

Modelling of Liquid Jet Break-up and Drop Formation

P. Pierron, M. Normandin, A. Soucemarianadin, J. R. Clermont and P. Attané
Laboratoire de Rhéologie - CNRS & Univ. J. Fourier, BP 53X 38041 Grenoble Cedex 9 (France)

Abstract

A liquid jet issuing from a nozzle may break-up into small drops of a variety of sizes when it is subjected to even minute disturbances due to the phenomenon of capillary instability. With the large number of parameters involved in the description of the jet instability, it should be of great interest to solve the governing equations numerically. Several attempts have been made in this direction and are still very active since a detailed numerical investigation of the break-up of a viscous jet requires a very accurate numerical technique. The present work proposes one such technique capable of an efficient computation of the unknown free surface. It is the stream tube method which uses a transformation of the physical domain. The governing equations are then solved by using an optimisation algorithm. An expected advantage of the method is the easiness in introducing elaborate rheological constitutive equations in order to account for complex fluid behaviour. In this paper, we will give the basic features of the stream tube method in the context of an unsteady jet flow and present the procedures allowing to obtain streamlines and kinematic quantities on the jet instability problem.

Introduction

The problem of modelling the break-up of liquid jets is a fairly old one. The first mathematical treatment is due to Rayleigh[1] using linear instability theory, where he considered an infinite jet, and examined the temporal behaviour of an axially periodical disturbance. This is not strictly the problem issuing from a nozzle, which was first considered by Keller et al[2]. Satellite formation is not predicted by the linear theory. Following this linear analysis there have been essentially three approaches to the problem of jet instability. The differences between these approaches consist in the accuracy by which the geometry and fluid mechanics of the problem are taken into account.

First is the non-linear perturbation analysis of Rayleigh's problem which results in analytical solutions. Analyses that fall in this category include that of Yuen[3] and Chaudhary and Redekopp[4] in which the viscosity was neglected. Moreover perturbation analyses assume that the free surface shape, while unknown deviates only slightly from a predetermined shape and thus are not really capable of representing correctly the late stages of the jet life.

The second approach to the jet break-up problem is the approximation of the full Navier-Stokes equations by use of a set of one dimensional jet equations which take into account the characteristic features of the flow, notably the jet slenderness as described by Yarin[5]. The simplest equations are those derived by Lee[6], neglecting radial inertia and viscous effects. The spatial one-dimensional equations derived by Lee were solved by Torpey[7] using a weighted residual method, in which a system of non-linear partial differential equations are reduced to a set of ordinary differential equations. Torpey limits his expansion to the two first Fourier modes and claims fair agreement with his experimental results. We have shown elsewhere[8] that this level of approximation can only be justified for low initial perturbations which is not the case of industrial applications[9]. Moreover Eggers and Dupont[10] have shown that inviscid models may become inconsistent much longer before break-up occurs.

Finally the last modelling approach to the problem of jet instability is the direct numerical solution of the Navier-Stokes equations. The first attempt was done by Shokoohi[11] who used a vorticity-stream function formulation to track the fluid surface. The computations were limited to low Reynolds numbers by numerical stability constraints and also by computer time requirements. Mansour and Lundgren[12] used a boundary integral method to study the instability of an inviscid jet. This method is very attractive since it involves only information about the surface of the fluid. Although being quite accurate boundary integral methods neglect either viscous or inertial forces, both of which become important asymptotically as emphasized by Eggers[13]. Moreover for these methods, both the surface tracking and the flow computations are quite complicated problems which have to be coupled appropriately. Very recently Ashgriz and Mashayek[14] proposed a temporal analysis of the capillary jet break-up problem based on a Galerkin finite-element method with penalty function formulations in order to solve the continuity, momentum and conservation equations. The free surface of the jet which is a priori unknown is determined using a special method developed by the authors[15]. This work is probably the most extensive study of jet break-up to date.

In this paper, our aim is to propose a robust numerical method which is able to deal with the highly non-linear problem of spatial jet instability which may lead to rather intricate free surface shapes. This method should also be versatile enough to model accurately different initial and/or boundary conditions such as non-sinusoidal perturbations[16].

Features of the Stream Tube Method

The basic elements of the stream tube method have been discussed in an exhaustive manner elsewhere[17] and therefore only the main features necessary for the understanding of the results given hereinafter will be presented in this sub-section.

In the case of an axisymmetric problem, the stream tube method defines a transformation function f which allows a physical domain D to be mapped into a simpler domain D* where the streamlines are parallel straight lines. This function f is an unknown of the problem to be solved in the new domain which is geometrically much simpler as shown below in figure 1.

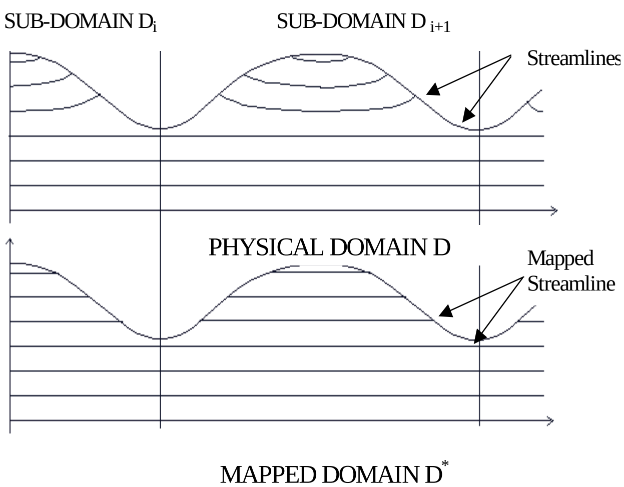


Figure 1. Representation of physical and mapped domains

In the case of the jet, as also shown in figure 1, the domain is sub-divided into sub-domains Di involving a one-to-one local transformation for obtaining the mapped sub-domain Di*. For every sub-domain, we have:

$$r = f_i(R, Z) \tag{1}$$

$$z = Z(r, z) \in Di \; (R, Z) \in Di^* \tag{2}$$

$$\Delta = \delta(r, z)/\delta(R, Z) = f`{}_i{}'_R \neq 0 \qquad \text{for all } i$$

where $f`{}_i{}'_R$ is the Jacobian of the function f_i. It will be noticed that there is necessity to consider boundary conditions between two different sub-domains. In order to formulate the basic equations of the stream tube analysis, reference sections are required for each sub-domain. In the problem under consideration, the reference section is that of maximum radius i.e. the swell section.

For the axisymmetric flow which is under study, the velocity vector can be written as:

$$\vec{V} = u(r,z,t)\vec{e}_r + w(r,z,t)\vec{e}_z \tag{3}$$

using cylindrical co-ordinates. Following Clermont[17], we can write the stream function at the section of reference as:

$$\varphi(r, z_b, t) = -\int_0^r \xi w(\xi, z_b, t)\, d\xi \tag{4}$$

The reference section z_b allows to construct a mapped sub-domain for which the maximum radius is *Rb* corresponding to that of the free surface for $z = z_b$. Let us now take

$$\phi^*(R,t) = -\varphi(r, z_b, t) \tag{5}$$

The derivation operators which come from the equations defining the function f can be written:

$$\frac{\delta}{\delta r} = \frac{1}{f_r'}\frac{\delta}{\delta R} \tag{6}$$

$$\frac{\delta}{\delta z} = -\frac{f_z'}{f_r'}\frac{\delta}{\delta R} + \frac{\delta}{\delta Z} \tag{7}$$

Following the same transformation, the velocities are of the form:

$$u = \frac{f_z'}{f\, f_R'}\frac{\delta}{\delta R}\left(\phi^*(R,t)\right) \tag{8}$$

$$w = \frac{1}{f\, f_R'}\frac{\delta}{\delta R}\left(\phi^*(R,t)\right) \tag{9}$$

In order to simplify let us take:

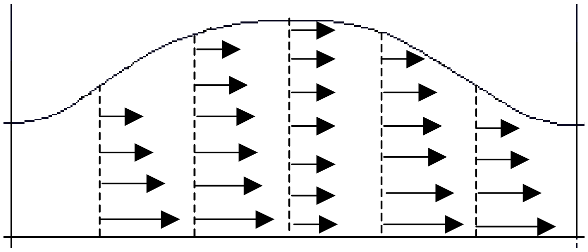
$$\bar{\phi}^*(R,t) = \frac{\delta}{\delta R}\left(\phi^*(R,t)\right) \tag{10}$$

from where we obtain:

$$u = \frac{f_Z'}{f\, f_R'}\bar{\phi}^*(R,t) \tag{11}$$

$$w = \frac{1}{f\, f_R'}\bar{\phi}^*(R,t) \tag{12}$$

From the velocities at the sections of reference and the mapping function, it is possible to compute the velocities in all the sub-domains of the jet as shown in figure 2. In this figure we have considered the radial velocity to be negligible.



VELOCITY FIELD

Figure 2. Map of the velocity field in a sub-domain

Within the context of the stream-tube method, it is necessary to compute the mapping function f from the mapped domain D* i.e. the domain of computation. Let us assume that the axial velocity is constant in every reference cross-section as shown in figure 2. This assumption leads to:

$$\frac{\delta w}{\delta r} = 0, \quad \forall z, \forall t \tag{13}$$

Using this relationship, we obtain the following partial differential equation for the mapping function f.

$$f\, f'_R - R\left[f'^{\,2}_R + f\, f''_{R^2} \right] = 0 \tag{14}$$

This partial differential equation with unknown f is true for every sub-domain and for all times. Moreover, on the axis of the jet we have the relationship f = 0.

We propose to choose a polynomial approximation of degree three for f in the following form:

$$\forall z \; f(R,t) = \alpha_1(Z,t)R + \alpha_2(Z,t)R^2 + \alpha_3(Z,t)R^3 + \ldots \tag{15}$$

The α_i are associated to one given section for each time t. They represent a subset of the unknowns of the problem.

Governing Equations of the Problem

As shown elsewhere[9] there are at least two possible methods to perturb the jet. Indeed the disturbance can be in the form of an electrohydrodynamic stimulation (radius perturbation) or a piezoelectric excitation which leads to a velocity pulsation. In this study, we consider a disturbance using the latter technique which gives an excitation of the following form:

$$w\,(r, Z_0^+, t) = w_0 \sin(\omega t + \varphi) \tag{16}$$

where w_0 is the initial amplitude and the φ phase shift.

The velocity profile at the nozzle exit is taken to be uniform but any other type of profile can be envisaged. In this investigation body forces and inertial forces are ignored.

The equilibrium equations in cylindrical co-ordinates can be written as:

$$\begin{cases} \dfrac{\delta \sigma_{rr}}{\delta r} + \dfrac{\delta \sigma_{rz}}{\delta z} + \dfrac{\sigma_{rr} - \sigma_{\theta\theta}}{r} = 0 \\ \dfrac{\delta \sigma_{rz}}{\delta r} + \dfrac{\delta \sigma_{zz}}{\delta z} + \dfrac{\sigma_{rz}}{r} = 0 \end{cases} \tag{17}$$

and with $\bar{\sigma} = -P\,\bar{I} + \bar{T}$, we obtain :

$$\begin{cases} -\dfrac{\delta P}{\delta r} + \dfrac{\delta T_{rr}}{\delta r} + \dfrac{\delta T_{rz}}{\delta z} + \dfrac{T_{rr} - T_{\theta\theta}}{r} = 0 \\ -\dfrac{\delta P}{\delta r} + \dfrac{\delta T_{rz}}{\delta r} + \dfrac{\delta T_{zz}}{\delta z} + \dfrac{T_{rz}}{r} = 0 \end{cases} \tag{18}$$

Now we can apply the derivation operators on each component of the extra-stress tensor with special attention to the points on the free surface. In the case of an inviscid fluid, on the free surface the pressure is not an unknown since it is defined by the following relationship:

$$P_{surf} = T\left(\frac{1}{r_n} + \frac{1}{r_t} \right) \tag{19}$$

where T is the surface tension coefficient and r_n and r_t are the usual radii taken into account for the calculation of the surface tension[6].

By applying the derivation operator on r_n and r_t we obtain:

$$P_{surf} = \frac{T}{f} \tag{20}$$

Finally to summarise, the equilibrium equations on the free surface are:

$$\begin{cases} \dfrac{T}{f^2} + \dfrac{1}{f'_R}\dfrac{\delta T_{rr}}{\delta R} + \dfrac{\delta T_{rz}}{\delta Z} - \dfrac{f'_Z}{f'_R}\dfrac{\delta T_{rz}}{\delta R} + \dfrac{T_{rr} - T_{\theta\theta}}{f} = 0 \\ \dfrac{1}{f'_R}\dfrac{\delta T_{rz}}{\delta R} + \dfrac{\delta T_{zz}}{\delta Z} - \dfrac{f'_Z}{f'_R}\dfrac{\delta T_{zz}}{\delta R} + \dfrac{T_{rz}}{f} = 0 \end{cases} \tag{21}$$

and for the points within the jet we have:

$$\begin{cases} -\dfrac{1}{f'_R}\dfrac{\delta P}{\delta R} + \dfrac{1}{f'_R}\dfrac{\delta T_{rr}}{\delta R} + \dfrac{\delta T_{rz}}{\delta Z} - \dfrac{f'_Z}{f'_R}\dfrac{\delta T_{rz}}{\delta R} + \dfrac{T_{rr} - T_{\theta\theta}}{f} = 0 \\ \dfrac{f'_Z}{f'_R}\dfrac{\delta P}{\delta R} - \dfrac{\delta P}{\delta Z} + \dfrac{1}{f'_R}\dfrac{\delta T_{rz}}{\delta R} + \dfrac{\delta T_{zz}}{\delta Z} - \dfrac{f'_Z}{f'_R}\dfrac{\delta T_{zz}}{\delta R} + \dfrac{T_{rz}}{f} = 0 \end{cases} \tag{22}$$

For a viscous jet there is no difference between free surface and interior points.

As stated earlier in the numerical features of the stream tube method, the jet is partitioned into sub-domains which leads to define two compatibility equations between the sub-domains with respect to continuity on the axial velocity and the mapping function.

$$w(R_i, Z_{C_i}) = w(R_{i+1}, Z_{C_{i+1}}) \tag{23}$$

$$f(R_i, Z_{C_i}) = f(R_{i+1}, Z_{C_{i+1}}) \tag{24}$$

where $\left(R_i, Z_{C_i}\right) \in D^*{}_i$ and $\left(R_{i+1}, Z_{C_{+1i}}\right) \in D^*{}_{i+1}$

The governing equations concern the interface pressure and can be written as:

$$\begin{cases} \vec{t} = \overline{\sigma}\vec{n} & (a) \\ \left|\vec{t}\right| = T\left(\dfrac{1}{R_1}+\dfrac{1}{R_2}\right) & (b) \\ \vec{t} \,//\, \vec{n} & (c) \end{cases} \tag{25}$$

The components of the normal vector are:

$$\vec{n} = n_R\vec{e}_R + n_Z\vec{e}_Z = \frac{1}{\sqrt{1+\left(f_Z^{'}\right)^2}}\vec{e}_R - \frac{f_Z^{'}}{\sqrt{1+\left(f_Z^{'}\right)^2}}\vec{e}_Z \tag{26}$$

with (a) and (b) we obtain:

$$\left(\left(\left(-P+T_{rr}\right)n_r + T_{rz}n_z\right)^2 + \left(T_{rz}n_r + \left(-P+T_{zz}\right)n_z\right)^2\right)^{1/2} = \frac{\sigma_s}{\left(1+\left(\dfrac{\delta r}{\delta z}\right)^2\right)^{1/2}}\left\{1-\frac{\dfrac{\delta^2 r}{\delta z^2}}{\left(1+\left(\dfrac{\delta r}{\delta z}\right)^2\right)}\right\} \tag{27}$$

And finally with (c) we have:

$$-f_Z^{'} = \frac{T_{rz}n_r + \left(-P+T_{zz}\right)n_z}{T_{rz}n_z + \left(-P+T_{rr}\right)n_z} \tag{28}$$

With this equation the mathematical formulation of the jet instability problem is completed.

Numerical Procedure

Mesh generation for the problem under study is an highly non-trivial part of the overall solution procedure. It is important to generate the most efficient mesh so as to reduce the computing time. Since we are considering the spatial instability it is necessary to mesh the whole jet. Moreover we are in the case of highly distorted domains where moving boundaries are involved and these domains experience successive evolutions as time advances. In the present work, we consider an adaptative mesh i.e. there are few points in the early stages of the jet and their number increases as time advances. The jet becomes more and more distorted in the last stages as shown in figure 3.

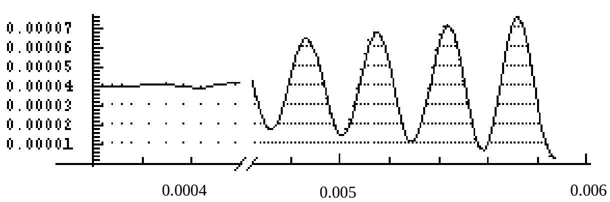


Figure 3. Adaptative Mesh

At each mesh point the pressure is unknown, and at every cross section, the three unknowns of the analytical form of the mapping function f are to be considered (see equation 15). On the interior points, we have to write the equilibrium equations, while for the boundary points, we have also to take into account the interface pressure. Finally, we are left with the sub-domain boundary conditions (continuity of axial velocity and mapping function). This leads to an over-determined system of equations which we solve using the Levenberg-Marquardt optimisation algorithm. For our preliminary numerical experiments, we have chosen to work with about 500 mesh points. The numerical code is implemented on a Pentium II micro-computer with double precision variables. At this time, the numerical simulations are carried out on the low initial perturbation regime[9] for a fluid of viscosity similar to that of an ink and for Reynolds and Weber numbers equal to that found in industrial ink-jet printing. Typical runs are of the order of 6 hours CPU time. As expected we find results close to linear perturbation analyses as given by Chaudhary and Redekopp[4]. This helps to show that the controlling parameters have been appropriately taken into account.

Concluding Remarks

In this paper, a numerical method based on the stream tube analysis has been developed to investigate the break-up of an unsteady Newtonian jet. Some distinguishing features of the stream tube formulation, mainly the transformation of streamlines into parallel lines in the mapped domain are shown to be of help in characterising the intricate free surface shapes in the jet break-up problem. The possibility of computing stresses and velocity fields within the jet by use of an adaptative mesh is also to be underlined.

Although the method presented in this paper is applied to Newtonian fluids, the elements given in this paper may be readily generalised for other fluids with complicated rheological behaviour.

This work is only in its initial stages since we have only considered the low initial perturbation regime. Results on the high initial perturbation regime are needed for full validation of the numerical simulations. We expect also in the near future to extend this method to the problem of drop on demand printing.

References

1. Rayleigh Lord, *Proc. Roy. Soc. Lond.* **29**, 71, (1879).
2. J. B. Keller**,** S. I. Rubinov and Y. O. Tu, *Phys. Fluids* **16**, 2052, (1973).
3. H. C. Yuen, *J. Fluid Mech.* **33**, 151, (1968).
4. K. C. Chaudhary and L. G. Redekopp, *J. Fluid Mech.* **96**, 257, (1980).
5. A. L. Yarin, Free liquid jets and films: hydrodynamics and rheology. Longman Scientific & Technical, (1993).
6. H. C. Lee, *IBM J. Res. Develop.***18**, 364, (1974).
7. P. A. Torpey, *Phy. Fluid A* **1**, 661, (1989).

8. J. H. Xing, A. Boguslawski, A. Soucemarianadin, P. Atten and P. Attané, *Exp. in Fluids* **20**, 302 (1996).
9. B. Lopez, A. Soucemarianadin and P. Attané, Proc. IS&T's XIIIth Intl. Cong. on Advances in Non Impact Printing Technol. 609, (1997).
10. J. Eggers and T. F. Dupont, *J. Fluid. Mech*. **262**, 205 (1994)
11. J. F . Shokoohi, PhD Thesis Columbia University New York (1976)
12. N. N. Mansour and T. S. Lundgren, *Phys. Fluids* A **2**,
13. J. Eggers, *Rev. Of Modern Phy.,* **69** 3, 865 (1997)
14. N. Ashgriz and F. Mashayek, *J.Fluid Mech.,***291,**163,(1995)
15. F. Mashayek and N. Ashgriz, *Intl. J. Numer.Meth. Fluids* **17**,1035, (1993)
16. A. Kalaaji, B. Lopez and A. Soucemarianadin and P. Attané, Proc. IS&T's XIVth Intl. Cong. on Advances in Non Impact Printing Technol. (1998).
17. J. R. Clermont, *C.R. Acad. Sci. Paris*, **297**, (1983)

Acknowledgements

The authors wish to thank the Région Rhône-Alpes for partial financial support of this work.

Biography

Arthur Soucemarianadin is Professor of Mechanics at the University Joseph Fourier in Grenoble, France. He is a member of IS&T and the American Society of Rheology.
e-mail: souce@ujf-grenoble.fr