THERMAL POLLUTION IN WATER

Lorin R. Davis
Professor Emeritus, Oregon State University, Corvallis, Oregon USA

Keywords: Thermal pollution, Plumes, Evaporation, Surface heat transfer, Water systems, Jets, Diffusers, Mixing zone

Contents

1. Introduction
2. Second Law of Thermodynamics and Thermal Pollution
3. Modeling Methods
4. Empirical Models
5. Numerical Models
6. Integral Models
    6.1. Eulerian Models
    6.1.1. Submerged Plumes
    6.1.2. Surface Discharge
    6.2. Lagrangian Models
7. Visual Plumes Model (VP)
8. Heat Loss Calculations
9. Restrictions and Accuracy
Glossary
Bibliography
Biographical Sketch

Summary

Thermal pollution occurs as a natural process required by the Second Law of Thermodynamics for any thermodynamic cycle to operate. Water is often used as a cooling medium, which is heated and returned to the environment warmer than when it started. The problem is to reduce the impact this has on the environment and to reduce the total amount of heat rejected. The amount of head rejected to water bodies can be reduced by conservation, using alternated disposal methods such and cooling towers, and more efficient plants and machines. The impact of heat rejection can be reduced by limiting the volume exposed to elevated temperatures or limiting it to regions that do not cause environmental harm. The volume of water exposed to elevated temperatures can be reduced through the proper design of submerged discharge diffusers. Discharge at the surface produces a large surface area exposed to elevated temperatures but increases the rate of heat transfer to the atmosphere and keeps high temperatures away from some deeper dwelling organisms that might otherwise be harmed. The ability to predict regions of impact is important in determining if a particular discharge is acceptable or not. Several methods of predicting thermal plume dispersion are discussed including physical, empirical, integral, and numerical. Selected computer models using these methods available for public use are presented.

1. Introduction

The injection of elevated temperature water into the environment is a controversial issue. In some cases, it is said that heat addition causes no harm and even improves conditions, whereas in others, the whole ecosystem is changed. Some fishermen say the best fishing is near a thermal outfall while others claim the hot water is killing the fry and larva. Whatever the stand, it is generally agreed that some species prefer warmer water while others prefer colder water. It is also known that some species, especially juveniles, can only stand elevated temperatures for a given period of time, the higher the temperature, the shorter of the lethal exposure time. In general, discharging warm water into a cooler body of water will result in the change of bio life in the neighborhood of the discharge from cold water species to warm water species. The size of the affected region can be from a few feet to several thousand feet from the discharge.

Since every thermal cycle must reject some heat to satisfy the 2nd Law of Thermodynamics, the human race would have to go back to before fire used to heat caves to live without some human impact on the environment due to thermal discharge. Only the extremists wish to go back that for. As a result, the problem becomes how to reduce the amount of heat rejected to the environment and the impact it has on it.

One of the biggest sources of thermal pollution in water comes from electric power plants where water passes through the condenser and returned to the environment at an elevated temperature. The amount of warm water discharged from power plants can be reduced through conservation of electrical use, increased plant efficiency, using cooling towers to cool the water and return it to the condenser rather than discharge it to the environment, and more use of alternate power sources. Evaporative cooling towers produce large plume clouds which are not desirable in some areas, require large quantities of makeup water and are difficult to use with salt water. Air cooled cooling towers are very large, expensive and reduce the overall efficiency of the plant. Alternate energy sources such as wind, solar, hydro, etc. are expected to become much more prevalent, but will probably never meet the demands for power in today’s world. Engineers have been trying to improve plant efficiencies for years for economic reasons and have felt lucky to increase efficiency by only a fraction of a percent. As a result, a considerable amount of warm water will be discharged to the environment for some time to come.

Warm water is discharged through either a submerged diffuser with one or more discharge ports or at the surface through a simple discharge channel. Port velocities from submerged diffusers are usually in the range of 3 m/s. Near the discharge port, these high velocities induce rapid entrainment of ambient water which quickly reduces the temperature of the plume. A multiple port diffuser with several small discharge ports spaced far enough apart of prevent interference dilutes the plume much faster than a larger single port discharge. Discharge at the surface causes very little mixing with the ambient water resulting in a large surface layer of warm water. This gives a maximum area impacted by elevated temperatures but the plume is at the surface where it can lose energy to the atmosphere and does not impact deeper bio-life. As a result, both types of discharge are used, depending on the location and requirements.

The ability to predict the extent of elevated temperatures for a particular discharge is necessary in order to determine if a particular discharge method satisfies regulations or not. This chapter addresses limitations imposed by the 2nd law of thermodynamics, different methods of calculating thermal plume dispersion for both submerged and surface discharge, and methods of calculating surface heat transfer.

2. Second Law of Thermodynamics

Various statements of the 2nd law of thermodynamics exist but the one that applies to the present problem is known as the Kelvin-Plank statement. One version of this law is: *No heat engine exists that can operate with only one heat reservoir.* In basic terms, this says no machine (power plant) can be 100% efficient. They will always reject some heat to the environment. The Clausius statement of the 2nd law implies that heat transfer by itself will always be from a high temperature to a lower temperature. This means that the waste heat rejected from the power plant will always be at a higher temperature than the environment. A question might be asked: Why not use a refrigerator to cool the waste water down. The problem with that is that a refrigerator is a heat engine operating in reverse. It absorbs energy at a low temperature and discharges this energy plus the energy required to run the refrigerator at a high temperature. So it discharges more energy at a high temperature than it absorbs in the cooling process.

Present thermal power plants have an efficiency of about 33%. Newer plants are a little better. This means that a 1000 MW (electrical) power plant discharges 2000 MW to the environment. If the cooling water passing through the condensers increases 20 C before being discharged back into the environment, the 1000 MW plant would need 50 m^3/s of cooling water. Assuming a flow velocity of 4 m/s in the cooling water pipes, the discharge pipe would have to be 5 m diameter. That is a lot of water. However, a discharge diffuser with 100 35 cm discharge ports placed at a depth of 30 m in open water would dilute the warm water to just 1.0 C above the ambient within 15 m of the discharge ports. As a result, the potential of impact from such a power plant is great, but the use of a properly designed discharge diffuser can limit the size of the effected region.

There are many other sources of thermal discharges into ambient waters but the same rules apply to them as with power plants.

3. Modeling Methods

Various methods exist to determine the size of a temperature field from a thermal discharge. Because of the difficulties in simulating actual turbulent jets into real environments, all of these methods are approximate. No exact method presently exists. They do, however, give answers that approach actual values and can be used to predict temperatures with reasonable accuracy.

Each method has its advantages and disadvantages. The problem is to know the limits of each method and when to use which one.

Physical modeling: Physical modeling is where a scale model of the receiving water and discharge system are created to make measurements that can be used to estimate the response in the actual discharge. Dynamic and geometric similarity laws are used to determine the size, shape, and operating conditions of the model. They are then used to predict the temperatures in the actual discharge from measured values in the model. Dynamic similarity requires that various dimensionless parameters such as Reynolds number, Froude number, Densimetric Froude number, and Nusselt number be the same for both the physical model and full sized discharge and environment. Geometric similarity requires that all dimensions in the model be the same as the values in the full sized project multiplied by a fixed scale ratio.

Because various lengths are contained within the dynamic similarity parameters, it is often impossible to have all dimensionless parameters the same in both the model the actual discharge and have the model smaller than the actual discharge. As a result, it is common practice to ignore some of the minor parameters. For example, with submerged, buoyant discharges, similarity is often based on the Densimetric Froude number. The Reynolds number is ignored as long as it is large enough to insure a turbulent jet. It is then assumed that as long as the densimetric Froude number is the same in the model as in the actual discharge, temperature ratios determined in the model will be the same as in the geometrically similar full scale discharge. This assumes that ignoring Reynolds number and other minor parameters will have no effect on the relationship between values in the model and the actual discharge. Unfortunately, turbulence is not the same in models as in full sized discharges and occasionally predicted values are in error.

In large shallow lakes, rivers and estuaries, geometric similarity results in very shallow depths in the model. These shallow depths may introduce effects such as surface tension that are not present in the full scale environment. This results in very large models or the use of a skewed model with different horizontal and vertical length scales. This requires special use of similarity relations.

The advantage of physical models is the ability to include time dependency and the complete geometry of the environment and discharge. This is very important when transients or geometry must be accounted for. The disadvantages are the high cost and the inability to look at variations with ease. It often takes weeks or months to set up a single new case. Many hydraulic laboratories and universities around the world have facilities and personnel to do physical modeling.

Empirical modeling: Empirical modeling is where laboratory or field data are used to generate algebraic equations that can be used to predict plume development on similar discharges. As in physical modeling, the major variables in the problem are arranged into dimensionless groups or into length scale parameters. These parameters are plotted and regression analyses are used to determine equations the best describe the results. These equations are then used to predict the response of any discharge that falls into the category as the measured data.

Often it takes several equations to cover all ranges. Figure 1 shows how this might work. L1, L2, L3, and L4 are length scales that contain discharge and ambient conditions. Dilution, Q/Q_o, is the ratio of flow in the plume to discharge flow. Equation (1) is a curve fit to the data in region 1. Equation (2) is a different curve fit to data in region 2. Unfortunately, these equations are only good within the region were the original data were taken and a large number of equations would need to be developed to cover all types of discharges and environments. Extrapolating beyond the limits of the original data can lead to large errors. In addition, it is often difficult to have equations continuous from one region to the next. Figure 1 shows how two equations might have a discontinuity at the boundary of the two regions where they apply.

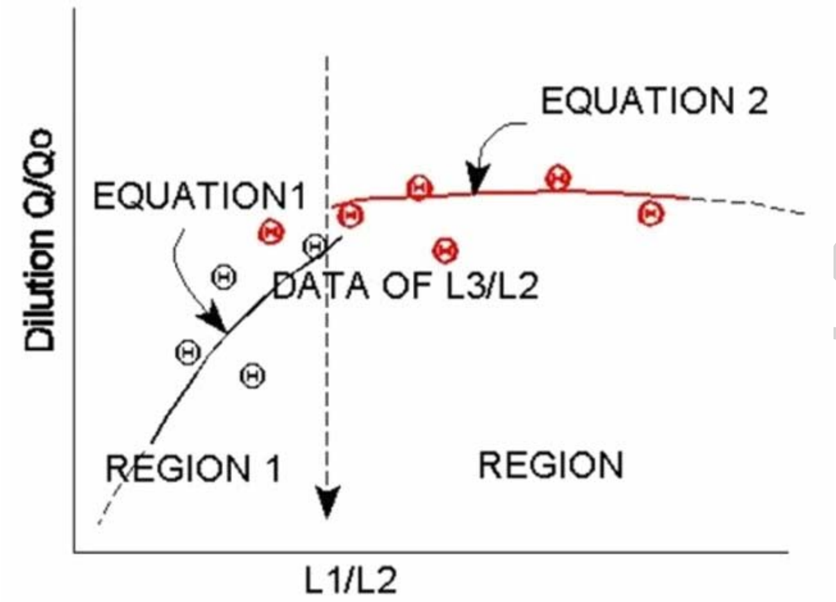

Figure 1: Sample graph showing dimensionless groups and data used to generate empirical equations.

The advantage of empirical methods is the ability to include the effects of complex geometries in the equations and that the equations can easily be evaluated. Once the equations are known, they can often be evaluated with a hand calculator. Usually they are coupled to a more complex interface that automatically selects the proper equations. The disadvantages are that an infinite number of equations would need to be developed to cover all possible types of discharges and environments and have continuous predictions when passing from one region to the next. This would require measurements in every possible environment and discharge configuration. Since this is not possible, a finite number of classes are usually developed to cover a range of variables. Discontinuities can exist when going from one class to the next. (See Section 4)

Numerical methods: Numerical methods require dividing the area of interest into a large number of finite volumes or elements. Fluid properties such as velocity, density, pollutant concentration, and temperature vary from one element to the next but are assumed constant within the element. There would be a small discontinuity in fluid properties when going from one element to the next, the smaller the element, the smaller the change. As a result, the region of interest could be made up of millions of small elements. As computer memory and computational speed has increased, the number of elements that a person could analyze has increased. Large elements lead to large computational errors but shorter computer run time.

The boundaries in a numerical methods problem are usually simulated to represent the actual boundaries in the problem. Three dimensional solid boundaries require special representation of the elements in contact with the boundary. Usually some simplifying assumptions are made to make the problem easier to set up. Fluid boundaries are taken far enough away that the discharge of concern will not affect them. At these boundaries, fluid properties can either be fixed at a known value or made variable with time with some known variation

.

The flow field must first be divided into the appropriate grid system. There are numerous programs available whose sole purpose is to generate grid systems for complicated problems. The differential equations of fluid motion are discretized and solved numerically for this system of elements with a computer. Various methods of discretization and solution are used. The most critical values are how the boundaries are simulated and the turbulence model used. Since turbulence is not an exact science, only simplified functions presently exist. One of the more popular functions is the k-epsilon $(k-\in)$ model. Computer run time for each case can vary from a few minutes to a few weeks depending on the type of problem and size of the computer used to make the computations. Output includes fluid properties at each element for a steady state problem or for each element at a finite number of time steps for a transient problem. These can be used to plot velocity, temperature, and concentration profiles in the flow field at any viewing plane. For transient problems, several plots can be generated for each time step and shown in animation.

The advantages of numerical methods are the ability to represent actual geometry and time dependency without having to go to the laboratory and set up a physical model. Disadvantages include the time required to set up and run a problem having a complicated boundary and the inability to accurately represent turbulence numerically. (See Numerical Models)

Integral methods: In integral methods, plume properties are integrated across the face of the plume, perpendicular to the plume centerline using an appropriate approximation to the actual profile. The three most popular profiles are the “top hat” profile which assumes plume properties are constant across the face of the plume, a statistical bell shaped profile based on a Gaussian curve with maximum values at the center, and a 3/2 power law profile that approximates the Gaussian curve. Variations from actual profiles are lost in the integration since the result is a function of only the plume centerline values and plume size. Since these profiles deal with values in excess to ambient conditions, integration need only be made from the plume centerline to its edge. Actual geometric boundaries are not included and are only accounted for in a secondary manner. When these integrated profiles are incorporated into the differential equations of motion, as system of ordinary differential equations result that can be solved step-wise along the centerline of the plume.

In order to solve the system of equations described above, an entrainment function must be developed and used that simulates the fluid drawn into the plume by turbulent shear and exchange at the plume boundary. It must also include the fluid forced into the plume by ambient current. This entrainment function is the key to success in integral methods. They have been developed in the past using field and laboratory experiments.

The step-wise integration of the equations yields fluid properties including plume size and location at each integration step along the trajectory of the plume. For Gaussian and 3/2 power law profiles, centerline values of temperature, concentration, and velocity calculated. From the assumed profile and plume size, off centerline values can be determined. For "top hat" profiles, average values are calculated. These can be used with an assumed profile and plume size to determine centerline values.

The advantage of integral models is that they can be evaluated in just a few seconds with present micro computers. They also give a good representation of the plume in the near field where physical boundaries are not important. As a result, a large number of cases can be analyzed within just a few minutes and get good answers in the region where temperature and concentration are the highest. The disadvantage is that they cannot be used where physical boundaries play a significant role in the dispersion process. In addition, transient plumes can only be analyzed as quasi-steady state problems with a series of steady state runs with different ambient conditions are used to simulate transient behavior. (See Surface Plumes and Submerged Plumes)

4. Empirical Models

The most widely used empirical model is the CORMIX model. It comes in several versions including CORMIX1-2-3, and CORMIX-GI. They are basically both the same except the latest version, GI, has a convenient WINDOWS based interface and graphic output. The 1-2-3 DOS based version uses interactive screen prompts to determine input values. The GI version uses pop-up windows and dialog boxes for data entry. Although it is listed under "Empirical Models", CORMIX also has a multiple port integral method model called CORJET that is used on simpler configurations.

CORMIX is a software system that incorporates an expert system interface with a number of hydraulic models. CORMIX is designed to handle a wide variety of discharge configurations and ambiences including single and multiple port diffusers, submerged and surface discharge, positive and negative buoyant discharges, diffusers with unidirectional, fanned, alternating ports or risers with multiple ports, open oceans, lakes, rivers, and estuaries. It can also consider tidal varying ambiences in a quasi-transient approach. CORMIX is inherently steady state, but it will consider the rate of change of ambient conditions and account for re-entrainment due to tidal reversal.

Once all the data entered into the dialog boxes, CORMIX determines a number of length scales. (See Empirical Methods). These are used to determine the flow class. Flow class is the system used by CORMIX to specify which type of plume is expected to develop as a result of the discharge configuration and ambient. For example: is the flow jet like or plume like, is stratification sufficient to dominate, is there a weak current or strong current, is the receiving water deep or shallow, is the alignment parallel or perpendicular, is discharge positively buoyant or negatively buoyant, is there coanda bottom attachment or wake attachment, is there an upstream wedge, etc. There is something like sixty six flow classes for submerged discharge along. CORMIX then runs the hydraulic models it has determined are appropriate for this flow class in sequence and patches them together.

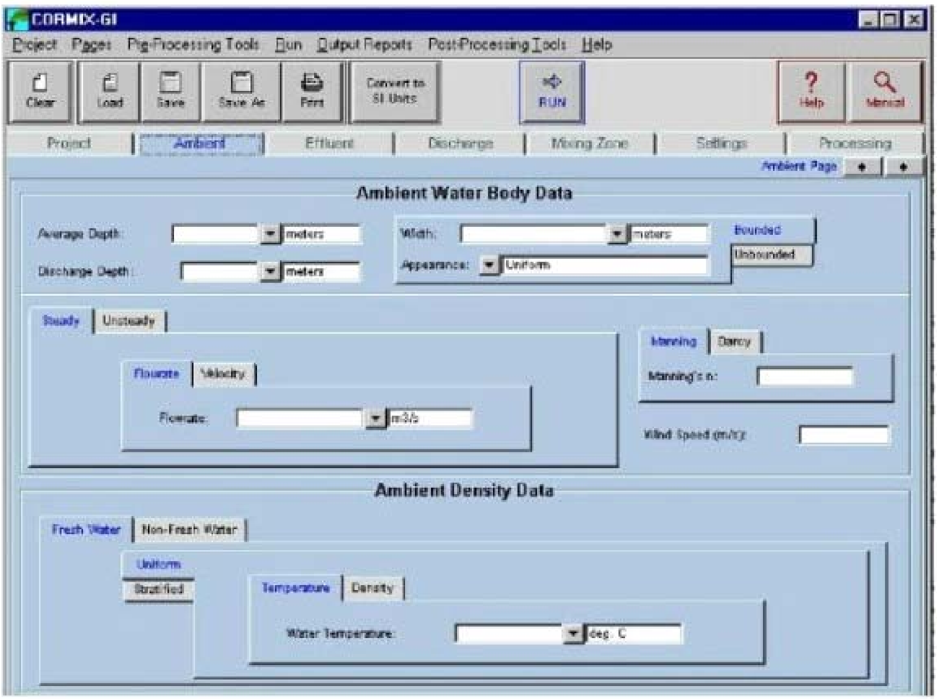

Figure 2: Sample input window in CORMIX-GI

Figure 2 shows an input window for the GI version with its several input tabs. The various windows created by each tab provide information on the Project, Ambient, Effluent, Discharge, Mixing zone regulations, Output settings, and a tab were you can validate the input and run the actual hydraulic models. Output includes both tabular and graphic forms. Figure 3 is a sample of one of the graphic output screens showing temperature as a function of distance for a sample case. Since the pollutant was selected as temperature, concentration is in degrees C above the ambient.

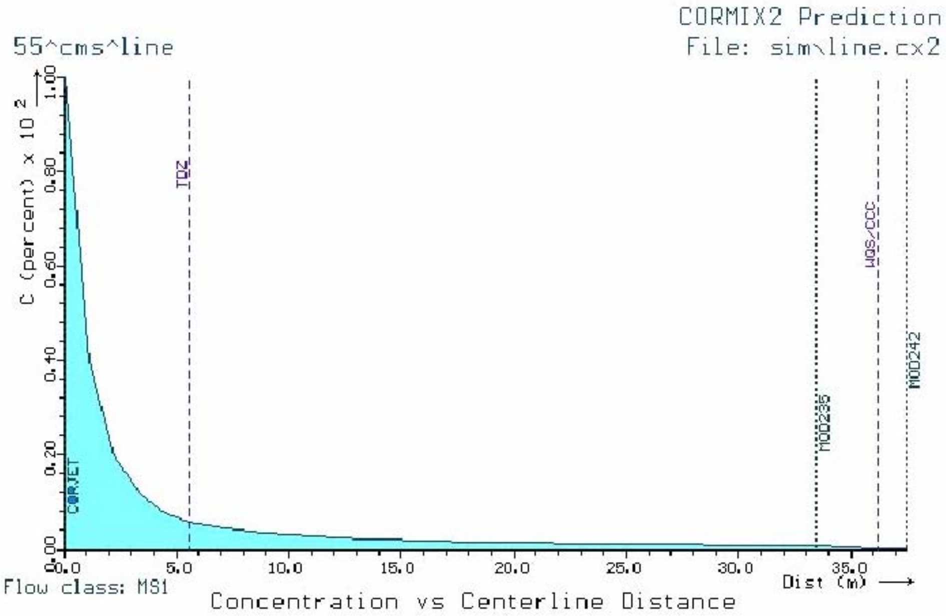

Figure 3: A sample plot developed by CORMIX for excess temperature (called

concentration on plot) versus distance downstream.

-
-
-

Bibliography

Davis L.R. (1999). *Fundamentals of Environmental Discharge Modeling,* Boca Raton, CRC Press [This book is good reference on all aspects of plume modeling]

Hinze, J.O., (1959) *Turbulence, an Introduction to Its Mechanism and Theory*, New York, McGraw-Hill [This is a good reference on fluid turbulence]

Hirst E.A. (1971). *Analysis of Round Turbulent Buoyant Jets Discharged to Flowing Stratified Ambients,* Oak Ridge National Laboratory Report ORNL 4685. [Has complete derivation of differential equations used in the Eulerian method]

Parsons, R.M. Laboratory for Water Resources and Hydrodynamics. (1971) *Temperature Prediction in Stratified Water* , Water Pollution Control Research Series Report 16130 DJH 04/71 MIT, Cambridge.[Reference on surface heat transfer]

Ptankar, S.V. (1980). *Numerical Heat Transfer and Fluid Flow*, Hemisphere Publishing Corporation [This is good reference on Computational Fluid Dynamics and Numerical Methods]

Van Wylen, G.M., and Sonntag, G.E.(1997) *Fundamentals of Thermodynamics, 5th Edition*, New York, Chichester, Brisbane, Toronto: John Wiley & Sons Publishing. [This is typical of a thermodynamic text that discusses the 2nd Law of Thermodynamics]

http://www.engr.orst.edu/~davisl/plumes.html [Both the UDKHG, and PDSG models can be downloaded from this web site]

http://steens.ese.ogi.edu [The CORMIX model can be obtained from this web site]

http://ftp.epa.gov/epa_ceam/wwwhtml/software.htm [The Visual Plumes model and user's manual can be downloaded from this web site]

Biographical Sketch

Lorin R. Davis, *Professor Emeritus. Rogers Hall, Oregon State University. Corvallis, Oregon USA, 97331*

Education : Ph.D. University of Illinois

Experience: 35 years teaching/research: San Jose State University, Oregon State University. 30 years consulting on environmental discharge modeling with USEPA, CH2M Hill, EXXON, International Paper, and others.

Author or coauthor of several computer programs used by the USEPA for mixing zone prediction including UDKHDEN, PDS, PSY, DKHW, AND VISUAL PLUMES (VP).

Publications: *Fundamentals of Environmental Discharge Modeling*, CRC Press, Boca Raton, 1999. *Workbook on Thermal Plume Prediction: Vol II Surface Discharge*, with M.A. Shirazi, USEPA, Washington, 1974. *Workbook on Thermal Plume Prediction: Vol I Submerged Discharge*, with M.A. Shirazi, USEPA, Washington, 1972 . 60 journal articles and reports.