Coating Flow Simulations Using Smooth Particle Hydrodynamics

P.J. Reichl, P. Morris, K. Hourigan, M.C. Thompson
Fluid-dynamics Laboratory for Aeronautical and Industrial Research (F.L.A.I.R.)
Dept. of Mechanical Engineering, Monash University, Clayton, Vic. 3168, Australia
and
S.A.T. Stoneman
Dept. of Civil Engineering, University of Wales, Swansea, United Kingdom.

ABSTRACT

The simulation of plane impinging jets using the Lagrangian technique of smooth particle hydrodynamics is presented. In the first section the impingement of a plane jet onto a stationary surface is considered for Reynolds numbers 800, 4000 and 8000. For each case, the height of the associated streams is measured and compared with both experimental and theoretical results. The agreement between these results is found to be close for most angles. In the second section, impingement onto a moving surface is considered for different plate velocities. The preliminary results show that the moving surface can lead to some spatial height variation in at least one of the streams.

NOMENCLATURE

c	is the speed of sound
$d\mathbf{r}$	volume element
F_{ex}	external force
Fr	Froude number
h	smoothing length
m_a	mass of particle a
P_0	constant in pressure equation
P_a	pressure at particle a
$\mathbf{r}_a$	position of particle a
Re	Reynolds number
t	time units
$\mathbf{v}_a$	velocity of particle a
$W(r,h)$	interpolating kernel
α	constant in artificial viscosity term
β	constant in artificial viscosity term
γ	constant in equation of state
μ_{ab}	term in artificial viscosity equation
η	constant in artificial viscosity term
∇	spatial gradient
∇_a	spatial gradient taken with respect to coordinates of particle a
ρ_0	initial particle density
ρ_a	density of particle a
ν_{ab}	viscosity acting between particle a and b

1. INTRODUCTION

Free surface flows occur in a wide variety of situations in both industry and the environment. However, due to the problems associated with the implementation of appropriate boundary conditions on a surface whose position is constantly changing, such flows are often arduous to model numerically.

Of specific importance is the study of impinging jets, the use of which is widespread throughout many industries. For example, impinging jets are frequently encountered in VTOL aircraft, fluid mixing, fuel filling, combustors and ejectors, ventilation, electronic component cooling, in the annealing of non ferrous sheets, and in the cutting of rock in the mining industry.

Of particular concern is the use of impinging jets in the coating of surfaces, which is often widespread in industries associated with the application of corrosion inhibiting agents, paints and dyes, magnetic films on tapes, and laminates on paper products. In extrusion coating, for example, a thin layer of fluid is deposited onto a moving substrate, with the liquid then solidifying as it passes through an oven. In these situations, the aim is to achieve a uniformity in the coating thickness, and thus ensure that no undulations or voids are present in the final product.

From a numerical stand-point, what often makes coating flows difficult to model is the fact that the region of space occupied by the flowing liquid is not always known in advance, but is itself part of the solution of the hydrodynamic equations. The plethora of different coating techniques only helps to reinforce some of the practical difficulties that are associated with coating flows. Ruschak (1985) describes in detail some of the problems that are often encountered.

The present work may be divided roughly into two parts, with the first looking at the heights of the two streams associated with the impingement of a plane jet onto a stationary flat surface, while the second involves the investigation of a plane jet impinging onto a moving surface. In both cases, the Lagrangian technique of Smooth Particle Hydrodynamics (SPH) was used. This technique was first utilised to investigate complex astrophysical phenomena (Lucy 1977, Gingold and Monaghan 1977). However, more recently it has been applied to almost incompressible flows, with Monaghan (1994), Thompson *et al.* (1994), and Takeda *et al.* (1994) considering SPH's applicability to a number of test problems, in which the fluid was assumed to be nearly incompressible (that is, fluctuations in density of order 1% were permitted).

Smooth particle hydrodynamics is a Lagrangian technique, and as a result it requires no grid. Its in-built ability to automatically track the position of the free surface, while also handling any surface gradient, may therefore make this scheme a useful tool in the study of free surface behaviour.

In addition, its roots in astrophysics have ensured its application to flows involving magnetic fields, and while such flows are not investigated here, the potential to exploit this knowledge for applications involving magnetic films is still untapped.

The problems to be examined here provide a further test of SPH's ability to handle free surface flows. In the first case, the heights of the two streams from an impinging plane jet onto a stationary plate will be compared with the results of experiment and theory. This problem is considered because it enables the incorporation of free surfaces, while still remaining relatively simple, and also allowing for both experimental and theoretical validation of the results. The second case is considered because of its importance to many industrial applications.

The following section highlights the basic formulation and governing equations, while section (3) will look at the results and their discussion.

2. FORMULATION

The governing equations in SPH determine the characteristics of each of the interpolating points or particles. These equations will now be presented.

2.1 Momentum

The momentum equation is obtained after applying the theory of integral interpolants to the modified Euler equation given below

$$\frac{d\mathbf{v}}{dt} = -\frac{1}{\rho}\nabla(P + P_V) + F_{ex}, \qquad (1)$$

where P_V is an artificial viscous pressure (Cloutman 1991).

The resulting momentum equation for each interpolating point is then given by

$$\frac{d\mathbf{v}_i}{dt} = -\sum_{j=1}^{N} m_j\left(\frac{P_i}{\rho_i^2} + \frac{P_j}{\rho_j^2} + \nu_{ij}\right)\nabla_i W_{ij} + F_{ex} \qquad (2)$$

where ν_{ij} is the artificial viscosity.

ARTIFICIAL VISCOSITY

The artificial viscosity used here is that given in Monaghan (1992), which is as follows:

$$\nu_{ij} = \begin{cases} \dfrac{-\alpha\, h\, c\mu_{ij} + \beta\, h^2\, \mu_{ij}}{\rho_{av}} & \mathbf{v}_{ij}\cdot\mathbf{r}_{ij} < 0 \\ 0 & \mathbf{v}_{ij}\cdot\mathbf{r}_{ij} > 0 \end{cases} \qquad (3)$$

where

$$\mu_{ij} = \frac{\mathbf{v}_{ij} \cdot \mathbf{r}_{ij}}{\left|\mathbf{r}_{ij}\right|^2 + \eta^2}.$$

which has been shown to conserve both discrete linear and angular momentum.

In the above expression, α and β are both constants. For the almost incompressible case being examined here, we may set $\beta = 0$. The other parameter, η, is simply a small constant chosen so as to ensure that the denominator remains non-zero; in this case it is selected to be $0.01h^2$. The effective kinematic viscosity of the fluid is then proportional to αhc, with the constant of proportionality being dependent upon the interpolation kernel; for the cubic spline and quintic spline kernels used here, this constant is found to be 1/8 (see Murray (1995)).

Alternative viscosity models are prescribed in Takeda *et al.* (1994) and Watkins *et al.* (1996).

2.2 Continuity

"Incompressible" SPH requires a small but finite degree of compressibility, and as a result the density at each point will vary accordingly.

It is therefore more desirable when attempting to model free surfaces to use the continuity equation that gives

$$\frac{d\rho_i}{dt} = -\sum_{j=1}^{N} \rho_j \mathbf{v}_{ij} \nabla_i W_{ij}, \qquad (4)$$

rather than the direct density formulation, since a discontinuous density change is permitted at the free surface.

2.3 Equation of State

In order to determine the value of the pressure at each point, an equation of state is required which links the pressure to a variable, or series of variables which are already known.

The equation of state used here is that given by Batchelor (1967, pp56), with the modifications given in Thompson *et al.* (1994)

$$P = P_0\left(\left(\frac{\rho}{\rho_0}\right)^{\gamma} - 1\right). \qquad (5)$$

In the investigation undertaken here, γ was, as in Monaghan (1994) and Thompson *et al.* (1994), chosen to be 7.

2.4 Kernel

From the momentum and continuity equations (2) and (4), it can be seen that the properties for each particle are dependent upon those of all of the other particles. However, since it is the choice of interpolating kernel that affects the degree to which the surrounding particles interact with one another, it is possible in practice to select an appropriate kernel, such that only neighbouring particles exert any influence.

Two different kernels were used in this investigation, both having the property of compact support (that is, the kernel has a finite range in which it possess a non zero value). The kernels used were: the cubic spline kernel which is given in Monaghan and Lattanzio (1985), and the quintic spline kernel of Morris (1996).

It should be noted that both kernels should produce results of accuracy $O(h^2)$.

2.5 Boundary Forces

The boundary forces used in the present study consisted of either a Lennard-Jones force (which is given in Monaghan, 1994), or a system of multiple layers of particles.

The Lennard-Jones force, in the absence of an additional viscous model for boundary behaviour, can only be used on its own in cases in which the boundaries are assumed to be free slip. For no slip boundary conditions, it is thus necessary to employ a particle boundary arrangement.

3. RESULTS AND DISCUSSION

<u>Part One : Stationary Plate</u>

The simulation of an impinging jet over a range of orientation angles of the jet to the vertical, was investigated for the following Reynolds number and Froude number combinations.

Re=800, Fr=4.0
Re=4000, Fr=4.0
Re=8000, Fr=4.0

The Froude number, which in this case is the dominant parameter governing the behaviour of the system, was chosen to be 4.0 to match the experimental results which were performed at $Re \approx 6400$ and $Fr = 4.0$.

The basic setup of the numerical system involved a series of particles being ejected from a non interfering nozzle with a uniform velocity profile. The profile was assumed to be uniform for ease of numerical implementation. The Reynolds and Froude number values were then based on the initial jet velocity and the nozzle width.

For cases in which the boundary consisted of a series of particles, either two or three layers of particles were arranged to fit the boundary, and these boundary particles were then fixed in space, with only their density being allowed to vary. The density of the boundary particles was permitted to change, so as to allow the boundary to respond to any fluctuation in the pressure associated with the motion of the fluid particles. In this case the boundary better captures the interface between the fluid and solid wall.

The resolution of the system is dependent upon the smoothing length and hence upon the number of particles. For all the cases investigated here, the smoothing length h, was chosen to be 1.2 times the particle separation.

For the simulations involving the stationary plate, approximately 25000 particles were used (50 particles across the inlet). In compressible flows, such as those that often occur in astrophysical situations, automatic resolution variation is possible; however, in the incompressible case it is not such an easy task.

A fixed resolution was used in the simulations presented here, but particles of varying smoothing lengths, and hence resolution, could have been used.

For the entire range of Reynolds numbers investigated, the boundary plate was modelled with a Lennard-Jones force, and the cubic spline kernel was used.

In addition for the Re=800 case, the boundary was also modelled with layers of particles, and both the cubic spline and quintic spline kernels were used. In this situation different kernels and boundaries were adopted, in order to test the sensitivity of the results to the type kernel and boundary employed. The results of such a comparison indicate that for the case of an impinging jet, the type of kernel and boundary used has little influence on the final result.

For each of the Reynolds number combinations, the heights of the upstream and downstream flows were measured for the angles of 10^o to 80^o, at 10^o increments.

Figure (1) shows an example of the experimental findings, which were undertaken at Swansea, for an injection angle of 40^o, with figure (2) illustrating the equivalent numerical result.

In order to validate the SPH results, a comparison with theory and experiment was undertaken. The results of such a comparison are highlighted in Figures (3)-(4) which show a good agreement between the numerical, experimental and theoretical results. Note that all of the numerical results in figures (3)-(4) have been time averaged so as to reduce any inherent transient behaviour.

The experimental results were measured directly from blown up photographs, and have as a consequence an error associated with them.

In order to measure the heights of the streams in the numerical case, each stream was divided up into a number of bins, and the maximum particle height in each bin was then averaged to give a final result, with the uncertainty of such a result being approximately $O(h)$.

The theoretical results were obtained via the application to a control volume of Bernoulli's equation, along with a simple mass and momentum balance. This theoretical result was then adjusted in order to take into account that the resulting wall impingement angle alters due to the effect of gravity. This adjustment was made on the basis of free fall arguments, and subsequently has some degree of error associated with it. (The equations used to obtain this result are given in appendix 1)

In order for SPH to accurately simulate a real fluid, a minimum number of particles is required so as to ensure that they adequately model a continuum. For the simulation of the jet at all Reynolds numbers, the number of particles in the downstream flow for angles greater than approximately 60^o was relatively small, and as a consequence the results for such cases have a higher degree of uncertainty associated with them. Indeed for the higher Reynolds number cases, Re=4000 and 8000, some break up or scattering was observed, and it was necessary in these situations to filter out the relatively few scattered particles, especially at larger angles.

The agreement between the results in figures (3)-(4) is close up until about 50^o, at which point for the thinner of the two streams, the experimental results tend to deviate away from those obtained from theory and numerical simulation.

A possible explanation for this is that the error associated with the measuring of the experimental results will tend to be magnified for cases in which the stream height is thinner, and as a consequence the experimental results become less meaningful in this situation. The effect of parallax errors will also become more dominant in situations in which the stream height is small. In addition, the numerical results are less meaningful when there are a small number of particles simulating the fluid, and hence the accuracy of both the experimental and numerical results diminish in this region.

Part two: Moving Plate

The investigation involving the moving plate was undertaken at a Reynolds number of 800 and a Froude number of 4.0. Numerically the setup was similar to the stationary case, except that the particles making up the boundary were driven with the desired velocity (the driving velocity being from right to left in figures (5)-(8)).

The behaviour of the impinging jet was then investigated for plate velocities of 0.5 and 1 times the initial jet velocity. Figures (5)-(8) show the behaviour of the jet for different angles and plate velocities. What appears to be evident from these preliminary simulations is the unsteady behaviour of the thinner of the two streams. This is particularly evident in figures (7) and (8).

An investigation involving a higher resolution in the thinner stream is required before any firm conclusions can be made (the current resolution being 60 particles across the inlet). It should be noted that these results are awaiting experimental validation.

For all cases listed above, the numerical results were obtained using a two dimensional SPH model.

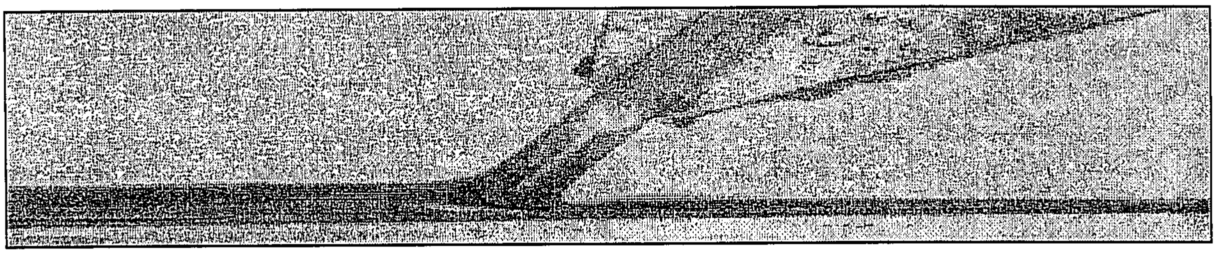

Figure 1. Experimental visualisation of an impinging jet onto a stationary surface, for an injection angle of 40 degrees.

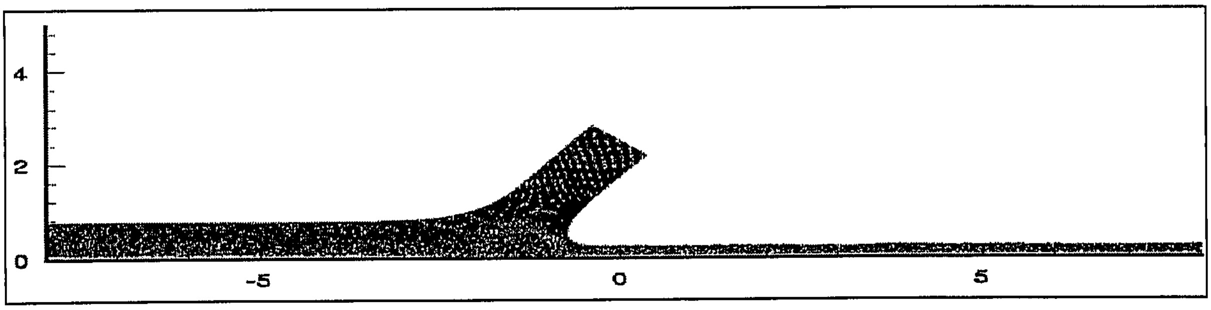

Figure 2. Predicted flow pattern from the SPH simulation of an impinging jet onto a stationary surface, at Re=800, θ =40 degrees

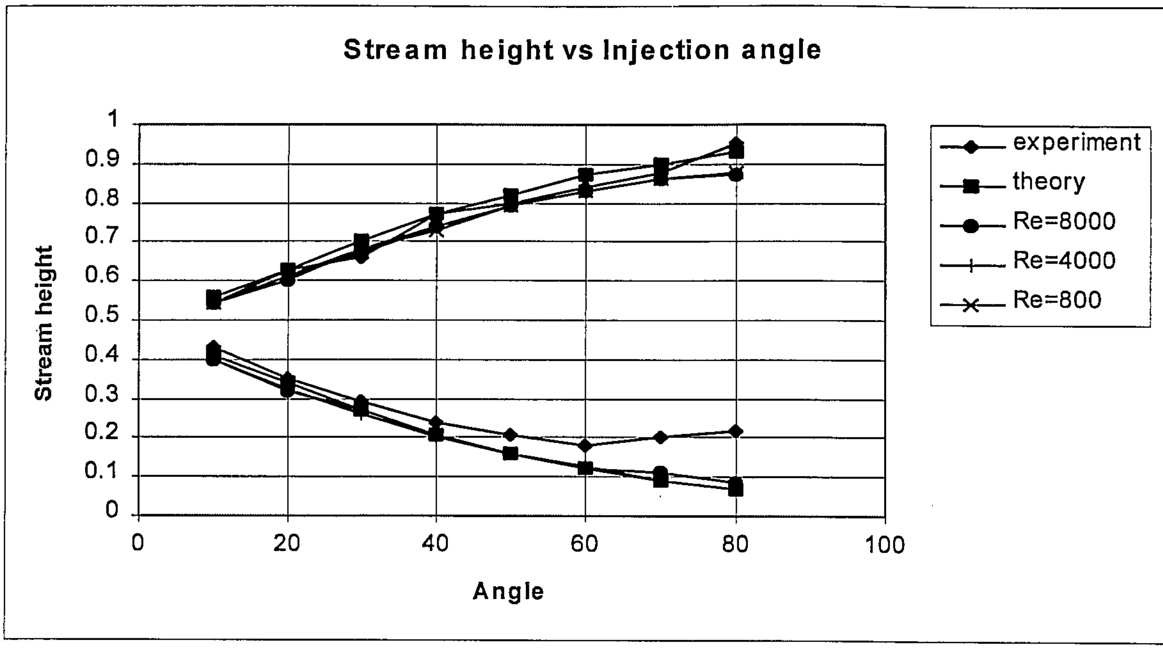

Figure 3. Comparison of stream heights for the numerical, experimental and theoretical results. Boundary = Lennard-Jones, Kernel = cubic spline.

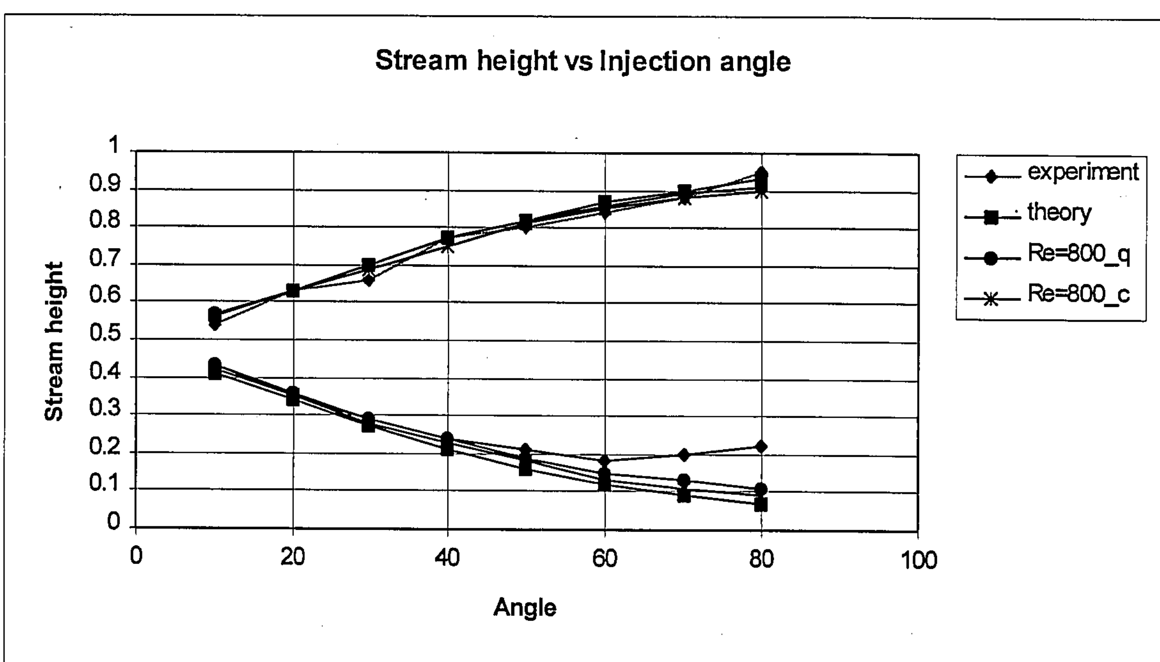

Figure 4. Comparison of stream heights for the numerical, experimental and theoretical results using different kernels. (_c = cubic spline, _q = quintic spline) Boundary = particle arrangement.

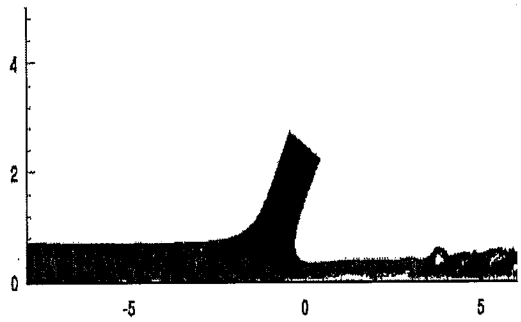

Figure 5. Predicted flow pattern from SPH simulation of an impinging jet onto a moving surface, at Re=800, plate velocity (right to left) 0.5 times nozzle velocity. θ =30 degrees. (Resolution approx 35000 particles).

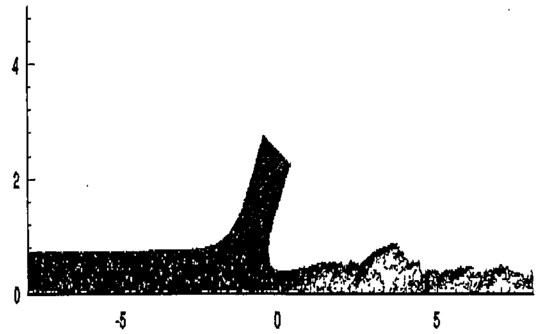

Figure 7. Predicted flow pattern from SPH simulation of an impinging jet onto a moving surface, at Re=800, plate velocity (right to left) 1.0 times nozzle velocity. θ =30 degrees. (Resolution approx 25000 particles)

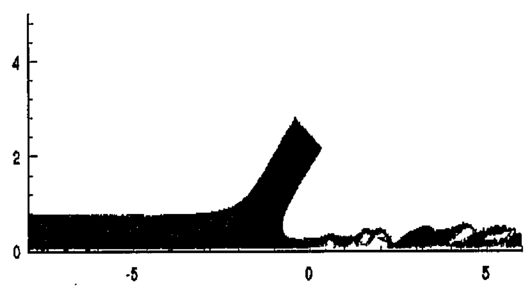

Figure 6. Predicted flow pattern from SPH simulation of an impinging jet onto a moving surface, at Re=800, plate velocity (right to left) 0.5 times nozzle velocity. θ =40 degrees. (Resolution approx 35000 particles).

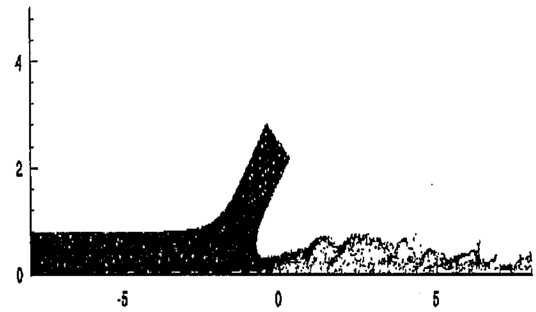

Figure 8. Predicted flow pattern from SPH simulation of an impinging jet onto a moving surface, at Re=800, plate velocity (right to left) 1.0 times nozzle velocity. θ =40 degrees. (Resolution approx 25000 particles).

CONCLUSIONS

SPH has been shown to predict well the heights of the streams associated with the impingement of a plane jet onto a stationary plate. The preliminary results, involving the impingement onto a moving plate, suggest that under such conditions spatial variation in the height of at least one of streams can be observed.

ACKNOWLEDGMENTS

Paul Reichl would like to thank Monash University, and the Australian Postgraduate Award, which has funded this research.

REFERENCES

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

Cloutman, L.D. (1991). "An evaluation of Smoothed Particle Hydrodynamics", Eds: Trease, H.E., Fritts, M.J., Crowley, W.P. In *Advances in the Free-Lagrange Method*, Springer-Verlag. pp 229-247.

Crank, J. (1984) *Free and Moving Boundary Problems*. Oxford University Press.

Gingold, R.A. and Monaghan, J.J. (1977). "Smoothed Particle Hydrodynamics: theory and application to non spherical stars", *Monthly Notices of the Royal Astrophysical Society*, **181**, pp 375-389.

Lucy, L. (1977). "A numerical approach to the testing of the fission hypothesis", *Astron. J.*, **82**, pp 1013-1024.

Monaghan, J.J. and Lattanzio, J.C. (1985). "A refined particle method for astrophysical problems", *Astronomy and Astrophysics,* **149,** pp 135-143.

Monaghan, J.J. (1992). "Smoothed Particle Hydrodynamics", *Annu. Rev. Astron. Astrophys*, **30**, pp 543-574.

Monaghan, J.J. (1988). "An introduction to SPH", *Computer Physics Communications*, **48**, pp 89-96.

Morris, J.P.A. (1996). "An overview on the method of smooth particle hydrodynamics", (*Private Communication*).

Murray, J.R. (1995). "SPH simulation of accretion disks in cataclysmic variables", *PhD Thesis*, Monash University.

Ruschak, K.J. (1985). " Coating Flows", *Ann. Rev. Fluid Mech.*, **17**, pp 65-89.

Takeda, H., Miyama, S.M. and Sekiya, M. (1994). "Numerical simulation of viscous flow by smoothed particle hydrodynamics", *Progress of Theoretical Physics*, **92**, pp 939-960.

Thompson, M.C., Hourigan, K. and Monaghan, J.J. (1994). "Simulation of free surface flows with SPH", *Advances in Computational Methods in Fluid Mechanics 1994, The 1994 ASME Fluids Engineering Division Summer Meeting*, Lake Tahoe, Nevada, June 19-23.

Watkins, S.J., Bhattal, A.S., Francis, N., Turner, J.A., and Whitworth, A.P. (1996). "A new prescription for viscosity in SPH", *A&ASS*, **119**, pp 177-188.

APPENDIX 1

Bernoulli equation

$$\frac{P_a}{\rho_a g} + \frac{v_a^2}{2g} + z_a = \frac{P_b}{\rho_b g} + \frac{v_b^2}{2g} + z_b$$

Continuity equation

$$\rho_{in} A_{in} v_{in} = \sum_{outlets} \rho z v$$

x Momentum equation

$$\rho_{in} v_{in} A_{in} v_{in} \sin\theta_* = \sum_{outlets} \rho z v v$$

Here A_{in} is the width of the inlet.
Applying these equations to the jet system results in 4 non linear equations in 4 unknowns which can be solved numerically. Note that θ_* refers to the corrected angle of impingement.