Effect of initial turbulence intensity and velocity profile on liquid jets for IFE beamline protection

A. Konkachbaev *, N.B. Morley, M.A. Abdou

Mechanical and Aerospace Engineering Department, UCLA 43-133 Engineering IV, Los Angeles, CA 90095-1597, USA

Abstract

Current designs for thick liquid protection of heavy ion inertial fusion reactors utilize banks of liquid jets and vortex flow tubes to protect sensitive beam line components from neutrons and debris following target explosions. These hydraulic protection systems have issues regarding their formation and flow characteristics that need to be further understood as part of the ongoing chamber technology program for IFE. In this paper, modeling and experimental observations and quantitative data for rectangular and circular jets are compared in an effort to illuminate the critical phenomena involved. For rectangular jets a variety of wave patterns have been observed owing to competing effects of turbulence, surface tension and boundary layer formation in nozzles. Modeling of these jet flows using a Large Eddy Simulation technique is a useful tool for understanding the various origin of these waves, and experimenting with techniques to control their behavior. Recommendations for nozzle designs within IFE constraints are presented based on the numerical simulations.
© 2002 Elsevier Science B.V. All rights reserved.

Keywords: Free surface jet; Turbulence; Droplet ejection; Velocity profile; Surface ripple

1. Introduction

The important issues for IFE chamber design utilizing thick liquid wall protection include keeping beam lines clear for beam propagation while concomitantly providing protection to sensitive beamline components from neutrons and debris following target explosions. For beamline protection grids utilizing arrays of rectangular and/or cylindrical jets (see Fig. 1), an effort is made here to study numerically the major hydrodynamic phenomena leading to jet deformation—turbulent surface ripple, droplet ejection, and corner rounding—in order to develop recommendations on how to avoid such phenomena within IFE design constraints. While very high area contraction could laminarize the flow by pressure suppression of upstream turbulence, requirements on high line mass density (i.e. jet packing fraction) do not allow much change in the flow area contraction ratio of the nozzle, because jets must be close together. So we expect the jet will always be somewhat turbulent. The only parameters of the jet, which can be significantly varied, are the upstream flow conditioning and the nozzle shape, which affect the

---

* Corresponding author. Tel.: +1-310-206-1230; fax: +1-310-825-1715

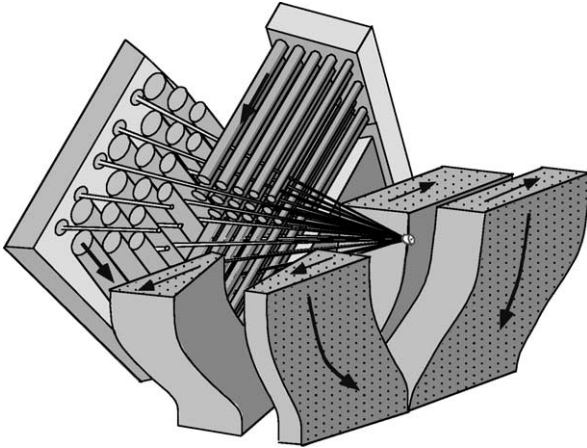*E-mail address:* askar@fusion.ucla.edu (A. Konkachbaev).

Fig. 1. Cut-away view of liquid pocket with beam paths (from Peterson [1]).

outlet velocity profile, relative turbulence intensity and initial jet shape. In the present study the influence of the velocity profile on the surface deformation and droplet ejection will be examined and discussed, and implications on nozzle design drawn.

2. Description of modeling tools

The calculations reported here were performed with a modified version of the FLOW3D computer code. This code is a 3D incompressible Navier–Stokes solver utilizing the volume-of-fluid methodology for tracking free surface movements and piece-wise planar free surface reconstruction. Volume-of-fluid refers to the technique where a fluid density function f is advected as a Lagrangian invariant around the computational space. The location of the free surface can be reconstructed from f, where:

$f = 0$ empty cell,

$f = 1$ full cell,

$0 < f < 1$ partially full cell.

Numerically calculated gradients of f are used to determine the surface normal vector for partially full cells, which is then used in assigning slopes to

planes representing the reconstructed free surface for visualization purposes. Surface curvature is also calculated from the normal vectors and used in the application of surface tension forces in the near-free-surface cells.

Modifications to the base program in the form of boundary condition subroutines have been added so that spatially and temporally varying boundary conditions on the inflow can be applied. Shown in Fig. 2 are samples of the velocity profiles being applied, where random fluctuations are added to simulate turbulent flows. It should be pointed out that no special care has been taken in these calculations to reproduce the exact spectrum of turbulent fluctuations, instead a purely random white noise generated with a random number generator is used. The inlet boundary condition subroutines can be used to read in actual turbulence seeds from either measured or directly simulated turbulence data.

3. Rectangular jet deformation

Jets with an initial rectangular cross-section are considered for both the oscillating jet and grid jet arrays pictured in Fig. 1. For a rectangular jet, there are surface tension waves originating from the corners and moving towards the center as the fluid moves downstream, producing the surface tension fan. There are also surface ripples in the center of the jet, between the two surface tension
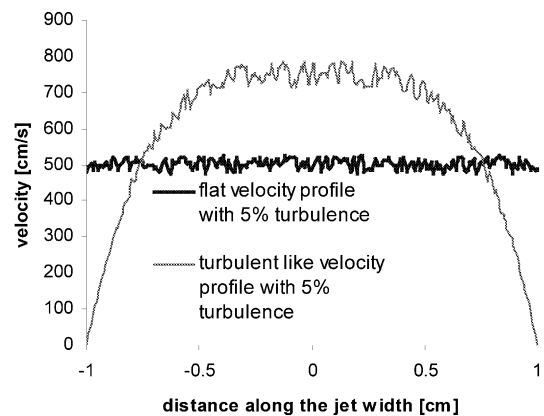

Fig. 2. Velocity profiles used in jet analyses.

fans. Such ripples are observed for any shape of free falling liquid jet and are qualitatively understood to be due to the combined effect of velocity profile relaxation and the impact of turbulence structures on the surface causing distention. Control of the velocity profile and the artificial perturbations added to the velocity profile lead in the numerical results to reproduction of these streaky structures seen in experimental jet simulations, (see Fig. 3). The initial turbulence of the jet is important because once the jet leaves the nozzle there is no sources of turbulence (unless the jet becomes extremely wavy), so only this initial turbulence is present and will decay downstream.

In Fig. 4, we see modeling result of the jet with the flat profile and no turbulence (equivalent to boundary layer cutting). The jet had $Re \sim 10^4$ and $We \sim 10^3-10^4$, to be able to compare to the experiment. This jet does not have any surface ripples. The flow is in the positive z direction, so that in these pictures, the fluid flows from the bottom to the top of the page. As we can see in
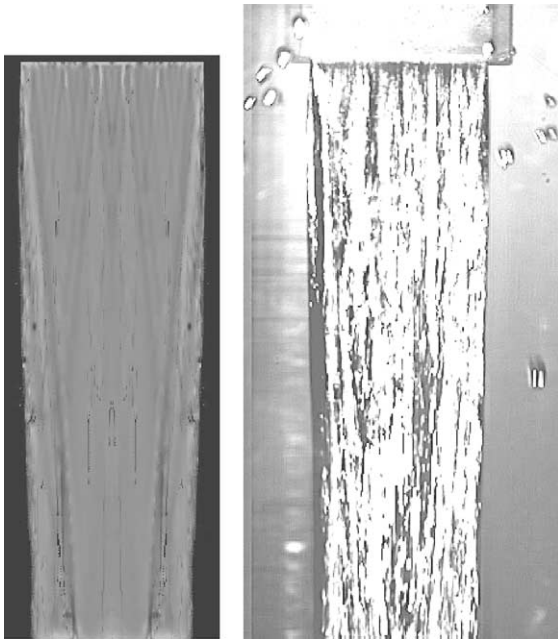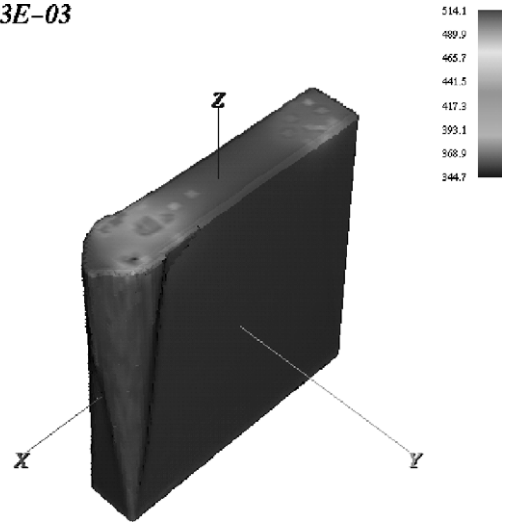
$t=4.003E-03$

Fig. 4. Rectangular jet with the flat profile and no turbulence (colored by z-velocity in cm/s).
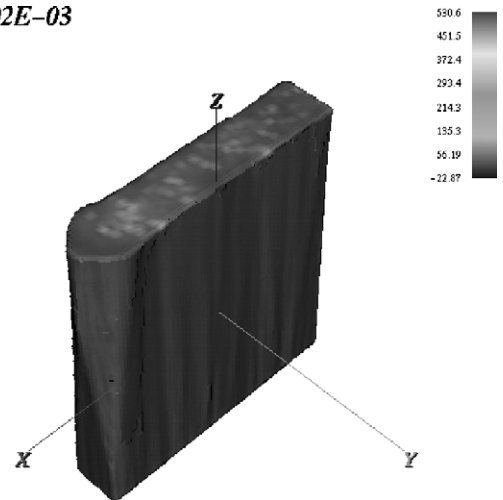
$t=4.002E-03$

Fig. 5. Rectangular jet with the flat velocity profile and 5% turbulence intensity (colored by z-velocity in cm/s).

Fig. 5, the flat profile with three directional fluctuations with peak value equal to 5% of the main flow velocity (see Fig. 2) exhibits the surface ripples/streaks described above. Seen in both figures are the surface tension waves emanating from the corners of the jet. So we can conclude that turbulent disturbance produces the surface ripples. For the jet with flat profile: no disturbance means no surface ripples, other than surface

Fig. 3. Comparison of numerical and experimental [2] image data for rectangular jet with $Re = 20\,000$ and $We = 2000$. Pictured length is 6 cm, liquid is modified Wood's metal, 5% peak fluctuations are used in simulation.

tension fan. Note that secondary flows can produce surface ripples in similar way. But we assume that the flow was conditioned so the secondary flows were eliminated.

In Fig. 6, we see try a velocity profile more closely resembling fully developed turbulent flow, constructed from a fourth order polynomial. The surface is disturbed slightly. In Fig. 7, the jet with the same velocity profile and with 5% turbulence is shown. We see more surface ripples with the turbulence than we see in the case with the same velocity profile, but without turbulence. So we can see that presence of profile by itself produces some of the surface ripples. It happens due to the shear instabilities in the jet. We see less ripples in the case with the fourth power profile, than in the case of the flat profile with 5% turbulence, compare Figs. 7, 6 and 5. So what is happening: the turbulence intensity is low near the surface, because the magnitude of the surface velocity is low near the surface and the turbulence level is proportional to the local velocity magnitude. So the velocity profile plays the dual role in the formation of surface ripples. In one role, the velocity profile is a source of shear instabilities, which causes the appearance of surface ripples. In another role, the presence of velocity profile decreases the
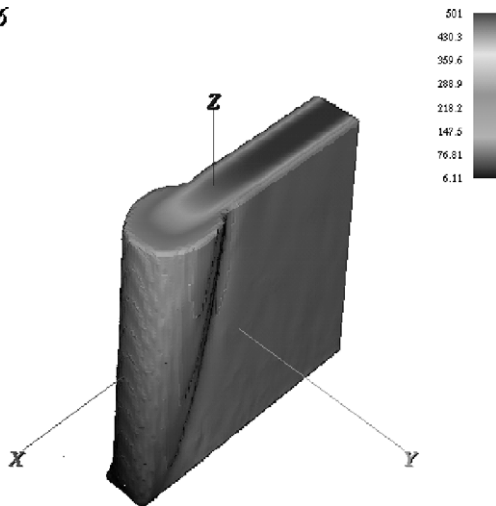
$t=.0136$

Fig. 6. Rectangular jet with fully-developed turbulent velocity profile and no turbulent fluctuations (colored by z-velocity in cm/s).
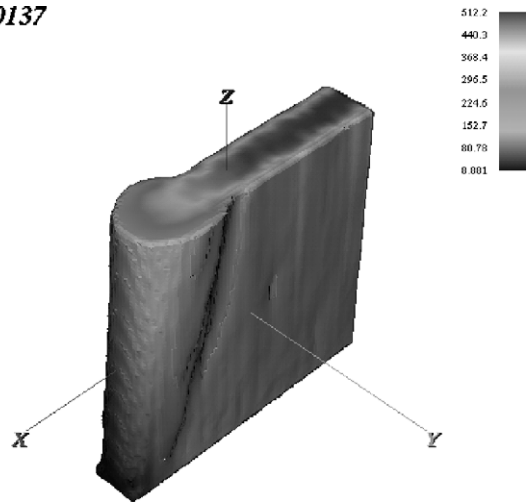
$t=.0137$

Fig. 7. Rectangular jet with fully-developed turbulent velocity profile and with 5% turbulence intensity (colored by z-velocity in cm/s).

magnitude of surface ripples by decreasing the turbulence magnitude near the surface of the jet.

Also modeling has shown that the jet with the flat profile has as high intensity of such disturbances on the surface as the jet with 'turbulent-like' power profile and 5% turbulence. While experiment results [3] shown that the jet with flat profile (via boundary layer cutting) is more stable. The difference is partially due to fact that the perturbations we apply are locally isotropic (same average magnitude in all three directions, while in real flows this may not always be true. For example in the case of tube flow the turbulence level along the direction of the flow is two to three times higher than that of transverse directions [4]. Similarly, nozzle contraction is known to suppress transverse turbulence, forcing the energy into the streamwise fluctuations [5]. The result of modeling with transversal turbulence three times less than longitudinal turbulence is shown in Fig. 8. We see that that the magnitude of the surface ripples decreased. Another reason is that boundary layer cutting erases the fluid past, by cutting off the parts of fluid that were in contact with surface, and where various kinds of instabilities were present. In this modeling we are considering preconditioned flow, in which such secondary motions are eliminated.
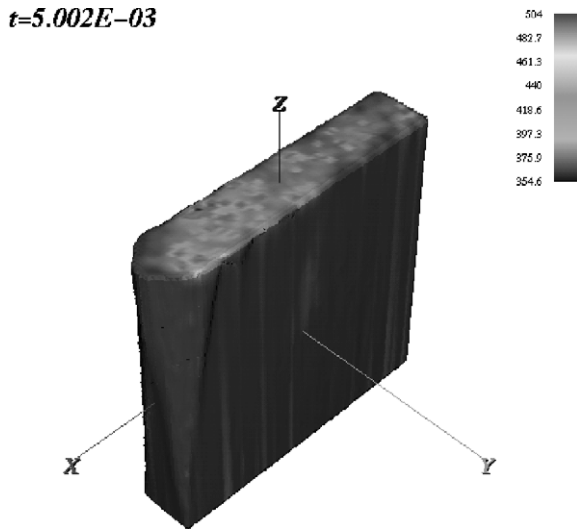
Fig. 8. Rectangular jet with flat velocity profile with 5% turbulence intensity along *z* axis and 1.7% in both transversal directions, (colored by *z* - velocity in cm/s).

Also let us comment on the influence of velocity profile on the surface tension fans at the jet corners. The velocity on the surface with fully developed turbulent velocity profile is zero right after the jet leaves the nozzle. It is increasing downstream due to the velocity profile relaxation—a process that is slower than that of flow development in a tube [6]. So to make the surface velocity comparable with bulk velocity it takes more than 50 hydraulic diameters. It means that in our cases the surface velocity of the jet with the fully developed turbulent profile was low in comparison with the surface velocity of the jet with the flat velocity profile. In this case, the surface tension forces have more time to act in the cases of the turbulent velocity profiles than in the case with the flat velocity profile, considering the same length of the flight, leading to larger 'bulging' of the corners, see Figs. 5 and 6.

4. Droplet ejection from circular jets

Circular jets are also considered for grid jet arrays, were a bank of such jets might replace a singe rectangular jet. One issue raised grid jets with high We and Re number jets is the possibility of droplet ejections that might obscure beamlines during the beam pulses. Some data on this phenomena exists for circular jets [7] in the range of $Re \sim We > 10^5$, which is typical for IFE applications as well. A study on the effect of the velocity profile and turbulent perturbations on the droplet formation is presented here. Three different velocity profiles were modeled: flat (ideal boundary layer cutting), parabolic (laminar) and fourth order polynomial (turbulent-like) profiles (see Fig. 2). No droplet ejection was observed without the addition of initial perturbations for any of these velocity profiles.

For the jet with the flat velocity profile and 5% perturbation level (see Fig. 9), droplets were not seen in the near nozzle region, although some surface distortions were observed. For the jets with parabolic velocity profile (see Fig. 10) and turbulent-like profile (see Fig. 11), both with 5% turbulence level, droplet ejection was observed. Although the inertial forces and the stability of the jet are the determining factors, the droplet ejection is caused by the turbulent disturbances and the shear instabilities. So the flat profile (highest inertia on the surface and the most stable) is found to be the most stable one. The parabolic profile appears to be similar to the more sharp 'turbulent-
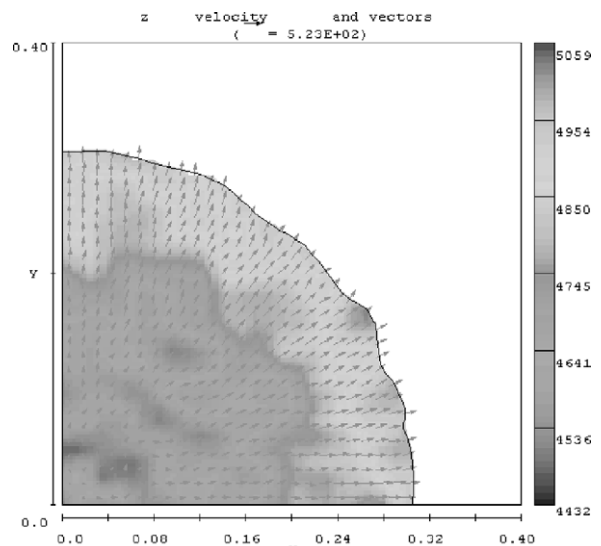

Fig. 9. Circular jet with flat velocity profile with 5% disturbance (at distance $x/\delta = 1/2$ from the nozzle).
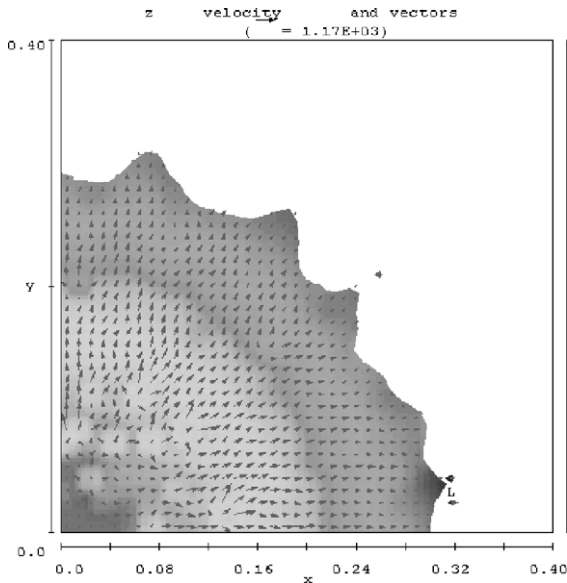
Fig. 10. Circular jet with parabolic velocity profile with 5% disturbance (at distance $x/\delta = 0.3$ from the nozzle).
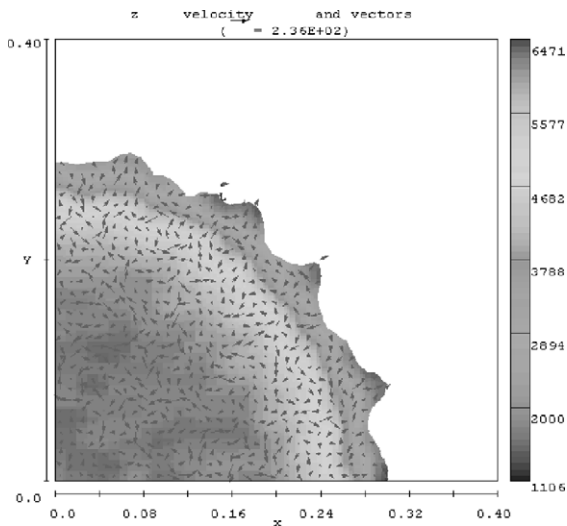

Fig. 11. Circular jet with fourth power velocity profile with 5% disturbance (at distance $x/\delta = 0.3$ from the nozzle, contours colored by axial velocity in cm/s).

like' power profile. These results agree with the results by Wu [3], who observed the droplet ejection at the similar non-dimensional numbers. He has seen in his experiments that cutting of the boundary layer significantly decreases the droplet ejection.

5. Conclusions

The objective of these numerical studies was to develop recommendations on how to avoid unwanted hydrodynamic phenomena (excess rippling, deformation and droplet ejection) for the free surface jet in a vacuum, which are used for the beam line protection in the designs of the heavy ion inertial fusion reactors. Such recommendations were developed by qualitative understanding of the physical mechanisms that caused these hydrodynamic phenomena:

1) The corner bulging is caused by the surface tension forces due to high curvature at the corners. By rounding the corners, we can decrease and avoid the jet thickness that exceeds the original thickness. Also the magnitude of the corner bulging depends on the inertial forces near the surface, which depends on the velocity profile. The jet with the flat velocity profile has the highest velocity on the surface (highest inertia). So the flat velocity profile will have the smallest bulging for a given flow length.
2) The surface ripples (except near the corners) are caused mostly by the turbulent disturbances on the surface, the magnitude of which depends on the velocity profile.
3) The droplet ejection is caused by the turbulent disturbances and shear instabilities. The ratio of the inertial forces to the disturbance is important also. So the flat profile (highest inertia on the surface and the most stable) is found to be the most stable one. The parabolic profile appears to be similar to the more sharp 'turbulent-like' power profile.

Considering all the above, the jet should have the flat velocity profile, which is equivalent to boundary layer cutting, in order to decrease or completely eliminate the droplet ejection and the corners bulging. But such profile might increase the intensity of the surface ripples depending on the

turbulence pre-conditioning. So the nozzle designs should push towards the uniform profiles with a narrow boundary layer to avoid the droplet ejection and bulging, while keeping the surface ripples minimal. How narrow is a trade off between causing the droplet ejection and the bulging against the surface ripples. Also the turbulent intensity should be minimized before the fluid enters the nozzle. These conclusions are in general agreement with more standard circular nozzle design guidelines set out by Hoyt and Taylor, [8] based on empirical studies at high contraction ratio.

References

[1] P.F. Peterson, Introduction to Liquid-Wall Chamber Configurations and Phenomena, IFE Chamber Technology Program Review, April 26–27, 2001.

[2] A. Konkachbaev, N.B. Morley, et al., Stability and contraction of a rectangular liquid metal jet in vacuum environment, Fusion Eng. Des. 51–52 (2000) 1109–1114.

[3] P.K. Wu, R.F. Miranda, et al., Effect of initial flow conditions on primary breakup of nonturbulent and turbulent round liquid jets, Atom Sprays 5 (1995) 175–196.

[4] G.K. Batchelor, The Theory of Homogeneous Turbulence, The Cambridge University Press, 1960.

[5] A.K. Hussain, V. Ramjee, Effects of axisimmetric contraction shape on incompressible turbulent flow, Transactions of the ASME, March 1976.

[6] L.D. Soderberg, P.H. Alfredson, Experimental and theoretical stability investigations of plane liquid jets, J. Mech. B/Fluids 17 (5) (1998) 689–737.

[7] P.K. Wu, G.M. Faeth, Onset and end of drop formation along the surface of turbulent liquid jets in still gases, Phys. Fluid 7 (1995) 2915–2917.

[8] J.W. Hoyt, J.J. Taylor, Effect of nozzle shape and polymer additives on water jet appearance, ASME, 101, September 1979.