Available online at www.sciencedirect.com
SCIENCE DIRECT
ELSEVIER
Nuclear Instruments and Methods in Physics Research A 544 (2005) 337–341
NUCLEAR INSTRUMENTS & METHODS IN PHYSICS RESEARCH
Section A
www.elsevier.com/locate/nima

Three-dimensional modeling of slab liquid jets used for heavy-ion fusion for beam line protection

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

Mechanical and Aerospace Engineering Department, 43-1134 Eng. IV, University of California, 420 Westwood Pza, Los Angeles, CA 90095-1597, USA

Available online 2 March 2005

Abstract

IFE designs for thick liquid protection of heavy-ion inertial fusion reactors utilize banks of liquid jets to protect sensitive beam line components from neutrons and debris following target explosions. IFE designers must have knowledge of the surface quality of these jets in order to determine the distance between the jets and the ion beams that must propagate through the void spaces between them. Here, numerical simulations of such jet flows performed with the customized Flow3D solver are reported. These numerical simulations predicted no significant jet breakup in the region of interest, but did show surface and shape deformation that may end up determining the minimum standoff distance between jets and driver beams. The simulations also show small-droplet ejection that may adversely affect beam propagation characteristics. The intrusion distance of liquid into the beam lines was determined to be below 10% of the original jet thickness throughout the computational domain (up to 1 m downstream from the nozzle). Recommendations on how to avoid or minimize unwanted hydrodynamic phenomena (surface rippling and droplet ejection) by upstream conditioning and nozzle design are developed for free surface jets in vacuum in the context of a qualitative understanding of the physical mechanisms at play.

Keywords: Free surface flow; Turbulence; IFE beamline protection; Liquid jet

1. Introduction

The "grid" jets, as pictured in Fig. 1, protect the driver hardware and first solid surfaces in the reactor chamber from debris and X-rays given off by the exploding fusion target. For rectangular-slab- grid jets, the objective of these simulations was to study surface characteristics in the characteristic IFE design parameter range—with typical non-dimensional parameters: Reynolds (Re) and Weber (We) numbers on the order of 10^5. Computational fluid dynamics (CFD) modeling was used to study the development of the jet and to develop recommendations on how to

*Corresponding author. Tel.: +1 310 206 1230; fax: +1 310 825 2599.
E-mail address: morley@fusion.ucla.edu (N.B. Morley).

0168-9002/$ - see front matter
doi:10.1016/j.nima.2005.01.230

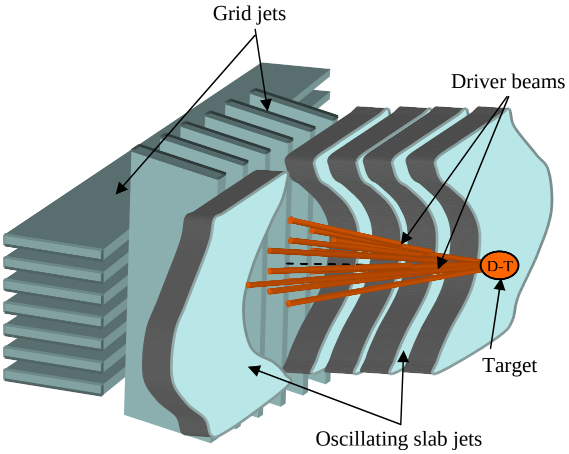

Fig. 1. Schematic view of liquid protection system in heavy-ion chamber.

minimize unwanted phenomena within the IFE design constraints. This work was meant to complement experimental investigations of these phenomena [1].

While it is generally accepted that very-high-area contraction in nozzles can *laminarize* the flow by pressure suppression of upstream turbulence, IFE design constraints resulting from the need for close packing of jets do not allow a large flow area contraction ratio. So this technique cannot be utilized beyond a contraction ratio of about 2:1 and so jets with high *Re* will always be turbulent. There are two design parameters, though, that can be manipulated to control jet characteristics: the upstream flow conditioning and the nozzle shape. Changes in designs can then be utilized to affect the outlet velocity profile, relative turbulence intensity and initial jet shape.

2. Numerical simulation of IFE grid jet at high *Re* and *We* numbers

The numerical simulation of typical-size grid jets potentially used in IFE, with width and thickness ranging from 10 to 40 cm, was performed using the LES turbulent model with seeded spatial and temporal fluctuations of initial jet velocity to model turbulent perturbations. The commercial code Flow3D with customized subroutines was used to perform these simulations. The details of the method and benchmarking of the simulations for certain control cases are described in detail in Refs. [2,3].

Fig. 2. Simulation of typical IFE grid jets with *We* and *Re* $\sim 10^5$ with inlet conditions: (A) flat, average-velocity profile and (B) "turbulent-like" flatter average-velocity profile.

A 3-D view of modeling results of typical IFE rectangular grid jet is shown in Fig. 2, where the surface ripples, which are primarily driven by turbulent disturbances, are easily seen as long streaks. The pictured length is 1 m (limited by numerical resolution restrictions); the rectangular jet has a width of 40 cm, a thickness of 10 cm, and a velocity of 12 m/s (*Z*-direction is downstream), which gives *Re* and *We* $\sim 10^5$ (when material properties of Flibe are used). Note that only half of the jet is shown because the symmetry boundary condition along the thickness of the jet is used to save computational time. Although the symmetry boundary condition can be used along the width of the jet too (i.e. cutting computational domain to only one quarter of the jet), it is not used in order to allow asymmetrical surface deformation. The jet shown in Fig. 2A has a flat average velocity profile, while the jet in Fig. 2B is initiated with more "turbulent-like" flatter average-velocity profile (see Ref. [2]). As is well known, the jet released from the nozzle has non-isotropic turbulence [4]. Therefore, non-isotropic spatial and temporal velocity perturbations were introduced for both velocity profiles using customized inlet boundary condition subroutines, with 10% relative fluctuation in the main stream direction and 5% in two other directions. In Fig. 3, the evolution of surface waves driven by the turbulent perturbations at different location downstream can be seen for these cases.

The surface ripples grow larger downstream, but even at 1 m downstream there are no significant gross shape changes from the original rectangular cross-section (the acceptable ripple size will be discussed later). There is no evidence of droplet ejection. As was found in Ref. [5], a flat velocity profile obtained by boundary layer cutting is the most stable one with respect to droplet ejection in the IFE range of high *Re* and *We* ($\sim 10^5$). As for surface waves, the flat velocity profile is not a source of shear instabilities, which would cause the appearance of surface ripples. In addition, the high speed at the surface layer results in the surface material moving through the region of interest in a short time, reducing the time available for temporal instability growth. However, grid jets with a flat velocity profile (requiring cutting) may be too difficult to incorporate in the IFE design due to spacing problems.

For comparison, modeling results of the jet with a more turbulent-like boundary profile are shown in Fig. 2B. This velocity profile has a flatter, $\frac{1}{4}$-power dependence on the distance from the wall. In Fig. 4, the cross-section of the jet is shown at 2 cm downstream. Small droplets can be seen leaving the surface in this case. Of course, the separation of droplets is highly sensitive to the resolution of the grid in the free surface region. To check the ability of the CFD solver to resolve droplet ejection, the jet used in past experimental work [5] was modeled [2,3]. The results agreed well qualitatively and the same grid resolutions were used in modeling IFE jet since *Re* was the same in both simulations. As a standard procedure, the grid resolution was tripled to check for any changes. Similar droplet ejection is still observed and the physical structure of the jet is still the same as it was in the case with lower grid resolution.

The search for droplets has shown a dozen or so droplets ejected in the first 10 cm downstream. After 10 cm downstream, no new droplet ejections were found within the computational domain. Note that the ejected droplets did not disappear, after leaving the computational domain they would still travel downstream, unless absorbed

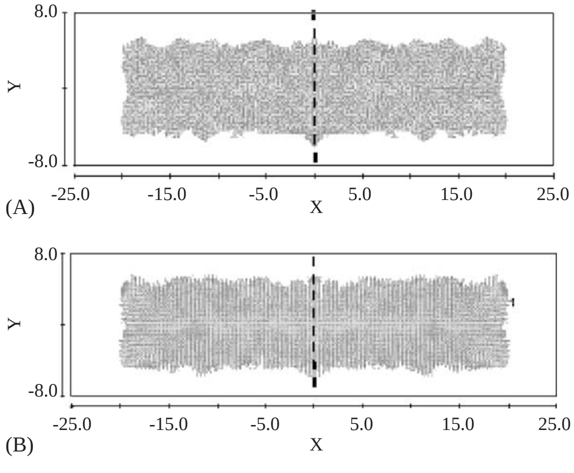

Fig. 3. Cross-sectional view (Fig. 2A) at (A) ~50 cm and (B) 100 cm downstream (symmetry boundary shown as a dashed line).

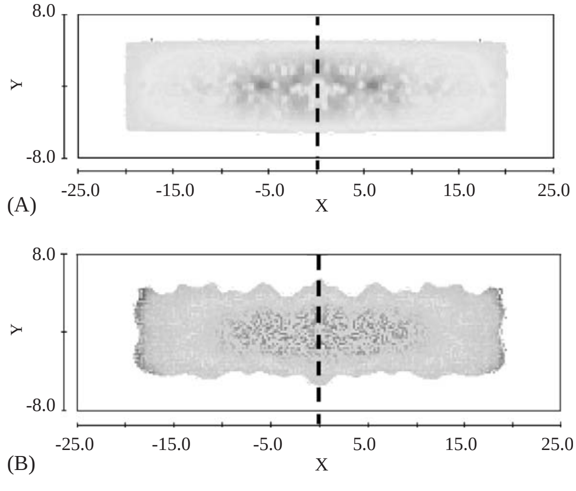

Fig. 4. Cross-sectional view (Fig. 2B) at (A) 2 cm and (B) 100 cm downstream.

by another jet. So we can conclude that the modeling predicts some droplets ejection within the first 10 cm downstream (up to 1 m downstream from the nozzle). It is postulated that the initial turbulence intensity is decreasing with downstream distance, as viscous dissipation is a sink of turbulent energy, and no large-scale shear exists at boundaries to regenerate turbulence. In addition, these simulations have vacuum outside the jet, and further ‘‘striping’’ of protuberances (liquid fingers) by wind shear is not occurring as it does in experimental jet simulations in atmosphere [5].

As was discussed above, surface waves grow with downstream distance. We can see additional confirmation in Fig. 4, where the cross-section of the jet with the turbulent-like velocity profiles are shown at different distances downstream, at 2 cm and 1 m downstream. The observed ‘‘intrusion’’ distance of the surface waves is defined as the furthest distance inward from the original position of the free surface where liquid is found. In Fig. 5, we can see that the intrusion distance depends on the distance downstream, but that it did not exceed 1 cm even at 1 m downstream. This 1 cm is approximately 10% of the original thickness of 10 cm. This distance would be the acceptable standoff of a driver beam from the initial jet location. An intrusion distance of 5–10% is expected to be acceptable for IFE chambers, as specified in Ref. [6].

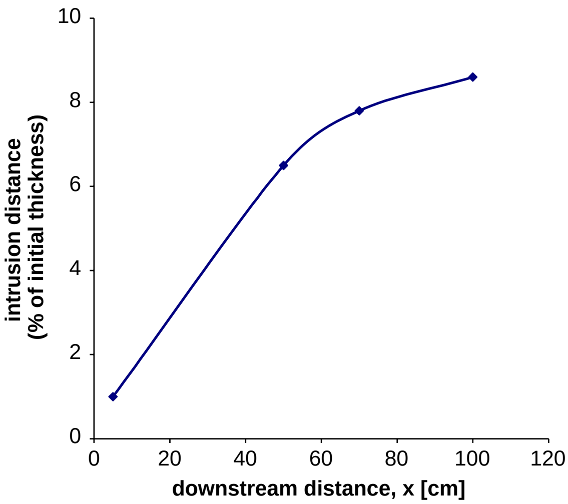

Fig. 5. Maximum intrusion distance of the surface waves for the jet shown in Fig. 2B as a function of distance downstream.

3. Discussion and conclusions

These numerical simulations predict no significant jet breakup in the parameter range and flow region of interest for the typical IFE grid jet, but do show surface and shape deformation that might end up determining the minimum standoff distance between jets and driver beams, as well as some small droplet ejection that may adversely affect beam propagation characteristics. The intrusion distance of liquid into the beamlines was determined to be below 10% of the original jet thickness throughout the computational domain, up to 1 m downstream from the nozzle where it appears to be saturating. So the IFE design can take these predicted surface characteristics for the typical grid jet into account.

Another objective of this work is to develop recommendations on how to avoid or minimize unwanted hydrodynamic phenomena (surface rippling and droplet ejection) for the free surface jet in a vacuum. Such recommendations were developed by a qualitative understanding of the physical mechanisms, which lead to existence of these hydrodynamic phenomena. Concerning droplet ejection, it was found that the turbulent-like disturbances combined with shear instabilities (which are imposed by the presence of the boundary layer) lead to droplet ejection. However, turbulence perturbations of typical intensity or the shear instabilities by themselves do not lead to droplet formation in the previously discussed range of parameters within 20 hydraulic diameters downstream. The modeling results qualitatively agree well with the experimental results of Ref. [5], where the effects of the boundary layer cutting on the droplet ejection were studied (boundary layer cutting results in nearly flat velocity profile). So the recommendations to avoid or minimize the droplet ejection would be to have the velocity profile as flat as possible while trying to minimize the level of turbulence.

Future work should include the use of more realistic turbulent perturbations and the effects of complex geometry of the flow nozzle on resultant velocity profiles. In addition, the development of better surface tracking models, customized specifically for droplet ejections at high *Re* and *We*, should be explored.

Acknowledgements

This work was performed with the support of Department of Energy grant no DE-FG03-94ER54287.

References

[1] S.G. Durbin, M. Yoda, S.I. Abdel-Khalik, D.L. Sadowski, T.P. Koehler, ARIES Team, Fusion Sci. Technol. (2004), to appear.
[2] A. Konkachbaev, N.B. Morley, M.A. Abdou, Fusion Eng. Des. 63–64 (2002) 619.
[3] A.I. Konkachbaev, Ph.D. Thesis, University of California, Los Angeles, 2004.
[4] H. Schlichting, Boundary Layer Theory, seventh ed., McGraw-Hill, New York, 1979.
[5] P.K. Wu, et al., Atom. Sprays 5 (1995) 175.
[6] N.B. Morley, A.Y. Ying, A. Gaizer, T. Sketchley, A.I. Konkachbaev, M.A. Abdou, Fusion Technol. 34 (1998) 1035.