[4]Memory, S. B., and Rose, J. W., "Free Convection Laminar Film Condensation on a Horizontal Tube with Variable Wall Temperature," *International Journal of Heat and Mass Transfer*, Vol. 34, No. 11, 1991, pp. 2775–2778.

[5]Lee, W. C., Rahbar, S., and Rose, J. W., "Film Condensation of Refrigerant 113 and Ethanediol on a Horizontal Tube—Effect of Vapor Velocity," *Journal of Heat Transfer*, Vol. 106, No. 3, 1984, pp. 524–530.

[6]Karimi, A., "Laminar Film Condensation on Helical Reflux Condensers and Related Configurations," *International Journal of Heat and Mass Transfer*, Vol. 20, No. 11, 1977, pp. 1137–1144.

Transient Thermal Mixing Following Sudden Transverse Injection of a Fluid

Fariborz Khodabakhsh,* Ramesh Konduri,† Sastry Munukutla,‡ and Stephen Idem§
Tennessee Technological University, Cookeville, Tennessee 38505

Introduction

THIS Note presents the results of an experiment designed to measure thermal transients induced by a sudden transverse injection of a fluid into a main flow of different temperature. Mixing problems of this nature are commonly encountered in such industrial applications as chemical processing, petroleum refining, and electrical power production. For example, in a boiling water reactor (BWR) of a nuclear power plant, reactor cleanup water is injected into the main feedwater flow. The cleanup water line is mounted transversely to the feedwater line. The temperatures of the main flow and injection flow are, in general, different from each other. A mixing region exists downstream of the injection point, wherein radial, circumferential, and longitudinal temperature gradients exist in the flow. During startup and shutdown operation, large temperature and flow rate changes occur in the mixing region. The transient fluid temperatures are strongly coupled with the pipe wall thermal response.

A search of the existing literature indicates that although a large number of references have been published on thermal mixing problems, very little work has been conducted on transient thermal mixing for transverse injection in cylindrical geometry. Pryputniewicz and Bowley[1] presented the temperature characteristics of a hot rising plume as a function of discharge Froude number and discharge depth. Salzman and Schwartz[2] investigated the trajectory and dispersion of a solid-gas jet injected into a transverse stream. Kamotani and Greber[3] and Chassaing et al.[4] examined the longitudinal and transverse distributions of velocity, temperature, and turbulence intensity of turbulent circular jets issuing into a crossflow. Wu et al.[5] studied four slotted cross section jets in crossflow and compared their behavior with circular jets. Unsteady jets were also investigated. Oosthuizen[6] studied the behavior of heated air jets discharged vertically from "shaped" circular nozzles into still air. Sherif and Pletcher[7] measured mean and fluctuating temperature fields of a heated jet discharging into cold water. Andreopoulos[8] presented mean temperature measurements and velocity-temperature fluctuations in a heated jet issuing perpendicularly into a cold stream at various velocity ratios. Andreopoulos[9] performed spectral analysis and flow visualization on a jet issuing perpendicularly into a developing crossflow. Sherif and Pletcher,[10] Andreopoulos,[11] and Andreopoulos and Rodi[12] measured turbulence characteristics of round jets in crossflow. Krausche and Fearn[13] studied the influence of injection angle of a round jet in a crossflow on vortex properties.

Amos and Moody[14] presented a procedure for predicting temperature loadings for thermal stress calculations in thick-walled pipes. It was shown that the longitudinal temperature gradient in a thick-walled pipe will cause less thermal stress than would be predicted from thin-walled theory. Konduri et al.[15] described an experiment on a one-fourth scale model designed to simulate the mixing process that occurs in the feedwater line of a BWR nuclear powerplant following transverse injection of hot reactor cleanup water. The transient temperature response of the pipe wall in the mixing region was found to vary in a first-order manner.

Experimental Program

A water flow loop was constructed in order to investigate the effects of suddenly mixing two fluid streams of different temperatures. The scale model experiment was designed to simulate the thermal mixing process which occurs in the main feedwater line of a BWR power plant. The flow rates in the main feedwater line and in the injection line were determined by means of orifice plates which had been calibrated in situ. Wall temperatures were monitored at different locations in the test section by means of copper-constantan thermocouples mounted flush with the pipe wall. For a complete description of the test setup and a discussion of experimental uncertainties, refer to Konduri et al.[15]

Two types of transient flows were studied. In the first type, water at room temperature was flowing steadily in the main feedwater line while the "hot" water was injected suddenly through the injection line. This type of test is referred to as hot injection. In the second type, the main feedwater flow was initially shut off by a valve. Hot injection water flow was pumped through the injection line, thereby inducing a steady flow rate through the test section of the main line. The transient was created by suddenly starting the main feedwater flow. This type of test is referred to as cold injection.

Thermal Model

In the mixing region, the transient pipe wall response was found to be first-order in nature, being described by a time constant τ. For a step change in fluid temperature, the pipe wall temperature varied exponentially with time t as

$$[(T - T_0)/(T_1 - T_0)] = \exp[-(t/\tau)] \tag{1}$$

In this case, T is the local pipe wall temperature. Note that a subscript 0 implies a feedwater line mixed value, whereas subscripts 1 and 2 denote initial feedwater and injection line values, respectively. The dimensionless time constant is defined by

$$\tau^* = [(V\cdot\tau)/X][(T_0 - T_1)/(T_2 - T_1)] \tag{2}$$

where V is the mean fluid velocity, and X is the axial distance from the injection point. The transient mixing in the mixing region depends on the turbulence level at that point. Because this is a transverse injection problem, the turbulence level depends on both the main feedwater flow and injection water flow velocities. The Reynolds number ratio Re^* is defined as the ratio of the Reynolds numbers of the main flow to that of the injection flow

$$Re^* = \frac{\rho_1 V_1 D_1/\mu_1}{\rho_2 V_2 D_2/\mu_2} \tag{3}$$

Received June 19, 1992; presented as Paper 92-2926 at the AIAA 27th Thermophysics Conference, Nashville, TN, July 6–8, 1992; revision received Dec. 15, 1992; accepted for publication Dec. 15, 1992.

*Research Engineer, Center for Electric Power. Member AIAA.
†Research Assistant, Center for Electric Power.
‡Professor, Mechanical Engineering. Associate Fellow AIAA.
§Assistant Professor, Mechanical Engineering. Member AIAA.

where μ is the fluid viscosity, and D is the pipe diameter. The mixing is dependent on the Reynolds numbers of both flows, therefore, in this study the functional relationship is proposed as follows:

$$\tau^* = C(Re^*)^m Pr^n \tag{4}$$

where Pr is the Prandtl number.

Results

The transient temperatures climbed sharply to their final mixed mean temperatures at axial locations nearest to the injection point. Downstream from the injection point, the thermal response was slower, such that the temperatures appeared to change exponentially with time. Hence, the first-order response model appears to describe the data in the mixed region of the test section. This mixed region occurs after a development length of approximately three pipe diameters.

The Reynolds number ratio was calculated for all tests. A polynomial curve was used to relate Re^* and the average time constant in the mixing region $\bar{\tau}^*$. For hot injection tests at 93 and 66°C injection flow temperature, the resulting equation obtained by a least-squares curve fit is given by

$$\bar{\tau}^* = 0.528(Re^*)^{-0.3972}Pr^{0.457} \tag{5}$$

This relation is plotted in Fig. 1. For cold injection tests at 93 and 66°C injection flow temperature, the resulting curve fit equation is given by

$$\bar{\tau}^* = 0.130(Re^*)^{-0.592}Pr^{1.2811} \tag{6}$$

This curve is plotted in Fig. 2. In either type of test, the Prandtl number was evaluated at the final mixed temperature of the fluid T_0. The Reynolds number ratio was evaluated at the initial temperatures in the feedwater line and injection line, T_1 and T_2, respectively.

Conclusions

Significant circumferential and axial temperature gradients were observed in the mixing region for all flow rate combinations. Pipe wall temperatures opposite the point of hot water injection were higher than other circumferential locations prior to mixing. The time constants of pipe wall temperature transient response increased in the direction of flow, but varied inversely with the velocity of the mixed flow in the feedwater line.

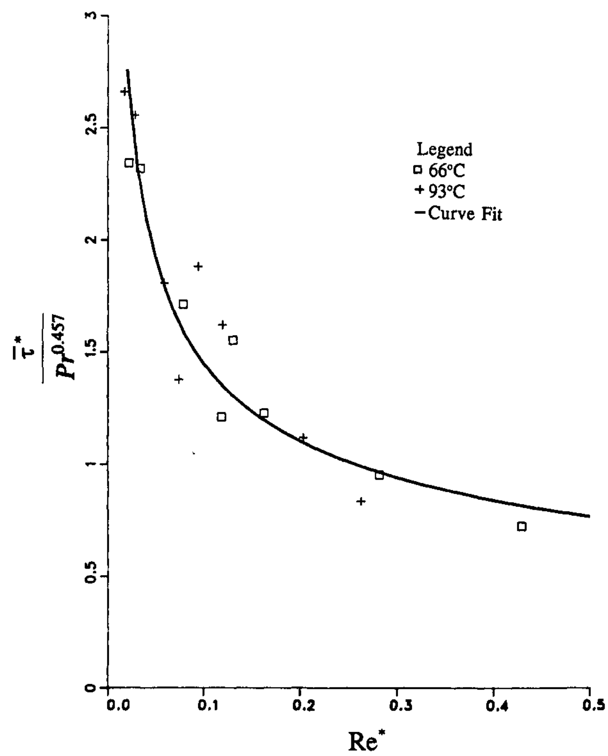

Fig. 1 $\bar{\tau}^*$ **vs** Re^* **for hot injection at 66 and 93°C.**

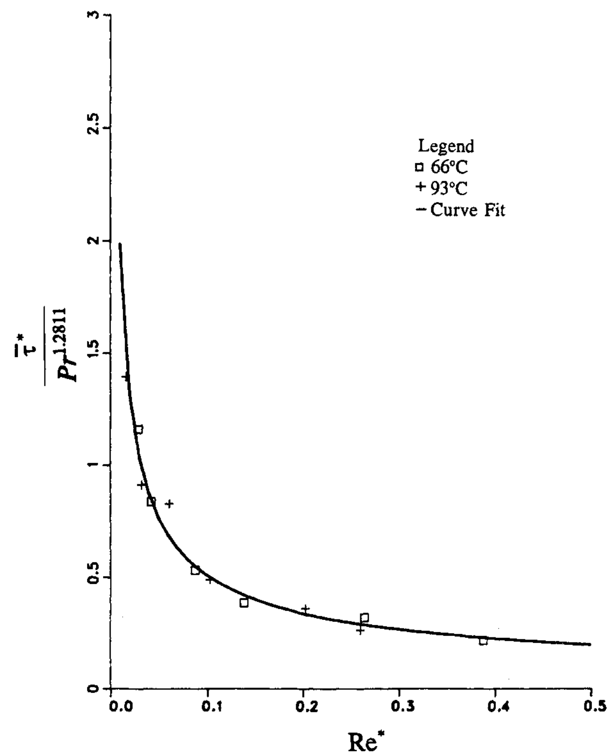

Fig. 2 $\bar{\tau}^*$ **vs** Re^* **for cold injection at 66 and 93°C.**

Based upon the final mixed state, the actual Reynolds numbers in the test section ranged from 6.5×10^4 to 3.4×10^5 for all water experiments. The high momentum of the combined flows precluded the possibility that buoyancy effects were significant in the initially stratified flow. This is supported by calculation of the Grashoff numbers in the mixing region.

It was found that the dimensionless time constants for sites on the pipe wall located more than three pipe diameters downstream from the injection point varied about a narrow band of values. Therefore, after three pipe diameters the mixing was primarily axial in nature. Appropriate scaling collapses the time constant data onto a single curve in the axial mixing region over the Reynolds number ratio range $0.01 < Re^* < 0.45$. A one-dimensional exponential response model does not describe the time constant data in the development region upstream of three pipe diameters, wherein circumferential fluid mixing is intense.

References

[1]Pryputniewicz, R. J., and Bowley, W. W., "An Experimental Study of Vertical Buoyant Jets Discharged into Water of Finite Depth," *Journal of Heat Transfer*, Vol. 97, No. 2, 1975, pp. 274–281.

[2]Salzman, R. N., and Schwartz, S. H., "Experimental Study of Solid-Gas Jet Issuing into a Transverse Stream," *Journal of Fluids Engineering*, Vol. 100, No. 3, 1978, pp. 333–338.

[3]Kamotani, Y., and Greber, I., "Experiments on a Turbulent Jet in a Cross Flow," *AIAA Journal*, Vol. 10, No. 11, 1972, pp. 1425–1429.

[4]Chassaing, P., George, J., Claria, A., and Sananes, F., "Physical Characteristics of Subsonic Jets in a Cross-Stream," *Journal of Fluid Mechanics*, Vol. 62, Pt. 1, 1974, pp. 41–64.

[5]Wu, J. M., Vakili, A. D., and Wu, F. M., "Investigation of the Interacting Flow of Nonsymmetric Jets in Crossflow," *AIAA Journal*, Vol. 26, No. 8, 1988, pp. 940–947.

[6]Oosthuizen, P. H., "Profile Measurements in Vertical Axisymmetric Buoyant Air Jets," 6th International Heat Transfer Conf., Vol. 1, Toronto, Canada, 1978, pp. 103–108.

[7]Sherif, S. A., and Pletcher, R. H., "Measurements of the Thermal Characteristics of Heated Turbulent Jets in Cross Flow," *Journal of Heat Transfer*, Vol. 111, No. 4, 1989, pp. 897–903.

[8]Andreopoulos, J., "Heat Transfer Measurements in a Heated Jet-Pipe Flow Issuing into a Cold Cross Stream," *Physics of Fluids*, Vol. 26, No. 11, 1983, pp. 3201–3210.

[9]Andreopoulos, J., "On the Structure of Jets in a Crossflow," *Journal of Fluid Mechanics*, Vol. 157, 1985, pp. 163–197.

[10]Sherif, S. A., and Pletcher, R. H., "Measurements of the Flow and Turbulence Characteristics of Round Jets in Crossflow," *Journal of Fluids Engineering*, Vol. 111, No. 2, 1989, pp. 165–171.

[11]Andreopoulos, J., "Measurements in a Pipe Flow Issuing Perpendicularly into a Cross Stream," *Journal of Fluids Engineering*, Vol. 104, No. 4, 1982, pp. 493–499.

[12]Andreopoulos, J., and Rodi, W., "Experimental Investigation of Jets in a Crossflow," *Journal of Fluid Mechanics*, Vol. 138, 1984, pp. 93–127.

[13]Krausche, D., and Fearn, R. L., "Round Jet in a Cross Flow: Influence of Injection Angle on Vortex Properties," *AIAA Journal*, Vol. 16, No. 6, 1978, pp. 636, 637.

[14]Moody, F. J., and Amos, B. T., "A Procedure for Predicting Temperature Loadings for Thermal Stress Calculations in Thick-Walled Pipes," ASME Special Publication PVP-91, 1984, pp. 45, 46.

[15]Konduri, R., Khodabakhsh-Sharifabad, F., Idem, S., Munukutla, S., Bosi, D., Kirkendall, S., and Shrivastava, H., "Transient Thermal Mixing in the Feedwater Line of a Nuclear Power Plant," International Power Generation Conf., Paper 91-JPGC-Pwr-16, San Diego, CA, 1991.

Thermal and Hydrodynamic Behavior in Flow Networks

Wen-Jei Yang* and Nengli Zhang†
University of Michigan, Ann Arbor, Michigan 48109
and
S. Umeda‡
Fukuyama University, Fukuyama, Japan

Nomenclature

D	=	hydraulic diameter of ducts, cm
H_1, H_2	=	water levels in upper tank and lower tank, respectively, cm
Nu	=	Nusselt number
q	=	heat flux, W/m^2
Re	=	Reynolds number
V	=	average velocity of the fluid in ducts, cm/s
V_i	=	local velocity of the fluid in ducts, cm/s
x	=	distance from inlet, cm
θ	=	intersection angle, deg

Subscripts

b	=	bottom and top
e	=	entrance
in	=	inner
L	=	left duct
o	=	outlet
out	=	out
R	=	right duct

Received Aug. 25, 1992; revision received Dec. 22, 1992; accepted for publication Dec. 22, 1992.

*Professor, Department of Mechanical Engineering and Applied Mechanics. Associate Fellow AIAA.

†Research Associate, Department of Mechanical Engineering and Applied Mechanics.

‡Professor, Department of Civil Engineering.

Introduction

A SYSTEM of mutually intersecting flow passages inside a plate or an assembly of plates, as shown in Fig. 1, is called a flow network. It has been discovered that a ramming of mutually intersecting flows results in a significant increase in convective heat transfer performance. Therefore, flow networks may serve as effective heat transfer devices with potential applications in industry, i.e., plate-type heat exchangers. Unfortunately, both theoretical and experimental investigations dealing with fluid flow and heat transfer in the networks are not available. Recently, a number of studies have been conducted to determine the pressure drop and heat transfer performance inside flow networks: Zhang et al.[1] conducted an experimental study on flow characteristics in flow networks with various geometries. In it, the intersecting pressure loss coefficient was defined and its magnitude determined as a function of geometric and flow conditions and a physical model was developed to predict the intersection pressure loss. The study concluded that a network with an intersection angle of 60 deg is the optimal geometry for the minimum intersecting pressure loss. Two additional investigations followed to aid in further understanding the flow behavior and heat transfer performance in the optimal flow network. The first used the hydrogen bubble method to observe flow patterns and mixing behavior in the intersection channel.[2] The second was a systematic measurement on the local heat transfer performance.[3] Both equal and unequal isoheat fluxes between the walls were treated. Recently, a hydraulic test setup of flow networks was constructed by Umeda et al.[4] for 1) a flow visualization study using both the dye injection and hydrogen bubble methods, 2) flow measurements by means of the laser Doppler velocimetry, and 3) pressure measurements using a piezometer.

This Note combines and compiles the results from the above-mentioned four investigations and explains in detail the mechanics of fluid flow and heat transfer in the flow networks.

Hydrodynamics in Flow Networks

Figure 1 is a schematic of a hydraulic recirculating flow system for testing flow inside two intersecting square ducts. The flow network was placed between two head tanks separated by a distance of 30 cm. The water level in the upper tank was maintained at a higher head of H_1 and that in the lower tank at a lower head of H_2. The difference $(H_1 - H_2)$ caused a flow through the two ducts with an intersection angle of q. A pump was employed to recirculate the flow. Each duct was square, 1.5 × 1.5 cm in cross section, and made of

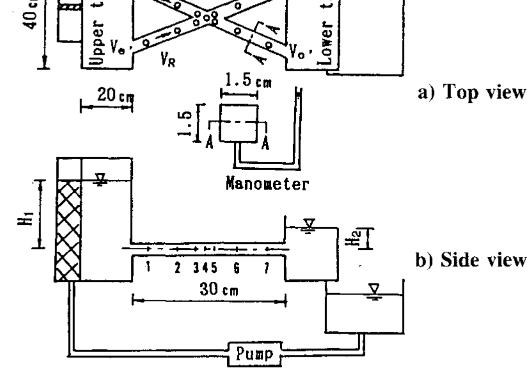

Fig. 1 Schematic of hydraulic recirculating flow system.