ELSEVIER

Journal of Nuclear Materials 258–263 (1998) 440–445

journal of nuclear materials

Experimental and analytical studies on high-speed plane jet along concave wall simulating IFMIF Li target flow

Hideo Nakamura [a,*], Kazuhiro Itoh [b], Yutaka Kukita [b], Mizuho Ida [a], Yoshio Kato [a], Hiroshi Maekawa [a], Hiroji Katsuta [a]

[a] *Japan Atomic Energy Research Institute (JAERI), Tokai-mura, Ibaraki-ken 319-1195, Japan*
[b] *Nagoya University, Furo-cho, Chikusa-ku, Nagoya-shi 464-8603, Japan*

Abstract

As part of the conceptual design activity (CDA) of the international fusion materials irradiation facility (IFMIF), the characteristics of the high-speed liquid lithium (Li) plane jet target flow have been studied by water experiments and numerical analyses for both heating and non-heating conditions. The simulated prototypal-size water jet flows were stable over the entire length of ~130 mm at the average velocity up to 17 m/s. The jet flow had a specific radial velocity profile, close to that of free-vortex flow, because of a static pressure distribution in the jet thickness due to centrifugal force. Detailed velocity measurement revealed that this flow condition is penetrating into the upstream reducer nozzle up to a distance ≈ the jet thickness. The numerical analyses using a two-dimensional Cartesian-coordinate model were successful to predict the velocity profile transient around the nozzle exit, though underestimated the development of the velocity profile and the jet thickness. © 1998 Elsevier Science B.V. All rights reserved.

1. Introduction

The IFMIF will provide a high-flux, high-energy (~14 MeV) neutron irradiation field by d-Li stripping reaction for testing and development of low-activation and damage-resistant fusion materials. Two deuteron (d) beams of 125 mA, 30–40 MeV are focused onto a rectangular region (50 mm-high × 200 mm-wide) on a high-speed (10–20 m/s) liquid Li target flowing along a concave back-wall in a vacuum as schematically shown in Fig. 1. The total beam power of ~10 MW is to be absorbed within the jet thickness of ~25 mm, which is slightly larger than the beam trajectory range.

The characteristics of liquid plane jet flowing along a concave wall have been studied in JAERI, by means of water simulation experiments, as part of the CDA of the IFMIF [1–3] that started in February 1995. The intensity and stability of the generated neutron beam and the integrity of the target assembly depend on the stability of the Li target flow during the beam irradiation. The water experiments simulated the Li target flows [4] to investigate fundamental non-heating flow conditions. A tentative but practical design of the IFMIF target assembly proposed by JAERI was employed in the water experiments by referring to a design work for the fusion materials irradiation test facility (FMIT) [5–7]. The size of the test section was prototypal to preserve the Reynolds number, Re, and the Froude number, Fr, in the simulated flows. The influence of the surface tension on the jet free surface condition that would be presented by the Weber number, We, was studied by changing the jet average velocity.

In parallel with the water experiments, multi-dimensional numerical analyses have been performed in JAERI to study both the thermal and hydraulic response of the IFMIF target flow [8,9]. The temperature margins in the IFMIF target were found to be enough to avoid significant vaporization and voiding respectively at the free-surface and the peak temperature location in the jet flow for the tested conditions; back-wall radius

---

[*] Corresponding author. Address: Department of Reactor Safety Research, Japan Atomic Energy Research Institute (JAERI), Tokai-mura, Ibaraki-ken 319-1195, Japan. Tel.: +81 29 282 5846; fax: +81 29 282 6728; e-mail: nakam@lstf3.tokai.jaeri.go.jp.

0022-3115/98/$19.00 © 1998 Elsevier Science B.V. All rights reserved.
PII: S0022-3115(98)00235-9

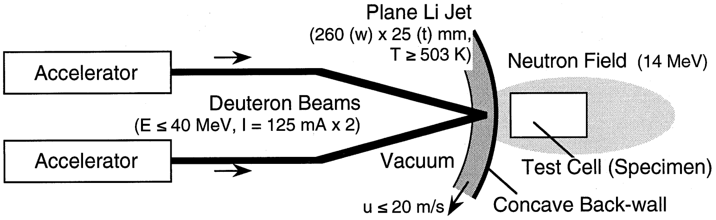

Fig. 1. Schematic illustration of target configuration for neutron generation.

from 100 to 1000 mm and average jet velocities from 10 to 20 m/s.

This paper summarizes the previous results and recent efforts to clarify the IFMIF jet characteristics especially on the flow response around the reducer nozzle which determines the quality of the jet flow.

2. Experiments

2.1. Test facility

Details of the experimental setup are described in Ref. [4]. Fig. 2 indicates a side view of the test section which was made of transparent acrylic resin for a visual observation of the flow. Demineralized water in a 10 m^3 water tank was circulated through the test section by a large pump, obtaining an average jet velocity up to ~17 m/s. To decrease turbulence intensity in the jet, a two-dimensional double-reducer nozzle was proposed to achieve a large contraction ratio with no flow separation in the nozzle by employing Shima's reducer nozzle model [10] based on the potential flow theory. A 12 mm-long straight section was placed at the nozzle exit to have a uniform velocity profile based on the Shima's measurement [10]. The reducer nozzle discharged the jet flow tangentially onto the concave back-wall (240 mm-wide, 250 mm-radius) which had an angle range ±11.3° giving a 98.6 mm-long wall. A 40 mm-long straight section was added at the downstream end of the back-wall, resulting in a total free-surface length of 129 mm.

The interfacial structure of the jet free surface was observed with a high-speed video camera (HSV-1000, nac). The jet thickness was measured with a ultrasonic flaw detector (USIP 11, Krautkrämer GmbH) from the outside of the back-wall. The jet velocity profiles were measured with a Laser-Doppler velocimeter (RSA-2000, Aerometrics/TSI) employing a back-scattering-type fiber-probe.

2.2. Experimental results

2.2.1. Summary of previous results [4]

The water jet flows were stable over the entire length of ~130 mm without droplet entrainment from the free surface at the average velocity up to 17 m/s. Interfacial waves with a wave length of ~1 mm were found to appear only when the average jet velocity exceeded ~3 m/s. The characteristics of waves depended highly on the average jet velocity. Three subsequent regions of

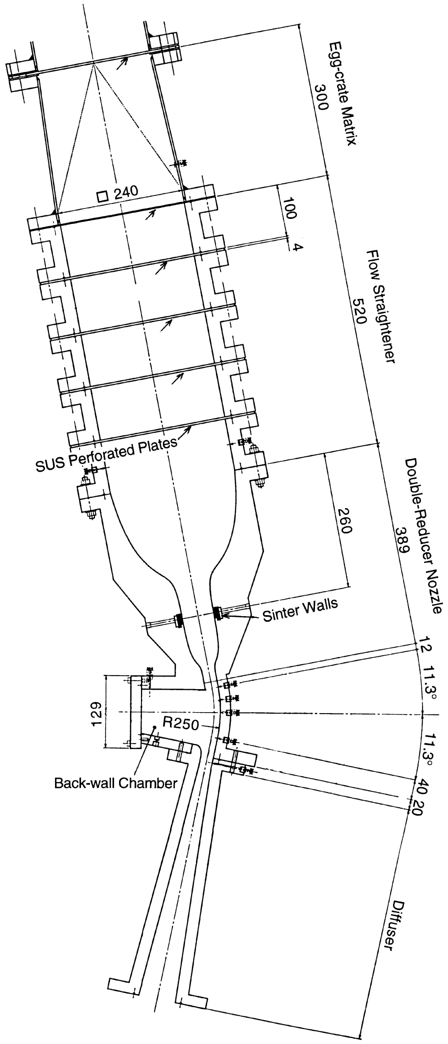

Fig. 2. Schematic of target test section for water experiments.

interfacial structures appeared; smooth surface immediately downstream of the nozzle exit, two-dimensional ripple-like waves and finally three-dimensional waves. The axial lengths of both the smooth region and two-dimensional wave region were shortened as the jet velocity was increased. The three-dimensional waves covered the whole surface when the average velocity exceeded ~10 m/s. The wave amplitudes did not increase much along the length of the jet, probably because of a stabilizing effect of the centrifugal force.

The change in the thickness along the jet flow was small for a relatively high average jet velocity (>4.8 m/s). The measured jet thickness increased by ~1 mm around the nozzle exit due to the change in the velocity profile. The change in the jet thickness was insignificant over the length of the jet at the velocity of 9.6 m/s.

The jet velocity profile closely agreed with that of irrotational free-vortex flow; u_r r = constant, where u_r is the stream-wise velocity at a given radius r. The velocity was thus maximum at the free surface and decreased in the jet thickness. This specific flow condition appeared in the jet because the radial static pressure developed in the jet thickness by the centrifugal force immediately after the jet exited the nozzle. This flow condition is beneficial to limit the interfacial temperature during the IFMIF operation.

2.2.2. Velocity profiles

Recently performed detailed velocity measurement revealed that the above specific velocity profile also appears in the nozzle. Fig. 3(a) and (b) respectively indicate the jet velocity profiles measured in the nozzle and in the jet at an average velocity of ~9.6 m/s. Both X and Y axes are normalized to the average velocity and the flow thickness, respectively. The velocity profiles at the other flow rate were essentially similar to those shown in Fig. 3. The data clearly show that the region with the radial static pressure distribution formed in the jet penetrated into the upstream nozzle being accompanied by the specific velocity profile. The velocity profile became insignificant with a distance from the nozzle exit.

The boundary layer formed on the upstream nozzle front wall resulted in a thin low-velocity region beneath the jet free surface as shown in Fig. 3(b). The velocity in the thin region increased along the length of the jet, being accelerated by fluid in an adjacent high velocity

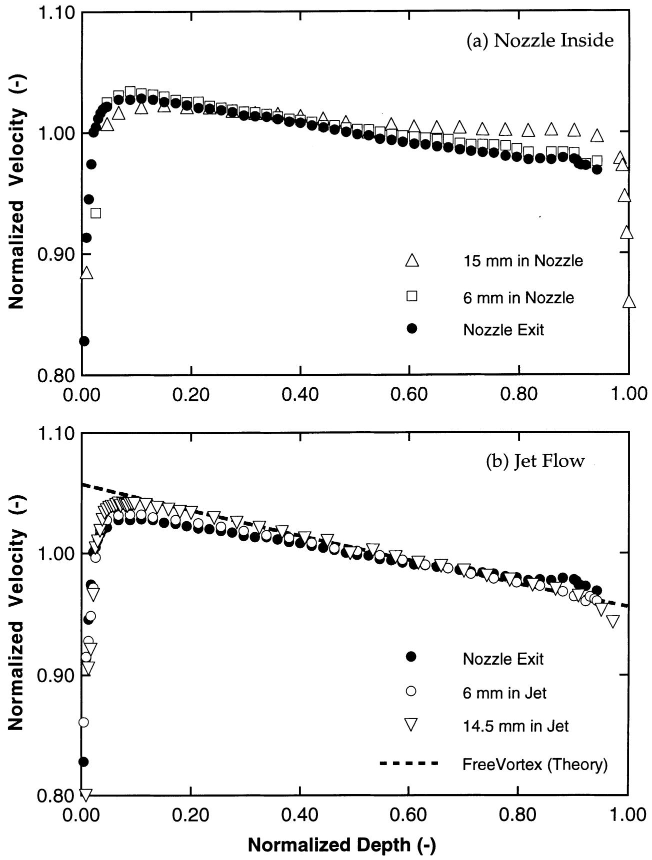

Fig. 3. Measured velocity profiles: (a) Nozzle inside; (b) jet flow.

region. A distance between the deuteron beam and reducer nozzle would thus have an effect to suppress the local temperature increase at the jet free surface.

3. Numerical analysis

Concerning the experimentally observed specific velocity distributions around the nozzle exit, the FMIT two-dimensional analyses revealed that a cylindrical-coordinate model covering only the jet flow on the concave wall fails to represent the velocity profile change at the nozzle exit [7]. Inlet velocity profile was just preserved along the length of the jet. The free-vortex-like velocity profile appears in the jet because the static pressure distribution changes at the exit of the nozzle. The rigorous prediction of the velocity profile transient around the nozzle was attempted using a two-dimensional Cartesian-coordinate model.

3.1. Calculation conditions

A multi-dimensional multi-fluid flow simulation code FLOW-3D v.6. [11] was used, because this code can deal with a gas–liquid interface using the "volume of fluid" (VOF) model, being beneficial to predict the jet behaviors with free surface. Two-dimensional Cartesian-coordinate model with a square mesh (0.5×0.5 mm) shown in Fig. 4 was used to simulate the flow response that greatly changes at around the nozzle exit. Second-order monotonicity preserving method was used for the numerical approximation of momentum equation. For simplicity, a uniform velocity profile with an average velocity of 10 m/s was assumed for an inlet flow to a straight nozzle. Furthermore, the gravity acceleration, wall shear and flow turbulence were ignored. The ~50 mm-long curved back-wall was modeled as an obstacle by the FLOW-3D "FAVOR" method [12].

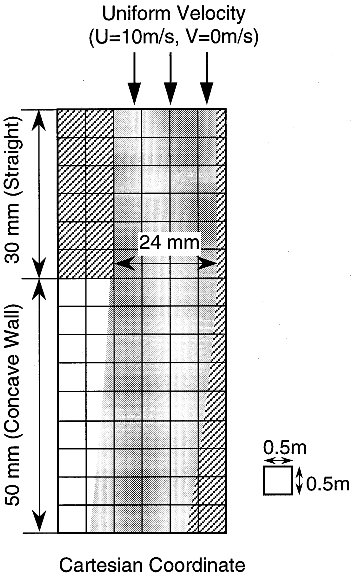

Fig. 4. Schematic of calculation model.

3.2. Calculated results

Fig. 5 shows the calculated pressure distribution in the flow. The pressure distribution in the nozzle is magnified in a separate figure. This result clearly indicates that the edge of the co-axially developed pressure distribution in the jet extends into the nozzle up to a distance ≈ the nozzle exit width. The depth of the "influenced" region into the nozzle was confirmed by parameter analyses using long nozzles (results not shown). The pressure distribution near the downstream end became highly oscillative numerically, probably because of the assumed "continuous" boundary condition at the bottom of the present model. The velocity profile and jet thickness to be used hereafter are thus limited to the region of the jet flow calculated with smooth co-axial pressure distribution.

Fig. 6 shows the velocity profiles around the nozzle exit, being compared with that of the free vortex flow. Both X and Y axes are normalized to the average velocity and the flow thickness, respectively. The velocity profile changed gradually along the flow axis. At the nozzle exit, the velocity around the front wall increased greatly partly because the wall shear was ignored. The calculated results did not reach the free vortex velocity profile at 6 mm from the nozzle exit, though the data shown in Fig. 3(b) agreed well with the theoretical prediction. Fig. 7 compares the calculated results with the

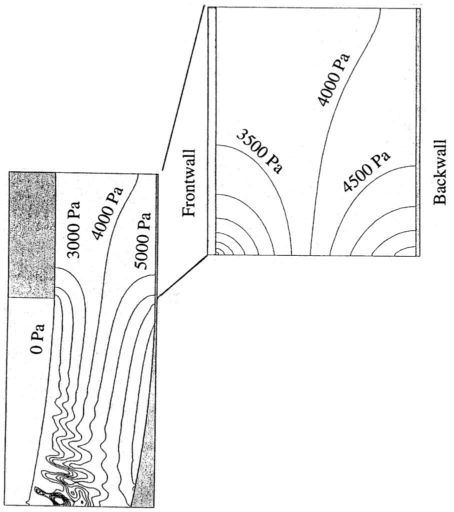

Fig. 5. Pressure contours.

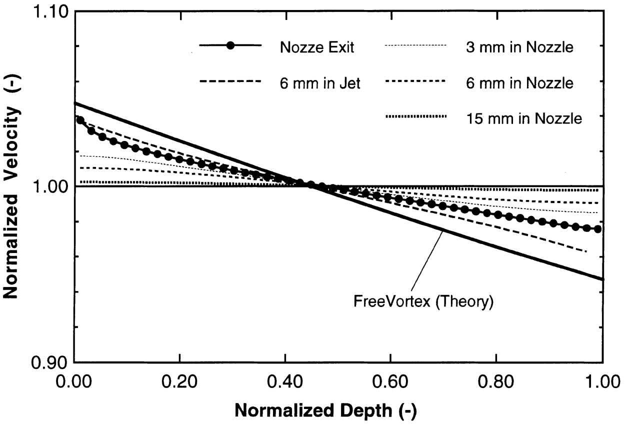

Fig. 6. Calculated velocity profiles around nozzle exit.

data at five locations along the flow axis. The predicted velocity profile underestimated the inclination especially in the nozzle. The absence of the boundary layer because of the slip-wall assumption may contribute to such underestimation especially for the front wall side. The influence of static pressure distribution in the jet penetrated deeper into the nozzle than that predicted.

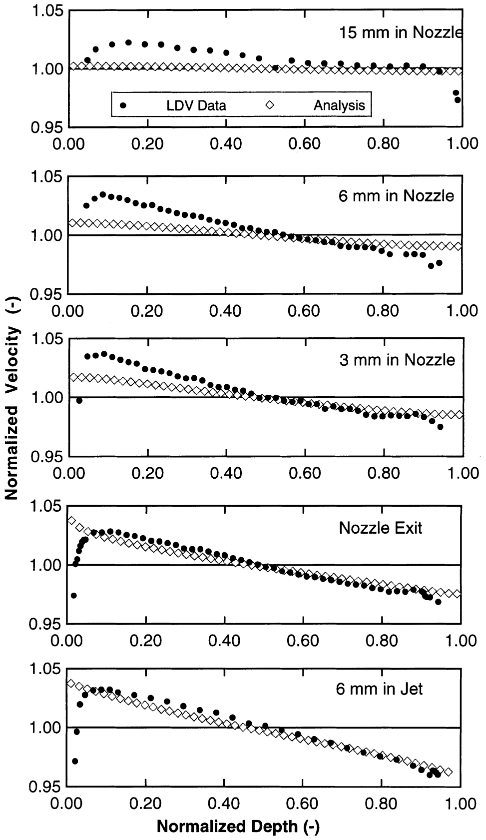

Fig. 7. Comparison of velocity profiles around nozzle exit.

The increase in the calculated jet thickness was as much as 0.2 mm (not shown), and was far smaller than the measured value of ~1 mm near the nozzle exit, probably because the slip-wall assumption caused a lack of the boundary layer development along the walls.

4. Conclusions

The characteristics of a high-velocity plane jet flowing on a concave wall for the IFMIF studied experimentally by water simulation experiments and numerically by multi-dimensional hydrodynamic analyses were summarized. The experiments revealed that the edge of the static pressure distribution in the jet penetrated into the upstream nozzle. The Cartesian coordinate model successfully simulated the trend of the measured velocity profile transient around the nozzle exit which is necessary to the rigorous prediction of the jet flow characteristics. The analyses, however, underestimated the development of velocity profile in both the jet and upstream nozzle, and the jet thickness.

In the future, close comparison between the experiment and numerical analyses will be performed further especially to clarify the interfacial stability based on the fine measurement of the wave forms and velocity profiles near the jet free surface, both of which are necessary to realize stable and long jet target flows.

Acknowledgements

The authors are indebted to members of Nuclear Engineering Co. for their performing the water flow experiments.

References

[1] IFMIF CDA Team, IFMIF International Fusion Materials Irradiation Facility, Conceptual Design Activity, Final Report, RT/ERG/FUS/96/11, ISSN/1120-5563, 1997.

[2] Y. Kato, H. Nakamura, M. Ida, H. Maekawa, H. Katsuta, T. Hua, S. Cevolani, Conceptual design study of IFMIF target system, in: Proceedings of the 8th International Topical Mtg. Nucl. Reactor Th-Hydraulics (NURETH-8), Kyoto, 1997, p. 1260.

[3] H. Katsuta, Y. Kato, S. Konishi, Y. Miyauchi, D. Smith, T. Hua, L. Green, G. Benamati, S. Cevolani, H. Roehrig, W. Schutz, these Proceedings.

[4] H. Nakamura, K. Ito, Y. Kukita, M. Ida, Y. Kato, H. Maekawa, Water experiment of high-speed, free-surface, plane jet along concave wall, in: Proc. NURETH-8, Kyoto, 1997, p. 1268.

[5] W.C. Miller, C.E. Annese, J.D. Berg, R.R. Miles, Testing of the prototype FMIT target with liquid lithium, in: Proceedings of the 3rd International Conference Liquid Metal Engineering and Technology, vol. 2, London, 1981, p. 333.

[6] J.A. Hassberger, Stability of the FMIT high speed, free surface liquid jet flowing along a curved back wall, in: Proceedings of the 10th Symposium on Fusion Engineering, vol. 2, Philadelphia, 1983, p. 1849.

[7] J.A. Hassberger, Velocity Distributions in the FMIT Liquid-Lithium Target, HEDL-TME 81-32 UC-20, 1981.

[8] M. Ida, Y. Kato, H. Nakamura, H. Maekawa, Preliminary Analyses of Li Jet Flows for The IFMIF Target, JAERI-Research 97-030, 1997.

[9] M. Ida, H. Nakamura, A. Ito, Y. Kato, H. Maekawa, Thermal and fluid analysis for the IFMIF lithium target jet flow, in: Proc. NURETH-8, Kyoto, 1997, p. 1276.

[10] A. Shima, Theory of direct and inverse methods to obtain nozzle shape, Mem. Inst. High Speed Mech., Japan 17, 1961/62, No. 164 (in Japanese).

[11] Flow Science, FLOW-3D Theory Manual (1996).

[12] C.W. Hirt, J.M. Sicilian, A porosity technique for the definition of obstacles in rectangular cell meshes, in: Proceedings of the 4th Int. Conf. Ship Hydro., National Academy of Science, Washington, DC, 1985.