International Colloquium on Jets, Wakes and Shear Layers
Melbourne, Australia, 18–20 April 1994

A SIMULATION OF FIXED AND MOVING JETS

Joe Monaghan
Department of Mathematics, Monash University,
Clayton, Victoria, 3190, AUSTRALIA
and
Mark Thompson and Kerry Hourigan
CSIRO Division of Building, Construction and Engineering,
P.O. Box 56, Highett, Victoria, 3190, AUSTRALIA

ABSTRACT

The smoothed particle hydrodynamics method (SPH) has been used to simulate several free surface flows. We show that the method can also be applied to simulate fixed and moving jets. We benchmark the method by applying it to steady jets emerging from static pipes for which, in the two dimensional case, complex variable methods can be used to calculate very accurate results which are found to agree closely with the SPH results. We then apply the method to moving two dimensional jets. In particular, we simulate the rotation of sprinklers in both the normal mode, and in the reverse mode (sucking fluid into a sprinkler immersed in a tank).

INTRODUCTION

Free surface flows of incompressible, or nearly incompressible fluids, occur in a wide range of industrial and enviromental problems. The industrial problems include sloshing in tanks, splashing, continuous casting and fuel filling. In the enviroment they include surface wave propagation and breaking, and the flow of dense fluids (mud or volcanic magma) into the ocean.

The simulation of these flows is difficult because boundary conditions are required on an arbitrarily moving surface. The MAC method (Harlow and Welch 1965), which uses particles to define the surface, and finite differences to solve the hydrodynamic equations, is the most flexible and robust of the available numerical methods. It has been applied to a wide variety of problems and extended to deal with moving boundaries and simplified (for references see Harlow (1988)), but it remains complicated to implement.

An attractive alternative for free surface problems is the Lagrangian particle method SPH (smoothed particle hydrodynamics). SPH does not require a grid and, because it uses moving particles, it is extremely robust and flexible. SPH has been used to simulate compressible fluids in astrophysics (Lucy 1977, Gingold and Monaghan 1977, Monaghan 1988, 1992), and the fracturing of elastic solids (private communication Benz and Melosh 1993).

Recently it has been shown that SPH can be extended to simulate fluids like water which are nearly incompressible (Monaghan 1994). For these fluids the normal approach is to replace them by an artificial fluid which is incompressible. The SPH simulation replaces the real fluid by an artificial fluid which is more compressible than the real fluid. The artificial fluid still has a speed of sound which is much larger than the bulk speed of the fluid and the density fluctuations are small. For the simulations we describe here the fluctuations are typically one to two percent.

Boundaries may be modelled very simply by using boundary particles which impose forces on the fluid (Peskin 1977, Sulsky and Brackbill 1991). This idea is based on the fact that real boundaries are produced by atoms or molecules which exert forces on the fluid. The real boundary force can be approximated by an artificial force with a range close to the resolution length of the calculation. If a viscous boundary condition is required we include the boundary particles in the calculation of the viscous stress. The boundary particles can be set up to follow any fixed or moving boundary.

As an example we simulate the motion of a sprinkler where the arms of the sprinkler rotate as a rigid body. This problem became notorious when Feynman asked what would happen if the sprinkler was run in the reverse mode with the sprinkler in a tank and the water sucked through the sprinkler arms. Which way would it rotate? We answer the question by simulating the system.

THE SPH EQUATIONS

The SPH equations for a compressible gas are described in detail by Monaghan (1992). They are obtained from the continuum equations of fluid dynamics by interpolating from a set of points which may be disordered. The interpolation is based on the theory of integral interpolants using interpolation kernels which approximate a delta function. The interpolants are analytic functions which can be differentiated without the use of grids. Any interpolated functon can therefore be differentiated analytically: finite differences are unnecessary. If the points are fixed in position the equations are identical to finite difference equations with different forms depending on the interpolation kernel.

The SPH equations describe the motion of the interpolating points which can be thought of as particles. Each particle carries a mass m, a velocity $\mathbf{v}$ and other properties depending on the problem.

The momentum equation for particle a becomes

$$\frac{d\mathbf{v}_a}{dt} = -\sum_b m_b \left(\frac{P_a}{\rho_a^2} + \frac{P_b}{\rho_b^2} + \Pi_{ab} \right) \nabla_a W_{ab} + \mathbf{F}_a, \qquad 2.1$$

where the summation is over all particles other than particle a (though in practice only near neighbours contribute), P is the pressure and ρ is the density, Π_{ab} produces a shear and bulk viscosity, $\mathbf{F}_a$ is a body force (for the problems considered here this is gravity), W_{ab} is the interpolating kernel (see for example (2.6)), and ∇_a denotes the gradient of the kernel taken with respect to the coordinates of particle a. The terms involving the pressure are derived from the pressure gradient. They are written in symmetrized form to conserve linear and angular momentum when the kernel is symmetric.

In this paper we use the spline based kernel (Monaghan and Lattanzio 1985, Monaghan 1992). The kernel depends on a length h which determines the resolution (see (2.6)). For separations $> 2h$ the kernel vanishes so that the summations only involve near neighbours. Typically h is 1.2ℓ where ℓ is the initial particle separation. It is possible to have a different resolution length for each particle, but this should not be necessary for incompressible flow.

The viscous term Π_{ab} has the general form

$$\Pi_{ab} = \begin{cases} \dfrac{-\alpha \bar{c}_{ab} \mu_{ab} + \beta \mu_{ab}^2}{\bar{\rho}_{ab}} & ; \quad \mathbf{v}_{ab} \cdot \mathbf{r}_{ab} < 0 \\ 0 & ; \quad \mathbf{v}_{ab} \cdot \mathbf{r}_{ab} > 0 \end{cases}$$

where

$$\mu_{ab} = \frac{h\mathbf{v}_{ab} \cdot \mathbf{r}_{ab}}{\mathbf{r}_{ab}^2 + \eta^2}.$$

In these expressions the notation $\mathbf{A}_{ab} = \mathbf{A}_a - \mathbf{A}_b$, and $\overline{A}_{ab} = (A_a + A_b)/2$ has been used, and c is the speed of sound. Because of its symmetry the viscous term conserves linear and angular momentum. The viscosity vanishes for rigid rotation. For the problems described here we take $\beta = 0$. The term involving α introduces both shear and bulk viscosity into incompressible flow. In the present case, with negligible changes in the density, the viscosity is almost entirely shear viscosity. In two dimensions the shear viscosity coefficient is approximately $\rho\alpha hc/8$. In most of the calculations we take $0.01 < \alpha < 1$.

We calculate the density from the continuity equation which, when converted to SPH form, becomes

$$\frac{d\rho_a}{dt} = \sum_b m_b(\mathbf{v}_a - \mathbf{v}_b) \cdot \nabla_a W_{ab}. \qquad 2.2$$

The thermal energy per unit mass changes according to

$$\frac{du_a}{dt} = \frac{1}{2}\sum_b m_b \left(\frac{P_a}{\rho_a^2} + \frac{P_b}{\rho_b^2} + \Pi_{ab} \right) \mathbf{v}_{ab} \nabla_a W_{ab}. \qquad 2.3$$

The rate of change of particle position is

$$\frac{d\mathbf{r}_a}{dt} = \mathbf{v}_a, \qquad 2.4$$

but it is often important for free surface problems to use the XSPH variant which involves adding the following correction to the right hand side of (2.4)

$$\Delta_a = \epsilon \sum_b \frac{m_b(\mathbf{v}_b - \mathbf{v}_a)}{\overline{\rho}_{ab}} W_{ab}. \qquad 2.5$$

This correction to the velocity keeps the particles more orderly and, in high speed flow, prevents the penetration of one fluid by another (Monaghan 1989).

In this paper we choose $\epsilon = 0.5$. For consistency the velocity used in the continuity equation should be the velocity used for stepping the particles however, this is not necessary for the problems we consider here. Note that each particle has effectively two velocities. One of these, $\mathbf{v}$, comes from the momentum equation while the other is the corrected velocity used for moving the particles.

For the reader unfamiliar with the SPH equations they may be interpreted conveniently using a gaussian kernel. In two dimensions the gaussian kernel has the form

$$W_{ab} = exp(-(\mathbf{r}_a - \mathbf{r}_b)^2/h^2)/(\pi h^2), \qquad 2.6$$

and the contribution of particle b to the acceleration of particle a is easily seen to be a symmetric central force. From this fact it follows that the method conserves linear and angular momentum. In the same way it can be seen that the density of particle a increases when particle b is moving towards it.

THE EQUATION OF STATE

We use the equation of state given by Batchelor (1967). This equation of state, modified to give a smaller speed of sound, is suitable for the simulation of the bulk flow of the fluid. The equation of state has the form

$$P = B\left(\left(\frac{\rho}{\rho_0}\right)^\gamma - 1\right), \qquad 3.1$$

with $\gamma = 7$. The choice of B determines the speed of sound. For example, when a dam of height H collapses, an approximate upper bound to the speed v of the water is given by

$$v^2 = 2gH, \tag{3.2}$$

and the coefficient B in the equation of state should be taken as

$$B = \frac{200gH}{\rho\gamma}, \tag{3.3}$$

with a speed of sound c approximately $\sqrt{200gH}$ and $M \sim 0.1$. Numerical experiments confirm that the density variations are consistent with this estimate. In a similar way an equation of state can be set up for any flow problem. Other examples will be given later.

BOUNDARY CONDITIONS

Most of the calculations to be described here were made with boundary particles which exert central forces on fluid particles. The form for the force was guided by the known forces between molecules. For a boundary and fluid particle separated by a distance r the force per unit mass $f(r)$ has the Lennard-Jones form

$$f(r) = D\left(\left(\frac{r_0}{r}\right)^{p_1} - \left(\frac{r_0}{r}\right)^{p_2}\right)\frac{\mathbf{r}}{r^2}, \tag{4.1}$$

but is set to zero if $r > r_0$ so that the force is purely repulsive. The calculations described here use $p_1 = 12$ and $p_2 = 6$. Other choices of p_1 and p_2 give similar results.

The length scale r_0 is taken to be the initial spacing between the particles, and the coefficient D (with dimensions v^2) was chosen by considering the physical configuration. For problems involving dams, bores or weirs with fluid of depth H, we take $D = 5gH$, where H is the depth of the water, but $D = 10gH$ or $D = gH$ give similar results.

Peskin (1977) constructed boundary forces by assuming that there was a concentrated force at the boundary which could be described by a delta function. This idea can be implemented in different ways according to the way the delta function is approximated. However, in the calculations described here, the use of forces based on known molecular forces worked better than forces based on approximations of the delta function.

IMPLEMENTATION ISSUES

For the two dimensional calculations described here the particles can be set up initially on a cartesian lattice, or preferably, on an hexagonal lattice. The mass of particle b is given by $m_b = \rho_b \Delta A$ where ΔA is the area per particle. Particles that extend beyond boundaries are removed.

The summations can be evaluated efficiently using link lists to access neighbouring particles (Monaghan 1985). The link list uses a grid of book-keeping cells with size $2h$. Only particles in neighbouring cells can then contribute to the properties of particles in a given cell.

Time stepping is carried out using a predictor corrector scheme. The time step is largely controlled by the Courant condition but we use the general time step control (Monaghan and Lattanzio 1985, Monaghan 1992) which takes into account viscosity and body forces.

THE EVOLUTION OF AN ELLIPTICAL DROP

A simple test of the SPH formulation is the flow of an elliptical drop in two dimensions when the initial velocity field is linear in the coordinates. The condition that the drop remains elliptical with time varying axes a and b is that

$$v_x = \frac{x}{a}\frac{da}{dt} \text{ and } v_y = \frac{y}{b}\frac{db}{dt} \qquad 6.1$$

where, for example, v_x is the x component of velocity. The condition that the fluid remains incompressible is that ab is constant which also follows from the vanishing of $\nabla \cdot \mathbf{v}$.

¿From the momentum equation, and condition that the pressure is constant on the elliptical surface, we deduce the following equations

$$\frac{dA}{dt} = \frac{A^2(a^4 - \omega^4)}{a^4 + \omega^4}, \qquad 6.2$$

where A is defined by

$$\frac{da}{dt} = -aA, \qquad 6.3$$

and ω is the initial value of ab.

These equations can be solved to high accuracy and the results compared with the SPH simulation. The initial velocity field was (-100x, 100y) and the initial fluid configuration was a circle of radius 1m. The pressure was computed using a coefficient that gave the normal pressure for water and a sound speed of 1400m/s. In this case, as expected, the density fluctuations were $< 1\%$. In Fig. 1 we show two fluid particle configurations from a simulation using 1884 particles. It is apparent that the particle configuration preserves a smooth outer boundary with no tendency to break up or become ragged. The errors in the calculation are $< 2\%$.

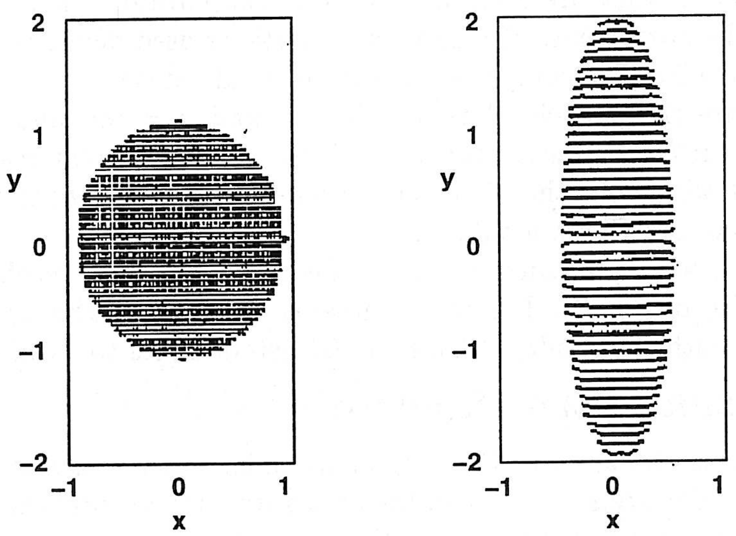


Figure 1: Particle positions for the evolution of an elliptical drop evolving from a circle to an ellipse. The initial speed is 100 m/sec and the initial radius is 1 m.

JETS FALLING UNDER GRAVITY

Solutions for the steady two-dimensional jets of inviscid, incompressible fluid emerging from nozzles in two dimensions have been calculated by Dias and Christodoulides (1991) using

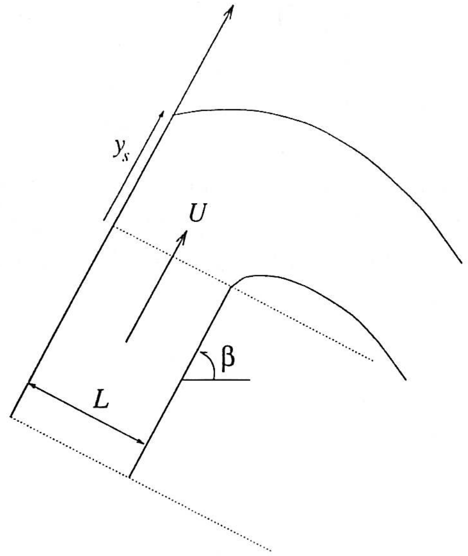


Figure 2: The idealized nozzle for the static jets.

a combined complex variable and expansion technique. The idealized nozzle which they treated is shown in Fig. 2. They calculated a family of solutions depending on the Froude number $Fr = U/\sqrt{gL}$ and the angle of the nozzle from the horizontal (β).

For the SPH simulations the boundaries are modelled using boundary particles. The viscosity parameter $\alpha = 0.1$ but it is not expected that viscous effects will be very important. Fluid SPH particles are fed in through the inlet at constant initial velocity and separation. To limit particle numbers they are removed from the computation when they have moved sufficiently far from the nozzle exit. The number of particles used depends on Fr and β but it is typically 2000 to 5000. The computations run to steady state.

Figures 3 and 4 are the particle plots for $Fr = 1$ and $\beta = 90°$ and $60°$ respectively. Harlow and Amsden (1971) simulated the vertical jet and the current results are in good qualitative agreement with both the MAC simulation and in good quantitative agreement with the results of Dias and Christodoulides.

A more demanding test is to compare the height which the fluid reaches on the centre line through the jet (y_s of Fig. 2). Figure 5 compares the SPH results for selected Froude numbers. They agree with the results of Dias and Christodoulides to within a few percent.

THE FEYNMAN SPRINKLER PROBLEM

The fluid emerging from the arms of the normal sprinkler has angular momentum in one direction and the sprinkler arms rotate with the oppostite angular momentum. The motion can also be understood in terms of the acceleration of the fluid as it flows around the curved arm. The arms must provide the acceleration and the reaction force is in the sense of producing the observed motion of the arms. In the reverse mode the sprinkler is in a pool and water flows into the arms. The acceleration of the fluid in the arm is exactly as before and so are the reaction forces. It would appear that the arm should rotate as before but then the angular momentum may not be conserved. The analysis of the system is complicated because the force on the arm includes the effect of the fluid outside the arm, and the angular momentum of the fluid includes all the fluid in the pool and the sprinkler arms. The reader is urged to think about this problem.

The sprinkler we simulate is a two dimensional sprinkler with arms formed by boundary particles which move as a rigid body. The arms consist of straight sections which end in

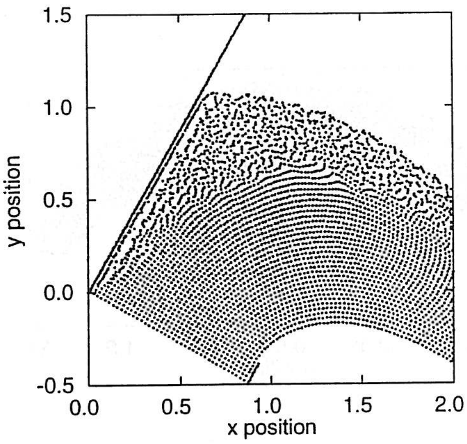


Figure 3: The particle positions for the SPH simulation of the jet inclined at 60 deg after the configuration has been run to reach a steady state.

quarter circle sections. The total torque exerted by the fluid particles on the boundary particles is calculated and the rotation of the sprinkler arms is calculated from the equation of motion for a rigid body. For the normal sprinkler the fluid was injected in the inner region across a line perpendicular to the arm. The particles are forced to remain at their initial speed for a distance of $2h$ from the injection line. During this time their velocity vectors rotate with the sprinkler. The viscosity parameter $\alpha = 0.01$ and a typical simulation has 10 rows of injected particles. As the simulation proceeds the number of particles increases from approximately 2000 to 5000.

The sprinkler arms then rotate in the direction which conserves angular momentum. The integration conserves linear and angular momentum with errors 3×10^{-6}. A typical configuration is shown in Fig. 6.

For the sprinkler in the reverse mode the arms were placed in an artificial circular pool. To move the fluid towards the sprinkler arms a radial force was imposed. This simulates approximately the effect you would get if a can with arms was allowed to descend into a pool of water (see the experiment of Berg and Collier 1989). The effect of gravity, through the pressure head, causes water to flow into the can through the arms. The particles flowing into the arms are removed at an inner boundary.

The flow of the particles entering one arm of the sprinkler are shown in Figs 7 and 8. The flow is similar to the flow into a sink except for the loss of symmetry produced by the curved arm and the radial force.

The sense in which the sprinkler arms rotate will be revealed at the conference.

DISCUSSION AND CONCLUSIONS

The results of the computations show that SPH can be used to simulate fixed and moving two dimensional jets. The applications to static jets have been compared with accurate results found by complex variable methods and the agreement is very good. The simulation of moving jets is straightforward regardless of the number of dimensions. The only disadvantage of the method is that, since it uses a stiff equation of state, the time step is much smaller

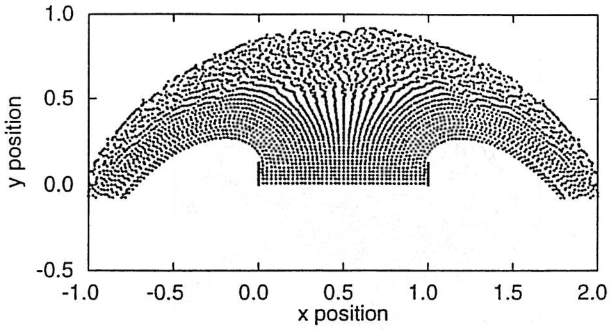


Figure 4: The SPH particle postions for the vertical jet in a steady state.

than for other methods. However this is compensated by the ease with which different configurations can be treated. It would, for example, be easy to deal with the flow of an inhomogeneous fluid in a container with moving boundaries.

REFERENCES

Harlow, F. and Welch 1965, *Physics of Fluids* **9**, 842.

Harlow, F. and Amsden J. J. 1971, *Fluid Dynamics* Los Alamos National Lab.

Harlow, F. 1988, *Comp. Phys. Commun.* **48**, 1.

Lucy, L. B. 1977, *Astron. J.* **82**, 1013.

Gingold, R. A. and Monaghan, J. J. 1977, *Mon. Not. Roy. Astr. Soc.* **181**, 375.

Monaghan, J. J. 1985 *Comput. Phys. Rep.* bf 3 71.

Monaghan, J. J. 1988, *Comp. Phys. Commun.* **48**, 89.

Monaghan, J. J. 1989, *J. Comput. Phys.* **82**, 1.

Monaghan, J. J. 1992, *Ann. Rev. Astron. Astrophys.* **9**, 942.

Monaghan, J. J. 1994, *J. Comput. Phys.* **10**

Monaghan, J. J. and Lattanzio, J. C. 1985, *Astron. and Astrophys.* **149** 135.

Peskin, C. S. 1977, *J. Comput. Phys.* **25** 220.

Sulsky, D and Brackbill, J. U. 1991, *J. Comput. Phys.* **96** 339.

Batchelor, G. K. 1967, An Introduction to Fluid Mechanics Cambridge Press.

Berg, R. E. and Collier M. R. 1989, *Amer. J. Phys.* **57** 654.

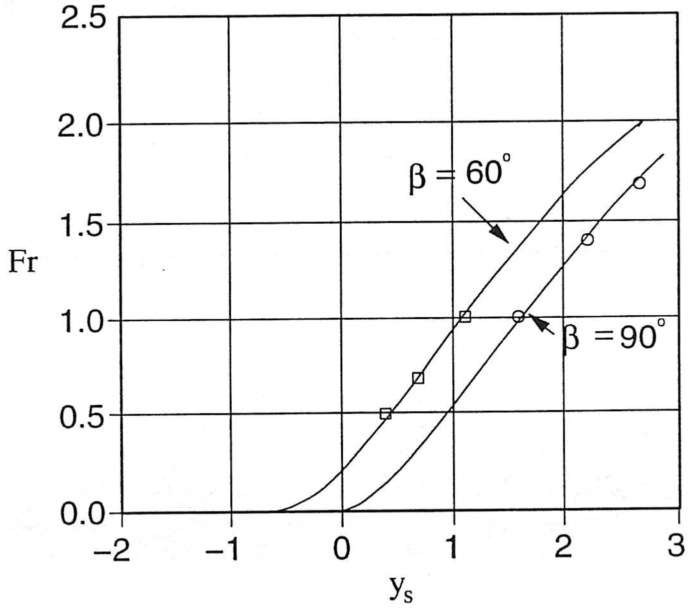
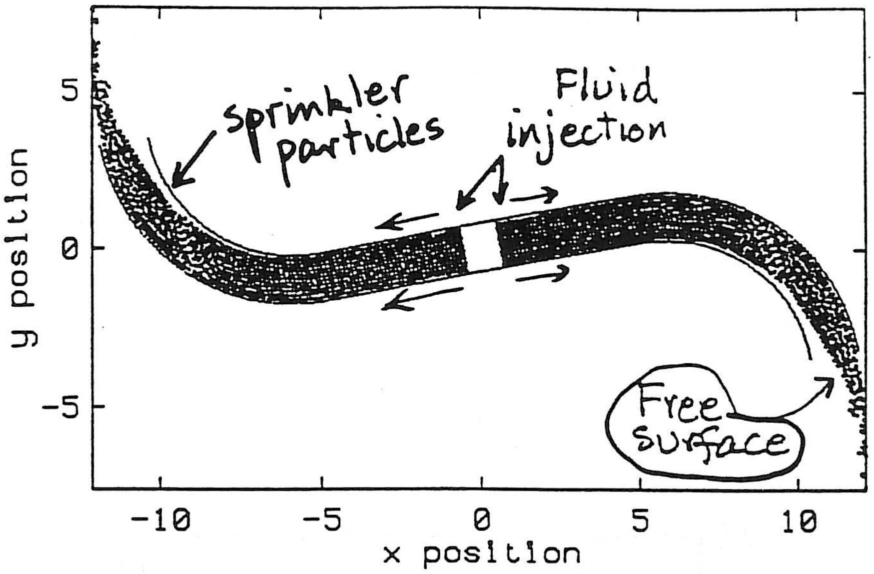


Figure 5: The height reached by jets along the left hand surface of the inclined jet and along the central line for the vertical jet. The parameter y_s is defined by Fig. 2.



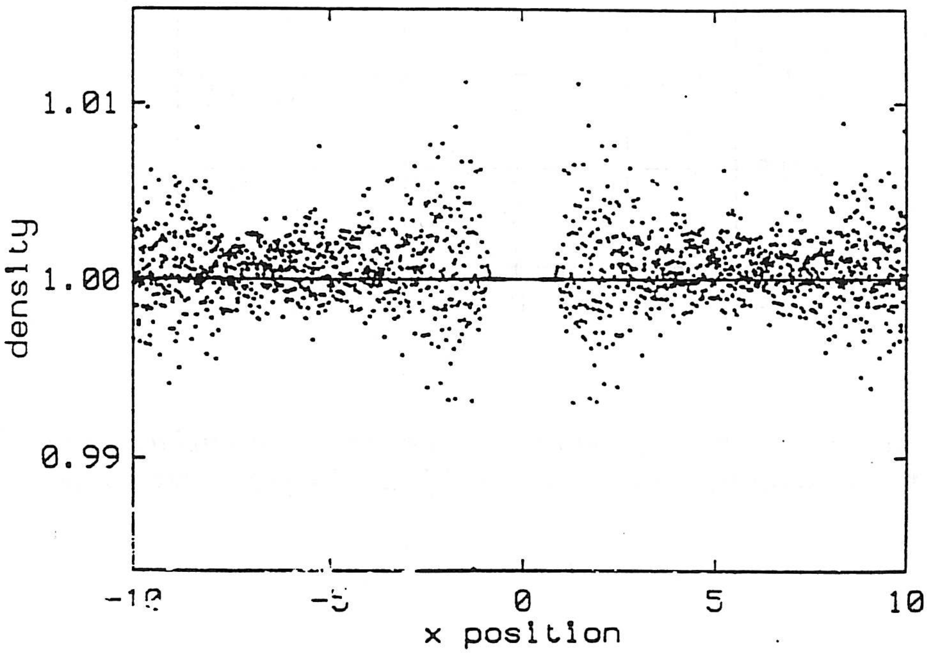


Figure 6: The SPH and boundary particles of the sprinkler in the normal mode. The lower frame shows the density of each particle. The fluctuations are approximately one percent.

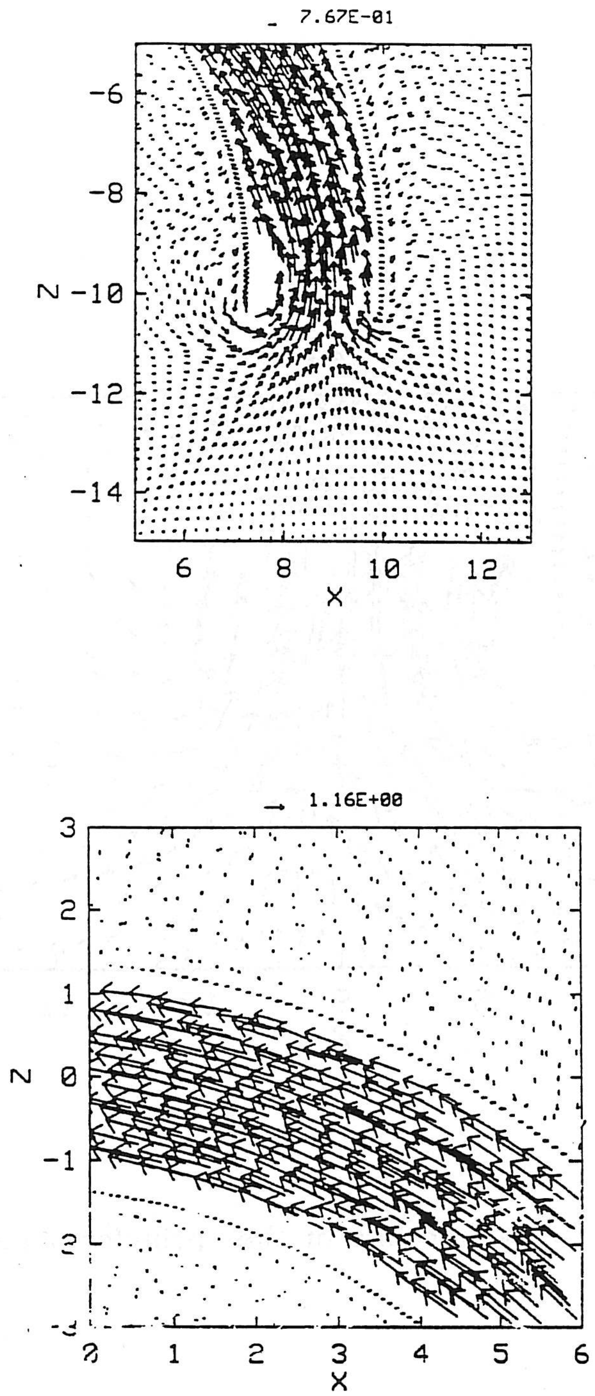


Figure 7: Velocity vectors of the SPH particles for the sprinkler placed in a pool and operated in the reverse mode. Only part of one arm is shown.

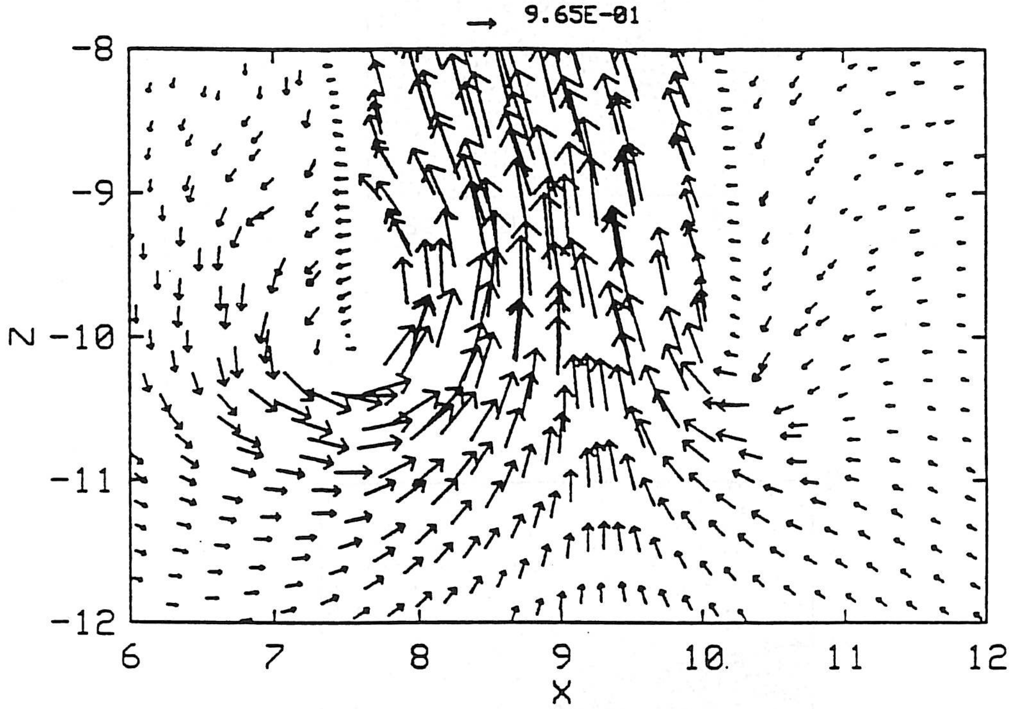


Figure 8: Details of the flow into one of the sprinkler arms operated in the reverse mode.