DIFFUSERS FOR HEATED WATER DISPOSAL FROM POWER PLANTS

Philip J. W. Roberts

AUTHOR: School of Civil and Environmental Engineering, Georgia Institute of Technology, Atlanta, GA 30332

REFERENCE: *Proceedings of the 2011 Georgia Water Resources Conference*, held April 11–13, 2011, at the University of Georgia.

Abstract. Power plant cooling is a major consumer of water. The two major methods of cooling that use water are once-through or closed-loop with cooling towers. Once-through extracts huge amounts of water but returns almost all of it with little consumed; cooling tower systems extract much less water but consume more. Both systems discharge heated water that can have environmental impacts. To meet receiving water temperature requirements, discharge is often through a diffuser.

The major types of thermal diffusers for the main water body types are briefly reviewed and examples are given of the design of two diffusers. The first is a once-through discharge into a shallow, tidal, estuary. It consists of an alternating diffuser with jets inclined upwards. Dilution is mainly a result of the momentum of the discharging jets. The second is for a power plant with a cooling tower discharging into a river. It is a unidirectional diffuser with multiple horizontal buoyant jets. Dilution is mainly due to the momentum and buoyancy of the discharge. Simulations with the mathematical model UM3 of Visual Plumes showed that the surface temperature regulations would be met.

A diffuser system is an effective means of disposing of heated water from power plants. They mix and dilute the heated water so that temperature rises are rapidly reduced with little environmental impacts.

To optimize water use and consumption, it may be possible to design plants to operate with once-through cooling and cooling towers at different times of the year.

INTRODUCTION

Steam-cycle power generation, which is how approximately 90% of non-hydro electrical power is produced, requires extensive cooling of the steam condensers. The rate at which this heat must be dissipated is huge, of the order of twice the rate of useful energy production (power) of the plant. The cooling can be either wet cooling, by water, or dry cooling. Wet cooling in the production of electrical power is one of the largest uses of water in the US and worldwide. Surface water was the source for more than 99% of total thermoelectric-power withdrawals. In Georgia, the power generation sources in 2008 were: 74% coal, 19% nuclear, 6% oil and gas, and 1% hydro. Nuclear plants require slightly more water than coal or oil-fired plants.

There are three main types of cooling systems used to dissipate the excess heat:

- Once-through. If the power plant is near a large water source such as a major river, lake, estuary, or coastal water, a large quantity of water can be passed through the condensers in a single pass and discharged back into the water body, warmer and without much loss from the amount withdrawn. The water can be salt or fresh. Some small amount of evaporation will occur off site due to the water being a few degrees warmer.
- Recirculating. Has two main ways:
  - Recirculating with cooling ponds.
  - Closed-loop with cooling towers. Water cools the condenser and is then passed through a cooling tower, where an updraught of air through water droplets cools the water. The cooling tower evaporates up to 5% of the flow and the cooled water is returned to the condenser. The evaporated water must be continually replaced. In addition, some “blowdown” water is removed to prevent the buildup of contaminants as the water evaporates. The heated blowdown water must be disposed of.
- Dry. A few power plants are cooled simply by air, without relying on evaporation. This may involve cooling towers with a closed circuit, or high forced draft air flow through a finned assembly.

Dry cooling uses little water, but is not commonly used in the U.S. Cooling ponds can require a very large land surface area and are also not common. Therefore, the two most common types of cooling are “wet:” Once-through, or closed-loop with a cooling tower.

The quantities of water extracted and consumed (not returned to the water body) differ considerably for the two methods. Once-through systems withdraw huge flows, but most is returned to the water body and the loss, due mainly to increased evaporation in the water body, is relatively small. If discharged into a river, the amount of heating depends on the supply of water available for dilution. Cooling tower systems extract much less water, but consume more. The cooling tower blowdown is returned to the river hotter than in once-through cooling, but the smaller flow can be more easily diluted and cooled.

Consider, for example, a 2,000 MWe power plant. Once-through cooling would require about 90 m^3/s (2,100 mgd) but the water consumed is relatively small, around 1

m^3/s (23 mgd). For the same example with a cooling tower system, withdrawal is about 4 m^3/s (90 mgd), but consumptive use doubles to about 2 m^3/s (45 mgd).

The water that is discharged by both methods is heated by about 14°C (25°F). The heated water can have deleterious effects on the water body that receives it, so the temperature rise they cause are usually limited by environmental regulations. They typically specify a maximum temperature rise, for example, $\Delta T < 3°C$ or 5°F, and a maximum absolute temperature, for example 30°C or 90°F. The temperature rise criterion is probably more critical in winter, but maximum temperatures are more likely a problem in summer.

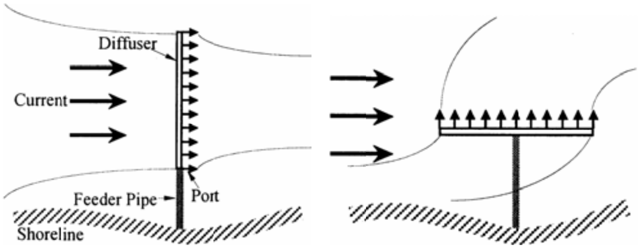
Other environmental impacts include the amount of water abstracted and the effects upon organisms in the aquatic environment, particularly fish and crustaceans. This latter includes both kills due to abstraction and the change in ecosystem conditions brought about by the increase in temperature.

The system for hot water disposal must minimize any environmental effects of the discharges. It can be returned to the environment either through a diffuser that results in rapid dilution and reduction in temperature rise, or a surface channel. The surface channel results in higher surface temperatures and more rapid dissipation of the excess heat to the atmosphere, but is probably prohibited by environmental temperature rise regulations. Therefore we here focus on diffuser systems which can dilute the heated water and reduce the temperature rise to allowable levels. We review the common diffuser types and give an example of designs for a plant into a tidal bay and for a proposed power plant.

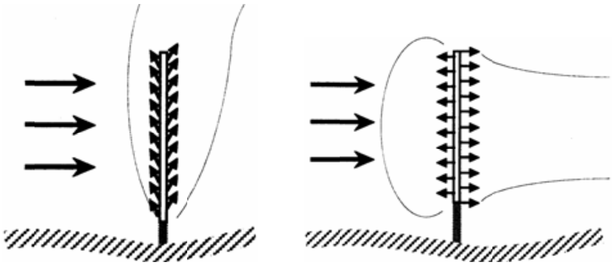
DIFFUSER DESIGNS

The design of thermal diffusers is discussed in several publications, for example Jirka (1982). Generally speaking, the discharge is from multiple round nozzles as high velocity jets that entrain ambient water and dilute and cool the hot water.

The main types of multiport thermal diffusers are shown in Figure 1. The diffuser design depends on the water body type. Types a and b are “unidirectional” diffusers in which the nozzles all point in one direction. The coflowing diffuser (Type a) with the nozzles pointing downstream is used if the flow is predominantly in one direction, such as in a river. The tee diffuser (Type b) is parallel to shore with the nozzles pointing offshore and is used in large water bodies with reversing currents, for example estuaries and open coasts. An alternative for that situation is the staged diffuser (Type c) which is perpendicular to the shore with the nozzles pointing offshore. Types b and c generate offshore momentum that prevents recirculation of the discharge back over the diffuser. Alternating diffusers (Type d) have nozzles discharging on both sides of the diffuser so there is no net horizontal momentum flux. This is less common for thermal diffusers, but may be applicable in situations where currents are weak, such as a lake.



a) Coflowing diffuser b) Tee diffuser



c) Staged diffuser d) Alternating diffuser

Figure 1. Basic types of multiport diffusers for thermal discharges (After Seo et al., 2001)

The flow and mixing induced by thermal discharges are complex. Hydrodynamic phenomena include the dynamics of turbulent jets, their modification by buoyancy and flowing currents, their interaction with the free surface and merging, restrictions on entrainment due to shallow water, collapse of turbulence due to gravitational effects, and spreading of the diluted effluent as a surface density current.

These effects and the consequent temperature rises are predicted by either mathematical or physical models. Physical modeling may be required due to the complexity of the flows and their three-dimensional nature. The mathematical models are usually either of the entrainment type, for example UM3 of Visual Plumes, or semi-empirical, such as CORMIX. More recently, computational fluid dynamics (CFD) models have begun to be used (Kuang and Lee, 2006; Kuang et al., 2006; Tang et al., 2008) but these are difficult to apply for routine engineering predictions due to the wide variation of length scales, complexity of the simulation grids, and very lengthy computation times.

DIFFUSER EXAMPLES

In the following we give examples of diffuser designs for two cases, a once-through discharge into a tidal estuary, and a cooling tower with blowdown discharge into a river.

Tidal bays. A diffuser was proposed for a power plant expansion that would take in cooling water from a shallow bay and discharge it back into the bay. The currents in the bay are strongly tidal and reverse back and forth over the diffuser. The temperature rise of the cooling water as it passes through the plant is 11°C (20°F), and the allowable temperature rise in the receiving waters is 2.2°C (4°F). The diffuser must therefore achieve an initial dilution of at least 5:1 in order to meet the allowable temperature rise.

A proposed design consisted of two outfalls, each terminating in a Y with a diffuser at the end of each leg. The diffusers were 25.9 m (85 ft) long with 18 ports oriented upwards at an angle of 45° (Figure 2). The port diameters are 254 mm (10 in) and the flow rate per diffuser is 5 m^3/s (114 mgd), resulting in a nozzle exit velocity of 5.5 m/s (18 ft/s). The receiving water is fairly shallow, ranging from 5.8 to 7.3 m (19 to 24 ft). The currents are mainly tidal up to around 1 m/s. They flow mostly parallel to shore so the diffusers are approximately perpendicular to the predominant currents. The ports are on opposite sides of the diffuser and are perpendicular to the diffuser axis so there is no net horizontal momentum flux, i.e. it is an alternating diffuser.

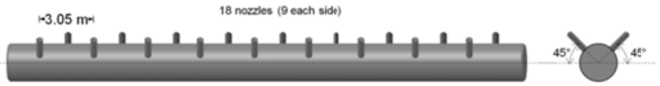


Figure 2. Proposed diffuser

Of particular interest was the low “worst-case” dilution that occurs at slack water, which is assumed to last for 5 to 10 minutes. Dilution estimates were made with the mathematical model UM3, which is a component of the U.S. EPA Visual PLUMES package (Frick et al., 2003; Frick, 2004).

Visual Plumes consists of a suite of models intended for various purposes. UM3 is a three-dimensional Lagrangian entrainment model for jets and plumes. The general characteristics of entrainment models are discussed in Fischer et al. (1979) and Roberts et al. (2010). External fluid is assumed to be entrained into the plume at a rate proportional to the local plume centerline velocity. The local profiles of velocity, density deficiency, and tracer concentrations are assumed to be self-similar, and the equations for conservation of mass, momentum, are integrated over the plume cross-section. For this reason, entrainment models are also sometimes called integral models. The equations are solved numerically to predict plume conditions, including dilution and plume width, along the jet trajectory. If the ports are close together, the plumes may merge. This merging is incorporated into UM3. Entrainment models are widely used in engineering to predict a wide variety of flows related to wastewater and atmospheric discharges. For further details, see Jirka (2004, 2006).

UM3 predicted that the proposed design would meet the regulatory surface temperature requirement. Questions were subsequently raised, however, about the stability of the discharge due to the high source momentum flux and shallow water, and whether an entrainment model like UM3 is applicable in an unstable flow where the supply of entrainment water may be restricted. Dilution estimates were then made with CORMIX, which predicted that the diffuser would not meet the regulatory temperature requirement.

In order to resolve these discrepancies, to provide insight into the mixing processes of the discharge, and to measure the expected dilutions, scale model experiments were performed in the Environmental Fluid Mechanics Laboratory at the Georgia Institute of Technology. The experiments were fully three-dimensional, i.e. the length of the diffuser was finite and less than the width of the test channel. A three-dimensional Laser-Induced Fluorescence system (3DLIF) was used to measure the mixing field and the resulting dilutions and temperature risers. Experiments were performed in stationary and flowing environments and are described in Roberts et al. (2011).

Flow visualization was first done by adding blue food coloring to the effluent; close-up and wide-angle side-view photographs are shown in Figure 3. The jets issuing from the nozzles are clearly visible. After impacting the water surface, the jets make a transition to horizontal flow but they are still turbulent, resulting in additional entrainment, mixing, and further dilution. This causes the layer thickness to increase with distance from the diffuser until it appears to extend over most of the water depth. The flow is “stable,” with no reentrainment or recirculation back into the rising jets

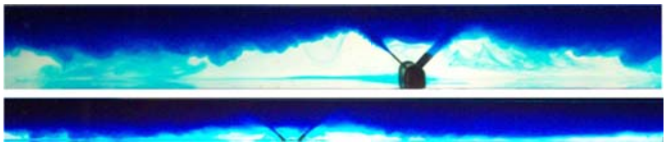
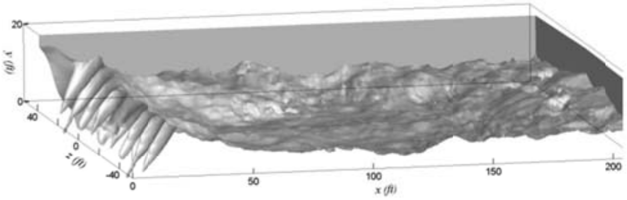


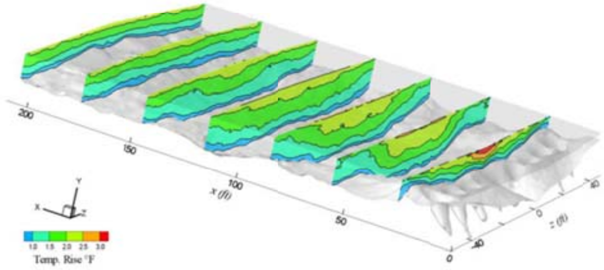
Figure 3. Photographs of diffuser plumes

The three-dimensional effluent field and mixing processes that were obtained by 3DLIF are shown in various ways in Figure 4. In Figures 4b and 4c, the water surface and the boundaries of the effluent field are shown as transparent gray surfaces, which make the temperature rises, that are shown as false colors in various planes, visible.

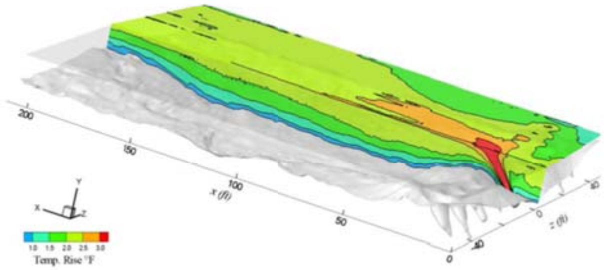
Figure 4a is a perspective view looking upwards of the topography of the bottom of the spreading layer and the jets. This adds three-dimensional information to the photographs of Figure 3. The jets are visible at the left side and after impacting the bottom the flow first thickens due to turbulent entrainment then spreads laterally due to gravitational spreading. The layer is thicker near the centerline and thinner towards the edges.



a) Bottom view of spreading layer topography



b) Lateral temperature profiles



c) Longitudinal profile along centerline and surface temperatures

Figure 4. 3DLIF images of thermal discharge

The dye tracer concentrations converted to temperature rises are shown in Figures 4b and 4c. Figure 4b shows lateral profiles in vertical planes that are parallel to the diffuser axis at varying downstream distances. These show more clearly that the spreading layer is initially thicker at the centerline and thinner towards the edges. The layer extends over most of the water depth near the centerline between about 21 and 46 m (70 and 150 ft) from the diffuser. But it becomes two-layered farther downstream and thins due to buoyant surface spreading. The temperatures are higher on the centerline, and decrease towards the edges. Again, this is a consequence of the three-dimensional nature of the mixing process caused by the finite diffuser length. This is particularly true closer to the diffuser; farther downstream, lateral spreading and mixing causes the temperature rise to decrease and to become more laterally uniform, until the isotherms become essentially horizontal.

These features are also evident in Figure 4c, which shows temperature variations in the vertical plane parallel to the current direction through the diffuser centerline and also on the surface. The vertical plane passes through the center nozzle, and its jet is clearly visible as the red high temperature region. Lowest surface dilutions occur where the jet centerlines impact the water surface. The surface dilution is lowest on the diffuser centerline, and increases towards the layer edges. The surface dilution also increases downstream from the jet impaction points due to further mixing and entrainment in the turbulent spreading layer.

The lowest, or minimum, dilution (and therefore highest temperature rise), occurs where the jet centerline impacts the water surface. The dilution at this point is 6:1. Therefore, the maximum surface temperature rise is expected to be about 11/6 ≈ 1.8°C (3.3°F), although this occurs only over a very small surface area (see Figure 4c). The proposed diffuser should therefore meet the requirement of maximum temperature rise of 2.2°C.

Dilution increases beyond the impact point, reaching a value of about 9.5:1 at $x/H \approx 10$. This is the near field dilution. The increase in dilution is due to the fact that the flow is still initially turbulent. It is stably stratified, however, which causes the turbulence to ultimately collapse. This marks the end of the near field (in the hydrodynamic sense) and mixing and dilution is considerably suppressed beyond this point.

The dilutions in flowing currents were even higher, but the regulations were based on worst-case, zero currents speeds.

River diffuser. The second example is a river diffuser for the blowdown flow from a proposed power plant with mechanical cooling towers. The flow is much lower than the previous example, but the temperature rise higher. The discharge is into a run-of-the-river reservoir.

The blowdown flow ranges from 9 to 64 cfs depending on the plant operation mode. This is much smaller than the average annual flow in the river, which is 2,500 cfs but can be more than 60,000 cfs. Measurements of velocities in the river were made by means by a downward-looking Acoustic Doppler Current Profiler (ADCP) attached to a boat. The flow rate in the river when the measurements were conducted was about 691 cfs. The measured velocity vectors are shown in Figure 5.

The discharge regulations state that the temperature rise above natural ambient conditions in the river must be less than 5°F, and the maximum river temperature must be less than 90°F at all times.

Temperatures were measured at hourly intervals in the river. The time series of temperature and estimated monthly blowdown temperatures are shown in Figure 6. The river and blowdown temperatures vary considerably through the year, and by several degrees over a few days. The river temperatures were averaged by month to estimate the mean difference between the river and blowdown. The maximum anticipated blowdown temperature is 91.0°F.

Figure 5. ADCP current measurements

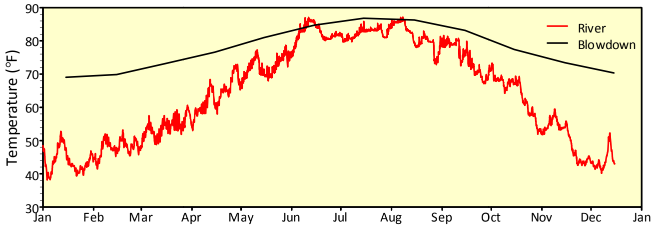


Figure 6. Blowdown and river temperatures (2008)

The highest river temperature occurs in summer, in June and July, and is about 82.3°F, but the largest temperature difference occurs in winter, in December, when it is about 26.3°F. Therefore, the critical time for the absolute temperature difference to be exceeded is winter, but the likeliest time for the absolute temperature to be exceeded is summer.

The maximum temperature difference between the blowdown and river is 26.3°F, so to achieve a temperature rise ΔT in the river of less than 5°F requires a dilution greater than 26.3/5 = 5.3:1. This can readily be accomplished by discharging the blowdown as jets from a multiport diffuser. This is a standard method of discharging wastewaters and power plant cooling waters because it can result in rapid and efficient mixing and dilution (Fischer et al., 1979). Because the blowdown is heated, there is a density difference between it and the river water that creates a buoyancy force on the jets, which are therefore termed buoyant jets. The general characteristics of a horizontal buoyant jet in a stationary unstratified water body are shown in Figure 7.

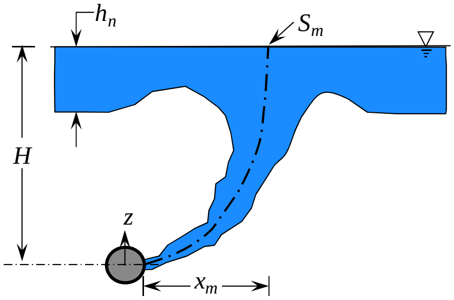


Figure 7. Definition diagram for horizontal buoyant jets

The general flow characteristics are as follows. The jet follows a curved trajectory to the water surface. As it rises, it entrains ambient water that mixes with and dilutes the discharge and reduces the temperature rise. The maximum surface temperature occurs where the jet centerline impacts the water surface. This corresponds to the minimum surface dilution, denoted by S_m in Figure 7. It occurs at a horizontal distance x_m from the nozzle. After impacting the water surface the diluted effluent spreads horizontally as a density current with a thickness h_n. Some additional mixing and dilution occur in the transition to horizontal flow at the water surface, so the surface dilution is everywhere actually greater than S_m. The dilution and impact distance depend primarily on the source momentum and buoyancy fluxes and the water depth. The fluxes depend on jet exit velocity, the nozzle diameter, and the density difference between the effluent and receiving water. As the waters here are essentially fresh, their densities depend only on temperature.

To compute the dilution, and therefore the temperature rise at the water surface, UM3 was again used. It was run for various combinations of flows, port spacings, port diameters, and port depths. The flows were 9, 18, and 64 cfs, port spacings of 10, 2, and 1 ft, port diameters 3 and 4 inches, port depths 6 and 8 ft, and the maximum (worst case) temperature difference in winter when the blowdown temperature is 70.4° and ΔT = 26.3°F.

It was found that the dilution requirements would be met by a diffuser consisting of 64 four inch diameter ports spaced 1.4 ft apart with their centerlines submerged six ft below the normal pool level water surface. The ports are assumed to discharge horizontally, which is a standard configuration that maximizes dilution. For this design, simulations were then run to estimate the maximum temperature rise that could occur in summer. It was predicted that the maximum surface temperature would be 84.0°F and would never exceed 90°F.

The diffuser is fixed to the face of the dam so its axis is parallel to the predominant currents (Figure 5). However, zero current speed was assumed in the simulations. This is a conservative assumption, as the current will always increase dilution (and therefore reduce temperature rise). The diffuser is unidirectional, with the nozzles all pointing in the same direction. The heated water will form a surface layer a few feet thick that will not be directly entrained into the hydroelectric plant intakes.

CONCLUSIONS

Water for cooling plants used for the generation of power is a major consumer of water in the US. The two major methods of cooling using water are once-through where large quantities of water are extracted from a water body or closed-loop with cooling towers. The amount of water extracted and consumed is very different for the two types. Once-through extracts huge amounts of water but returns almost all of it, with little consumed; cooling tower systems extract much less water but consume more.

Both systems discharge heated water which can have deleterious effects on the environment. So the temperature rise and absolute temperature in the receiving water body are usually limited by environmental regulations. To meet these temperature requirements, discharge is often from a multiport diffuser as high velocity jets that entrain ambient water and mix with it, resulting in rapid and efficient cooling of the heated water.

The major types of thermal diffusers are briefly reviewed along with the mathematical models often used to predict dilution and to design the diffusers. The diffuser type depends on the receiving water body type: river, lake, estuary, or coastal water.

Examples are given of the design of two diffusers. The first is an alternating diffuser for once-through discharge into a shallow, tidal, estuary. The second is for blowdown from a cooling tower system for a power plant discharging into a river.

The bay outfall consists of an alternating diffuser with jets inclined upwards. Dilution is mainly a result of the momentum of the discharging jets. Hydraulic model tests are described that illustrate the complexity of the hydrodynamic processes involved. Despite this complexity, however, the maximum surface temperature rise can be predicted by simple jet models.

The blowdown diffuser consists of multiple horizontal jets in a unidirectional configuration. Dilution was predicted by the US EPA mathematical model UM3 of Visual Plumes. The simulations showed that the surface temperature regulations would be met. The diffuser is fixed to the face of the dam and so is parallel to the prevailing river flow. The effect of the currents on dilution was neglected however, and so dilutions would be expected to be actually higher than predicted.

Diffuser systems are effective means of disposing of heated water from power plants. They can effectively mix and dilute the heated water so that temperature rises are rapidly reduced with little environmental impacts.

To optimize water use and consumption, it may be possible to design plants to operate with once-through cooling in winter and cooling towers in summer.

REFERENCES

Jirka, G. (1982). "Multiport Diffusers for Heat Disposal." *Journal of the Hydraulics Division, ASCE*, 108(HY12).

Jirka, G. (2004). "Integral Model for Turbulent Buoyant Jets in Unbounded Stratified Flows. Part I: Single Round Jet." *Environmental Fluid Mechanics*, 4(1), 1-56.

Jirka, G. (2006). "Integral Model for Turbulent Buoyant Jets in Unbounded Stratified Flows. Part II: Plane Jet Dynamics Resulting from Multiport Diffuser Jets." *Environmental Fluid Mechanics*, 6, 43-100

Fischer, H. B., List, E. J., Koh, R. C. Y., Imberger, J., and Brooks, N. H. (1979). *Mixing in Inland and Coastal Waters.* Academic Press, New York.

Frick, W. E., Roberts, P. J. W., Davis, L. R., Keyes, J., Baumgartner, D. J., and George, K. P. (2003). "Dilution Models for Effluent Discharges, 4th Edition (Visual Plumes)." U.S. Environmental Protection Agency, Environmental Research Division, NERL, Standards and Applied Science Division, Office of Science and Technology, Athens, Georgia.

Frick, W. E. (2004). "Visual Plumes mixing zone modeling software." *Environmental Modelling & Software*, 19, 645-654.

Kuang, C., and Lee, J. (2006). "Stability and Mixing of a Vertical Axisymmetric Buoyant Jet in Shallow Water." *Environmental Fluid Mechanics*, 6(2), 153-180.

Kuang, C. P., Lee, J. H. W., Liu, S. G., and Gu, J. (2006). "Numerical study on plume interaction above an alternating diffuser in stagnant water." *China Ocean Engrg.*, 20(2), 289-302.

Roberts, P. J. W., Salas, H. J., Reiff, F. M., Libhaber, M., Labbe, A., and Thomson, J. C. (2010). *Marine Wastewater Outfalls and Treatment Systems*, Published by the International Water Association, London.

Roberts, P. J. W., Tian, X., and Jung, Y. (2011). "Physical Model Study of an Alternating Diffuser for Thermal Discharge." *J. Hydraul. Eng.*, in Press.

Seo, I. W., Kim, H. S., Yu, D., and Kim, D. S. (2001). "Performance of Tee diffusers in shallow water with crossflow." *J. Hydraul. Eng.*, 127(1), 53-61.

Tang, H. S., Paik, J., Sotiropoulos, F., and Khangaonkar, T. (2008). "Three-Dimensional Numerical Modeling of Initial Mixing of Thermal Discharges at Real-Life Configurations." *J. Hydraul. Eng.*, 134(9), 1210-1224.