same conditions than in section *2.2*, a turbulent Poiseuille flow was simulated with SPH using the Langevin model previously described. The boundary conditions consist of a wall function for the boundary particles velocities, as defined by (8).

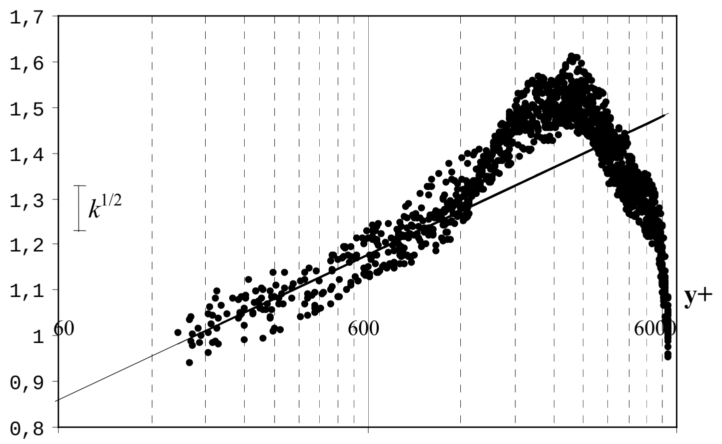

Fig. 3 - Turbulent Poiseuille flow in a pipe with SPH, using the Generalized Langevin Model. Computed mean velocity profiles after 1 s (solid circles) for Re = 64,000, compared with the log-law (solid line). The horizontal axis represents the dimensionless distance to the lower wall in log-scale.

The velocity profiles, presented in figure 3, show again a log-law near the wall. Large fluctuations are visible, due to the fact that the mean operator given by (16) is not really a Reynolds average, but a kind of Large Eddy Simulation (LES) filter. On figure 3, it is visible that the order of magnitude of these fluctuations is $k^{1/2}$. On the other side, in contrast with the method described in section **2**, the GLM method applies for a number of different flows.

4 Conclusions and further work

Two turbulence models for the Smoothed Particle Hydrodynamics (SPH) numerical method were developed and presented here. In the particular case of a Poiseuille flow in a pipe, they show a qualitatively correct behaviour of the turbulent velocity profiles. The Langevin model seems very attractive and more appropriate. However, the model can be improved by specifying a better Reynolds average operator within the SPH formalism. On the other hand, the eddy viscosity model is probably the best balance between speed and accuracy.

We must highlight the fact that, to the authors knowledge, literature about turbulence for SPH is still very poor. Some authors, like Gotoh (2001) and Hoogerbrugge & Koelman. (1992), have investigated turbulence modelling in the framework of other particle methods, but it seems that the present work is presented for the first time, though Welton (1998) attempted the introduction of a Simplified Langevin Model (SLM) in an SPH code.

Further work will include a 3-dimensional modelling of turbulence, based on a LES approach.

Acknowledgements

The authors gratefully acknowledge Prof. D. Laurence, Senior Engineer at Electricité de France and Professor at the University of Manchester (UMIST), for his helpful contribution.

References

1. P.A. Durbin and C.G. Speziale, Realizability of second moment closure via stochastic analysis (CTR Manuscript 184, 1993).
2. H. Gotoh, Sub-particle scale turbulence model for the MPS method, Comput. Fluid Dynamics J. **2**(4), 339,347 (2001).
3. P.J. Hoogerbrugge and J.M.V.A. Koelman, Simulating macroscopic hydrodynamic phenomena with Dissipative Particle Dynamics, *EuroPhys. Lett.*, **19**(1992), 155-160.
4. B.E. Launder, G.J. Reece and W. Rodi, Progress in the development of Reynolds stress turbulence closure. *Journ. Fluid Mech.* **68**, 537 (1975).
5. J.P. Minier, Lagrangian stochastic modelling of turbulent flows (lecture note of the Von Karman Institute, 1998).
6. J.J. Monaghan, Smoothed Particles Hydrodynamics. *Annu. Rev. of Astroph.* **30**, 543-74 (1992).
7. J.J. Monaghan, Simulating free surface flows with SPH. *Journ. Comput. Phys.* **110**, 399-406 (1994).
8. J.P. Morris, P.J. Fox and Y. Zhu, Modelling low Reynolds number incompressible flows using SPH, *Journ. Comput. Phys.* **136**, 214-226 (1997).
9. S.B. Pope, Lagrangian PDF methods for turbulent flows. *Annu. Rev. Fluid. Mech*. **26**, 23-63 (1994).
10. S.B. Pope, Turbulent flows (Cambridge University Press, 2000).
11. D. Violeau, C. Chauvelier-Alario, A. Foucart, M. Benoit, Development of a numerical wave flume using Smoothed Particle Hydrodynamics, 27th ICCE conf., Sydney, 2000.
12. W.C. Welton, Two-dimensional PDF/SPH simulations of compressible turbulent flows, *Journ. Comput. Phys*. **139**, 410-443 (1998).