Facultad de Ingeniería
División de Estudios de Posgrado
Departamento de Ingeniería Ambiental

Universidad Nacional Autónoma de México

MODELACION MATEMATICA DE DESCARGAS AMBIENTALES

Dr. Lorin Davis–Expositor
Dra. Rina Aguirre–Cordinadora

(The Mathematical Modeling of Environmental Discharges)

5 - 9 de Septiembre 1994

MODELACION MATEMATICA DE DESCARGAS AMBIENTALES

CONTENIDO

ANEXOS

1. GENERAL ASPECTS OF DILUTION MODELING.

DILUTION MODELS FOR EFFLUENT DISCHARGES

by

D.J. Baumgartner*, W.E. Frick**, P.J.W. Roberts***, and C.A. Bodeen****

* Environmental Research Laboratory
University of Arizona, Tucson, AZ 98706

** Pacific Ecosystems Branch, ERL-N
Newport, OR 97365-5260

*** Georgia Institute of Technology
Atlanta, GA 30332

**** AScI Corporation
Newport, OR 97365-5260

July 1, 1992

U.S. Environmental Protection Agency
Pacific Ecosystems Branch, ERL-N
2111 S.E. Marine Science Drive
Newport, Oregon 97365-5260

ABSTRACT

This report describes two initial dilution plume models, RSB and UM, and a model interface and manager, PLUMES, for preparing common model input and running the models. Two farfield algorithms are automatically initiated beyond the zone of initial dilution. In addition, PLUMES incorporates the flow classification scheme of the Cornell Mixing Zone Models (CORMIX), with recommendations for model usage, thereby providing a linkage between two existing EPA systems.

The PLUMES models are intended for use with plumes discharged to marine and some freshwater bodies. Both buoyant and dense plumes, single sources and many diffuser outfall configurations may be modeled.

The PLUMES software accompanies this manuscript. The program, intended for an IBM compatible PC, requires approximately 200K of memory and a color monitor. The use of the model interface is explained in detail, including a user's guide and a detailed tutorial. Other examples of RSB and UM usage are also provided.

This is Document No. N210a of the Environmental Research Laboratory, Narragansett. The accompanying software carries No. N210b.

DISCLAIMER

This document is intended for internal Agency use only. Mention of trade names or commercial products does not constitute endorsement or recommendations for use.

ACKNOWLEDGEMENTS

We acknowledge the leadership roles of Hiranmay Biswas, EPA Office of Science and Technology, and Barry Burgan, Craig Vogt, and Karen Klima, EPA Office of Wetlands, Oceans and Watersheds. They helped to formulate the concepts in the manual in broad terms, allocated resources, and provided opportunities to increase the scope of our efforts.

Also, we appreciate and recognize the technical advice and assistance of Edward Dettmann, Kenwyn George, Norm Glenn, Gerhard Jirka, and Mills Soldate. Other contributors include Michael Dowling, Karen Gourdine, George Loeb, and Chung Ki Yee. Their comments and suggestions contributed significantly to the content of this work.

The day-to-day support of Norbert Jaworski, Harvey Holm, and David Young of EPA is also gratefully acknowledged.

CONTENTS

GENERAL ASPECTS OF DILUTION MODELING

INTRODUCTION

Pollution control authorities frequently employ buoyant plume models to simulate expected concentrations of effluent contaminants in ambient receiving waters. During the decade of the 1980s a great deal of attention was given to the subject because of the U.S. Environmental Protection Agency's (EPA) regulation of publicly owned municipal wastewater discharges to marine waters (USEPA, 1982). The central feature of this regulation was a modified permit based on an applicant demonstrating the environmental acceptability of less than secondary treatment, consistent with criteria listed in section 301(h) of the federal Clean Water Act.

A number of models and other methods, e.g., field data, were used in this context, primarily to demonstrate compliance with a variety of applicable regulatory requirements of local, regional, state, and federal agencies. In addition, models were used to aid in the design of marine monitoring programs and in the design of new or modified ocean outfall pipelines and diffuser systems. In 1985 EPA published a user's guide to five models used in these activities (Muellenhoff et al., 1985) although the models had been distributed and used for years in many previous applications.

Possibly because of the popularity and the endorsement associated with the EPA user's guide, the models were applied by regulatory agencies, designers, and dischargers to problems beyond those for which they were originally intended. Some applications involved industrial wastes, drilling fluids from offshore oil exploration and development projects, and effluent discharge into freshwater systems, both lakes and rivers. Staff in the EPA offices were asked frequently to assist with these applications, and many users requested EPA to develop a more general model, or specific models for each situation. As a result of these requests, this user's guide and revised computer programs are provided. With respect to the 1985 models (Muellenhoff et al., 1985), UOUTPLM and UDKHDEN are neither reissued nor addressed herein, UPLUME is provided as a separate file but neither recommended nor addressed, ULINE is provided as a separate file and is recommended under some circumstances as an extension of RSB, which otherwise replaces it, and UMERGE is modified, extended, and replaced by the resident model UM. To the extent that PLUMES, described immediately below, facilitates UDF file generation, all earlier models are supported by PLUMES.

Both RSB and UM are contained in and managed by the interface program PLUMES. In addition, PLUMES contains two farfield algorithms and the CORMIX1 flow categorization scheme (Hinton & Jirka, 1992). In Appendix 1, the CORMIX flow categorization method is used as a framework for making provisional model recommendations.

The model UM is described subsequently in the manuscript, as is RSB, a model based on hydraulic model studies by Roberts (1977) and Roberts, Snyder, and Baumgartner (1989 a,b,c).

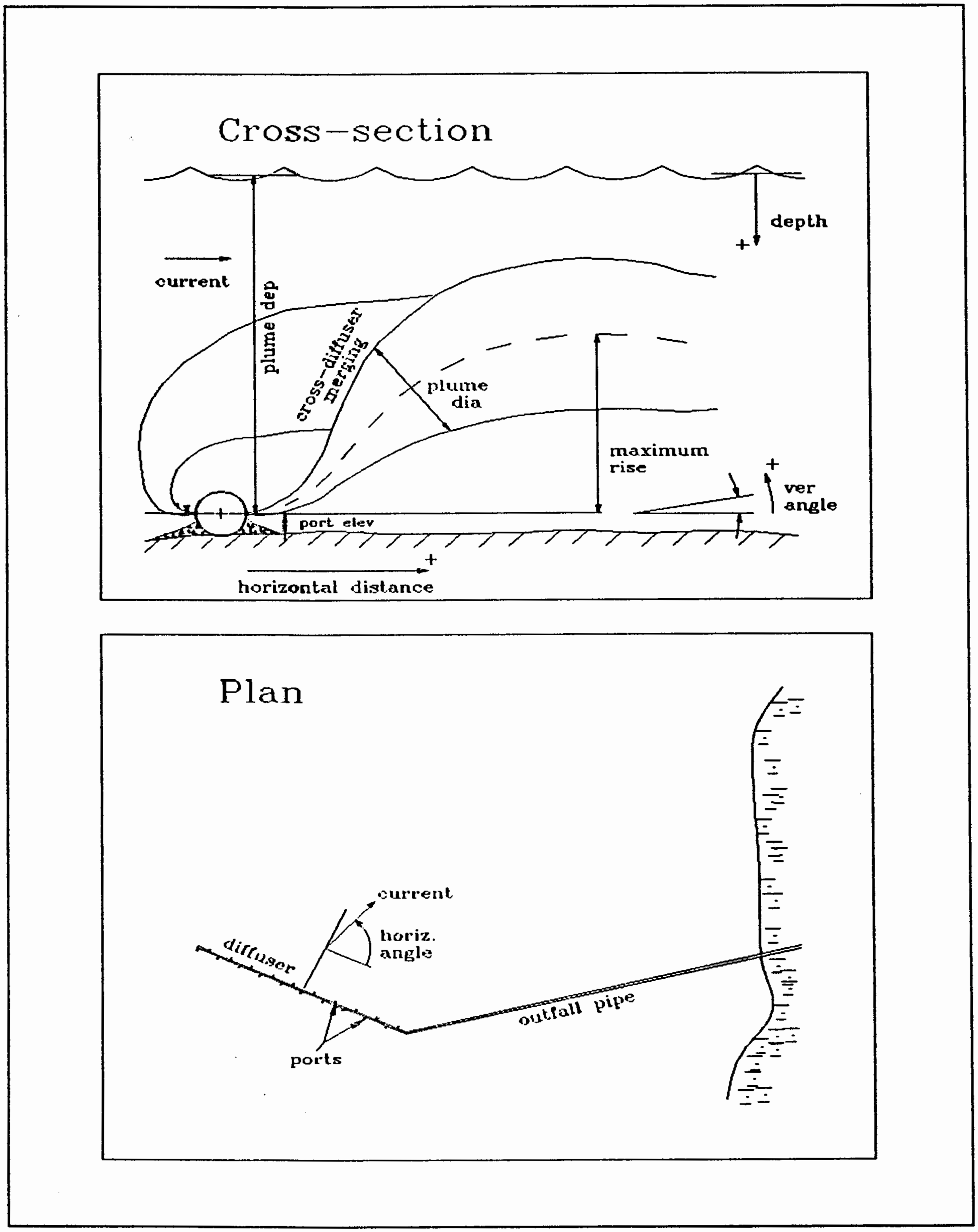
The new UM model provides essentially equivalent results as UMERGE, in fact, UM may be interpreted to mean "Updated Merge". However, UM possesses considerably more capabilities than its predecessor.

New subjects treated in this report include effluent material discharged at an arbitrary vertical angle to address the cases of positively buoyant material discharged downward, and negatively buoyant material discharged upward. These situations are handled by PLUMES. Discussion is provided on the problem of particulates in the waste stream, as this is becoming recognized as one of the more insidious problems of water pollution control, and on the possible use of the models in freshwater systems. Verification based on field and laboratory data is addressed as is information on uncertainty of predictions.

Subjects such as mixing zones and initial dilution concepts discussed in the 1985 report are repeated, sometimes verbatim, and updated with current interpretations. Discussion of the physical basis of models is expanded.

Readers of the earlier report (Muellenhoff et al., 1985) will also notice some deletions and changes. The computer codes for the programs are not included in the manuscript nor in the diskettes generally provided. Another is that the executable models are to be provided on diskette by the EPA marine research laboratory in Newport, Oregon, rather than by NTIS. (They may also be made available on the CEAM, Athens Bulletin Board Service.) These procedural changes are related. Due to user experiences as well as work conducted by EPA it is at times necessary to correct or improve the computer codes. It now appears that changes will occur sufficiently frequently so that it will be more effective to provide current models to users directly from EPA rather than from NTIS. New diskettes distributed by EPA will be accompanied by brief narratives describing the improvements in the physics or the computational routines that take place following publication of this report. These adjustments are judged to be too difficult to arrange on a timely basis through NTIS.

The authors assume readers will have some familiarity with terminology of buoyant plume mechanics, either as applied in regulatory practice or in fluid mechanics generally. Terms used in equations are defined in the text, frequently using different symbols than in original works cited. In different parts of the document, a symbol may represent different quantities, however, the meaning should be clear from the context. Terms and relationships are also explained in the "Help" screens of the interface program PLUMES. General definition sketches are shown in Figure 1.



Ilustr. 1. Definition sketch.

REGULATORY ADAPTATION OF PHYSICAL PROPERTIES OF PLUME BEHAVIOR

Initial Dilution

Initial dilution is the dilution achieved in a plume due to the combined effects of momentum and buoyancy of the fluid discharged from an orifice, and due to ambient turbulent mixing in the vicinity of the plume. The rate of dilution is quite rapid in the first few minutes after exiting the orifice and decreases markedly after the momentum and buoyancy are dissipated. Figure

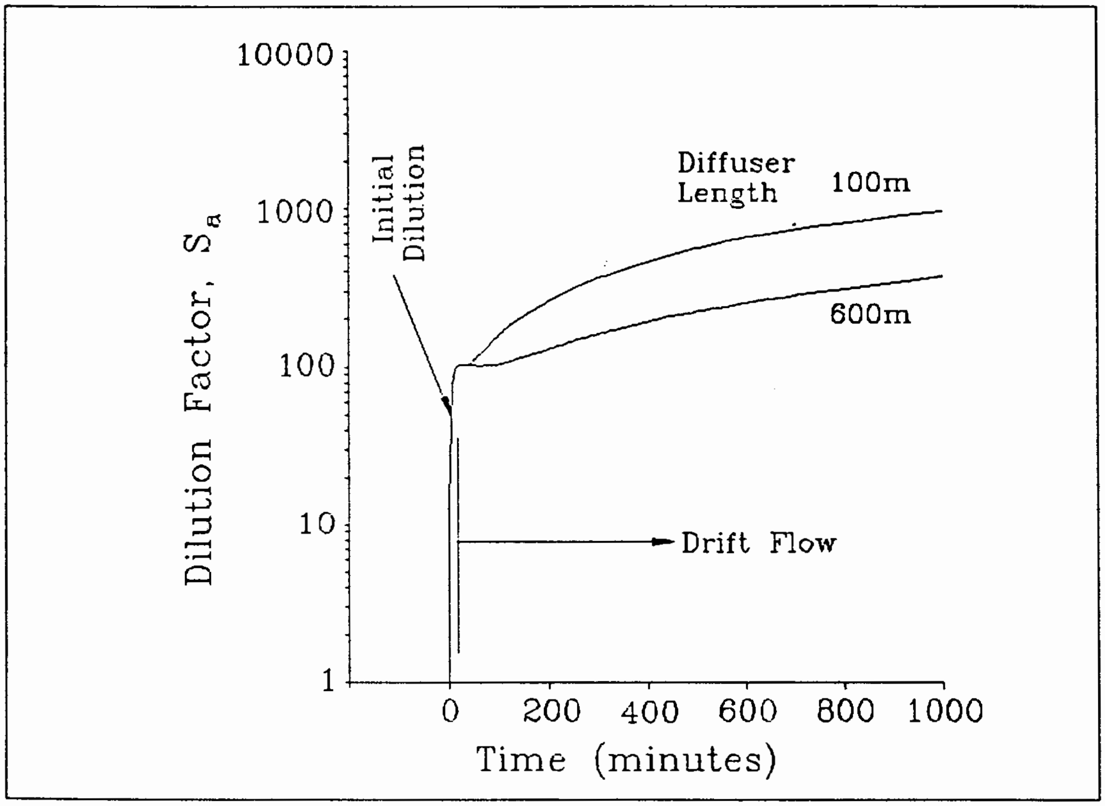


Figure 2. Plume dilution as a function of time.

2 schematically represents the relative dilution factors achieved in buoyant plumes and in the subsequent drift flow region under low to moderate current conditions.

Ambient currents will also influence the rate of dilution during the buoyant rise of the plume irrespective of jet momentum and buoyancy. As current speed increases so does initial dilution. This is shown in Figure 3 from Baumgartner et al. (1986) for certain west coast conditions using the models in Muellenhoff et al. (1985). UPLUME, not including current, gives constant dilution.

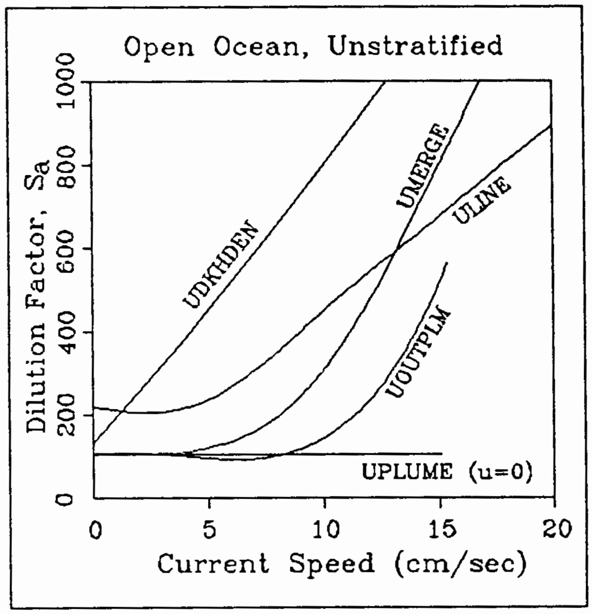


Figure 3. Dilution as a function of current speed.

It is useful to compute expected dilutions and plume locations under the vast range of current regimes likely to be encountered near an outfall. The information would be useful in optimizing monitoring programs intended to sample the distribution of ambient values of effluent constituents in analyzing the effectiveness of regulatory controls. Given sufficient data on environmental impacts in the region and accurate exposure data, one could imagine that regulatory agencies might evaluate the societal benefits derived from modifying the definition of critical initial dilution. For example, perhaps the twenty or thirty percentile value of current might be employed, rather than zero current or the ten percentile current, if data show only a slightly increased adverse effect! The increased uncertainty, and risk, associated with calculated values based on these still developing physical models of turbulent dispersion mechanics is not always recognized. It is a cost of attempting to describe more completely the behavior of the plume under actual conditions.

Critical Initial Dilution

The models described in this report are not constrained by any **regulatory** definition of allowable current speed, although there are limiting current conditions that each model can simulate. In relation to permit requirements of regulatory agencies it is necessary to think of "allowable" initial dilution factors, or "critical" initial dilution factors based on conservative values of parameters in addition to current speed. "Critical" values in terms of EPA's 301(h) permit requirements (USEPA, 1982) include consideration of current direction as well as speed, and other environmental and wastewater factors. The importance of current direction will be discussed subsequently in the report.

The California Ocean Plan (State Water Resources Control Board, 1988) requires zero current speed to be used in computing initial dilution values intended to predict compliance with permit conditions. Whether intended or not, this regulatory approach results in a predicted initial dilution that is less uncertain than would be obtained when the effects of current are included. In the EPA regulations for a permit modified by section 301(h) of the Clean Water Act (USEPA, 1982), EPA allowed the lowest ten percentile current to be used in computation of the critical initial dilution value. In many coastal settings the ten percentile value is below

5 centimeters per second (cm sec-1), i.e., 0.16 ft sec-1, or less than 0.1 knot. At current speeds this low there is essentially no effect on the rate of dilution.

Other environmental and wastewater flow considerations are not discussed, primarily because the models are generalized to the extent that any set of regulatory constraints may be handled in use of the models. Furthermore, these parameters do not influence the physics of plume behavior.

Mixing Zone

Permit conditions of regulatory agencies usually allow exceptions within a mixing zone adjacent to the point of discharge. With respect to EPA's 301(h) regulations, the rationale and the precautions associated with mixing zones, and the relationships to initial dilution are described in Muellenhoff et al. (1985). The use of the initial dilution models since 1985 in defining mixing zones and in computing allowable discharge concentrations has suggested the need for additional discussion.

In nature, regulatory restrictions notwithstanding, the initial dilution process occurs over a wide spatial range compared to the length of an outfall diffuser or the depth of water at the discharge site. The effect of current on the scale of the initial dilution process is portrayed in Figure 4. Under low current conditions, e.g. U = 0.1 m/sec, initial dilution is virtually completed before the plume is carried downcurrent a distance X_i equal to the water depth, for example 30 meters when the buoyancy frequency N, a measure of density stratification, is 0.03 per sec. In a strong current the process can extend downcurrent a distance equal to multiples of diffuser lengths (Roberts et al., 1989b). At a current speed of 1m/sec X_i would be 300 meters.

Recognizing this, what might a regulatory agency prescribe as a mixing zone, that is, a zone in which water quality criteria are permitted to be exceeded? If a conservative posture is adopted, the agency would allow a mixing zone of 30 meters on both sides of the diffuser. If a more liberal view prevails a distance of 100 meters could be established. With the possible exception of riverine settings, it is necessary in most cases to describe the zone on both sides

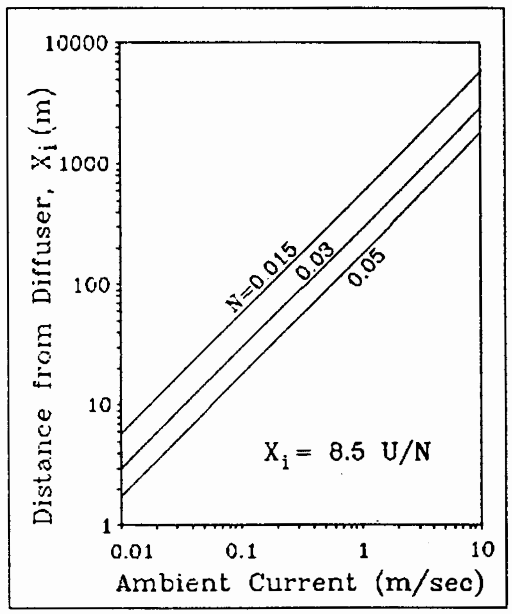


Figure 4. Length of the zone of initial dilution as a function of current speed.

of the diffuser because the currents during one part of a day are likely to be about 180 degrees opposite those six hours later.

EPA has adopted the conservative posture, at least for marine outfall problems regulated under section 301(h). Thus a smaller area of the environment is removed from the general region protected for unlimited use. Organisms entrained into the plume would be exposed to rapidly decreasing concentrations of pollutants and within minutes, e.g., three, would be in an environment containing pollutants at concentrations below the safe limit. The expectation is that most of the time, e.g., 90% of the time or more, currents are sufficiently high to cause even a greater rate of dilution. Under high currents the concentrations at the boundary of the mixing zone would be expected to be less than the specified criteria values and quite possibly a good portion of the mixing zone would actually meet the necessary criteria.

This expectation has not been rigorously tested. Hydraulic model tests conducted by Roberts et al. (1989 a, b, c) suggested that situations might exist where the expectation is not realized. The model UM can be used to generate simulated data that might be useful to test this assumption. A hypothetical outfall situation is described as follows:

EXAMPLE PROBLEM

Flow: 4.47 cubic meters per second
Number of 8.5 cm ports: 143
Port spacing: 7.3 m
Discharge angle: horizontal
Water depth: 76 m

A sample of the input and output for this problem is shown in Appendix 2. Model UM was run for a range of currents, and the plume concentrations at a downcurrent distance of 30 m were interpolated from the output data. (The Zone of Initial Dilution, or ZID, defined in the 301(h) regulations, would be larger but, in general, mixing zone regulations vary from state to state.) The data shown graphically in Figure 5 demonstrate that, as currents increase, the dilution increases to a maximum but then begins to decrease.

(Three noteworthy inflections appear in Figure 5. At current speeds lower than those marked (a) the plumes reach maximum height inside the mixing zone and impinge on the surface. Adjustments have been made to cause the simulation to reach the mixing zone boundary. All cases with currents less than (b) encounter the overlap condition to be described subsequently. Finally, at speeds higher than (c) the plumes no longer merge.)

Assuming this example is somewhat representative, what importance should be attached to the concentrations above a standard level at the boundary when the currents exceed a relatively large value? Organisms entrained into the plume will have traveled with the rapidly diluting wastefield for only a couple of minutes before the concentration is reduced below the standard, whereas with a small current the exposure time in the mixing zone is approximately 10 minutes.

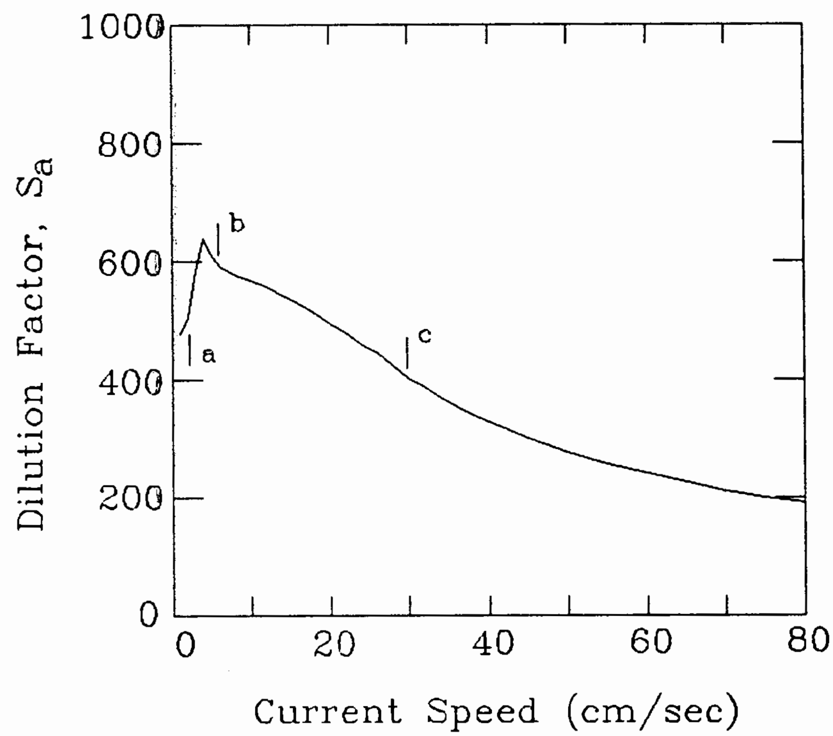


Figure 5. Dilution at the mixing zone boundary as a function of current speed.

Organisms at and beyond the boundary will then be more greatly stressed than entrained organisms in low current conditions. If for example the regulatory authority established the mixing zone boundary to protect a community of benthic organisms from being exposed to concentrations above the standard, then the standard will be abrogated when currents are large. Even in unstratified ambients it is possible that high current speeds will cause effluent streams to hug the seabed thus placing benthic resources at greater risk. Under low currents the plumes will rise and be retained closer to the diffuser. Entrained organisms and near-surface resources are more at risk under this scenario. Regulatory agencies may effectively incorporate this knowledge into mixing zone boundaries which are narrower near the surface and wider at depth based on these model simulations.

The term "near field" was adopted in narratives associated with the 301(h) regulations to describe the region near the outfall inside the zone of critical initial dilution, and "farfield" was similarly meant to apply to areas possibly impacted beyond this zone. For most cases "near field" would be consistent with the term "mixing zone".

Dilution Factor

The dilution factor, S_a , used in some regulatory applications, including the EPA model UM is the reciprocal of the volume fraction of effluent, V_e , contained in the diluted plume. An equivalent way of expressing this term is the ratio of effluent volume plus volume of ambient dilution water, V_a , to the effluent volume, as in Equation 1.

$$S_a = \frac{1}{\frac{v_e}{(v_e+v_a)}} = \frac{(v_e+v_a)}{v_e} \tag{1}$$

Thus in the region immediately outside the discharge orifice the volumetric dilution factor is very nearly 1. In some discussions of this term in other works, e.g. the California Ocean Plan (State Water Resources Control Board, 1988), the factor is considered to be the ratio of the volume of ambient dilution water, V_a , to the volume of effluent discharged, V_e. In this definition the volumetric dilution factor approaches zero near the orifice. Above a value of 30 the difference in the two definitions is progressively less than 3%, an inconsequential amount for most regulatory purposes.

The former definition, i.e., Equation 1 is used in this report. This is not an arbitrary decision, but rather is based on the general equation used to calculate the contaminant concentration in the plume. Using the continuity equation,

$$c_p\, v_p = c_e\, v_e + c_a\, v_a \tag{2}$$

where

c_p = Cross sectional average concentration in the plume,
V_p = Volume flux of the plume,
c_e = Concentration in the effluent,
V_e = Volume flux of the effluent,
c_a = Concentration in the ambient dilution water, and
V_a = Volume flux of the ambient dilution water.

Substituting $V_a + V_e$ for V_p and rearranging,

$$c_p = \frac{c_e\, v_e + c_a\, v_a}{v_e + v_a} \tag{3}$$

The volume fraction, Equation 1, is a useful approximation of the concentration of a pollutant in the diluted plume only if the pollutant concentration in the ambient dilution water is very low compared to the concentration in the effluent. Thus if S_a = 30 (which means the effluent is diluted with 29 volumes of ambient water), the concentration of any volumetric tracer or conservative pollutant in the effluent is one thirtieth the concentration in the effluent only if the ambient concentration is zero. In the case of zero ambient concentration Equation 3 reduces to:

$$c_p = \frac{c_e \, v_e}{v_e + v_a} \tag{4}$$

Dividing both sides by c_e and inverting,

$$\frac{c_e}{c_p} = \frac{v_e + v_a}{v_e} = S_a \tag{5}$$

Equation 5 demonstrates that for the special case of zero ambient concentration the volumetric dilution factor also describes the dilution of a pollutant. In most regulatory uses of the plume models, however, it is necessary to consider the actual, nonzero, ambient concentration of the suite of pollutants in the effluent. In the remainder of this report the term "effective dilution factor" (S_{aei}) is used to describe the dilution achieved for each pollutant in a plume. That is,

$$S_{ae_i} = \frac{c_{e_i}}{c_{p_i}} \tag{6}$$

where the index, i, is used to demonstrate that in determining the final concentration of a pollutant in the diluted effluent the effective dilution must be determined for each pollutant individually.

Effective Dilution Factor

It is instructive to recognize that S_{aei} is not necessarily constant for a suite of pollutants in a discharge for any given volumetric dilution factor, S_a. This is so because the ratios c_{ei} / c_{ai} are not necessarily constant, and the volumetric dilution factor is determined only by the density of the plume irrespective of the contribution made by any of the pollutants individually. The effective dilution factor, S_{aei}, can be determined from Equation 6 for each pollutant by first determining the concentration of each pollutant in the plume. The general solution is related to the volumetric dilution factor, S_a, through Equation 3. First, multiply the right side of Equation 3 by v_e / v_e, giving

$$c_{p_i} = \left(\frac{c_{e_i} \, v_e}{v_e} + \frac{c_{a_i} \, v_a}{v_e} \right) \frac{v_e}{v_e + v_a} \tag{7}$$

Next, recalling Equation 5, substitute $S_a - 1$ for v_a / v_e, and $1 / S_a$ for $v_e / (v_e + v_a)$, Equation 7 becomes

$$c_{p_i} = \frac{c_{e_i} + c_{a_i}(S_a - 1)}{S_a} \tag{8}$$

This is simplified to

$$c_{p_i} = \frac{c_{e_i}}{S_a} - \frac{c_{a_i}}{S_a} + c_{a_i} \tag{9}$$

which is analogous to equation (1) given in Muellenhoff et al. (1985).

The advantage of Equation 9 is that for many situations the computer program for a plume model needs to be run only once, that is, to obtain S_a. With S_a in hand c_{pi} can be computed repeatedly using paired values for c_{ei} and c_{ai}. If c_{ai} is not uniform over the depth through which the plume rises, an average value can be used to provide an estimate of c_{pi}. However, this is only an estimate as entrainment is not generally a linear function of the vertical position of the plume in the receiving water. The new model, UM, described in this report accepts a tabular input of the vertical distribution of ambient concentration and computes the actual, effective diluted concentration. Since this model is quick and easy to run, there is only a modest advantage in using Equation 9 to obtain subsequent estimates of c_{pi}.

However with the CORMIX models and with RSB the dilution factors and plume concentrations provided are based strictly on volumetric dilutions and must be corrected for the ambient background. For a first order correction it is possible to assume the rate of dilution is uniform over the rise to the trapping level so that if the ambient concentration is uniform over that depth a simple correction can be applied using Equation 9. In the simple example problem given above, the ambient pollutant concentration is given as 1.6 concentration units, thus a volumetric dilution factor of 1000 results in a plume concentration of 1.698, or an effective dilution factor of only 58.9! The influence of background on effective dilution is apparent.

Spatial and Temporal Variation of Plume Concentrations

The concentrations of water quality indicators, such as contaminants and desired constituents (e.g., dissolved oxygen) are neither uniform nor steady with respect to the space and time scales involved in regulating the concentrations at the end of the mixing zone. The nonuniformity of constituents in the horizontal extent of an outfall diffuser is generally not investigated and is usually assumed to be uniform, as is the incremental volumetric flux. If nonuniformities along the length of the diffuser are encountered the dilution model can be run for each segment of the diffuser that may be assumed uniform. A separate hydraulic model described in Appendix 3 is included in the software. Vertical nonuniformity is more commonly encountered in design, performance analysis, and compliance monitoring.

Vertical nonuniformity is important to consider from the standpoint of the constituent concentrations in the ambient receiving water, i.e., the dilution water mixed with the effluent being discharged. The variations in the vertical are due to physical processes influencing the advection of ambient water into the region of the discharge, and, for some constituents, antecedent biological and chemical processes that have changed the form or concentration of the constituent. Typically, field observations during synoptic surveys are relied on to provide vertical profiles of the water quality indicators. Dissolved oxygen (DO) is an example of one water quality indicator that exhibits vertical nonuniformity in many lake, estuarine, and coastal situations. The concentration of DO in a plume is important to determine because of direct biological effects, and because the strategy for effective regulation of DO at the end of the mixing zone is strongly dependent on the relative influence of effluent constituents and the vertical profile of receiving water constituents. The way in which the dilution models are used to analyze the plume DO concentration illustrates a method for dealing with other ambient nonuniformities.

The Dissolved Oxygen Problem

The DO concentration in a plume is affected by the DO in the effluent, the chemical and biological constituents in the effluent which exert a DO demand, chemical and biological demand factors in the seabed, and by oxygen demand in the water column carried by currents into the region of mixing. The DO demand in the effluent is conveniently represented by the effluent parameter called the Immediate Dissolved Oxygen Demand, IDOD. According to Standard Methods for the Examination of Water and Wastewater (APHA, 1975), IDOD is the amount of oxygen consumed in a 15 minute reaction time. (Later additions of Standard Methods do not include this method because the authors were not able to interpret the significance of the measurement in relation to total oxygen demand.) Since mixing zones established under the EPA regulations for 301(h) permits represent travel times generally of the order of less than 10 minutes, IDOD is a conservative estimate of the mixing zone demand. On this time scale chemical and biological demands in the ambient are inconsequential although for farfield water quality considerations after initial dilution they are frequently decisive. Under these assumptions the concentration of DO in the plume, c_{DO}, is found using the equivalent of Equation 9 with an additional term to represent the immediate demand, viz:

$$c_{DO} = c_{DOa} + \frac{c_{DOe} - IDOD - c_{DOa}}{S_a} \tag{10}$$

To solve this equation it is necessary to have field data on the c_{DOa} profile; the values of c_{DOe} and IDOD being derived from laboratory analyses. In many cases the c_{DOa} is low near the seabed due to benthic demand, reaches a maximum at an intermediate depth in the water column, and then is constant or slightly decreasing in the near surface layer of the receiving water.

In some coastal regions there are deep permanently anoxic or hypoxic basins. Lakes and reservoirs may also have such basins, perhaps only seasonally. If an outfall is placed in an oxygen-poor basin and the vertical density structure is such that the plume rises into near surface waters, the resulting DO in the plume will be very nearly the same as the deep water, thus quite likely abrogating a desired DO standard irrespective of the amount of oxygen demand in the effluent. While the violation of the standard is not due to the pollutant discharged in this case, it is due to the discharge of effluent! If aquatic organisms in the surface layers are sensitive to low oxygen concentrations it will matter little to them if the deficit is due to effluent or deep oxygen-poor water forced to the surface by the buoyant effluent. The potential for "forced upwelling" or "effluent pumping", as it has at times been labeled, should be considered in the design of outfalls, both from a standpoint of selecting the site, and of the mechanics influencing the height of rise of the plume. By careful balancing of those design factors which influence final plume concentration, optimum strategies can be developed for achieving ambient standards.

Equation 10 is analogous to Equation VI-7 in EPA's Revised Section 301(h) Technical Support Document (USEPA, 1982). However it is not stated that the tabular listing (page VI-21) of IDOD contributions to the final plume dissolved oxygen concentration are <u>negative</u> contributions.

Recirculation, Quiescent Periods, and Other Temporal Variations

The models reported in Muellenhoff et al.(1985) were steady state models, as are the models used in this report and, as such, they do not take into account temporal variations in any of the variables. For most applications this limitation should not be a problem. In the EPA 301(h) regulations the effective initial dilution is determined for a set of effluent and receiving water conditions that approaches a worst case scenario, that is, there is only a very low probability that there would be physical circumstances under which a predicted final plume concentration would be exceeded. The models can be used repeatedly however to generate a data set for a range of values expected or observed in nature, as done for example to construct Figure 3 showing the effect of different current speeds on volumetric dilution. Although this result is not a time-variable solution to a buoyant plume problem the rate of change in dilution between two current speeds is not an important consideration in regulatory practice, because the effect of current on plume behavior is nearly instantaneous. Thus it is eminently satisfactory to use the steady state model at discrete time steps.

Data sets can be generated to show the frequency distribution of currents and associated dilutions at a discharge site, as in Figure 6 (Baumgartner et al., 1986). From an environmental management perspective it may be important to investigate the distribution of dilutions achieved as a result of seasonal changes. Figure 7 shows the monthly distribution of initial dilution values calculated by UMERGE (Muellenhoff et al., 1985) for incremental changes in tidal currents superimposed on a steady longshore current for a typical U.S. west coast discharge site.

The dramatic effect of current speed, in this case the effect of tidal current, shown in Figure

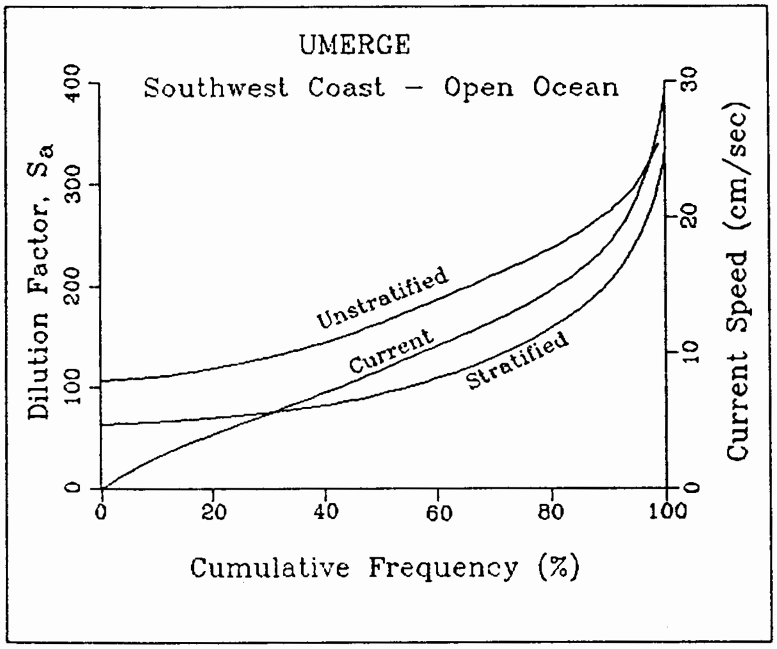


Figure 6. Dilution and current frequency.

6 demonstrates that most of the time dilutions at the end of the buoyant plume phase will be greater than in the critical case used to specify permit conditions. In this example the critical dilution appears to be about 35 whereas every month there are values greater than 100. When field monitoring of the effluent concentrations is required it is important to recognize the range of values that might be expected due to the range of ambient currents occurring as well as variations in the effluent as it leaves the treatment plant.

Just as easily as Figure 7 was generated, a computer code external to the model could be developed to generate a data set of dilutions resulting from variations of other variables, such as wastewater flow and density stratification in the receiving water. Figure 8 is an example of a response surface developed using UMERGE (Muellenhoff et al., 1985) to show the effect either singly or combined variation in the densimetric Froude number, *F*, and the stratification number, *SP*. The Froude number is a convenient way to independently investigate variation in either wastewater flow or the design diameter of the effluent ports in this graphic.

Similar graphical representations of any three selected variables can be useful in analysis, however, there are limitations of unknown importance if the variables chosen are not independent. For example, although it would be possible to construct a response surface for dilution, current speed, and density gradient, which would be accurate in the abstract sense of these variables being independent in the model formulation, it is not likely that the range of density gradients chosen would in nature be entirely independent of the current speed.

Figure 7. Simulated annual variation in dilution.

Effect of Wastewater Flow on Dilution

Depending on the densimetric Froude number at the discharge port, the effect of increased effluent flow per port on dilution can be shown to be detrimental, insignificant, or favorable. With low Froude numbers as frequently found with municipal ocean outfalls, an increase in flow causes a decrease in dilution, while at higher Froude numbers, as might be found with modern power plant cooling water discharges, an increase in discharge results in an increase in dilution.

Figure 8. UMERGE dilution response surface as a function of Froude number and stratification.

According to Rawn, Bowerman, and Brooks (1960), the 1930 data from the Los Angeles outfall provided a guide to the conditions under which the transition occurs (see Figure 9).

If density stratification or shallow water prevents the plume from rising very far, the transition to increased dilution is seen in this graph to occur at lower Froude numbers. This reflects the importance of high jet-like plume velocity near the discharge causing an increased rate of entrainment and a greater horizontal travel before reaching the trapping level or the surface. In deep water the vertical travel of the plume and the entrainment caused by buoyancy over the major portion of the travel distance play an increasingly greater role than conditions near the port in determining the final dilution. In deep water the transition to increased dilutions would be seen only at very high effluent flows.

An example of the effect of wastewater discharge flow on dilution can be seen in Figure 10 also. In this graphic the negligible effect of low current speeds as simulated by the model UMERGE (Muellenhoff et al., 1985) is shown in the Roberts' Froude number (Roberts, 1977). Increased effluent flow causes the densimetric Froude number to increase from 0.7 to 7,

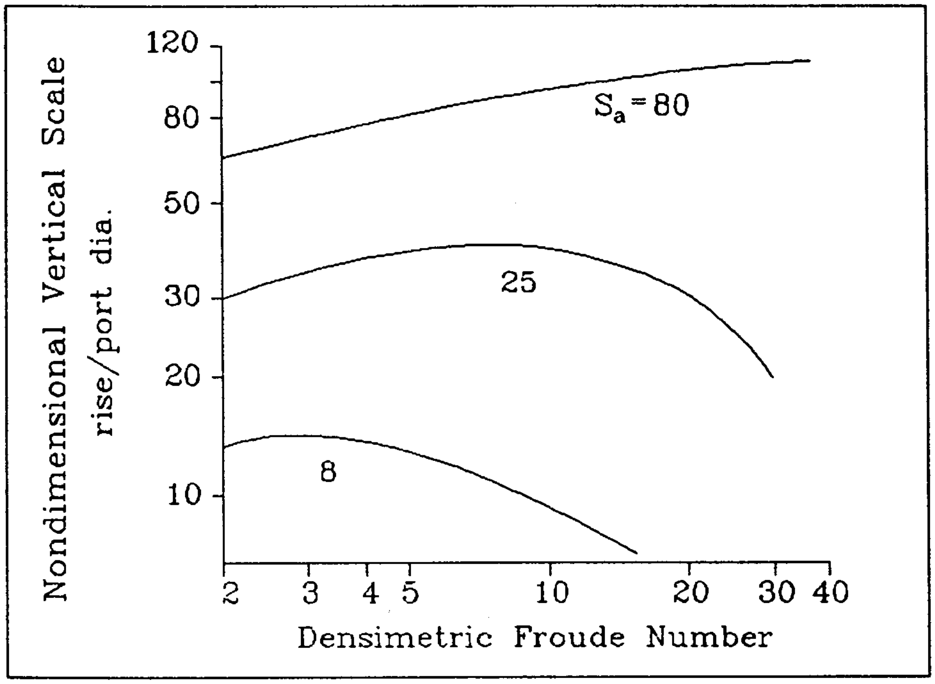


Figure 9. Examples of plume rise, dilution (S_a), and densimetric Froude number (effluent flow) relationships.

resulting in a decrease in dilution in deep water from about 250 to 150.

Table I shows the effect of increased effluent flows calculated by the UM and RSB models described in this report for the outfall characteristics described in more detail in Appendix 2. UM predicts a more substantial negative effect on dilution for an increase in flow from 100 MGD to 275 MGD. The volumetric dilution (S_a) at the surface at 100 MGD is 314 while at 275 MGD it is 187. The volumetric dilution is not calculated at the surface by RSB; instead the RSB model calculates the volumetric dilution at the end of the buoyancy dominated region assuming the water depth is sufficiently great to accommodate the complete initial dilution regime. At a flow of 100 MGD through this outfall the water depth would have to be greater than 65 meters to achieve a dilution of 394. At a flow of 275 MGD, the water depth would have to be greater than 90 meters to achieve a dilution of 382. Later in this report example problems which are more appropriately defined will be provided to examine the differences between the two models and

Table I. Dilution factor, S_a, predicted by UM and RSB vs. effluent flow

Q	Dilution Factor	
(MGD)	UM	RSB
100	314	394
137	259	387
183	222	384
228	201	383
275	187	382

Figure 10. Dilution response surface as a function of Roberts Froude number and the densimetric Froude number.

the dilution values they calculate. The effects of port spacing and density stratification will be explored in these examples also.

Depth as a Factor

Depth as a governing factor in the effective placement of ocean outfalls has taken on significance that is not always warranted. It is true that all other things being equal, the greater the extent of vertical travel experienced by the plume, the greater is the amount of entrainment. If a location is chosen with greater depth but poorer circulation, the net result may be less effective dilution of wastes than placement in a shallower but more open coastal area. This is the major concern with placement of outfalls in fjords, embayments, and, in some cases, estuaries, but this consideration must also be kept in mind when canyons, trenches, and deep basins offshore are considered as outfall sites. The implications for seabed accumulation of effluent particulate matter may be more important in the long run than the water column implications of re-entrained effluent.

Offshore Distance and Depth

The rationale for great depth as a factor in design of ocean outfalls seems to have been recognized empirically as a result of observations by A. M. Rawn on the Los Angeles outfall built in 1937 (Pomeroy, 1960). The primary consideration evidently was to reduce nearshore pollutant (coliform) concentrations through greater travel times, and thus more die-off, associated with outfalls further offshore. Greater depth, at least in the Southern California Bight was a gratuitous benefit of offshore distance. Through thoughtful analysis of monitoring data Rawn and coworkers recognized that lower beach coliform counts in the summer were in large part related to summer density stratification at the discharge site. In designs for subsequent outfalls submergence of the diluted sewage field was a conscious objective in addition to distance from shore (Brooks, 1956). This dependence on depth took on unique significance in the early legislative history of the 301(h) amendment, and was even proposed as the basis for granting waivers in estuaries! EPA scientists suggested that physical criteria relating to effective seaward displacement of pollutants from estuaries would be necessary in addition to depth and these were then included in the final language.

Submerged Driftflow, Upwelling, Wind Drift

The practice of designing diffusers to retain the drift field in the pycnocline, a region of large vertical gradient in density, below a surface layer may result in adverse implications for nearshore water quality due to characteristic upwelling of deep water along some major continental margins. This may not be a problem in the Southern California Bight, but needs to be considered when exporting southern California technology to other locations. It has been mentioned as a factor to be considered in outfall designs for the Oregon coast (Behlke and Burgess, 1964). The concentration of contaminants carried nearshore may be higher than if the outfall had been designed to take advantage of greater dilution offered by the full depth of water. This is a tradeoff to be considered in light of the potential damage caused by onshore drift of surface waters under prevailing winds in certain parts of the year.

By careful attention to wind, current and density patterns, it may be possible to design an outfall so that the plume is submerged when there is the least chance of upwelling, and above the pycnocline when there is the least chance of onshore winds. Most outfalls do not have the design or operational luxury to allow for opening or closing some of the ports. For those that do there is an additional option for adjusting the height of rise of the diluted plume.

Dye Tracing of Plumes

Dye tracing is a well known technique used in hydraulic models and prototype outfall settings, although the cost of added tracers in prototype situations is considerable because of the large volumetric flow rates and large dilutions usually achieved within several tidal cycles. The rate of dye addition (Q_d) to the effluent flow V_e needed to provide a dye concentration of c_d following dilution of S_a is:

$$Q_d = \frac{V_e \, c_d \, \alpha_a \, S_a}{W \, \alpha_d} \qquad (11)$$

where

α_a = specific gravity of ambient water
α_d = specific gravity of dye solution
W = weight fraction of dye in stock solution.

The required dye rate in gallons per minute is shown in Figure 11 for various dilution factors and effluent flows in MGD to achieve an ambient dye concentration of 1 ppb. Rhodamine WT, typically used in dye studies, is available as a 20% solution (α_a = 1.19) in small (15 gallon) drums.

Spatial Averages and Discrete Values

Some buoyant plume models produce dilution factors in terms of the centerline concentration, sometimes referred to as the "minimum" dilution for the cross section of the plume at a given distance downstream from the orifice. As the plume radius continues to expand with increasing distance, the minimum dilution progressively increases. For example the centerline (minimum) dilution at a distance of 6 meters from the diffuser port may be 6 while 10 meters from the orifice the minimum dilution would be more like 9. Some models calculate an average dilution for the cross section of the plume and this of course also increases downstream. The average dilution is always larger than the minimum dilution. The appropriate average is termed the flux-average dilution found by weighting the concentration distribution by the velocity distribution over the cross section of the plume.

The physics of the dilution process is frequently based on the centerline mass concentration so that the resulting calculation of average dilution is external to the physics. That is, if a modeler assumes the effective width of a single round plume is defined by the five percentile value of a Gaussian distribution, the average dilution will be less than if the 33 percentile value is chosen. In either case the centerline concentration would be the same. For this reason modelers would prefer to compare model results in terms of the centerline value rather than average values. Both values need to be considered in field or lab verification studies, and both values may be useful for regulatory purposes. Some models assume a uniform cross sectional concentration (referred to as a "top hat" profile) equivalent to the centerline concentration.

UM uses an assumed profile to help establish minimum dilutions from predicted model average dilutions.An opportunity to discuss minimum and average dilutions further is given in the following chapter: "Example: A CORMIX1 Comparison, Density, Stability, and Profiles". It should be noted here that while minimum dilutions are often of interest to regulators, average, or "top hat", dilutions are more consistent with the dynamic requirements of plume theory.

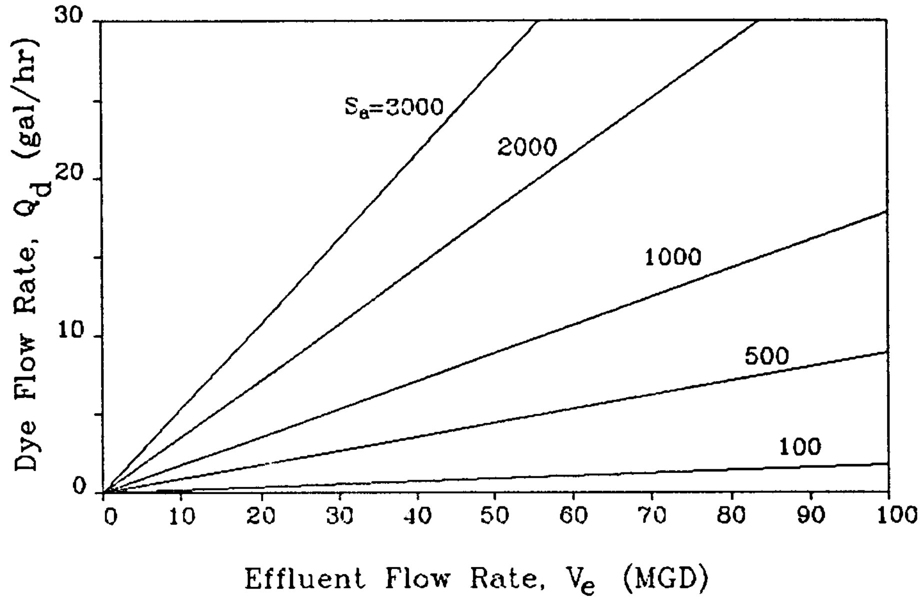


Figure 11. Dye flow rate to achieve 1 ppb in seawater with 20% Rhodamine WT.

Regulatory Use

Regulatory interest may be appropriately directed toward both average values and discrete values. Unfortunately the state of the art of regulatory practice is not as sophisticated as plume modeling and is generally constrained by lack of information on the temporal and spatial scales of aquatic organisms' responses to exposure conditions in natural settings. For some parameters California (State Water Resources Control Board, 1988) and the USEPA (1986) specify maximum allowable instantaneous and several temporal average values. If an applicable criterion for a certain biological resource near the outfall is an instantaneous value, a discrete value obtained over 5 to 30 seconds, as could be achieved by sampling methods used for plume studies in the field, would be appropriate. Many such samples would be taken to attempt to find the highest concentration of pollutants, i.e., the centerline value.

Additionally it might be argued that a biological resource at risk at any moment is appropriately evaluated over an expanse of space so that a spatial average is required, again evaluated in a short time period. The time period over which this averaging would take place is unfortunately not easily defined in relation to "instantaneous". It certainly is not seconds because it is impractical to acquire these data synoptically across the expanse of even one plume diameter let alone a multiport diffuser. If the data are obtained in an hour or two during slack tide, calm seas, and low currents, it is possible that the values will not be greatly different from one plume to the next in the same diffuser. Depending on the biology of the resource, either the maximum concentration (the minimum dilution) or the flux average dilution might be the appropriate value to use in determining compliance with "instantaneous" criteria applied to a spatial resource expanse.

Criteria that are expressed in terms of temporal averages (daily to semi-annual) suggest that plume concentrations be assessed extensively in three dimensions, both at the boundary of the mixing zone and in some cases at sensitive biological resource locations down-current. Current speed and direction play significant roles when assessing the concentrations at the boundary.

By incorporating data on the cyclical variation of effluent composition, density profiles, and current direction it is possible to construct a running six month average (or median) for a number of points on the mixing zone boundary. The six month average is expected to be quite variable at these points, and the point with the highest exposure frequency may not have the highest average concentration.

Beyond the mixing zone there may be regions where current streams of diluted effluent, leaving the zone at different times in different directions, would converge over a reef, a kelp

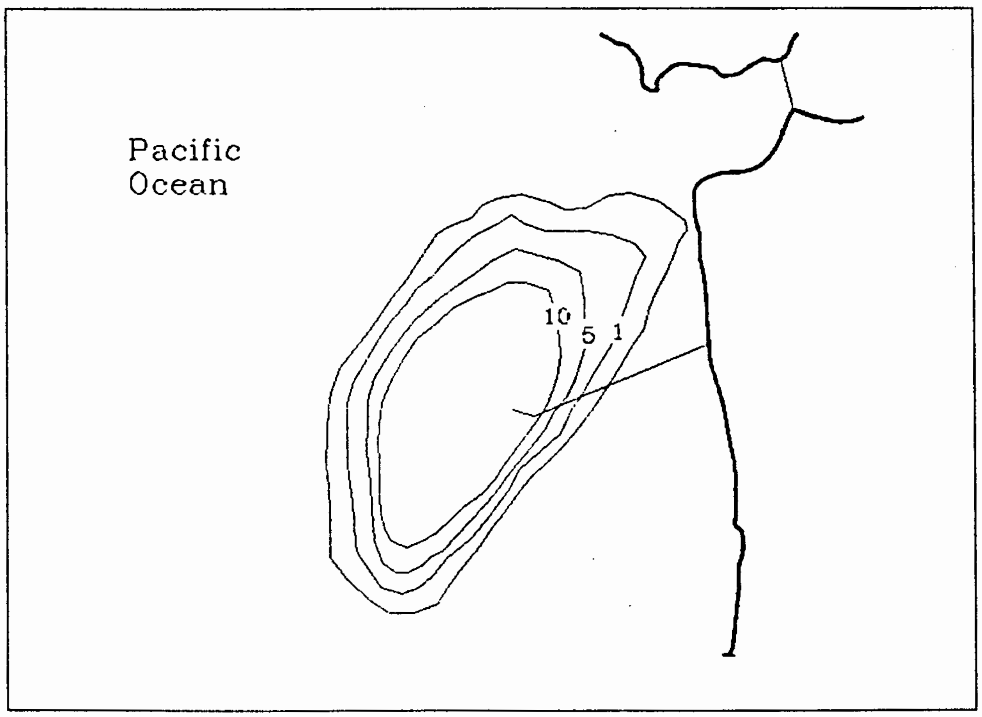


Figure 12. Visitation frequency (percent) of effluent about the San Francisco Southwest Ocean Outfall.

forest, or a swimming area. Thus if frequency and duration are important exposure characteristics in resource response, the exposure may be more critical even if the concentration (intensity) is lower, as it almost surely will be. In this case current direction is important to understand on a larger scale so that circulation patterns are evaluated. Some formal applications of this "visitation frequency" approach (Figure 12) have been used in regulatory assessment of criteria that are presumably "instantaneous" (Roberts, 1990). Depending on the size and nature of the resource to be protected either discrete or spatially averaged values might be appropriate.

The regulatory authority may not need to prescribe specific criteria for each of several segments along the mixing zone boundary. More likely they will be interested only in the highest six month average concentration wherever and whenever it occurs. Thus the formal methods for determining a relationship between frequency of occurrence, intensity of the stress (concentration), and duration of the exposure for plume performance at the mixing zone boundary are not rigorously established. However, designers, environmental scientists, and regulators should assess these performance characteristics conceptually, and possibly with a well chosen suite of model simulations, to conscientiously achieve responsible regulations and to guide improvements in the state of the art. USEPA (1986) provides a method to evaluate the appropriate relationship for ammonia in freshwater streams, which may be taken as an indication that frequency, intensity, duration relationships developed for evaluating outfall performance would be useful in improving regulatory practice.

Aside from the question of whether discrete values or cross sectional averages are used to test compliance with criteria, the way in which field samples are used to verify or compare with model results is an important consideration.

Verification Sampling

In laboratory or field verification studies of plume performance the average value is measured or captured in a sample bottle only by chance. Characteristically the field value measured is from a very small spatial region and represents a signal over a certain time span. A large number of samples is sought from the same cross section in order to arithmetically compute an average. In the laboratory, using a single plume, this is relatively easy to do. But in the field where multiple plumes are usually involved, and a moving flow field too deep below the surface to see is being sampled by a moving sampler from a moving boat, it is quite uncertain what portion of the cross section the value represents. Attempts to acquire a large number of samples from a different radial position of the same cross section are frustrated because of the relative horizontal motions involved. Surface waves and possibly internal waves in the pycnocline can also cause the sample to be obtained from a shallower or deeper cross section.

For these reasons field verification studies are best attempted for a cross section as far from the orifice as practical as long as the region is still within the range where the buoyant plume physics apply. Nearer to the orifice the values are changing more rapidly and the dimensions of the plume are much smaller, making it much harder to get the sampler in the right place, or even in the plume. In addition it is best to conduct the study when currents are low so that the plume rises nearest to the surface, shortening the interval between samples, as the sampling device need not be lowered so far. Placement of the sampling device may be improved because it may even be possible to see the plume. Aside from the use of the data for verification of the physics, samples taken during low currents may be especially useful for verification of regulatory compliance. Field verification data taken near the end of the initial dilution region can be compared with controlled laboratory simulations for similar conditions, and then, if necessary, the laboratory verification data can be relied upon for estimation of field values closer to the orifice.

ENTRAINMENT FROM OTHER SOURCES AND RE-ENTRAINMENT

Regulatory Background

In drafting modifications to the Federal Water Pollution Control Act (Anon., 1982), the United States Senate (Anon., 1983) proposed strengthening the authority of the Environmental Protection Agency (EPA) to deny waivers from secondary treatment for publicly owned treatment works (POTWs) discharging partially treated wastes into estuaries. Concern was expressed for re-entrainment of contaminants discharged previously from the POTW under consideration, and also for entrainment of contaminants discharged by other sources. Amendments to section 301(h) of the Act appearing in section 303 of the Water Quality Act (WQA) of 1987 (Anon., 1987) addressed these concerns:

> Section 301(h) is amended by striking out "such modified requirements will not interfere" and inserting in lieu thereof "....will not interfere, alone or in combination with pollutants from other sources..."

and further on:

> Section 301(h) is further amended by adding "....marine waters must exhibit characteristics assuring that water providing dilution does not contain significant amounts of previously discharged effluent from such treatment works."

These amendments suggested that EPA would need to revise the methods used to calculate compliance with water quality standards at and beyond the boundary of a mixing zone. Three topics needed to be addressed:

1. Definition of "significant amounts"
2. Entrainment of contaminants from other sources
3. Re-entrainment of contaminants from the proposed discharge

The water quality standard to be met is most easily assessed if it is expressed in terms of a concentration of a pollutant, i.e., a numerical criterion. For example, the California Ocean Plan (State Water Resources Control Board, 1988) contains such limitations, a few of which are listed in Table II, along with background seawater concentrations. The questions raised by the 1987 WQA amendments concern the proper value to use for the ambient (background) concentration for certain environmental settings, and how much is too much for a given discharge.

Table II. Concentrations of contaminants in coastal waters of California.

Contaminant	Allowable Instantaneous Maximum, C_{si}	Background Seawater Concentration
Arsenic	80 ug/l	3 ug/l
Mercury	0.4 ug/l	0.0005 ug/l
Silver	7 ug/l	0.16

Significant Amounts

The definition of significant amounts is easily resolved by use of mathematical models such as UM. That is, significance, in the sense of "importance", rather than a statistically computed value, is eloquently expressed in the test of compliance against a numerical standard in this model. If for a given setting Equation 9 provides values of c_{pi} that are lower than the values of c_{si}, then indeed the diluting water does not contain significant amounts of previously discharged effluent. Thus the question of how much re-entrained effluent is allowable is operationally defined with the types of models that were already in use in 1987, and at least for this purpose the 1987 revisions did not require a change in the models or their application. The major question is, "What is the proper value to use for each c_{ui}?"

Relationship of Ambient Dilution Water to Plume Concentrations

The following discussions is intended to show that the amount of effluent that is allowed to be re-entrained is a variable amount depending on the value of the standard, the amount of the contaminant in the effluent, and the volume of entrained diluting water. This can be seen by rearranging the terms of Equation 8 as follows:
The requirement that the plume concentration of contaminant be less than the standard for each contaminant can be expressed in the following inequality:
where c_{si} is the numerical value for the ith standard. Substituting the expression for c_{pi} from

$$c_{p_i} = \frac{c_{e_i}}{S_a} + \frac{c_{a_i}(S_a - 1)}{S_a} \tag{12}$$

$$c_{p_i} < c_{s_i} \tag{13}$$

Equation 12 into Equation 13

$$\frac{c_{e_i}}{S_a} + \frac{c_{a_i}(S_a - 1)}{S_a} < c_{s_i} \tag{14}$$

For cases where the dilution factor S_a is greater than 30 there would be less than a 3% error in the second term of Equation 14 by approximating $c_{ai}(S_a-1) / S_a$ by c_{ai}. Then Equation 14 can be written as:

$$\frac{c_{e_i}}{S_a} < c_{s_i} - c_{a_i} \tag{15}$$

Whether Equation 14 or Equation 15 is used, it is helpful to visualize the initial dilution requirement in this form for three reasons. First, it clearly shows that a certain standard may be met with different sets of values for c_{ei} and c_{ai}. For example, if one effluent has an ammonia nitrogen concentration of 120 mg/l and the local ambient is 3.9 mg/l, the California instantaneous allowable maximum of 6 mg/l would be met if an S_a of 60 were achieved. Another outfall, or the same outfall at a different time, achieving an S_a of 60 could meet the standard with an effluent value of 305 mg/l if the local ambient were 0.9 mg/l!.

Second, the value of the ambient concentration is seen to be of the same relative importance as the designated standard value in determining compliance. Thus if one locality has a standard one unit higher than another, but the ambient is also one higher, the necessary ratio of c_{ei} / S_a is the same. In other words, both dischargers have theoretically the identical options of reducing c_{ei} or building a more efficient diffuser or any favorable combination of these options. And if one locality has a standard one unit higher, and an ambient one unit lower, the discharger at this location would have to meet a less stringent ratio of c_{ei} / S_a, i.e., it is two units higher. This relationship is shown in Figure 13.

Third, notice that S_a is not subscripted with an "*i*" meaning that S_a is not dependent on the contaminant under consideration, as explained previously. It may be helpful to think of a "contaminant specific effective initial dilution" as the ratio of the concentration of a specific contaminant in the effluent to the concentration resulting after the volumetric process of critical initial dilution is achieved, i.e., c_{ei} / c_{pi}. By rearranging Equation 12 and again accepting an error no greater than 3% for dilution factors greater than 30, Equation 12 becomes:

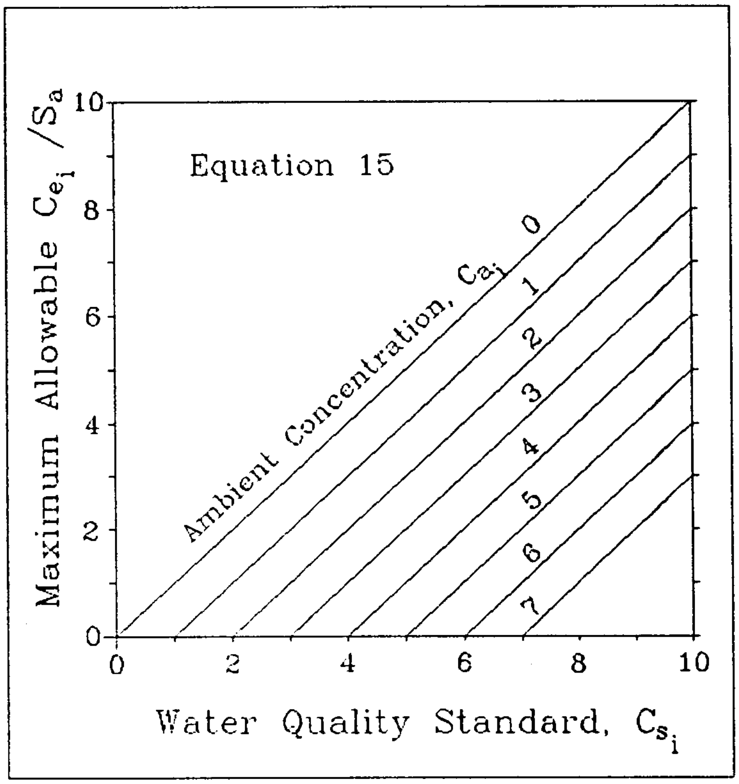


Figure 13. Maximum ratio of effluent concentration to S_a for standard compliance and dependence on ambient concentration.

$$\frac{c_{e_i}}{c_{p_i}} = \frac{1}{\frac{1}{S_a} + \frac{c_{a_i}}{c_{e_i}}} \tag{16}$$

Expressed in this way it is clear that the effective dilution of the specific contaminant, limited by regulation to less than a given numerical standard, depends on both S_a and the ratio c_{ai} / c_{ei}. Figure 13 graphically depicts that the ratio c_{ei} / c_{pi}, the contaminant specific effective initial dilution, is dramatically reduced below S_a as the ratio c_{ai} / c_{ei} increases.

This analysis has shown that the computational technique employed to test compliance with numerical water quality standards does take into consideration the entrainment of contaminants existing in the ambient dilution water. Thus the Senate revisions, contrary to first impressions, did not require a change in the EPA evaluation procedures to determine "significant amounts" of previously discharged effluents.

What is yet to be shown is how the value of c_{ai} may be determined or estimated to reflect the influence of other discharges nearby. The first requirement is for regulatory instructions to explain clearly that c_{ai} must accurately reflect the quality of the water entrained, i.e., the water adjacent to the diffuser, not the water at some remote, pristine location. Thus, for example, the "ambient" values in Table II are not likely to be generally useful, and may be inaccurate for California coastal discharges.

Entrainment From Other Sources

In the case of existing discharges it is not necessary to employ mathematical models to assess the amount of entrainment from other sources, and the amount of re-entrainment of previously discharged effluent, because field monitoring data will reflect the combined result of these factors. A priori assessment is needed in cases where a major change in effluent quality is proposed, or the outfall is to be modified or relocated, and models are useful for this purpose.

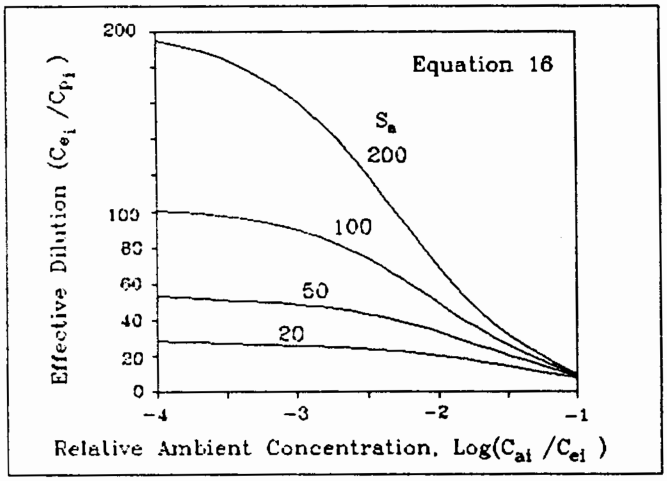


Figure 14. Effect of ambient concentration on effective dilution.

In the preceding sections it is shown that the effect of entrainment from other sources is properly incorporated in mathematical models such as UM as long as a proper data set for the ambient concentration of specific contaminants is used for input. Data available for an existing outfall may be useful for the relocated site if it is within the region covered by sampling stations, and sufficient vertical detail is provided in the data set. The presumption is that the new site or the modified outfall (e.g., longer or more ports) would provide better critical initial dilution. Since the data set would reflect both entrainment from other sources as well as re-entrainment of effluent, the data set would provide a conservative estimate. If the new site is outside the region sampled, new monitoring stations could be established and coastal circulation models could be employed to assess transport of pollutants from known sources in the region.

Entrainment into the plume of an outfall from other point and nonpoint sources is not generally a problem in the open ocean because of many factors. In most cases there is a large distance between point sources, providing ample opportunity for diluted waste to be dispersed and carried away from the region of entrainment of another outfall. Also, the volume of nonpoint sources of pollutants discharged directly to the ocean is small. Greater care is now given to locate modern ocean outfalls in well-flushed offshore environments rather than near shore. The volume of coastal waters available for dilution of point and nonpoint sources is great. For example, a 100 km section of coastal shelf out to a distance of 10 km with an average depth of 25 meters contains 25 x 10^9 m^3 of water, about 5000 times the daily effluent flow that might be generated by a municipality of 10 million people.

For a simple generalized case of contaminants transported from a source, for example another outfall, the concentration contributing to the ambient at the new site can be determined from Equation 17 (Brooks, 1960):

$$c_{p_i} = \frac{c_{max}}{erf\sqrt{\dfrac{U\ b^2}{16\ \epsilon_o\ X}}} \tag{17}$$

where

c_{pi} = Plume concentration at the end of initial dilution
c_{max} = Centerline (maximum) concentration at distance X
$erf()$ = Standard error function of ( )
U = Current speed in the X direction
b = Width in the Y direction (orthogonal to X)
ϵ_o = Constant Horizontal (Y direction) eddy diffusivity
X = Travel distance

The model UM automatically computes the farfield dilution according to this equation (labeled TSD Equation VI.21) based on data entered into the model interface by the user, and according to a similar equation based on an eddy diffusivity that increases longitudinally in proportion to the 4/3 power of the plume field width (TSD Equation VI.20). (TSD refers to EPA, 1982).

These dilution factors assume negligible contribution from contaminants in the ambient water, thus they must be reduced to represent the effective dilution at the down-current site. Figure 14 can be used for this purpose, substituting c_{pi} for c_{ei} in the abscissa term. These dilution factors are minimums, that is, a cross-field functional form such as a Gaussian curve should be used to estimate the cross sectional average. The total dilution is the product of S_a and the cross sectional average dilution of the drift flow. The model has not been proposed for analysis of diffuse sources or riverine inputs.

It must be recognized that the dispersing plume from one outfall will contaminate near surface waters while the principal source of entrainment for another plume is the deeper waters. Verified two-layer circulation models for the coastal segment under consideration may be useful to estimate the vertical exchange of contaminants as well as horizontal migration, thus providing an estimate of distant deep water quality.

Diffuse source inputs and episodic events are difficult to deal with in assessing the quality of ambient water expected to be entrained into new outfalls. During major storms that may occur as frequently as two or three times per year in the northeast and northwest, annually in the southeast, and perhaps once in ten years in the southwest, (1983 in the Los Angeles Bight, 1988 in Hawaii), storm runoff flushes riverine and estuarine contaminants into the coastal waters. Wind driven currents and waves re-suspend coastal sediments and distribute contaminants throughout the waters of the nearshore continental shelf, in many cases causing impairment of water quality entrained into ocean outfall plumes.

Mathematical models of coastal circulation may be able to predict dispersion of a given slug of contaminants washed out of an estuary up the coast from an outfall. Under storm conditions large dilution factors would be expected, however it is unlikely data are available to quantify contaminant levels in estuarine discharges. Direct land runoff and runoff from combined and storm sewers discharging directly to the ocean complicate both the analysis of transport and dispersion calculations as well as specification of contaminant levels.

Single-layer circulation models are likely to be inadequate in assessing runoff related effects. Depending on the concentration of dissolved and suspended materials, the bulk density of the runoff-contaminated coastal waters may be sufficiently low so that a short time after subsidence of the storm, deep denser offshore water will gradually move in toward shore and the turbid storm water will be carried in a thinner lens on or near the surface. Since a large percentage of the water entrained into the plume occurs at depth, there may be considerably less entrainment of contaminated storm water into the plume than would appear to be the case as one views the situation from the surface (or from the air). Mathematical models of coastal circulation may not be as useful for the period just on the heels of the storm event because of the difficulty in dealing with multi-layer flows in the high energy coastal environments. Because of the importance of entrainment at depth in achieving the proper degree of initial dilution before reaching the level of buoyant equilibrium, it is not appropriate to use a one-layer model which assumes the water column is completely well mixed under conditions of low currents.

During the storm event it is reasonable to expect that water quality values related to human use of the marine resource in the vicinity of the outfall might well be suspended de facto. For example, sport fishing and scuba diving are not likely to be engaged in near the outfall during a coastal storm. Consequently no harm is expected to be done to this use if effective dilution during the storm is impaired by entrainment of poor quality ambient water.

No references have been identified describing the behavior of marine organisms during storm events and their response to the mixture of effluent and runoff constituents. Their sensitivity must be considered irrespective of the suspension of human uses. There may be sufficient resiliency in coastal ecosystems so that short period perturbations can be accommodated. The incremental perturbation due to entrainment of runoff-contaminated ambient may be either small or large compared to average shelf conditions, depending on the circumstances of each event and each locality. It should be recognized, however, that even with entrainment of contaminated dilution water, the amount of dilution will be significantly increased over that predicted by conservative plume assessments specified by EPA due to the much greater energy dissipation occurring during storms. The net effect may be that organisms will experience a much lower concentration of pollutants during a storm than in the average case.

Given the concern over the inapplicability of models for the complex cases of shelf advection of pollutants in a variety of conditions, monitoring data may be the best option for estimating ambient quality under all conditions. In light of the generally poor water quality data base available in coastal shelf areas, if there is indeed a national priority for improvement of methods to estimate entrainment of other sources into extant outfall dilution fields, there is an opportunity to build a monitoring network that will serve a host of other highly important coastal resource issues. A report of a panel convened by the Marine Board (Eichbaum et al., 1990) contains recommendations for improvements in this area.

One important advantage of the use of field data to determine the quality of dilution water over the use of model simulations is that it is an operationally responsive approach. As new data are obtained, management options for control of the point source or the remote source, or both, can be balanced.

Re-entrainment from Existing Discharge

In addition to contamination of dilution water from other sources there are circumstances under which an existing discharge can re-entrain a portion of previously discharged effluent. However, the farther offshore an outfall is located the less this is likely to be a problem. Coastal currents and winds, which dominate replenishment of coastal waters with relatively clean offshore water, are not likely to be suppressed to the extent that flushing of diluted effluents is materially impeded for long periods of time. Under critical conditions of low wind and current, diluted effluents rise to the surface or to a level of buoyant equilibrium in the pycnocline. Water which is entrained between the discharge on the seabed and the spreading layer is not contaminated with previously discharged effluent due to the density stratification, thus C_a is not increasing with time. Tidal currents typically have a rotational character so that previously discharged effluent is carried some distance inshore on one reversal past the discharge point and offshore past the diffuser on the next reversal. Again, under stratified, low current conditions the effluent rises nearly to the surface or at least into the upper mixed layer. It does not remain at depth where the majority of entrainment takes place.

In shallow coastal settings where some outfalls historically had been placed, vertical turbulence is sufficient to reduce the degree of density stratification. If the discharge site happens to be between headlands the replenishment of shelf water by deep ocean water may be significantly restricted. In either of these settings partially diluted effluent can be returned to the deeper water levels and effective dilution can be substantially reduced. EPA has provided the model DECAL (Tetra Tech, 1987) to deal with this problem in a general coastal setting, i.e., not necessarily near shore, however it is restricted to cases where vertical turbulence is sufficient to cause complete vertical mixing near the outfall. Coastal circulation models and monitoring data as discussed in preceding sections may be used for these cases as well.

Relocation of the terminal end of an outfall to a site further offshore is frequently considered among the options to reduce environmental impacts of wastewater disposal. Another possible scenario for relocation of an outfall is lateral displacement upcoast or downcoast from the present location at about the same distance offshore. The rationale might be to minimize distance to the location of a new treatment plant, or any number of water and sediment quality considerations. If topographic and bathymetric features are similar at the former and proposed site, the circulation features will be similar. Re-entrainment could then be estimated taking into account any differences associated with the characteristics of the new diffuser. Monitoring data on conditions around the outfall to be replaced would be useful in estimating the degree of re-entrainment.

Entrainment and Re-entrainment in Estuarine Discharges

The above discussion focuses on open ocean conditions. For estuarine discharges the use of Equation 17 may not be appropriate as advection and turbulent mixing is not so conveniently described by this simple model. Monitoring data and estuarine circulation models may be useful, although point and diffuse sources may not be well characterized.

Compared to waste discharges along a stretch of open coastline, discharge of effluents into an estuary almost surely guarantees recirculation to other points in the system, and the entrainment of effluents from other sources into the plume generated by the outfall in question. Estuarine water quality analysis techniques have improved steadily since an EPA resource management assessment was made in 1971 (Ward and Espey, 1971). The assessment of research needs to support a national estuarine research strategy (Menzie and Associates, 1986) cites examples of additional model development that is still needed, but the state of the art is sufficient already for many management purposes. It is possible to adapt available models to many if not most estuarine problems and to conduct simulations with computers available to every modern regulatory program.

EPA maintains an estuarine modeling repertoire and provides computer programs and documentation manuals to potential users. These can be used to estimate the steady state concentration of contaminants at a variety of sites in the estuary given the mass loadings and input locations. Some models may be able to simulate varying concentrations of pollutants within a period of critical conditions such as portions of a tidal cycle. As water quality criteria become sophisticated enough to address short time variations the demand for detailed data on time varying mass inputs will begin to limit the utility of the models. Simulations conducted for all source inputs except the extant outfall, compared to simulated water quality in the absence of inputs, will show the effect of "other sources" on the quality of water entrained in the outfall.

Monitoring data would be useful for verification of the modeling results except for the fact that monitoring data will include the contribution from the extant outfall. For example, if several of the other sources contribute nitrogen, monitoring data could not partition the estuary-wide distribution of nitrogen since a municipal outfall also contributes nitrogen. It would be rare, and extremely valuable, if baseline monitoring data were available over long enough periods of time to provide some verification of the pristine case.

Use of an Intrinsic Tracer

There is a possibility, though unlikely, that a surrogate approach to partitioning of contemporary monitoring data may be useful. If the effluent were the unique (ambient effectively zero) source of any water quality constituent whose physical, chemical and biological fate mechanisms were known, or could reasonably be assumed to be inconsequential, the distribution of this tracer throughout the estuary could serve as a proportional marker for any other constituent in the outfall.

Thus if the tracer was at concentration 10 in the outfall and contaminant "X" was at concentration 4, then at some point in the estuary where the concentration of tracer was found to be 0.1 and the concentration of "X" was found to be 3, the amount of "X" from other sources could be found by solving Equation 9 first for S_a using the tracer data and then solving Equation 9 for c_a using the contaminant data and the value found for S_a. Of course the behavior of the surrogate and the contaminant "X" must be the same or adjustments to the correction have to be made to account for any differences in coagulation, adsorption, decay etc. While easily stated, this environmental behavior question may limit the practical use of the approach. No literature citations have been found that report use of this technique although in a practical sense it is of the same form of approach as injecting dye or some other tracer to determine the estuarine distribution of outfall constituents generally.

Salinity as a Surrogate Effluent Tracer

Under some specialized situations the distribution of salinity, which is more easily verified than nonconservative pollutants, can be an effective surrogate for a nominal effluent constituent in the water column. The simplest case is when an effluent is proposed to be discharged near the major freshwater inflow to the estuary.

In the case of a discharge near the entrance, salinity may be an approximate surrogate only if the wastewater flow is very much smaller than the incoming seawater volumetric flux during periods of small tidal exchange.

Unfortunately, neither case deals with the question of environmental fate factors (adsorption, speciation, decay), and surrogate values based on salinity have to be modified to account for evaporation, direct rainfall, and other influences on the salinity value. Nor are salinity distribution patterns useful for estimating particulate sedimentation values, which may be the most important consideration because the 301(h) modified permit usually results in greater suspended solids emissions than would be achieved with full secondary treatment.

FRESHWATER DISCHARGES OF BUOYANT EFFLUENTS

The buoyant plume problems of major interest to scientists and regulators have typically involved the discharge of lighter material into a denser environment, such as a smoke plume in the atmosphere or freshwater sewage effluent discharged into the marine environment. The models developed for these cases are also able to handle the discharge of heated water into a colder lake because of the slight density difference associated with temperature differences.

The models may be employed in some riverine situations as well as in lakes. That is, if the effluent is warmer than the river and is discharged at depth, the effluent would be expected to behave as a buoyant plume. The relative size of the diffuser ports in relation to the depth of the river may be important in achieving the dilution factors predicted by the models. Muellenhoff et al. (1985) recommended the depth be greater than ten times the port diameter, although there is no strong experimental or observational basis for this rule. Rather it is based on the knowledge that plume models were developed for deep water discharges and modelers are not confident in extrapolating verification data from deep water situations to shallow water applications.

For riverine situations in which the effluent is discharged through a multiport diffuser placed along the stream bed in the direction of flow rather than across the current, only the RSB (line source) model in this report may be applicable for analysis of the dilution field.

Industrial wastes discharged to rivers or lakes may have bulk densities greater than the receiving water due to high concentrations of dissolved contaminants. But if an effluent is substantially warmer than the lake or river the net result might be a lesser density and a positively buoyant plume would develop from a discharge at depth. However, modelers should be aware of the nonconservative nature of heat in describing the density of an effluent at the discharge point. The wastewater temperature at the diffuser port may be significantly lower than at the treatment plant due to heat lost as the effluent runs through an underground and underwater sewer.

Because most rivers will not have density gradients it is likely that warm water plumes will reach the surface of the receiving stream, and the surface plume will be subject to heat exchange with the atmosphere. The models in this guide do not incorporate atmospheric heat transfer functions so that any temperature output generated after the water surface is encountered must be accepted with caution. For short time periods atmospheric heat exchange will not make a large difference.

The subjects of subsurface and surface discharges of large heated effluent flows as for example from thermal electric power plants are treated in many reports.

The special phenomenon of nascent dense plumes, initially buoyant thermal plumes discharged into near-freezing freshwater, which rise briefly before becoming dense and sinking to the bottom are discussed in the next section.

NEGATIVELY BUOYANT PLUMES

Many industrial wastes whether discharged to fresh or marine waters have sufficient dissolved or suspended solids concentrations so that the bulk density is greater than the receiving waters into which they are discharged. The cases can include wastes discharged horizontally or at an angle (including 90 degrees) downward from the surface or upward from the seabed. Simple plume models such as UPLUME (Muellenhoff et al., 1985) have been used to fashion a surrogate solution to the problem of predicting trajectories and dilution factors for vertical discharges of negatively buoyant wastes. This has been accomplished by recasting the problem in terms of an analogous positively buoyant case.

It may help the reader to appreciate this approach by pointing out that many laboratory experimental data sets, and photographs, of positively buoyant plumes rising from the bottom of a simulated stably stratified ocean are in fact results from a negatively buoyant plume discharged from the surface, sinking toward the bottom! The laboratory experiment is set up this way for the physical convenience of the modelers. The photographs are typically presented in published reports upside down so that they visually depict the conceptual problem being addressed. The proper analogy is effected by due regard to the density differences between the plume elements and the local ambient so that the forces acting on the plume element are the same regardless of the direction of motion. Thus a freshwater plume rising from the seabed is simulated physically by a heavy liquid sinking in a lighter fluid. The mathematical simulation is analogous, and the printout from the computer program is an equivalent, surrogate solution.

An example of the above approach is the simulation of dilution factors computed for near surface, downward discharge of drilling fluids into a marine ambient by Ozretich and Baumgartner (1990). In this example the mathematical models PLUME, OUTPLM, and DKHPLM, which would accept only positively buoyant discharges directed up from the seabed, were provided input for a surrogate freshwater discharge into an ambient having an initial density difference and a density gradient equal and opposite to the prototype situation. The mathematically simulated results were comparable to data from a physical model of heavy fluids discharged downward from the surface, i.e., exactly as in the prototype.

Extrapolation of the usual plume model results to cases of very large solids concentrations, and slurries or solutions with very high specific gravities compared to the ambient fluid may violate the Boussinesq approximation which is generally assumed. This assumption, incorporated in plume models to simplify calculations, requires that density differences between the plume and the ambient must be small compared to the density of the fluid. For example, the specific gravity difference between sewage and seawater compared to seawater is approximately 0.02. Sewage sludge is about the same, whereas drilling fluids used in offshore oil exploration could have a ratio of as high as 0.5! Clearly 0.5 is not a small difference compared to 0.02, but there has not been a rigorous examination of the importance of the Boussinesq assumption in plume modeling, or for that matter what a useful criterion is for judging "small." Morton (1959) pointed out that density differences are rapidly dissipated within a short distance from the orifice, suggesting that violation of the Boussinesq approximation is not very serious for the major flow region. Fluid modeling studies by Roberts (1977), and by Roberts, Snyder, and Baumgartner (1989 a, b, c) show no effect of the ratio over a wide range.

In the hydraulic model studies of drilling fluids reported by Ozretich and Baumgartner (1990), drilling muds with specific gravities as high as 2.17 were adequately modeled by the model PLUME (Teeter and Baumgartner, 1979) as judged by comparison to measured depth of penetration to the level of buoyant equilibrium. The ratio of predicted to observed depths averaged 0.93 (S_e = 0.03) for 27 trials.

The model UM described in this report will accept direct input matching all physically observed positively or negatively buoyant plumes discharged at any angle from either the surface or the seabed. Furthermore it does not depend on the Boussinesq assumption. Other models accessed through the UM interface may or may not produce output for certain negatively buoyant cases, and output which appears complete for other than positively buoyant plumes discharged from the seabed must be considered carefully by the user.

Nascent Density: Thermal Discharges to Cold Water

A special class of negatively buoyant plumes are nascent dense plumes, plumes which begin as buoyant plumes but reverse buoyancy, becoming dense and sinking to the bottom or to some more deeply submerged trapping level. The best known examples are thermal freshwater plumes discharged to freezing ambient freshwater (Frick and Winiarski, 1978; Frick, 1980). The behavior, which can also occur in brackish water up to a salinity of approximately 14 o/oo, occurs because the plume, as its temperature cools by mixing with water near the freezing point, becomes denser than the ambient because the maximum density of freshwater is around 4 C. Thus, if the temperature of the ambient is less than 4 C, the potential for the nascent dense plume phenomenon exists.

The non-linear equation of state used in UM may be used to model nascent dense plumes, as explained in the chapter entitled: "A CORMIX1 comparison, density, stability, and profiles".

PARTICULATE DISCHARGES

Particulates in fluid discharges may vary from 10 ppm in municipal secondary effluent to over 100,000 ppm in drilling fluids. The mass of solids may contribute to the bulk density of the fluid, influencing the transient behavior of the plume and its equilibrium position. For municipal effluents this contribution is neglected because of the low concentration of particulates.

Simple plume models (e.g., UPLUME) have also been used to analyze the behavior of municipal sewage sludge in relation to alternative discharge methods such as pumping from barges. Comparison of the mathematically simulated results to small scale hydraulic models results demonstrated that sewage sludges containing between 2 to 6% suspended solids have essentially the same properties as aqueous solutions of the same bulk densities. As the buoyant equilibrium level is reached in a density stratified ambient fluid the particulates begin to separate

Figure 15. Separation of plume and flocculating particulates.

from the diluted sewage field, some rising, some settling, with or without flocculation. See Figure 15.

The physics of plume models does not attempt to describe the behavior of particulates within the buoyant plume region or following equilibrium, except to the extent they behave as part of the fluid continuum. Models are available (USEPA, 1987, Bodeen, et al., 1989) to simulate the dispersion and settling of sewage effluent particulates based on pioneering work of Hendricks (1982, 1983) in the Southern California Bight. These models may be applicable for analysis of other types of particulates. It should be borne in mind that the equations of state used in UM, RSB, and CORMIX are not necessarily appropriate for the fluids at hand. (Some additional amplification on this point is found in the section entitled: "Example: A CORMIX1 Comparison, Density, Stability, and Profiles.")

It may be possible to influence the behavior of particles in relation to the physics of sedimentation by adjusting the discharge conditions at the diffuser port, especially the exit speed. High exit speed may break up agglomerated particles causing them to behave as discrete particles at the equilibrium level. Low exit speeds may preserve the integrity of agglomerated particles and enhance the flocculation of others prior to arrival at the equilibrium level. This is a separate area of research beginning to be questioned. Attention so far has been focused primarily on the interactions of particulates following the transition from plume mixing to ambient turbulent transport (Hunt, 1990). Whether or not discrete or agglomerated particle are the more environmentally benign form has not been rigorously established, although a task force report of the Marine Board suggests dispersal is preferred to seabed accumulation (NRC, 1986). This recommendation is based on broad physical considerations rather than detailed ecological considerations which may be preemptory.

2. USER´S GUIDE TO THE PLUME MODEL INTERFACE, "PLUMES"

USER'S GUIDE TO THE PLUME MODEL INTERFACE, "PLUMES"

INTRODUCTION

UM and RSB are mathematical models for analyzing and predicting the performance and behavior of plumes formed by fluids discharged from orifices and diffusers into aquatic environments. RSB and UM are offered as options to users of the PLUMES interface, a master program system that the user manipulates. PLUMES also supports UPLUME and ULINE (Muellenhoff et al., 1985) by providing a way to create UDF files from PLUMES input data. An essential element of the interface is the data entry scoreboard-appearing interface, something akin to an annotated spreadsheet in Lotus, that allows you to describe effluent parameters, environmental conditions, diffuser design features, and computer controls.

Several separate programs comprise the system: the initial dilution models UM and RSB, the Brooks farfield dispersion equations, the flow classification algorithm of CORMIX1, the interface PLUMES, and a program for estimating the hydraulic characteristics of the outfall, PLUMEHYD. All except PLUMEHYD are integrated into the PLUMES interface to work together and help reduce the amount of time required to analyze various plume problems, or cases. With the resident CORMIX flow classification algorithm, PLUMES can also offer recommendations on model usage that go beyond the built-in models found in the interface, including EPA CORMIX1 (Doneker and Jirka, 1990), CORMIX2 (Akar and Jirka, 1990), and CORMIX3 (Jones, 1990): for single port discharges, diffusers, and surface discharges respectively. More comprehensive recommendations on model usage are provided in Appendix 1. The ultimate goal is to make it easier to explore options, conduct sensitivity analyses, and generally produce more in-depth project reviews, designs, or assessments in the limited amount of time available to you. The interface provides limited control over output format to help in writing reports.

Guidance in using UPLUME and ULINE may be found in Muellenhoff et al. (1985). PLUMEHYD is described in Appendix 3.

The interface has several main structures to activate the various PLUMES functions and resident models:

- the case (or record)
- cells
- pop-up menus
- dialogue windows
- help windows
- configuration string

In addition, various specialized built-in features are included to support the analytical process. Perhaps the most unique specialized capability is the conflict resolution feature which allows many ways of defining the problem, i.e. entering different sets of variables, and, consequently, must be able to detect instances of conflict when they occur and help to remedy them. The following tutorial chapter demonstrates the conflict resolution mode.

Another feature is a units conversion capability which helps minimize the need for a calculator.

The structure, commands, special capabilities, and the models work together to help you analyze initial dilution, mixing zone, and farfield dispersion problems. The level of refinement available in each of these zones varies considerably, being very high in the near field and becoming progressively simpler in the farfield.

PLUMES STRUCTURE

When PLUMES is started, introductory information is displayed which must be acknowledged by pressing any key. Once acknowledged, the main screen, often referred to as the **interface level** or simply the interface, appears. An example of the interface is given in Figure 16. The screen represents a single problem, or case, which, as the information in the upper right corner implies, could be just one record in a file of many cases.

To help enhance the readability of the interface, the display is in color.

```
 Jun 19, 1992,  11:35: 6  ERL-N PROGRAM PLUMES, Jun 10, 1992   Case:   2 of
 2
 Title   Sand Island validation: no blockage

  tot flow   # ports port flow   spacing  effl sal effl temp   far inc   far
dis
     4.469        285   0.01568      7.315       0.0         25        500
2000
  port dep  port dia plume dia total vel horiz vel vertl vel asp coeff print
frq
      70.1      0.085   0.08500      2.763      2.763     0.000      0.10
500
 port elev ver angle cont coef  effl den poll conc     decay  Froude # Roberts
F
      0.84        0.0         1.0     -2.893        100      55.26      18.40
2.044E-14
 hor angle red space p amb den p current   far dif   far vel K:vel/cur Stratif
#
        90        7.315        24.080.00001000   0.000453             0.15
2763000.00004871
     depth    current    density  salinity       temp  amb conc  N (freq) red
grav.
```

Figure 16. The PLUMES main screen, or interface level. (Software version is in color.)

The greater part of the interface is occupied by **cells**. In general, each cell has a short **label** and a space beneath it for **numeric data** (the value of the mathematical variable). The title cell, occupying the second line, is longer and is suited to alphanumeric input. In the main body of ambient cells used to define conditions in the receiving water, stacked cells share common labels.

The cells are organized into blocks of different colors. Hypothetically, outfall structure variable labels are on magenta background; effluent characteristics, brown; miscellaneous variables, gray; ambient variables, green; and specialized information, red. The actual colors depend on the brand and settings of the monitor in use. There is also a **pause cell** (here headed, unless it has been changed by you, by "hor dis>="), near the lower right hand corner of the interface, which may be used with UM to control output of information under specified conditions. The color of the numeric information in the cells is either displayed in yellow or in white, depending on whether the information was entered manually (or selected from a default value) or was computed by PLUMES. Yellow variables are independent variables; white ones are dependent. **Only some of the cells, which are selected to suit the problem (independent cells), need to be specified -- PLUMES computes the rest (dependent cells).** This flexibility makes it possible to define problems in a variety of ways.

At the top of the interface is a clock, the PLUMES version identification, and the case counter. At the bottom are three lines of data: the first is reserved for the **CORMIX flow classification** predictions and modeling recommendations, the second is the **dialogue line**, and the third contains basic help information, **program configuration identification**, and the name of the file of cases in use.

The dialogue line may be passive, displaying useful information that is relevant at various stages, or it can be active, awaiting instructions to continue. Sometimes you are alerted to new information in the window by sound. An example of a passive message explaining how to use the menus when the F1 key is pressed is shown in Figure 17. When action is required, the

```
Hit bolded letter or arrow keys and <CR>; use control sequences for speed
```

Figure 17. An example of the dialogue line.

options, typically consisting of keystrokes, will be displayed or a cell will be provided for inputting string information, such as a file name. The latter often display a default string which may be accepted or simply typed over, referred to as typeover input. At least one command, <fill New file>, uses its own special dialogue windows.

Except for some of the editing commands which are described only in the Miscellaneous Editing Commands section, the commands **can be selected from several menus**, the main one of which is shown in Figure 18 as it appears as a window on your screen. The menus are

```
 Jun 11, 1992,  19:57:49  ERL-N PROGRAM PLUMES, Jun 10, 1992   Case:   1 of
1
 Title   Sand Island validation: (no blockage) TRR case.

  tot flo┌───── Main menu ─────┐ spacing  effl sal effl temp   far inc   far
dis
    4.46│ run rsB program     │   7.315       0.0         25         500
2000
 plume de│ run Um program     │otal vel horiz vel vertl vel asp coeff print
frq
     70.│ show Independents   │   2.763      2.763      0.000       0.10
 500
 port ele│ units Konversion    │effl den poll conc       decay  Froude #
Roberts F
     0.8│ List equations      │  -2.893      6.1e8                     18.40
2.044E-14
 hor angl│ get Work file       │ current    far dif    far vel K:vel/cur
```

Figure 18. The main pop-up menu superimposed on the PLUMES interface.

provided mainly as a memory aid, and, in general, it is faster to use the keystroke form of the commands at the interface level. The ▸ symbol after some of the commands on the main menu indicates the presence of sub-menus. The implementation of the menus is explained below.

```
Help for variable:  den =  effl den  sigmat


    Effluent density.  When calculated from temperature and salinity,
    the salinity is assumed to have the composition of sea salt.  If the
    density is independent, a linear equation of state is assumed (see
    Example 2 in the manual for more detail).


Equations and variable definitions:
 den = (dena+1000)/(1.0 +vel*vel/(g*dia*abs(Fr)*Fr) - 1000
       { note single use of abs to retain sign }
     = (dena+1000)/(1.0+gp/g)-1000
     = dena -SP*(dena-dal)*dia/pdep
     = sigmat(s,t).
```

Figure 19. Example of a "cell definition window." The help window for the plume density cell.

The most pervasive help screens are the **cell definition windows**. These come up by issuing the **<List equations>** (^L) command on the main menu. The information provided is specific to the cell identified by the cursor and has one of two forms. An abbreviated form is used when the file EQNS is not in the current directory; it consists only of a definition of the cell and descriptive notes. With the file in the current directory, a second form adds the equations that are used by PLUMES to define dependent (white) variables. Figure 19 gives an example of the latter form, showing the terms involved in computing density by several methods. If file EQNS is not in the current directory because it was not copied or was deleted, it may be restored from the original disk.

The Configuration string appears in the middle of the bottom line of the interface. Each character in the string is a mnemonic for different program attributes. Changing the string will cause the program to work in one of several fundamentally different ways. For example, the "O" in "ATNOO" in Figure 16 indicates that the plume model UM, under overall control of the PLUMES interface, will terminate the initial dilution phase (near-field) if and when the mathematical condition of element overlap is attained.

The Configuration string can be different for each case in the case file.

INTERFACE CAPABILITIES

It is easy to be unaware of some of the special capabilities available in PLUMES because all are not controlled directly. However, understanding them will enhance the use of the system. The more notable ones are described below.

- an unstructured data input environment
- a conflict resolution mode for resolving many over-specified input conditions
- a configuration file
- selection from multiple solutions to governing equations
- display based on significant digits

Perhaps the most outstanding feature of the interface is its **unstructured data input environment**. The user is free to move about, skipping over cells, just as in a spreadsheet program. This facilitates "what if" inquiries.

The unstructured environment would not have much purpose if all the cells had to be filled in anyway. But, in fact, only some of the cells need ever be filled. The reason is that PLUMES provides redundant variables as a convenience. For example, there are cells for the total flow, number of ports, and port flow. Since it is assumed that all ports have equal flow, only the first two cells are necessary to specify the port flow. (Given that they are specified, the port flow should not have to be input, in fact, it would be potentially incorrect to do so because the value could be inconsistent with the total flow, which, as is explained below, would be brought to your attention by the conflict resolution algorithm.) In this case the total flow and number of ports are displayed as independent variables, i.e. in yellow, while the calculated (dependent) port flow cell is displayed in white.

For even more flexibility data can be entered into cells already containing information. If the superseded value was yellow (independent), the affected dependent (white) cells are simply recalculated. However, things are more complicated when a white cell is superseded with information you enter. In this case the overspecification alluded to above will, in general, cause the data set to be inconsistent. In the above example, the product of the port flow and the number of ports would no longer equal the total flow. **PLUMES detects the inconsistencies**[1] **and goes into a conflict resolution mode** in which you select (space bar to move to the selected variable, followed by "D" or delete keys) which variable is to be calculated (dependent). This capability facilitates sensitivity analyses.

PLUMES maintains a **configuration file** called **SETUP**, an ASCII file that is created if it is not present in the current directory. It stores information on the last use, including the location of the cursor, and the variables on the output table list. PLUMES attempts to find and read the file each time it is run.

Some of the equations used to define dependent cells in the interface have more than one solution. A good example is density as a function of temperature and salinity. It is well known that the greatest density for fresh water at standard temperature and pressure is around 4 C. Thus, there is a range of densities smaller than the maximum density with which temperatures less than 4 C and other temperatures greater than 4 C are compatible. Whenever this occurs, **PLUMES provides for the selection of the desired solution from the multiple solutions to governing equations**. The same occurs when the dependent variable is the solution to a square root, in which case the proper root, either positive or negative, must be selected.

The interface **displays numbers to 3 or 4 significant digits**. This capability assures that information is not lost due to formatting deficiencies. Numbers that cannot be displayed to the proper precision within the allotted space are converted to the "E" format of scientific notation, e.g. 1.4×10^{-8} is displayed as 1.4E-8. The "E" format may also be used to enter data. The **<cell Precision>** command may be used to show extra precision.

COMMANDS

Conventions

Control over the interface is exercised through a system of commands which may be issued at any time. The **commands are listed on a series of menus** and can be implemented by bringing up a menu or by striking the control key and the appropriate letter key. The former

[1] Strictly speaking, instances of over-specification are detected only when at least one of the defining variables of the offending cell is independent. However, a special command is available for checking the consistency of all variables, irrespective of their independent/dependent lineage.

is convenient for remembering the commands while the latter is faster. Thus, the "run rsB program" command, which is listed on the Main menu, can be issued at the interface level by simply holding down the control key and then pressing the letter B, also denoted by ^B. (The case of the letter is unimportant, i.e., ^B = ^b.)

There is only one way to access the Main menu, which is not to say the commands, directly and that is with the "F1" key. From the main menu the commands, which include bringing up the sub-menus, may be issued by hitting the chosen highlighted key, or, using the arrow keys to move to the chosen command and selecting it with the enter (carriage return) key or the space bar.

In the catalogue of commands to follow commands will be enclosed by < > brackets, to indicate they are keystrokes. Thus <run rsB program> or <B> are equivalent. For commands issued directly from the interface level without going through <F1>, sequences are harder to represent, for example, <control> <B> does not convey very well the fact that the keys are to be depressed simultaneously. For such cases the notation ^B is more useful and will be used extensively. For sub-menus, the chosen highlighted letter can be added to the key sequence. For example, to use the <cHeck consistency> command on the Miscellany menu from the interface press ^Y followed by ^H or <H>; this sequence is summarized as ^YH. If a command is issued which is invalid in context, PLUMES will send a reminder to the dialogue window. The commands are case insensitive.

In the following catalogue of commands, the name of the command as it appears on the menus is given, followed by the interface level keystroke command sequence and a brief description of the command itself.

The Main Menu

The Main menu (Help, <F1>) is shown in Figure 20.

```
        Main     menu
 | run rsB program
 | run Um program
 | show Independents
 | units Konversion
 | List equations
 | get Work file
```

Figure 20. The Main menu.

<run rsB program>, ^B:
Instructs PLUMES to run RSB, (Roberts, Snyder, Baumgartner, 1989 a,b,c). Subsequent dialogue window prompts ask you to specify the number of cases to run and the destination of the simulations. The console or monitor can be specified by typing the word "console" (without the quotes) or simply press space bar if the console is the default output device. Or type "prn" for printer or any legal DOS file name.

<run Um program>, ^U:
Run the UM model. Subsequent dialogue window prompts ask you to specify the number of cases to run

and the destination of the simulations (console, printer, or disk file). (See explanation of the ^B command above.)

<show Independents>, ^I:

Typically, PLUMES has several equations available for defining dependent cells. ^I examines each of these and, in turn, identifies all the potential defining variable sets for the cell in which the cursor is located. The cell's independent variables are revealed by black hatching of the background color of the cells' labels. ^I is useful for establishing which data (cells) will define the cell at the cursor for which data may be unavailable. (IMPORTANT: the variables in the defining set can themselves be either independent or dependent.)

<units Konversion>, ^K:

Allows you to change the input units of a cell to one of the units shown in the dialogue window. After the desired unit appears in the dialogue window, you may input the value in its native units. Upon leaving the cell the value is automatically converted to the system units (primarily SI, i.e. kg, m, sec, C). Subsequently, the conversions will appear in the dialogue window whenever the cursor is moved back into the cell.

<List equations>, ^L:

Provides a definition of the present cell. The header name is displayed at the top of the screen in the cell's interface color. An example is given in Figure 19. If the file EQNS is in the current directory, the set of equations that define the cell, together with variable explanation, is also provided.

<get Work file>, ^W:

Used to specify a new working file of records, or cases. A typeover window is provided for file name input. The < ↓ > key may be used to cycle through existing .VAR filenames in the present directory. The existing active file is stored and the new file is opened. The new file name replaces the old one at the bottom of the interface after the word "FILE". If the file does not exist it is created and filled with default data. The length of the new file is checked to help ascertain that the appropriate format exists.

<fill New file>, ^N:

Directs the interface to create a new file of records from the current file of records. You are asked for a new file name (existing files are rejected). The filename extension, .VAR, is recommended (see the <get Work file> command above). You must specify which records are to be copied to the new file. The numbers of the cases must be separated by blanks (spaces, not commas) but may be in any order. Sequential cases may be specified by connecting their beginning and end members with "..", e.g. the sequence 5 3..7 1 causes the cases 5, 3, 4, 5, 6, 7, and 1, in that order, to be copied to the new file. The command is useful for reorganizing your case files.

<add to Output>, ^O:

For the UM model ^O allows cells to be added to the list of cells that are displayed as output. Affected cells are highlighted by a blue rectangle in the first character of the cell label. Certain auxilliary variables like centerline dilution may be subjected to the ^O command by first using the ^YS command on the Miscellany menu.

<cell Precision>, ^P:

Increases the precision to which the cell value is expressed. (Up to six significant digits.)

<configuRe models ▸ >, ^Rx:

Displays the Configuration menu. The "x" indicates another key is to follow. If ^R is pressed at the interface level, the Configuration menu will appear after a timed delay if the "x" has not followed in the allotted time.

<moVement commands ▸ >, ^Vx:

Displays the Movement menu. The "x" indicates another key is to follow. If ^V is pressed at the interface level, the Movement menu will appear after a timed delay if the "x" has not followed in the allotted time. Some mnemonics of some of the editorial commands are also displayed.

<miscellanY menu ▸ >, ^Yx:

Displays the Miscellany menu. The "x" indicates another key is to follow. If ^Y is pressed at the interface level, the Miscellany menu will appear after a timed delay if the "x" has not followed in the allotted time.

<esc>

The null command. Returns the interface level.

The Configuration Menu

The configuration describes one of several possible running modes for the interface, UM, and RSB. The settings are identified in capital letters and numbers after under the word "configuration". Defaults are provided if the file SETUP is missing, otherwise they are taken from SETUP. The menu is shown in Figure 21. Unlike other menus which disappear after a command is selected, the configuration menu remains on the screen until <esc> is hit, allowing the entire configuration string to be edited in one pass. It can be accessed with <F1> followed by <R>, or ^R. Once the commands are known, it is more convenient to use the command sequences given below.

```
Configuration Menu
Auto ambient
Brooks eqn input
Cormixl categories
Farfield start
Reversal set
Show configuration
<esc>
```

Figure 21. The Configuration menu.

<Auto ambient>, ^RA:

Possible settings are A (on) and N (off) in the first character of the configuration string, e.g. ATNO0 or NTNO0. In the ambient block starting with the line below the surface ambient line, while moving from cell to cell, Auto ambient (on) will fill the cell with the value immediately above it if that value is independent (yellow). This is a convenient way of filling out the ambient block when many of the values are similar. The provided values can be edited in the usual ways.

<Brooks eqn input>, ^RB:

The command toggles between two options. Possible settings of T or R are identified by the second character in the configuration string at the bottom of the interface level. The R setting (reset), e.g. NRCO0, indicates that PLUMES will prompt you to approve or change the inputs (wastefield width and origin distance). This allows you to essentially run the farfield model independently of the initial dilution models. It is, for example, one way to reestablish the values tabulated by Tetra Tech (1982). The T setting (transmitted) will establish the initial dilution model results as the farfield model inputs. The default value is T (transmitted).

<Cormix1 categories>, ^RC:

The command toggles between two options. Possible settings of C or N are identified by the third character in the configuration string at the bottom of the interface level. The C setting, e.g. NTCO0, indicates that PLUMES will attempt to define CORMIX1 flow class corresponding to the input conditions. Recommendations for model usage are also presented. The N setting, e.g. NTNO0, specifies that no classification is attempted. The default mode is N.

<Farfield start>, ^RF:

Used to configure the UM model, it is identified by the second character in the configuration string at the bottom of the interface level. This command specifies at which point the farfield dispersion model is initiated following the use of UM in the

```
Start far-field at Max-rise, Overlap, or Pause criterion?
```

Figure 22. Farfield configuration options.

initial dilution phase, i.e. in the near field. When the command is issued the prompt shown in Figure 22 appears in the dialogue window. Using the M, or Max-rise option, e.g. AMNO0, the initial dilution phase is terminated when the plume reaches maximum rise (or the surface), after which the farfield model is initiated. The default value is O (Overlap), e.g. AONO0, which specifies the farfield model begins when the plume element can no longer be consistently defined due to geometric constraints (Frick, Fox, and Baumgartner, 1991). This condition, sufficiently pronounced, has been associated with upstream anvil formation (Frick et al., 1990). The P (Pause criterion) option, e.g. APNO0, initiates the farfield model when the condition in the pause cell is met.

<Reversals set>, ^RR:

Plumes rising in stratified receiving waters frequently trap at some intermediate level, a level of zero net buoyancy, in the water column. Generally, plumes will traverse, or overshoot, this level and perform wavelike motion because they still have vertical momentum. Thus, above and below the trapping level the buoyancy will switch from positive to negative or vice versa. This reversal in buoyancy will ultimately slow the vertical motion to a standstill before reversing again. Each reversal point is a crest or trough of the wave.

The <Reversals set> command specifies how many extrema are to be modeled before the farfield model takes control. If the number of reversals (the last character in the configuration string) is set to zero, e.g. AONO0, PLUMES will determine the number of reversals to be one, 1, for buoyant plumes and two, 2, for negatively buoyant plumes. The reason for this option is that normally rising plumes usually entrain much more vigorously between discharge and maximum rise than they do in the farfield, thus the initial dilution region is confined to the region between discharge and the first reversal (i.e. maximum rise). Negatively buoyant discharges are frequently discharged upwards and pass through maximum rise before their turbulence is dissipated, hence it is appropriate to continue relatively active entrainment through the subsequent sinking region. In any case, by specifying a nonzero integer between 1 and 9, the user can specify the number of oscillations which will be modeled. The 0 value is generally recommended but may be altered for the sake of being conservative or for special considerations.

<Show configuration>, ^RS:

Interprets the configuration string shown at the bottom of the interface screen. The other

```
                        PLUMES Configuration

A: Automatic ambient fill is on
T: Brooks equation input transmitted
N: The CORMIX flow categorization algorithm is inactive
O: UM far-field predictions begin at element overlap
   Farfield model initiation choices are:
   M: maximum rise; O: element overlap; P: pause criterion.
   Other criteria, such as surface interaction, will override these choices.


0: Brunt-Vaisala reversals determined by UM as 1 or 2
```

Figure 23. The Show configuration command window.

options are also summarized. An example configuration is shown in Figure 23.

<esc>

The null command. Returns the interface level.

The Movement Commands Menu

The Movement Commands menu is shown in Figure 24. It can be accessed with <F1> followed by ^V or <V>. Once the commands are known, it is more convenient to use the commands given below. Note that even though they appear on a submenu, to use the movement commands from the interface level it is NOT necessary to first use the ^V key.

The movement keys given on the Movement Commands menu are augmented by other editing commands described in the next section: Miscellaneous Editing Commands. They are basic and useful and should be learned thoroughly.

```
Movement commands
A cell left
S char left
D char right
F cell right <space>
E cell up
X cell down
go to next Case
Jump cell blocks
P (return last cell)
<esc>
<bckspc> del left
^t del word right
^ql sorry key
```

Figure 24. The Movement menu.

<A cell left>, ^A:

In the title cell, ^A moves the cursor to the beginning of any word in which the cursor is located. If the cursor is at the beginning of the string, it moves the cursor to the [tot flow] cell.

In the other cells, ^A moves the cursor to the beginning of the number in a cell, or, to the previous cell if the cursor is already at the beginning.

<S char left>, ^S, or < ← >:

Moves the cursor one character to the left of its present position. If it is already at the beginning of the number or string, it moves the cursor to the previous cell.

<D char right>, ^D, or < → >:

Moves the cursor one character to the right of its present position. If the cursor is at the end of the number or string, it moves the cursor to the next cell.

<F cell right>, ^F:

In the title cell, ^F moves the cursor to the end of any word in which the cursor is located. At the end of the title cell it moves the cursor to the [tot flow] cell. In all other cells, ^F moves the cursor to the right side of the value in cell or to the next cell if the cursor is already on the right side.

<Space bar> works normally in the title cell but moves the cursor to the next cell in the rest of the interface.

<E cell up>, ^E, or < ↑ >:

Moves the cursor up one cell in the interface. If the cursor is in the uppermost row of cells, the cursor is moved to one line below the deepest defined line in the ambient block or to the bottom of the column of cells.

<X cell down>, ^X, or < ↓ >:

Moves the cursor down one cell. If the cursor is in the row of cells in the ambient block one below the lowest defined depth, or is at the bottom of a column of cells, the cursor is moved to the top of the column of cells.

<go to Case>, ^C:

Directs PLUMES to go to another case specified in response to a typeover prompt in the dialogue window. The next case is always offered as a default and can be accepted with <space bar> or <Enter>. Otherwise, the default may be overridden by typing any other number followed by <space bar> or <Enter>.

If the specified case number is one greater than the number of cases that currently exist in the file of cases, a new case, is appended and filled with the same information contained in the case from which the ^C command is issued. Any number less than one or greater than the number of cases plus one is ignored.

See the <Page Up> and <Page Down> commands below.

<Jump cell blocks>, ^J:

Moves the cursor into the next colored block of the interface. ^J is a fast way to move about in the interface and the only way to move the cursor into the [title] cell.

<P (return last cell)>, ^VP:

This command is useful after a variable is selected for deletion in the conflict resolution mode. When a deletion is made the cursor normally returns to the cell in which the cursor was located after the value that caused the conflict was entered. The ^YP command returns the cursor to the cell which was deleted. Also works after the ^J, ^E, and ^X commands. NOTE: Due to the presence of the ^P (cell precision) command on the main menu, this command can only be accessed by using the ^V prefix.

<esc>

The null command. Returns the interface level.

Mnemonics:

The Movement commands menu lists a few editing commands which also may be issued at the interface level. These are described in the next section.

Other Useful Editing Commands

The following commands perform useful editing functions in the interface. Many of the commands are similar to those in the WordStar (trademark) word processing program and in the Borland Pascal editor. Some common WordPerfect commands are also used.

<space bar>, or <Enter>:
Moves the cursor to the next cell, except in the title cell, where it works normally.

<backspace>, or ^H:
Erases the character or digit to the left of the cursor.

<Delete>, or ^G:
Erases the character or digit under the cursor.

^T:
Erases the rest of the word or number to the right of the cursor.

<Page Down>:
Directs PLUMES to go to the previous case of the case file. When used in Case 1, the highest numbered case is brought into the interface.

For skipping over many cases or for creating new cases using an intermediate case as a template, use the ^C command.

<Page Up>:
Directs PLUMES to go to the next case of the case file. When it is the last case in the case file, a beep is issued to alert you to the fact that a new case will be created if the command is issued again. This is a fast way for browsing the case data file and for creating new cases using the last case as a template.

For skipping over many cases or for creating new cases using an intermediate case as a template, use the ^C command.

^QD:
Moves the cursor to the right of the last character or digit in the cell.

^QY:
Erases everything in the cell to the right of the cursor, all of it.

^QL:
"Sorry-I-changed-it-command". Restores the original value of a cell providing the cursor has not left the cell. Some conditions cause exceptions to this rule and data may have to be re-entered.

<~>:

May be used in many situations to dump whatever is on the screen to an ASCII file called DUMPALL. Subsequent uses of the command will cause the DUMPALL file to be appended so that occasional examination or deletion of the file may be appropriate. Intended for debugging and documentation purposes.

The Miscellany Menu

The Miscellany Menu (^Y) is shown in Figure 25.

```
─ Miscellany menu ─
ambient column Fill
Interpolate amb cell
Copy ambient line
Delete ambient line
Beget new cases
cHeck consistency
Notes
clear Output cells
Purge cases
construct Udf file
pauSe cell
cormiX category
Zap most variables
```

Figure 25. The Miscellany menu.

<ambient column Fill>, ^YF:

If variables in an ambient column are all the same, it is often useful to fill only the surface cell for that column and use the <space bar> to skip over successive cells in that column. After all the depths are entered (i.e. all the [depth] cells are filled with the appropriate depths), move to the surface cell in the empty column and issue the ^YF command. All the remaining cells in that column down to the deepest depth will be filled with the same value.

The <Auto ambient> command on the Configuration menu is useful for achieving these results on a continuous basis.

<Interpolate amb cell>, ^YI:

This command is used to place depth interpolated ambient values into intermediate empty cells in a given column in the ambient block. For example, similar to the ^YI command, you could specify a surface current of 0.10 m/sec and a bottom current of 0.20 m/sec. Then, from the surface cell in the [current] column, issue the ^YI command. The empty intermediate cells will be filled with depth interpolated values.

<Delete ambient line>, ^YD:

Used to delete the line in which the cursor is located from the ambient block.

<Copy ambient line>, ^YC:

Used to insert a copy of the line in the ambient block in which the cursor is located, i.e., between it and the next line in the ambient block.

<Beget new cases>, ^YB:

Used to copy the cell in which the cursor is located to the same cell in a specified number of subsequent cases. The number of cases involved is specified in a dialogue window which is provided.

<cHeck consistency>, ^YH:

Instructs PLUMES to evaluate all possible equations for the cell from the set of equations which may be displayed with the ^L command. The results are compared and any difference greater than a tenth of one percent is reported in the dialogue line.

Not all differences reported are cause for concern. In particular, very small values, which are for all practical purposes identical to zero, can occasionally differ by more than the criterion. Also, if the defining equation has more than one solution, as for example is the case when the horizontal velocity [hor vel] is computed from the total velocity [total vel] and the vertical velocity [ver vel], the signs of the reported values may differ. Nevertheless, any reported differences should be contemplated.

<Notes>, ^YN:

Reports the previous messages, up to ten, that have displayed in the dialogue window,

<clear Output cells>, ^YO:

Just as cells may be added to the list of variables to be printed or displayed by UM at run time, cells already on the list may be cleared using ^YO. The dialogue window gives a choice for clearing all cells from the table or for returning to the default list of variables. After the command is used the ^O command may be used to establish a different list.

<Purge case>, ^YP:

All cases after the one shown on the interface may be deleted from the case file. The command is especially useful when terminal cases have been added to the file inadvertently by the use of the < ↑ > command.

<construct Udf file>, ^YU:

Used to translate the cases specified in the dialogue window into the UDF format used in the 1985 plume models (Muellenhoff et al., 1985). This makes PLUMES operationally compatible with the earlier models. The intent is to support ULINE and users who may not have adopted the resident models. The interpreted cases are appended to an ASCII file called UDF.IN.

<pauSe cell>, ^YS:

Used to edit and set up the pause cell located near the lower right hand corner of the interface. After typing ^YS the dialogue line shown in Figure 26 is displayed. The cell

```
Back, Inequalities, Output, Variables(space), or <esc>,
```

Figure 26. The pause cell dialogue window.

is the only way to access selected model variables not present on the interface screen, i.e. average dilution, **centerline concentration**, time, density difference, and horizontal distance. The program control function of the pause cell works in conjunction with the <Farfield start> command on the Configuration menu. (Other cells that can be controlled, via conditions given below, include [port dep], [plume dia], [effl sal], [effl temp], [horiz vel], [vertl vel], and [p amb den].) The capital letters in the window are highlighted; their functions are:

<Back> or <B>:
Moves backwards through the list of model variables, including those listed above.

<Inequalities> or <I>:
Selects the possible inequality conditions or criteria. The idea is to set up conditions under which UM will be forced to output data or terminate before initiations of the farfield algorithm. For example, if the pause cell is the horizontal distance (travelled) [hor dis] cell, with a numeric value of 10 m, the inequality >=, and the Farfield start character in the Configuration string is set to "P" for Pause criterion, then UM will output a dilution immediately after 10 m is reached and initiate the farfield algorithm. If the Configuration string is not set to the Pause criterion, then UM will simply output a value at that point and terminate. This is a convenient way to establish output at desired points (like the mixing zone boundary) or criteria. The inequalities include >=, <=, =.

<Output> or <O>:
Adds the variable to the output table (see the ^O command).

<Variables> or <V> or <space bar>:
Moves forward through the list of variables.

<esc>
The null command. Returns the interface level.

<cormiX category>, ^YX:
Used to determine the CORMIX1 flow classification for the current values in the interface. The command is equivalent to setting the CORMIX program configuration to C for a single instance (see the ^RC command.)

<Zap most variables>, ^YZ:
Used to clear most of the variables on the interface screen. The cells not affected include the aspiration coefficient, the print frequency, the decay constant, and the farfield diffusion coefficient.

<esc>
The null command. Returns the interface level.

3. TUTORIAL OF THE INTERFACE

A TUTORIAL OF THE INTERFACE

PRELIMINARY INFORMATION

System Requirements and Setup

PLUMES is designed to be used on IBM compatible PCs. The program does not make use of graphics but does require a color monitor. The memory requirements of PLUMES are modest, less than 200K, and should not interfere with other resident programs. The latest advisories are contained in a file called **READ1st.exe**, which, as its name implies, should be read first. READ1st.exe contains information on how to unzip the program and document files. Information describing a few supplementary files is also found there.

PLUMES can be run from the A: prompt using the diskette provided, however, the tutorial notation assumes that it is installed on a hard drive, generally Drive C. We suggest that you create a new directory on which to install PLUMES. If this new directory is a sub-directory of the root directory the following procedure could be used at the C:> prompt to create a sub-directory called PLUMES and to change to the new directory. At the prompt type "mkdir PLUMES" followed by a carriage return (i.e. the Enter key: <enter>). The installation commands might look like this:

```
C:> mkdir PLUMES
C:> chdir PLUMES
C:\PLUMES> copy a: * . *
```

After each command an <enter> is implied. The mkdir (or md) command makes the PLUMES sub-directory, the chdir (or cd) command moves you to the new PLUMES sub-directory, and the copy command copies the PLUMES software from the diskette. At this point, the program, PLUMES.exe, may be run by typing at the prompt:

```
C:\PLUMES> plumes
```

The case of the command is unimportant.

If further guidance is needed on setting up the directories the DOS reference manual should be consulted. Keystroke conventions used in the tutorial are described in the preceding chapter.

EXAMPLE: PROPOSED SAND ISLAND WWTP EXPANSION

Introduction

This example is a step-by-step development, or tutorial, of the kind of problems encountered in applying 301(h) regulations. It is designed to make you familiar with the use of PLUMES and to give you a feel for its capabilities and limitations. Several figures are given along the way to allow you to compare your progress with a prepared example. These figures do not adequately convey what is a full **color** display on the computer monitor. Consequently, **the tutorial is most effective if it is used as a guide while filling out the PLUMES interface input form.**

The Sand Island example is intended to be realistic, not only as being representative of the problems encountered in practice but in terms of how analyses are not unique. In other words there is not a single right way, instead, an analysis is likely to be an evolutionary process. An examination of work and simulations already completed are likely to identify other factors that need to be considered. Thus, part of planning the analysis is to carefully examine modeling results already in hand to guide further changes which, fortunately, with PLUMES, are easily made. But, the greater flexibility available in PLUMES also requires vigilance on the part of the user because it is easy to overlook cells that, once filled, need to be changed.

The problem described here is based on a proposal by the Sand Island Waste Water Treatment Plant (WWTP) of the City and County of Honolulu, Hawaii which seeks to increase its permitted wet-weather flow capacity from 102 to 130 MGD. An increase in the design capacity of 202 MGD is also under consideration.

What will be the effects of the proposed actions on initial and farfield dilution? How are bacterial, turbidity, and other contaminant levels likely to change? Is the new discharge likely to meet water quality standards under the proposed operating changes? How do new techniques compare to earlier analytical procedures? These are some of the questions addressed.

The problem involves a diffuser with 285 ports located on both sides of the diffuser. Over time, the landward ports have become clogged with sand so that the discussion changes between 285 and 142 ports, 24 foot spacing and 12 foot spacing, which is confusing. This poses the question: "How do you compare the performance of the diffuser, now clogged, with the previously unclogged diffuser?" Ultimate answers are not provided and this analysis is incomplete. In fact, **THIS EXAMPLE CONTAINS DELIBERATE MISTAKES.**

Analysis

The problem can be broken down into five different parts:

(1) Collect pertinent information.
(2) Input information into the PLUMES interface.
(3) Run the PLUMES initial dilution and farfield plume models.
(4) Analyze the model results and make adjustments, if necessary.
(5) Use the results in the decision making process.

STEP 1: Collect Pertinent Information

One way to get a feel for the information needed is to run PLUMES and work an example. The first time you do so you may be dismayed by the number of cells displayed by the interface. It may seem imposing at first but only some of the variables, which you may choose, need to be defined -- the interface automatically calculates the rest, as soon as sufficient data is provided. You are free to pass over cells for which you have no data, filling those for which data is available.

If you create subsequent cases, and we will, the data already contained in the existing case may be used as a template for the new case by using the ^C command to simply move to the new case to be appended to the existing file from the case to be copied. Minor changes may then be made very quickly to only the affected variables.

STEP 2: Input the Sand Island Information

It is assumed the necessary data needed for Sand Island have been acquired; the appropriate references are given. Begin by entering the main menu using <F1> and set up a file for the example by pressing <W>, the <get Work file> command, or, more simply, press ^W without first pressing <F1>. The dialogue window appears requesting the work filename. Type in <Sandis.var> (which will not exist yet) followed by <space bar> or <enter>. Notice that the default filename can be overwritten without first deleting it. The .VAR filename extension is recommended because the <get Work file> command may be used to scan existing .VAR files in the current directory by simply using the < ↓ > key.

It is worth noting that the default name given in the dialogue box can be edited. For example, pressing < → >, or some other editing key like ^QD (move to the last character), before you type an ordinary character, will move the cursor into the field of the cell where it may be edited by adding or deleting characters.

When you are done the monitor will look somewhat like Figure 27, without the 4.469 values or title. The other values are default values which may be accepted or rejected as appropriate.

To start, give this case, identified as Case 1 in the upper right hand corner, a descriptive title, e.g. "Sand Island validation". First, of course, you must move into the title cell. You could go to the Movement menu using the <F1> key but it is faster to use the "jump" command, ^J, several times until the cursor moves into the title cell. Go ahead and type in the title. Push <enter>, ^J, or ^X when you are finished, in either case the cursor moves to the [tot flow] cell which is a good place to start filling out the rest of the interface.

```
  Jun 19, 1992,  11:26:42  ERL-N PROGRAM PLUMES, Jun 10, 1992   Case:   1 of
1
  Title   Sand Island validation
   tot flow    # ports port flow   spacing  effl sal effl temp   far inc    far
dis
     4.469          1      4.469      1000       0.0

   port dep  port dia plume dia total vel horiz vel vertl vel asp coeff print
frq
                                                                 0.10
500
  port elev ver angle cont coef  effl den poll conc     decay  Froude # Roberts
F
                0.0       1.0                 100

  hor angle red space p amb den p current   far dif   far vel K:vel/cur Stratif
#
        90    1000.0                        0.000453

      depth    current    density  salinity       temp  amb conc  N (freq) red
grav.
       0.0
```

Ilustr. 27 The PLUMES interface with the dialogue line showing units conversions of the total flow cell.

The total flow corresponding to the current permit is 102 MGD. It is the appropriate value for the [tot flow] cell in which the cursor should now be located. However, the dialogue line informs you that the primary units in this cell are m3/s, so a conversion is required. On the main menu we note there is a command called <units Konversion>. By toggling it (pressing ^K) we can change the input units to any that are shown on the dialogue line. Do so, then, when the dialogue line shows "MGD", enter 102 in the cell. Press the space bar to enter the value (in the process, moving the cursor to the next cell). Notice that, if you move the cursor back to the [tot flow] cell using ^A or <←>, the dialogue line will appear as shown at the bottom of Figure 27. In addition to showing the total flow in m^3/sec, MGD (million gallons per day), and cfs (cubic feet per second), the dialogue line also gives the recommended range of values for the cell, which, incidentally, is not enforced.

The cursor should now be in the [# ports] cell and the value shown in the [tot flow] cell should be 4.469. When the cursor was moved to the number of ports [# ports] cell, the third cell, labeled [port flow], acquired a white value equal to 4.469, even though we did not input a value in this cell. This is an example of how the interface is event driven, i.e. an event, your pressing the space bar, automatically initiated an action. We will have more to say about this shortly.

The number of ports is 285, but they are not uniform and the diffuser has sections of different diameters. Strictly speaking, a hydraulic model is needed to properly analyze the effluent velocities from the ports, but we will assume that the flow is uniformly distributed. If the diffuser is well designed, the deviations from this assumption will not be too great. If there is doubt a program such as PLUMEHYD.EXE (Appendix 3) may be run to give better estimates of the port flow distribution. In that case the total flow may not be consistent with the port flow and we may need to do a piecewise analysis of the diffuser. Alternatively, some other more conservative assumptions could be made. To simplify the analysis we will assume uniformity.

Do not worry about what to do about the "1" that is already in the cell, just type 285. As explained previously, the "1" disappears when you begin to type. Again, there are a number of editing commands explained in the previous chapter which allow you to modify the information that is previously contained in the cell.

After typing in 285 use the space bar to move the cursor to the [port flow] cell: it has now changed from 4.469 to 0.01568. The new value in m^3/sec is consistent with the flow from 285 equal ports producing a total flow of 102 MGD (4.469 m^3/sec). The [port flow] cell value is **white** (when the cursor is not in the cell) instead of yellow to remind you that this is a **dependent variable** which you did not input but was calculated by PLUMES from information you did input. Which cells are independent and which cells are dependent depends entirely on how you fill in the interface, i.e. whichever variables are most compatible with the available information. This gives you flexibility to use the data you have, not data you wish you had.

Before going on, note also that the new value is expressed as 0.01568 and not 0.016 (three decimal places to the right of the period) as might have been expected based on the formatting pattern established in the total flow [tot flow] cell. PLUMES reports data to three or four significant digits and up to six are accessible with the <cell Precision> command.

The spacing is 7.315 m (24 ft). It is also noted that the ports are opposed so that there are really two ports per 24 ft section. This presents an interesting problem because, if the plumes from both sides merge, as they would in a crosscurrent or as they might even in the absence of current, then this spacing is too large because this kind of merging is not modeled in the UM program, only side by side merging is. Thus, there is an intuitively appealing suggestion that we should use spacing of 12 instead of 24 feet. But for the moment we will ignore this complication. It will be easy to estimate its effect later when we develop additional cases. Input 7.315 or use the ^K command to input in feet.

In the case of Sand Island, we encounter another complication. Because the diffuser parallels the isobaths it acts as a barrier to sand moving seaward. This has apparently clogged the ports on one side and causes the port flow to double in the remaining ports. But, for now leave the spacing at 7.315.

It is important to note that with the spacing described in this way, the farfield predictions will not be correct unless the Configuration menu is used to input the correct length of the wastefield and the end of initial dilution. The reason is that, by ignoring cross diffuser merging, we have described the diffuser as if all ports are on one side of the diffuser. Consequently, the initial wastefield width needed for the farfield algorithm will be overestimated by approximately a factor of two. More will be said about this subsequently.

It may seem that we are following a rather cavalier path in defining the problem. However, in practice, it is common to first estimate parameters and play around. In effect, this represents a screening analysis. If it is found that the initial dilution is close to being inadequate for meeting water quality standards, then the analysis can be refined. In fact, as will be seen, the interface is ideally suited for this purpose because it is easy to change values anywhere in the interface without starting over. Thus, there is no disadvantage to first scoping out the problem and becoming aware of some of the potential pitfalls in the analysis beforehand.

When you are finished with the [spacing] cell you could move to the salinity cell. But wait a minute, we have just made an important observation about spacing so let's jump (repeated ^J's) back to the title and reflect this fact there. This will give you a chance to practice your editing. In the title cell you could use the arrow keys to move to the end of the string, but, for touch typists, it is easier to use the control key movement cluster. A few ^F's get you to the right place. If there are several words to jump over, the ^QD command does the move in one step. Type in ": no blockage" or something like that. Then return to where you were using the movement commands.

The header of the next cell shows "effl sal" and, because the cursor is now in the brown block, you may infer, correctly, that this refers to effluent salinity. If you are unsure about the content of any cell however, the ^L command (<List equations> on the main menu) may be used to define the cell, as shown in Figure 28.

```
 Help for variable:  s =  effl sal  o/oo


    Effluent salinity.  Sea salt composition is assumed.


Equations and variable definitions:

 s   = sigmasal(t,den).

   den : plume density
```

Ilustr. 28. A <List equations> screen for the effluent salinity cell.

The information given, that s = sigmasal(t,den), is somewhat more cryptic than most cells. It is an abbreviated way of showing that salinity is derived from a complicated function, in this case involving the Newton-Raphson method because the function cannot be solved analytically for salinity. Also, most cells have more than just one defining equation.

Information on the salinity of municipal effluent is not always available. We suspect that the effluent is largely fresh water and guess that it is close to 0.0. (If possible, this should be checked later.) The default is accepted by passing over the cell using the space bar and going to the effluent temperature cell [effl temp]. Being in Hawaii, the temperature is estimated to be about 25 C. As soon as both the salinity and temperature are specified, PLUMES calculates a value of -2.893 for plume density ([effl den] cell). This density is given in sigma-t units and translates to kg/m3 when 1000 is added. Thus, the approximated density of the effluent is 997.107 kg/m3. The conversion can be verified directly by taking an excursion to the [effl den] cell and consulting the dialogue line which gives the value in additional units. (Try the ^P command).

We can approximate for now the effluent salinity and temperature because the effluent is discharged to sea water with a much higher salinity. Thus, the greater part of the density difference, i.e. buoyancy, is due to salinity differences, and the temperature approximations are unlikely to affect the outcome by more than a few percent. However, in regulatory work you would try to define these variables more accurately. (See also the discussion in the Freshwater Discharges of Buoyant Plumes section of Chapter 1.)

The cursor should now be in the upper right corner of the interface in the Miscellaneous (gray) block of cells. Again using the <List equations> command, it is determined that the farfield increment cell [far inc] is the distance between points at which the farfield dilution estimates are reported during the simulation. Notice also that the header typeface is black, which means that cell input is not necessary to determine the initial dilution, (i.e. neither UM nor RSB require it for input). However, as we are interested in farfield bacterial concentration predictions, values for these should be established.

It is known that a surfing area is located within approximately 2000 m of the outfall and, therefore, this is considered to be an appropriate value to put in the farfield distance [far dis] cell. Since only the bacterial levels at this distance are of interest, the farfield increment, the [far inc] cell in which the cursor is presently located, can be rather large, 500 m will do. (An unnecessarily small value may give more output than you want causing previous information to scroll off the screen when UM is run.) Enter this value and follow this by putting the value 2000 m in the [far dis] cell.

Now enter the plume depth measured from a standard datum such as mean lower low water (MLLW). (This is an UM program variable which, like a few others, is initialized by the interfaces.) In this case we know the depth to the center of the ports from which the plumes emanate to be 70.1m. All ports are at essentially the same depth. The blue background before the first letter in [port dep] indicates the variable, (centerline) plume depth, will be an output variable when running UM.

The next cell, [port dia], is the actual physical diameter of the port (as opposed to the vena contracta plume diameter, the minimum diameter of the plume, in the following [plume dia] cell). The Sand Island diffuser has five different diameters to choose from, so which one should be used? Technically, a diffuser hydraulics model (Appendix 3) could be used to provide estimates for port flow from each port. Experience shows, however, that varying port flow over a limited range does not affect initial dilution radically. Nevertheless, it would be wise, especially for beginners, to do a sensitivity analysis by changing some of the values somewhat. The interface is ideal for this kind of exploration. A conservative value is always appropriate if the screening test is ultimately passed. (See also the Effect of Wastewater Flow on Dilution section.)

For now, enter 8.5 cm. Notice that the possible input units are in meters, feet, and inches, not centimeters. This time the proper conversion is not available through the ^K command. It is assumed that you are familiar with the fact that 8.5 cm is equal to 0.085 m. Notice that the leading 0 does not have to be entered. Notice also that after inputting the diameter many cells are starting to fill up with white values[1].

Now keep moving the cursor until the cursor is on the [print frq] cell. The print frequency cell [print frq] simply determines how many model steps there are between outputs. Except when the time step becomes too large, UM is designed to double dilution every 100 program steps. Thus a [print frq] cell value of 100 will cause UM to output dilutions of 1, 2, 4, 8..., approximately. This can be adjusted to taste, we will accept the default value for the time being. It is not critical in any case because the model outputs at important milestones, e.g. the trapping level. The performance of RSB is not affected by this cell.

---

[1] The basic idea behind filling empty cells in the interface is this: PLUMES can calculate cell values from input you provide because it is event driven and because it normally has many ways to calculate each cell. To give an example, move the cursor to the [total vel] cell. You may have wondered why some cell labels are displayed against a checkered background which changes as you move from cell to cell. These checkered labels tell you which other variables (cells) serve as independent variables for the cell in which the cursor is located. For example, right now the [port flow] and [plume dia] labels should be on a checkered background. That means that if [port flow] and [plume dia] are defined (either white or yellow), [total vel] will be calculated by PLUMES, as it apparently has been. This is a basic characteristic of the PLUMES interface that makes it act like a specialized spreadsheet. Essentially, most cells have one or several equations associated with it, just like spreadsheets, that allows unknown cells to be defined, providing the appropriate information is available.

But PLUMES provides more than the standard spreadsheet in this respect. If you will now push <F1> followed by <I> (or simply ^I at the interface level) for the <show Independents> command, you will see that other labels are now checkered: first [horiz vel] and [vertl vel], then [plume dia], [p amb den], [effl den], and [Froude #], etc.. Many cells have a multitude of ways of being calculated by PLUMES. The ^L command will reveal just how many there are and define them if the file EQNS resides in the current directory where it can be accessed by PLUMES. It is this ability of PLUMES to calculate variables in many different ways that helps assure that you will have to input only a minimum of information and that you do not have to be an expert and know how to provide specialized information. Your job is to keep finding cells that you know something about and fill them until the interface is completely defined. You can do this by moving directly to the cells you know, passing over the others. However, if at the end not all cell are filled, you will need to return to the remaining unfilled cells.

The port elevation cell [port elev] is used in calculating the CORMIX flow categories, and it also affects UM's prediction of when the plume hits bottom. Here we use the radius of the diffuser pipe, in this case enter 0.84 m.

Accept the default value in the vertical angle cell [ver angle] by skipping over it. A value of 0.0 indicates that the effluent is being discharged horizontally, which is the case with Sand Island and many modern diffusers. It should be becoming apparent that filling out the interface is not such a difficult task after all.

The contraction coefficient cell [cont coef] is normally used to compute the actual initial plume diameter by adjusting [port dia] on the basis of the differentiation between **bell shaped ports**, which have a coefficient approximately equal to 1.0, and **sharp-edged ports**, which have a value near 0.61. Sometimes this information is not provided in which case the value that yields the more conservative dilution could be used. Generally, its value does not affect dilution very much.

If salinity and temperature are specified as they have been the [effl den] (effluent density) cell is calculated using the equation of state found in Teeter and Baumgartner (1979). The computed values vary slightly from published values (see discussion at the end of the next chapter). If suspended or dissolved substances factor prominently into the effluent density it may be more appropriate to use a linear equation of state, invoked by defining the [effl den] cell while leaving the temperature and salinity cells empty.

Now move the cursor to the pollutant concentration [poll conc] cell. This cell is used to specify the concentration of a specific pollutant in the effluent and is used in combination with the ambient concentration cell [amb conc] to help determine the **effective dilution** achieved by the diffuser (see also the sections Dilution Factors, Effective Dilution Factor, and Relationship of Ambient Dilution Water to Plume Concentration). For example, if the ambient concentration is everywhere equal to zero then the effective dilution is identical to the effluent dilution. However, suppose we accept the default value of 100 given in the [poll conc] cell and all the ambient concentration cells have a concentration of 1.0. Then, no matter how great the volume dilution is, the effective dilution can never exceed 100. In this case the referenced values would be percentages.

Any **consistent units of concentration** may be used, which means that the units in the pollutant and ambient concentration cells must match. We will use a value of 6.1×10^8(colonies)/100ml for the bacterial concentration. In PLUMES format, **scientific notation is input in "e" format**, for example as 6.1e8. Note again, that to replace the default value we simply start typing in the value of 6.1e8 and <space bar> when done.

The cursor should now be on the decay cell [decay]. This is the simple first order decay constant, *k*, used in the equation

$$c = c_o e^{-kt} \qquad \textbf{(18)}$$

where c is the concentration time t after a concentration of c_0 is measured. For convenience, the primary unit is inverse days. Often, however, decay is expressed in terms of T90 values, which specifies how much time is required for 90 percent of the pollutant to decay, or how much time is required for 90 % of the bacteria to die. The T90 time must be input in hours; for Sand Island we use 1 hr. Thus, after one daylight hour of exposure in Hawaiian surface waters, 90 % of the bacteria have died. This unit is available by using the <units Konversion> command; when t90hr is indicated in the dialogue window enter the value 1.

As you move to the next cell you will notice that the space bar movement command bypasses the densimetric Froude number [Froude #] and Roberts Froude number [Roberts F] cells; the red block parameters are normally of interest only to researchers and designers. (When it is convenient to use them the ^J command may be used to get into this block.) These numbers will be calculated by the interface when all necessary input is entered.

The cursor should be in the horizontal diffuser angle cell [hor angle]. The outfall structure variables, effluent characteristics, and miscellaneous blocks are complete. The interface screen should now look like Figure 29.

```
  Jun 19, 1992,   11:32: 1  ERL-N PROGRAM PLUMES, Jun 10, 1992   Case:   1 of
1
  Title   Sand Island validation: no blockage

   tot flow   # ports port flow   spacing  effl sal effl temp   far inc   far
dis
      4.469        285   0.01568     7.315      0.0         25        500
2000
   port dep  port dia plume dia total vel horiz vel vertl vel asp coeff print
frq
       70.1     0.085   0.08500     2.763     2.763     0.000      0.10
500
  port elev ver angle cont coef  effl den poll conc    decay  Froude # Roberts
F
       0.84        0.0       1.0    -2.893     6.1e8     55.26

  hor angle red space p amb den p current   far dif   far vel K:vel/cur Stratif
#
         90     7.315                      0.000453

      depth    current    density  salinity      temp  amb conc  N (freq) red
grav.
        0.0
```

Ilustr. 29 A partially completed interface.

The cursor is now in the green ambient block, specifically, in the horizontal diffuser angle cell. An angle of 90 degrees (the default value) indicates that the current is perpendicular to the axis of the diffuser, i.e. it is flowing across the pipe and parallel (co-flowing) to the effluent plume. Notice that if 45 degrees were entered the value in the following reduced spacing cell [red space] would change from 7.315 m (the physical port spacing) to 5.172m, the geometrically projected spacing. **In UM, the effect of changing the direction of the current simply changes the reduced spacing.** The justification for this procedure is derived from Roberts (1977) and is valid over angles ranging from 45 to 135 degrees. While 90 degrees is the desired angle for now you may change it temporarily to see how it works. Values of 0 to 44 degrees, which are outside the recommended range shown in the dialogue window, would produce reduced spacings of 0 to 5.1m and should not be used for UM (but are appropriate for RSB). Similarly, values of 181 to 360 produce negative reduced spacings and should not be used.

Now skip the reduced spacing cell [red space] and move to the port ambient density [p amb den] cell. Notice that it is not one of the cells preferred for input (it has a white header) and we will not enter a value into this cell, even though we could, or the following port ambient current [p current] cell. Both cells will be calculated by the interface when the ambient depth, density (or temperature and salinity), and current are completed[2].

Now move to the farfield diffusion coefficient cell [far dif] and use the ^L command to get an explanation of this parameter. In general, unless there are good reasons to change it, the default value should be kept. For more information see Okubo (1962).

The next cell is the farfield velocity [far vel]. The cell label is black, therefore it is not required for initial dilution estimates. However, it is our goal to estimate farfield dispersion in order to determine maximum bacterial levels in areas where water contact activities occur. Although the dilution of contaminants in the farfield would be enhanced by greater current speeds we recognize that high current speed will also result in shorter travel times for the diluted wastes that are carried to the protected zone, thus resulting in less die-off of bacteria. However, the current speed should be realistic and take into consideration not only consistency with the near field current but also factors such as tidal reversals and the likelihood that high currents will persist for long periods of time. In the case of Sand Island, a current of 15 cm/sec is used corresponding to a travel time to affected areas 2000 m away of 3.7 hrs. Input 0.15 m/sec.

---

[2]. This is a good time to point out something you may have already noticed, some of the labels have yellow labels (yellow lettering in the label like the [tot flow] cell) while others have white ones (white lettering in the label like the port ambient density [p amb den] cell). In general, the yellow labels mark the variables that are recommended for input, in a sense, they are preferred variables. There are a variety of reasons why they are preferred which are rather technical and have to do with the math of the equations. For example, the program may be more easily confused about the sign of a calculated number if one of the secondary variables is input (e.g. the solution of a square root). There is even a possibility of inconsistencies in the input (refer to the manual for an explanation). The miscellanY submenu has a <cHeck consistency> command that should be consulted when it is suspected that there is an inconsistency. Normally, inconsistencies will not develop unless the user overrides a cell containing a white (not to be confused with the header lettering color described above) numeric dependent value with a yellow independent input value, a topic that has not been covered yet. Even under these circumstances, inconsistencies (or conflicts) will not usually arise. Also, to avoid alarm, in some cases the <cHeck consistency> command will report values of the same magnitude but different sign; this does not necessarily indicate the case is inconsistent. Finally, the check is based on a comparison of values of the same parameter calculated from each of the different equations that can be seen when issuing the ^L command. Sometimes it will report two very small values, both essentially equal to zero, which nevertheless differ by more than the fractional criterion.

The cursor is now in the main ambient block [depth]. This is where information on various layers of the ambient receiving water is input. The first depth cell [depth] should normally contain the default value of 0.0 m (water surface), so move to the ambient current [current] cell of the surface layer. We will input depth, salinity, and temperature data shown in Figure 30.

```
 Jun 19, 1992,   11:36: 5  ERL-N PROGRAM PLUMES, Jun 10, 1992   Case:   1 of
 1
 Title   Sand Island validation: no blockage
  tot flow   # ports port flow   spacing  effl sal effl temp   far inc   far
dis
     4.469        285   0.01568      7.315       0.0        25       500
2000
  port dep  port dia plume dia total vel horiz vel vertl vel asp coeff print
frq
      70.1     0.085   0.08500     2.763     2.763     0.000      0.10
 500
 port elev  ver angle  cont coef   effl den poll conc      decay  Froude #
Roberts F
      0.84        0.0         1.0    -2.893      6.1e8     55.26      18.40
2.044E-14
 hor angle red space p amb den p current    far dif    far vel K:vel/cur
Stratif #
        90       7.315        24.080.00001000   0.000453           0.15
2763000.00004871
     depth    current    density  salinity       temp  amb conc  N (freq) red
grav.
       0.0       1e-5      22.99     34.99      26.18          0   0.01217
```

Ilustr. 30 Completed interface.

Zero current is used to estimate minimum dilution, which we input in the surface ambient current [current] cell. Note that upon moving to the next cell, the 0 is replaced by a small, near-zero value of 1e-5, which is the e-form scientific notation for 0.00001 m/s. This is done to avoid a mathematical singularity elsewhere in the interface. Originally a 0.0 value was allowed but resulted in the creation of **quasi-defined cells** (identified by the background color of the cell turning cyan) which made this practice bothersome. For example, a zero current throughout the ambient block would make it impossible to define a value for the effluent to current ratio cell [K:vel/cur] because the ratio would involve a division by zero. Thus, a quasi-defined cell is one which would normally be defined (all the independent variables that are needed exist), however a singularity (division by zero, negative square root, etc.) keeps that from happening. This is now avoided. The value of 1e-5 is practically equivalent to zero but can be input as a smaller value still if necessary.

Note: Other quasi-defined cells can still be generated, if they are, usually the last cell entered caused the condition and can be changed to resolve it. In the case of the [Stratif #] cell, a non-zero density gradient in the ambient density cells will keep it from being quasi-defined.

To establish the minimum dilution it is necessary to also use the maximum density gradient. The appropriate values, as shown in Figure 30 in terms of the depth and density columns, are established by filling in the salinity and temperature columns for the depths shown.

For now, go to the surface ambient salinity cell [salinity], and then the surface ambient temperature cell [temp] and type in the appropriate values shown in the figure. As soon as you do, and follow it with <space bar>, the ambient density [density] value at the surface of 22.99 sigma-T units is computed. The cursor should now be in the ambient concentration [amb conc] cell. Here it is safe to input 0.0 since we expect the receiving water to be generally very pristine, the ambient currents carry the effluent out of the region of the diffuser, and, most importantly, the die-off is generally sufficiently rapid that even recirculated water is likely to contain negligible bacterial concentrations, but should not generally be assumed.

The cursor should now be in the next depth cell [depth]. Since data are given at 100 feet and every 50 feet thereafter, use the ^K command to bring up the ft units in the dialogue line and enter 100 ft. Move to the salinity and temperature cells and continue to fill in the ambient block as shown (the remaining depths are 150, 200, and 250). Because the Configuration string shows a leading "A" the auto-ambient mode is on, which means that default values are taken from the line above. Thus, none of the ambient current speeds or ambient concentrations below the surface need to be typed.

As the last cell in the ambient temperature [temp] column was filled the remaining red cells were automatically calculated by PLUMES and also filled in. The stratification parameter [Stratif #] characterizes the degree to which the ambient is stratified between the surface and seabed when a linear approximation is appropriate. Some technical references (e.g. Fisher et al., 1979) use the linear approach in estimating dilution factors and trapping levels. Like the Froude number, the stratification number is also used to determine similitude between prototype and hydraulic model representations of plume behavior. While useful especially for laboratory experiments, most environmental problems involve complex nonlinear density profiles. The RSB and UM models calculate plume variables, such as dilution and rise, based on the density gradients established by the gradient specified for the appropriate depth interval, rather than the overall average represented by the stratification parameter. You can demonstrate that the stratification parameter does not change when intermediate lines of ambient data are added, deleted, or changed, as long as the data that determine the average parameters are not changed. By running successive cases you will see that dilutions and geometric variables calculated by RSB and UM change appropriately.

STEP 3: Run Initial Dilution Models

The fact that all the cells (except for the elective Pause cell which is presently showing the horizontal distance [hor dis] cell, its default value) are filled is a sign that the plume model can now be run. Issue the ^U, or <run Um program>, command. The dialogue line will then query "From this case on, run how many cases" and offer a default of 1 in the dialogue window.

Since we still have only one case we can simply use the space bar to accept the default value. A second query asks "Write to ("prn" for printer, "console", or disk file name):" with a default value of "console". Accepting the default with a <space bar> routes the output to the monitor. The result is shown in Figure 31.

```
 plume dep plume dia poll conc  dilution   hor dis
         m         m                           m
     70.10   0.08500 610000000     1.000     0.000
     67.27     1.822  18800000     31.18     4.417
     61.20     3.682   6751000     84.58     6.463< trap level
     54.65     7.502   3744000     144.2     8.460< merging
     53.04     10.75   3406000     153.5     9.365< plume element overlap.
Farfield calculations based on Brooks (1960), see guide for details:
Farfield dispersion based on wastefield width of       2088m
```

Ilustr. 31 UM simulation of the first Sand Island case. Note that the initial wastefield width of 2088m is too large by a factor of two and the farfield predictions should be ignored.

The trapping level dilution is 84.58 which corresponds almost exactly to the dilution found by UMERGE (84.48) and UPLUME (Muellenhoff et al., 1985), and the earlier reported value of 84. Experience shows that under a large range of conditions (without current) UPLUME and UMERGE agree very closely (Baumgartner, Muellenhoff, and Frick, 1992). Therefore, it is not surprising that we obtain close agreement with UM. It gives us some confidence in the new methodology. Nevertheless, this degree of agreement should not be expected in general. For one thing, in comparing UM and UMERGE, the definition of the aspiration velocity has been simplified which can cause small differences depending on the relative importance of forced and aspiration entrainment. Also, some of the input was approximated and the values are subject to some adjustment. Later, you can make some of these adjustments.

The farfield bacteria concentration based on Equation VI-20 (Tetra Tech, 1982), which uses the less conservative eddy diffusivity factor appropriate to open coastal waters (4/3 power law), is 626.7, above the water quality standard of 400 colonies/100ml. However, this estimate may still be overly conservative. The plume is well submerged and it may not be appropriate to conclude that individuals in near-surface contact sport areas will be exposed to the wastefield at all. As a result of this run we could now adjust the T90 time to a value more appropriate for a submerged flow field, such as 10 hours. We would then see a bacterial concentration 1.5×10^6 colonies per 100 ml.

The message "plume element overlap", which is discussed further in the sections on model theory, means that dilution predictions beyond this point would degrade increasingly if UM (not the farfield algorithm) were continued to be used. It may not be significant if dilution increases little in the overlapped region, which can be established by running the simulation to maximum rise using the ^R command.

The UM simulation can be interrupted at any time, execution is then suspended until another keypress restarts or terminates it. After it is finished running, any key will reestablish the input screen, i.e. the interface. The same procedure can be used to run the program again. If we override the word "console" with "prn" (do not enter the quotation marks) on the dialogue line the output will go to the printer (be sure that it is properly connected). Given any other name, PLUMES will attempt to send the output to a disk file (created or appended). Notice that the output contains a copy of the interface screen so that there is an exact record of the input.

As has been indicated, the farfield predictions shown in Figure 31 are not correct because the length of the wastefield is overestimated owing to the assumption that all ports are on one side of the diffuser and are spaced 7.315m apart. The farfield simulation could be "corrected" without changing the near-field predictions by accessing the Configuration menu (^R) and toggling the <Brooks eqn input> option. The Configuration string will then change from, for example, "ATNO0" to "ARNO0", where the R stands for "reset" the farfield algorithm initial conditions. Then run UM or RSB as you normally would. After the initial dilution phase is completed PLUMES will prompt "Input wastefield width:" in the dialogue window. Enter an approximate width of 1040 m to override the default value of 2088. PLUMES then prompts "Input starting longitudinal coordinate", i.e., the horizontal travel distance. Here we will accept 9.36m which is the horizontal distance between the source and the end of the initial dilution zone.

```
 plume dep plume dia poll conc  dilution   hor dis

        m         m                              m

    70.10    0.08500 610000000     1.000     0.000

    67.27      1.822  18800000     31.18     4.417

    61.20      3.682   6751000     84.58     6.463< trap level

    54.65      7.502   3744000     144.2     8.460< merging

    53.04      10.75   3406000     153.5     9.365< plume element overlap.

Farfield calculations based on Brooks (1960), see guide for details:

Merge running, key to interrupt

Input starting longitudinal coordinate:
Farfield dispersion based on wastefield width of       1040m
```

Ilustr. 32 Using the Configuration menu to gain control over farfield input and output.

The results are shown in Figure 32. As was anticipated, the farfield concentration is now somewhat lower: 536.9. We hasten to add however that this underpredicts farfield concentration because the effect of cross diffuser merging is ignored. At least we have had the opportunity to demonstrate the Configuration menu, and, in any case, we now feel more certain that the 400 colonies/100ml standard will be exceeded. A better estimate of farfield concentration awaits a more complete analysis.

Go ahead and change the Configuration string back to "ATNO0" and run RSB by using the ^B command. The results are given in Figure 33. **It is important to understand that the PLUMES version of RSB uses a spacing value twice that shown in the interface because RSB assumes two ports per spacing distance.** Thus, to be comparable, non-PLUMES versions of RSB must be run with double the spacing to agree with PLUMES RSB.

```
                         RSB

        Written by Philip J. W. Roberts (12/12/89)

          (Adapted by Walter E. Frick (1/12/92))


Case:   2: Sand Island validation: no blockage


Lengthscale ratios are: s/lb =   3.42 lm/lb =   0.21

Froude number, u3/b =            0.00

Jet Froude number, Fj =         18.6


Rise height to top of wastefield, ze =  16.3

Wastefield submergence below surface =  53.8

Wastefield thickness, he =              12.2 m
Height to level of cmax, zm =           10.9 m

Length of initial mixing region, xi =    8.6 m


Minimum dilution,       Sm =    158
```

Ilustr. 33 The RSB simulation of the first Sand Island case. (Note the excessive estimate of the wastefield width.)

Notice that RSB does not report a trapping level or intermediate dilution. However, we may compare the average volume flux dilutions at the plume element overlap level: they are 153.6 and 182 for UM and RSB respectively. The corresponding wastefield thicknesses are 10.75 (see [plume dia]) and 12.2 meters respectively, varying by a similar amount. Finally, the respective centerline rises are 17.06 and 10.9 meters.

Once again, if the analysis allows the luxury, it is convincing to present the results of the most conservative conditions likely to be encountered for the variables even if they are unlikely to occur simultaneously.

(Note that if the simulated plume is allowed to develop to maximum rise, which is possible when the Configuration string is changed to, for example, "ATNM0" ("M" is maximum rise), the corresponding far-field dilution, diameter, and rise are 156.2, 16.5, and 17.23 respectively. This is characteristic of the overlap problem under which plume diameter is overestimated, which, if prolonged, feeds back and increases the initial dilution. Frick, Baumgartner, and Fox

(1992) show this problem is shared by Lagrangian and Eulerian integral flux plume models generally, due to inadequacies of the standard round plume assumption. It is unimportant in this case, the dilution increasing from only 153 to 156 in the overlapped region.)

PLUMES links the same Brooks farfield model to RSB as it does to UM. It may seem odd then that RSB predicts a farfield concentration almost equal to that of UM (612.2 vs. 626.7) even though the dilution is substantially higher (204.5 vs. 172.3). The reason is in this case, because of the small T90 time: the decay mechanism in UM is functional from discharge, while the pollutant is assumed to be conservative (non-decaying) in the initial dilution region in RSB.

```
  Jun 19, 1992,   12:11:52  ERL-N PROGRAM PLUMES, Jun 10, 1992   Case:   2 of
2
  Title   Sand Island validation, no blockage, min strat.
   tot flow   # ports port flow   spacing  effl sal effl temp   far inc   far
dis
      4.469        285   0.01568      7.315       0.0         25        500
2000
   port dep  port dia plume dia total vel horiz vel vertl vel asp coeff print
frq
       70.1      0.085   0.08500      2.763      2.763      0.000      0.10
500
  port elev ver angle cont coef  effl den poll conc    decay  Froude # Roberts
F
       0.84        0.0        1.0     -2.893      6.1e8      55.26      18.68
2.105E-14
  hor angle red space p amb den p current   far dif   far vel K:vel/cur Stratif
#
         90       7.315       23.290.00001000   0.000453            0.15
2763000.00001395
      depth    current    density  salinity      temp  amb conc   N (freq) red
grav.
        0.0       1e-5      22.99     34.99     26.18           0  0.006417
```

Ilustr. 34 Case 2: a weakly stratified Sand Island case.

As was suggested previously, it is perhaps appropriate to consider a weakly stratified case, as shown in Figure 34, in order to simulate a surfacing waste field that might impact recreational waters. Notice that this case is Case 2, as is shown in the upper right corner of Figure 34. To create a new case use the ^C command, <go to next Case> on the Movement menu. The new case will use the information contained in the present case from which the ^C command is given as a template. Once in the new case, it may be edited. In Figure 34, some of the ambient data has been changed: the case title, one line of ambient has been removed using the ^YD command, and changes shown in bold.

```
 plume dep plume dia poll conc  dilution   hor dis
         m         m                          m
     70.10   0.08500 610000000     1.000     0.000
     67.28     1.810  18800000     31.21     4.432
     44.72     7.382   1603000     331.5     8.008< merging
     29.51     11.73    936100     520.1     8.868< trap level
     19.57     37.57    727000     601.6     9.652< plume element overlap.
Farfield calculations based on Brooks (1960), see guide for details:
Farfield dispersion based on wastefield width of       2115m
   --4/3 Power Law--   -Const Eddy Diff-
      conc  dilution       conc  dilution  distance        Time
                                                  m     sec    hrs
     89800     602.0      89810     601.9     500.0    3269    0.9
     10510     610.5      10570     606.9      1000    6602    1.8
      1197     636.2       1224     621.8      1500    9936    2.8
     134.2     673.9      140.5     642.9      2000   13270    3.7
==========================================================================

 <key>
```

Ilustr. 35 UM and RSB predictions for Sand Island Case 2.

As before, you can now run UM and RSB, the results are given in Figure 35. Again, the predicted UM and RSB dilutions compare well, being 601.6 and 671 respectively. This time the UM plume diameter and the RSB wastefield thickness, which are not totally comparable quantities, also agree closely, being 37.57 and 36.6 meters respectively. The message warning **plume element overlap**, indicates upstream intrusion of the wastefield is possible (Frick et al., 1989). The rises are considerably different, being 50.53 (70.10 - 50.53) and 32.7 meters respectively. The UM farfield concentration is now 134.2 colonies/100ml, which is much less than the previous farfield concentration and would meet the water quality standard of 400 colonies/100ml.

STEP 4: Analyze the Model Results and Make Adjustments

In the previous section the RSB, UM, and historical model results were compared. Now we will delve into the implications of some of the findings and question some of the assumptions that were made. In doing so, we will change the program configuration to make it possible to find the CORMIX flow categories for the cases in question. We will also illustrate the **PLUMES conflict resolution capability**.

From the standpoint of assumptions made earlier, in Sand Island Case 3 we will first examine the implications of sand blockage of half of the diffuser ports. In Case 4 the focus shifts to the sensitivity of the models to the magnitude of the decay coefficient, to other assumptions and input data. Finally, in Case 5, we examine the effect of current on predictions.

```
 Jun 19, 1992,  12:17:37  ERL-N PROGRAM PLUMES, Jun 10, 1992   Case:   3 of
3
 Title   Sand Island validation, blockage, min strat.
 tot flow   # ports port flow   spacing  effl sal effl temp   far inc   far dis
    4.469       142   0.03147     7.315      0.0        25       500      2000
 port dep  port dia plume dia total vel horiz vel vertl vel asp coeff print frq
    70.1     0.085   0.08500     5.546     5.546     0.000      0.10       500
port elev ver angle cont coef  effl den poll conc      decay  Froude # Roberts
F
     0.84       0.0       1.0    -2.893     6.1e8     55.26     37.50 1.049E-14
hor angle red space p amb den p current    far dif   far vel K:vel/cur Stratif
#
       90     7.315    23.270.00001000  0.000453      0.15    554600 4.089E-06
    depth    current    density   salinity       temp   amb conc   N (freq)  red
grav.
      0.0       1e-5      23.19      35.13      25.90          0   0.003473
0.2574
       76       1e-5      23.28      35.15      25.64             0 buoy flux
puff-ther
                                                               0.008100
72.19
```

Ilustr. 36 Sand Island blocked ports case.

Instead of using the ^C command, in going from one case to the next it is easier to use the <Page Up> key. Use it to create Case 3. Now make the changes indicated in Figure 36 to the ambient block (remember to delete the middle lines using ^YD), the title, and the [# ports] cell. To change the PLUMES configuration use the ^R command to obtain the Configuration menu, then toggle the CORMIX flow categorization feature -- simply press <C>. Notice that the configuration string at the bottom of the interface changes from ATNO0 to ATCO0 after which the flow category is given above the dialogue line: "CORMIX1 one port flow s5 unattached".

The PLUMES CORMIX classification algorithm is presently limited to single ports, thus the classification applies only to the unmerged region of the plume. Also, CORMIX is limited to predicting plume behavior in, at most, two layer systems. Consequently, the interface will not predict the flow category unless there are at least two and not more than three lines of ambient input information. This is one reason why the middle lines in the ambient block in Figure 35 have been deleted. (Also, the surface salinity and temperature cells have been arbitrarily adjusted to give about the same density gradient found between the 30 and 76 m depths, ignoring the measured values at 61 m.) In this case this is not a significant simplification, especially since the original density structure is not entirely self-consistent as is evidenced by the unstable layer in the third line of ambient stratification in Figure 34 (denser fluid of 23.31 sigma-t units would appear to lie over less dense fluid of 23.28 sigma-t units). This could be the result of measurement anomalies or a real transient condition.

Run Case 3. Notice that neither the dilutions nor the farfield concentrations are significantly reduced (UM from 134.2 to 124.8 and RSB 184.0 to 140.7, see Figure 37).

While you are still in Case 3, use the <Page Up> command to create Case 4. Earlier it was assumed that the effluent temperature was 25 C, its more "correct" value is 25.1 C. While this is a trivial change, go ahead and enter it anyway. Also, an effluent of 0.99979 gm/cc is reported; do not enter it just yet. While the differences are seemingly trivial, it does provide an opportunity to demonstrate the conflict resolution capability.

By now you are familiar with the fact that PLUMES differentiates between independent (yellow) variables, or values, that you provide (or accept by default) and dependent (white) variables that PLUMES can create on its own from the information you type into the spreadsheet interface. You may have wondered, "What would happen if I move to a cell which contains a white value and I input a new value, thus overriding the old value?" This is the primary reason why other programs do not allow the input of redundant variables. The danger is that you will either create an inconsistency or, as it is called in mathematics, overspecify the system. PLUMES has the capability to resolve many of these conflicts.

Go ahead and move the cursor in the effluent density [effl den] cell and, ignoring the dependent value it contains, use the ^K command to obtain the units of gm/cm3 and enter 0.99997 kg/m^3.

As soon as you are finished entering the data the background in the effluent salinity and temperature cells ([salinity] and [temp]), the plume (port) depth cell [port dep] and the brown effluent plume density [effl den] cells acquire a magenta background color and the 70.1 value in [port dep] should begin to blink. PLUMES has detected the conflict that your overriding of the density value has caused. You are now confined to the conflict resolution mode until you complete the actions shown in the dialogue window. The <space bar> will move you from cell to cell, showing its location by blinking the value in each in turn. You must determine which of the conflicting independent variables you wish to delete and then do so. That is the only normal way to leave the mode. In this case, move the cursor to the [salinity] and press <D> or the delete key. Immediately, PLUMES replaces it with the value of 3.600 o/oo.

plume dep	plume dia	poll conc	dilution	hor dis	
m	m			m	
70.10	0.08500	610000000	1.000	0.000	
68.53	2.388	18860000	31.21	5.957	
49.04	7.377	2578000	211.2	13.38<	merging
2.752	19.19	790700	561.3	17.97<	trap level
0.1349	20.25	758700	577.1	18.17<	surface hit

Farfield calculations based on Brooks (1960), see guide for details:

Farfield dispersion based on wastefield width of 1052m

--4/3 Power Law--		-Const Eddy Diff-				
conc	dilution	conc	dilution	distance	Time	
				m	sec	hrs
96780	579.9	96960	578.8	500.0	3212	0.9
10840	614.9	11120	599.2	1000	6546	1.8
1169	678.0	1249	633.6	1500	9879	2.7
124.8	754.9	139.8	672.4	2000	13210	3.7

===

<key>

RSB

Written by Philip J. W. Roberts (12/12/89)

(Adapted by Walter E. Frick (1/12/92))

Case: 3: Sand Island validation: blockage, min strat.

Lengthscale ratios are: s/lb = 0.44 lm/lb = 0.07

Froude number, u3/b = 0.00

Ilustr. 37 UM and RSB predictions for Case 3.

This new value has interesting implications. The question might be asked whether the effluent is indeed so saline, or is it more likely that suspended or other dissolved contaminants contribute to the density or is it a case of analytical measurement errors? This question will not be resolved here but may be important to pursue if the reduction in dilution caused by reduced buoyancy results in a standards violation. In any case, by running UM you find that the farfield bacterial count has been raised only a few percent and is still below the critical value.

Now we will create Case 5 to provide another variation of Case 2, the case with minimum stratification and no blockage. Use ^C and <2> to return to Case 2 followed by ^C and <5> to establish the new case. Our main objective is to analyze the effect of current, but first we will look at another bacterial contaminant that is regulated, Enterococcus, which is found in the effluent at 6.3×10^6 colonies/100ml. When you make just this change in the [poll conc] cell and run UM and RSB you see that UM provides a farfield dilution of 673.9 which is exactly the same as Case 2 and a plume concentration of 1.386 which is 1/96.6 of 139, the concentration found with Case 2. Of course this is the same fraction as 6.3×10^6 is of 6.1×10^8, which we expect.

RSB provides a greater farfield dilution (993.9) than UM because the initial dilution is higher. However, the 1.901 colonies/100ml is greater than the UM concentration because UM includes die-off in the initial dilution region while RSB does so only in the farfield. However, both RSB and UM plume concentrations are proportional to the effluent concentrations.

Both RSB and UM are now in agreement: the discharge will meet the Enterococcus water quality criterion of 7 colonies/100ml, predicting 1.901 and 1.386 colonies/100ml respectively. However, predicted concentrations are very sensitive to the value of the decay constant. For example, if the T90 time for Enterococcus is 1 hour and 15 minutes, rather than one hour, an increase of only 25%, the Enterococcus bacteria concentration predicted by UM increases more than six-fold, to 8.044 colonies/100ml, a value close to the numerical value of the Enterococcus standard. The corresponding RSB concentration is 10.18.

Before wrapping this example up, we will make one more change. It may be argued that it is unrealistic to subject the plume to zero current in the initial dilution (or rise) region and then assume that the subsequent current is 15 cm/sec. We will now examine the effect of current on predictions of the two models. Since this will practically assure that cross diffuser merging will take place, we will consider the effect of restoring 285 ports at half the spacing and compare the two results.

From Case 3 create Case 6 using the ^C command. Add to the title the word "current". Move to the ambient current cell [current]. Now type .1, i.e. 10 cm/sec, over each of the currents. Also, change the concentration in the [poll conc] cell to 6.3e6. The interface for Case 6 should agree with Figure 38.

The abbreviated predictions for both UM and RSB are shown in Figure 39. The UM average dilutions at the end of initial mixing (overlap is no longer a problem) and at 2000 meters are higher than RSB, 1667 compared to 1480 and 2131 versus 1838. The UM farfield Enterococcus concentration is disproportionally lower than the RSB value, 0.5343 compared to 2.187 colonies/100ml. Nevertheless, both RSB and UM predict that the standard would be met under these conditions. However, at 1500m the models would be in disagreement on the standard being met.

```
  Jun 23, 1992,  12:50:31  ERL-N PROGRAM PLUMES, Jun 10, 1992   Case:    6 of
6
  Title   Sand Island validation: blockage, min strat., current
   tot flow    # ports port flow   spacing  effl sal effl temp   far inc    far
dis
      4.469         142   0.03147      7.315          0          25         500
2000
   port dep  port dia plume dia total vel horiz vel vertl vel asp coeff print
frq
       70.1     0.085   0.08500      5.546      5.546     0.000       0.10
500
  port elev ver angle cont coef  effl den poll conc     decay  Froude # Roberts
F
       0.84       0.0         1.0     -2.893     6.3e6      55.26      37.50
0.01049
  hor angle red space p amb den p current    far dif    far vel K:vel/cur Stratif
#
         90      7.315       23.27     0.1000  0.000453        0.15       55.46
4.089E-06
      depth     current    density  salinity       temp   amb conc   N (freq) red
grav.
        0.0        0.1       23.19      35.13     25.90            0  0.003473
```

Ilustr. 38 Case 6, with current.

As before, part of the discrepancy between the models can be explained by the fact that when using RSB it is assumed that the concentrations are conservative throughout the initial dilution region. Since this zone is 275 m long, the effluent takes the better part of an hour to traverse this distance. We also note that the flux of material flowing through the RSB plume cross-section is geometrically consistent with an average dilution of 1828 ((0.1x1031.5x79.2)/4.469). UM splits the two RSB values well and would be an appropriate choice.

All along we have been finessing the issue of port blockage and using the spacing on one side of the diffuser (half the number of ports) versus using half the spacing (all ports). Of course, PLUMES can be easily set up to do either. When the number of ports in Case 6 is restored to 285 and the spacing is reduced to 3.858m (12ft) the initial dilution for UM increases from 1667 to 1690 and for RSB it decreases from 1480 to 1427. Neither change is really significant, although it may be in other circumstances. If the most conservative analysis still shows the standards will be met, the "right" answer is really irrelevant. However, if standards are not met then refinements are in order.

```
plume dep plume dia poll conc  dilution   hor dis
       m          m                           m
    70.10   0.08500   6300000     1.000    0.000
    69.22     2.082    195300     31.21    5.830
    62.52     7.347     28050     208.3    16.46< merging
    42.47     34.54      4950     997.8    45.46
    35.52     48.89      3433      1335    57.70< trap level
    30.02     64.53      2546      1667    70.17< surface hit
Farfield calculations based on Brooks (1960), see guide for details:
Farfield dispersion based on wastefield width of       1096m
   --4/3 Power Law--    -Const Eddy Diff-
      conc  dilution        conc  dilution  distance          Time
                                                m          sec   hrs
     406.3    1670.4       406.7    1669.0     500.0       2866   0.8
     45.97    1753.4       46.91    1717.4      1000       6199   1.7
     4.987    1921.2       5.285    1810.3      1500       9532   2.6
    0.5343    2131.3      0.5924    1917.9      2000      12870   3.6

===========================================================================

 <key>

                                    RSB
                  Written by Philip J. W. Roberts (12/12/89)
                    (Adapted by Walter E. Frick (1/12/92))

 Case:   6: Sand Island validation: blockage, min strat., current
```

Ilustr. 39 UM and RSB predictions for Case 6.

STEP 5. Using the Results in the Decision Making Process.

As we said it might, the analysis evolved along several paths and examined several issues. Yet, the higher flow cases are still not analyzed and the analysis remains incomplete. Completing the job is left an as exercise. However, given that the data was reliable and appropriate, that our conclusions about the proper use of UM and RSB are correct, and that this is the only contaminant of concern, it seems that the proposed plant expansion should meet the state's water quality criterion for Enterococcus. Thus, even doubling the flow rate would allow the standard to be met according to the UM predictions.

But how good are the input data? In the case of the decay constant, we have observed extreme sensitivity of the bacterial concentration to minor changes in the decay constant. Bacterial survival in ocean water is known to depend strongly on solar insolation, protozoan predation, and other factors. We also saw that dilutions and concentrations were sensitive to ambient current speeds and ambient density, and, in this case, less sensitive to port spacing.

With these considerations in mind, it is important for the analyst to obtain the best data possible and to encourage regulatory agencies to acquire environmental data over a wide range of conditions. Even then, it is apparent that judgment is also likely to play a role in the decision making process.

4. EXAMPLE: A CORMIX1 COMPARISON, DENSITY, STABILITY AND PROFILES

EXAMPLE: A CORMIX1 COMPARISON, DENSITY, STABILITY, AND PROFILES

INTRODUCTION

Beginning in 1973, the U.S. EPA sponsored research which ultimately led to a succinct, untuned statement of forced entrainment, the Projected Area Entrainment (PAE) hypothesis (Frick and Winiarski, 1975; Winiarski and Frick, 1976, 1978; Teeter and Baumgartner, 1979; Frick, 1984; Frick, Baumgartner, and Fox, 1993). Models using PAE, all currently expressed in the Lagrangian formulation, include OUTPLM (Teeter and Baumgartner, 1979), UMERGE (Muellenhoff et al., 1985), UM, and JETLAG (Lee and Cheung, 1990). Sometimes criticized, the work was recently verified and justified by Lee and Cheung (1990) and Cheung (1991). Cheung (1991) shows that JETLAG, a three-dimensional plume model, clearly outperforms the highly regarded Chu (1975) and Schatzmann (1979) models in predicting trajectory and dilution constants in asymptotic flow regimes. It also indicates the correct power law dependence of the trajectory in different flow regimes. Frick, Baumgartner, and Fox (1993) demonstrate the similarity between UM and JETLAG for two-dimensional flow, showing that Cheung's conclusions concerning JETLAG apply to UM as well.

Nevertheless, while it should be possible to apply the PAE hypothesis to plume behavior in general, the EPA UM model is presently limited to simple merging geometries and surface interaction phenomena. Thus, it performs best when plumes are discharged in deep water. It is also a two-dimensional model, though an experimental three-dimensional vector version exists.

These limitation were a consideration in EPA's decision to develop the empirically based EPA CORMIX models, or expert systems (summarized by Hinton and Jirka, 1992). CORMIX stands for CORnell MIXing zone models. The idea was to exploit accumulated laboratory and field experience to compile a set of methods and empirical models to bridge the gaps evident in theoretical modeling.

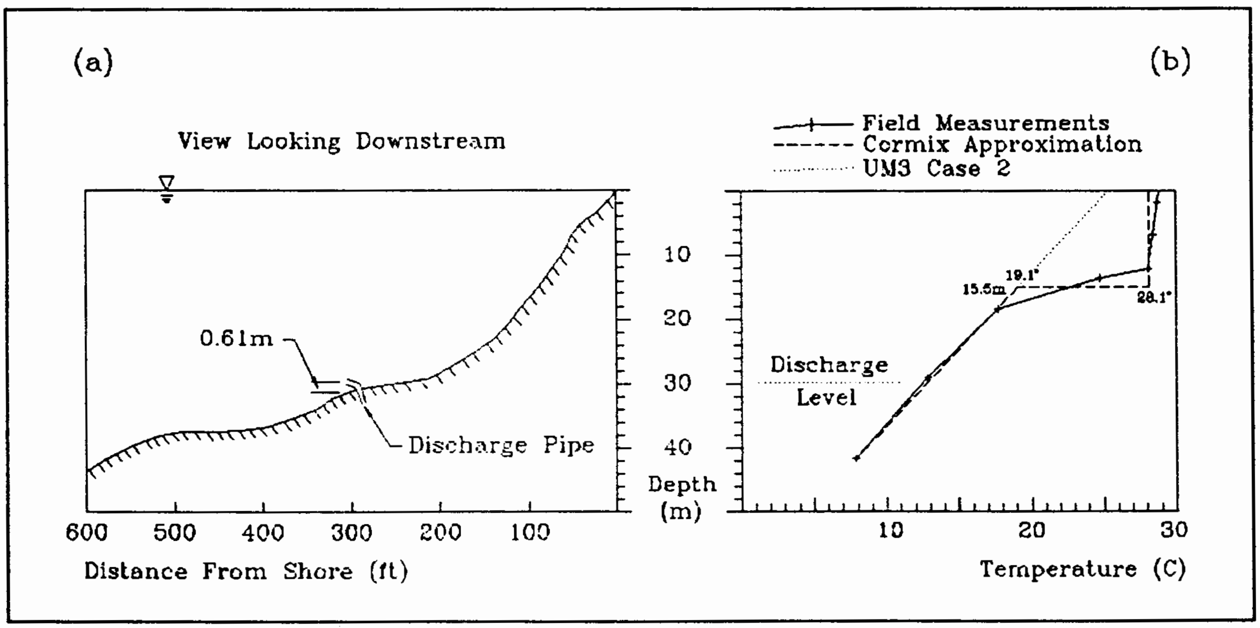
The Cornell initiative resulted in the development of CORMIX1, CORMIX2, and CORMIX3 for the analysis of submerged single port discharges, submerged diffusers, and surface discharges, respectively. About 80 different diffuser and ambient profile combinations, or **flow classifications** are represented.

Empirical models are most effective when prototype and model variables and conditions are highly similar. When these conditions are not met the predictions can degrade rapidly. In other words, it is often difficult to extrapolate to conditions which were not included in the experimental design on which the models are based. Since it is often not clear to the user when extrapolation occurs, this can be a real problem. This example demonstrates some of the pitfalls. They apply, though not to the same degree, to theoretical models as well.

Nevertheless, it is worthwhile to maintain and promote empirical plume modeling models along with theoretical approaches. The challenge is to find ways to make it possible to benefit from both while facilitating and enhancing the analysis process, perhaps even reducing the level of effort necessary in the long run. Combining the Cornell CORMIX flow classification schemes in the PLUMES interface is an attempt to achieve this goal. In principle, it makes a screening level analysis possible which provides information on its own efficacy, and recommendations for further analysis, if necessary. Ultimately, a way may be even found to include the CORMIX computational modules within PLUMES.

An example comparing the UM and CORMIX models is presented to give you an appreciation of how PLUMES may be used to help you assess the appropriate uses for CORMIX, RSB, and UM. At the same time it will help you understand the differences between the plume models, their strengths and weaknesses. The example chosen is from Appendix B of Hinton and Jirka (1992), in which a full statement of the problem and a description of the CORMIX1 solution is found. It is a single port problem (CORMIX1), chosen in part because, at the time of this writing, the CORMIX2 and CORMIX3 flow classification algorithms were not available.

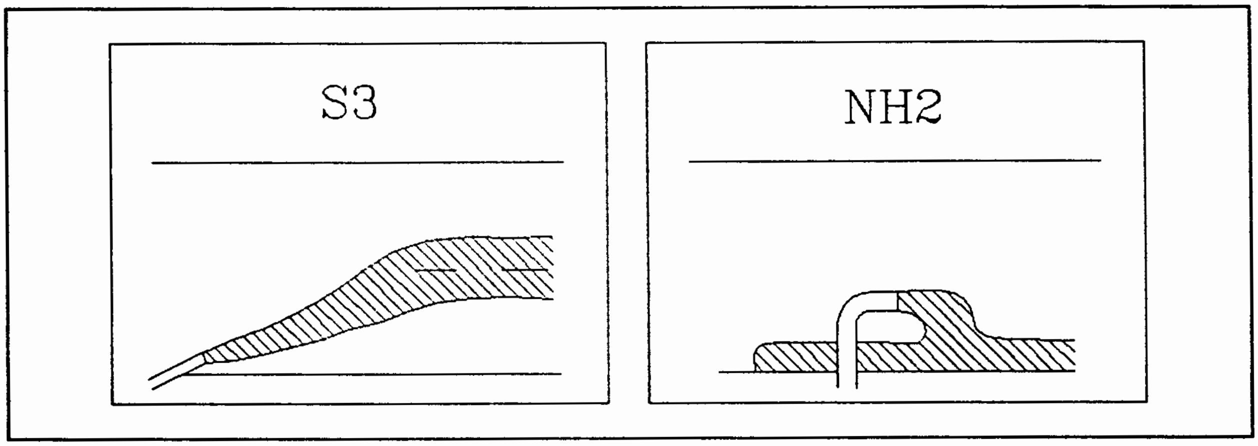
In addition to the references to CORMIX, this example provides an opportunity to explore the very important roles of density and stability in plume behavior and modeling. They are the sources of some of the pitfalls alluded to above. It also addresses the relationship between average and centerline plume properties.



Ilustr. 40. (a) Reservoir cross-section. (b) Temperature profile.

PROBLEM

A manufacturing plant is discharging effluent into a reservoir. The effluent of 3.5 MGD contains chlorides at a concentration of 3500 ppm (3.5 o/oo) and is released at a temperature of 20 C. The reservoir is large and deep, a cross-section is shown in Figure 40(a). The discharge is at a depth of 29.9 m, 0.6 m off the bottom, and is directed upward at an angle of 10 degrees, whose horizontal projection is perpendicular to the current. The port diameter is 25.4cm. In summer, the temperature profile (CORMIX approximation) of the lake is 29 C at the surface and 28.1, 17.5, and 11 C at depths of 15.5, 15.5, and 35 m respectively as shown in Figure 40(b). The current in the bottom layer is small: 0.015 m/s.



Ilustr. 41. Schematics of flow classifications S3 and NH2 (Hinton and Jirka, 1992).

The maximum allowable concentration is 1200 ppm of chloride and the allowable continuous concentration is 600 ppm. The mixing zone boundary is 60 m away from the port. CORMIX1 calculates an effluent density of 998.3872 kg/m^3 and an ambient density of 999.6476 kg/m^3.

Using a layer boundary depth of 15.5 m, CORMIX predicts the flow class S3 for this example. No bottom attachment is indicated. The dilution at the boundary of the specified regulatory mixing zone is predicted to be 11.9 at a depth of 27.5 m. This is a centerline dilution -- the average dilution would be significantly greater. The dilution is sufficient to meet both the maximum and continuous criteria.

Because the plume is expected to trap in the stable bottom layer, we expect the PLUMES UM model to simulate this case well, even though some of the underlying assumptions are not met. RSB, as a multiple port diffuser model, is not applicable. If issued, the RSB command will cause the message "Use RSB for multiple port diffusers" to appear in the dialogue window.

ANALYSIS

General Considerations

To begin this exploration of the relationship of the UM model to CORMIX and the issues of density, stability, and plume profiles, start PLUMES and type in the data as shown if Figure 42. (Since the PLUMES interpolation capability will be demonstrated, leave the blank cells in the ambient block as shown. Since the Configuration string shows that the auto ambient option is on, which will normally provide a default value for these cells, you can turn it off. Alternatively, you could move around the cells, or delete them the default values.)

```
  Jun 23, 1992,  20: 6:45  ERL-N PROGRAM PLUMES, July 1, 1992   Case:   1 of
1
  Title   CORMIX1 example, H&J 1992

   tot flow   # ports port flow   spacing  effl sal effl temp   far inc   far
dis
     0.1533          1    0.1533      1000        3.5        20        20
60
   port dep  port dia plume dia total vel horiz vel vertl vel asp coeff print
frq
       29.9    0.2540    0.2540     3.026     2.980    0.5255      0.10
500
  port elev ver angle cont coef  effl den poll conc     decay  Froude # Roberts
F
        0.6        10       1.0    0.9296      3500         0

  hor angle red space p amb den p current   far dif   far vel K:vel/cur Stratif
#
         90    1000.0             0.01500  0.000453     0.015     201.8

      depth   current   density  salinity      temp  amb conc  N (freq)  red
grav.
```

Ilustr. 42. First draft input for the CORMIX1 example.

Several assumptions and statements concerning the input should be clarified: **The default port spacing [spacing] of 1000m is appropriate. It means that merging will not occur because the plumes will never grow that large and thus UM will run as a point source model.** Also, as a first cut, the effluent salinity cell is input as 3.5 o/oo (3500 ppm) even though, since the effluent is neither fresh nor sea water, the PLUMES equation of state is not valid for for accurately estimating the density of the effluent. The ^K (units conversion) command has been used to convert units in several cells, e.g. to input 3.5 MGD in the total flow cell. The ambient depth of 29.9m has been entered into the ambient block while the density, salinity, and temperature cells have been left blank. PLUMES will be used to determine the ambient properties at port depth.

It is assumed the effluent is co-flowing, i.e. is discharged in the direction of the current, even though it contradicts the actual geometric flow configuration. This is necessary because for single ports UM is a two-dimensional model. A horizontal angle [hor angle] of 90 degrees indicates the plume will be modeled as a co-flowing case. It is a justifiable assumption because the current is small. Furthermore, it is a conservative assumption because entrainment will be underestimated and, therefore, dilution will be less than it would be otherwise. This is due to the fact that the plume will project less area to the current and therefore forced entrainment will be reduced. CORMIX1 works in a somewhat similar fashion, patching together different modules valid for different parts of the plume's trajectory.

Before proceeding, it is good practice to assure that the model configuration options are properly set. Use the <Show configuration> command, ^RS, to show the current settings. It shows that the CORMIX1 classification algorithm is currently inactive. Since we want to illustrate the association between PLUMES and CORMIX, use the <Cormix1 categories> command, or <C>, to activate the option. The third letter in the configuration string at the bottom of the screen will change from "N" to "C" (e.g., ATNO0 to ATCO0). The new configuration string is shown in Figure 44.

```
                         PLUMES Configuration

A: Automatic ambient fill is on
R: Brooks equation input deleted
C: The CORMIX flow categorization algorithm is active
O: UM farfield predictions begin at element overlap
   Farfield model initiation choices are:
   M: maximum rise; O: element overlap; P: pause criterion.
   Other criteria, such as surface interaction, will override these choices.

0: Brunt-Vaisala reversals determined by UM as 1 or 2

...
```

Ilustr. 43. The PLUMES configuration.

To establish the proper, interpolated temperature at the 29.9 m depth in the ambient block, put the cursor in the temperature cell at 35 m depth. Since the automatic ambient fill option is on you may have to use the ^T command to keep the 29.9 m temperature cell empty after the cursor is moved through it. From the 35 m temperature cell issue the command ^YI; the correct **interpolated** temperature, 13.0 C, appears immediately as shown in Figure 44. The same could be done for the salinity cell although it will be easier to simply move the cursor through the cell (or input 0 if the automatic ambient fill option is off). The ambient density will be calculated automatically upon moving from the salinity cell. While density issues will be discussed further, it is worth noting here that interpolating temperature and salinity value, will not result in the same density as interpolating on density directly. The interpolated values are also shown in Figure 44.

Notice that the **CORMIX window** near the bottom of the screen states: "CORMIX1 algorithm limited to three lines of ambient". This is a limitation of the PLUMES interface which does not yet implement the full CORMIX classification algorithm. (Furthermore, the abridged version that is implemented has not been reviewed by authors of the CORMIX models.) In some cases it is possible to circumvent this limitation. For example, if the plume remains in the bottom layer the details of the ambient temperature near the surface will be superfluous, making it possible to simplify the ambient profile in order to obtain the CORMIX flow class.

```
 Jun 23, 1992,  20:10:37  ERL-N PROGRAM PLUMES, July 1, 1992   Case:   1 of
 1
 Title   CORMIX1 example, H&J 1992

  tot flow   # ports port flow   spacing  effl sal effl temp   far inc   far
dis
    0.1533          1    0.1533       1000        3.5          20          20
 60
  port dep  port dia plume dia total vel horiz vel vertl vel asp coeff print
frq
      29.9    0.2540    0.2540     3.026     2.980    0.5255       0.10
 500
 port elev  ver angle  cont coef   effl den  poll conc       decay   Froude #
Roberts F
       0.6         10        1.0     0.9296       3500           0     -49.73
0.0003835
 hor angle  red space  p amb den  p current    far dif    far vel  K:vel/cur
Stratif #
        90     1000.0    -0.5584    0.01500   0.000453      0.015      201.8
-0.01961
     depth    current    density   salinity      temp  amb conc  N (freq)  red
grav.
```

Ilustr. 44. Interface with CORMIX flow category and interpolated temperature.

```
plume dep plume dia poll conc  dilution   hor dis

        m         m                              m

    29.90    0.2540      3500     1.000     0.000

    28.82     4.045     212.8     16.47     9.574

< only growth and aspiration entrainment after this point

< local maximum rise or fall

    30.50     7.458     109.4     32.05     19.49

    30.55     7.507     108.6     32.27     19.64< trap level

    31.37     8.660     93.91     37.32     22.92< bottom hit

    31.72     10.00     81.19     43.17     26.55< bottom hit

< only growth and aspiration entrainment after this point

< local maximum rise or fall< plume element overlap.
```

Ilustr. 45. UM and RSB output for the draft case (Case 1).

(In other cases, the two layer approximation used in CORMIX may be inappropriate.) Thus UM may be used to predict the rise of the plume which shows, after the fact, that the simplification is appropriate (i.e. the rise is limited to the bottom layer). The predictions are given in Figure 45.

UM predicts a plume concentration of 212.8 at maximum rise at a downstream distance of 9.574 m and a depth of 28.82 m. This is clearly in the bottom stratified layer and within the specified mixing zone of 60 m. Thus, the simplification of the ambient data to two lines of data, as is done in Case 2, is appropriate. Consistent with the predictive strategy for negatively buoyant plumes indicated by the configuration string, the UM prediction continues past the point of maximum rise. With the <Reversals set> option set to 0, UM determines the number of reversals, i.e. levels of maximum rise and fall, to be two if the effluent is negatively buoyant. The average concentration at maximum fall is 81.19.

UM may also be used to estimate plume centerline concentrations. For this purpose there is a centerline concentration [CL conc] cell which normally does not appear on the interface but may be revealed by manipulating the Pause cell located near the lower right hand corner of the interface, below the red block of cells. To get the centerline concentration into the Pause cell use the <pauSe cell> command, ^YS, on the Miscellany menu. When you do the dialogue

```
Back, Inequalities, Output, Variables(space), or <esc>
```

Ilustr. 46. The Pause cell dialogue window.

window shown in Figure 46 appears. Press space bar to move through the list of available cells until the centerline concentration [CL conc] cell appears. If you go too fast and pass it by you can return to it by pressing <B>. When the [Cl conc] cell appears press <O> to put the cell on the output table. The left byte of the cell will turn blue to indicate the variable will be output.

Run UM again. The results are shown in Figure 47. We see that the maximum rise average and centerline concentrations are 424.7 and 212.8 ppm respectively. At maximum fall, the corresponding concentrations are 171.1 and 81.19 ppm. Note that the ratio of the centerline to average concentration is not constant but increases from 1.0 at the source to 2.1 (171.1/81.19) at the end of the initial dilution zone. In some cases it will continue to increase to its theoretical limit, for single plumes, of 3.89. All concentrations are well below the 600 ppm standard.

Notice that a centerline concentrations is not given in the farfield. An estimate of the centerline concentration in the farfield could be obtained by multiplying the average concentration by the same factor, or peak-to-mean ratio, obtained in the initial dilution phase, i.e. 2.1.

plume dep	plume dia	poll conc	dilution	CL conc	hor dis
m	m				m
29.90	0.2540	3500	1.000	3500	0.000
28.82	4.045	212.8	16.47	424.7	9.574
< only growth and aspiration entrainment after this point					
< local maximum rise or fall					
30.50	7.458	109.4	32.05	225.6	19.49
30.55	7.507	108.6	32.27	224.1	19.64< trap level
31.37	8.660	93.91	37.32	195.7	22.92< bottom hit
31.72	10.00	81.19	43.17	171.1	26.55< bottom hit

Ilustr. 47. UM output with centerline concentrations.

If we were confident about the assumptions the analysis would be complete; after all, the standards are met under relatively conservative conditions (e.g. at the first maximum rise for a co-flowing plume). The same basic conclusion that the criteria will be met has been reached by the PLUMES and CORMIX1 analyses. However, it is instructive to continue with the analysis -- especially as it serves the purpose of further illustrating the subtleties of the modeling process.

Ambient Profile Simplification

If the assumption that the effluent brine density obeys the PLUMES equation of state were valid, there would be no reason to continue the analysis. However, it is clear that CORMIX uses another relationship and therefore the assumption is questionable. Further examination shows that the effluent density calculated by CORMIX1 corresponds closely to that of fresh water. Of course, the freshwater assumption is equally tenuous because chloride is a major constituent of denser sea water and the effluent should probably exhibit a greater density. In any case, it is sobering to see how little it takes to switch from one flow pattern shown in Figure 41 (S3) to another (NH2). Thus, **understanding the role that density plays in plume behavior and ambient stability is very important.**

Continue with the analysis by forming a new case, Case 2, using the ^C command or <Page Up> key. Then, delete the three intermediate lines from the ambient block using the ^YD command. Finally, move to the surface ambient temperature cell and type in 25.5 (the extension of the bottom temperature gradient shown as a dotted line in Figure 40). When you are done the interface screen should look like Figure 48.

PLUMES predicts a negatively buoyant flow classification type: NH3 with [bottom] attachment a5 (see Doneker and Jirka, 1990 for schematic descriptions of these classes). That the plume is negatively buoyant is also apparent from the fact that the effluent density (0.9296 sigmaT) in the brown block is greater than the ambient density (-0.7222 sigmaT) in the green block and the Froude # is negative. The NH3 is a classification similar to the NH2 classification given if Figure 41. It differs significantly in character from the S3 classification.

```
 Jun 24, 1992,   12:49:36  ERL-N PROGRAM PLUMES, July 1, 1992   Case:   2 of
2
  Title   CORMIX1 example, H&J 1992, reduced ambient lines
   tot flow   # ports port flow   spacing  effl sal effl temp   far inc   far
dis
     0.1533          1   0.1533      1000        3.5         20         20
60
   port dep  port dia plume dia total vel horiz vel vertl vel asp coeff print
frq
       29.9    0.2540    0.2540     3.025     2.979    0.5254       0.10
500
  port elev ver angle cont coef  effl den poll conc     decay  Froude # Roberts
F
        0.6          10         1.0    0.9296       3500          0     -47.19
0.0003455
  hor angle red space p amb den p current   far dif   far vel K:vel/cur Stratif
#
         90     1000.0    -0.7222    0.01500  0.000453      0.015       201.7
-0.01183
      depth    current    density  salinity       temp  amb conc  N (freq) red
grav.
        0.0      0.015     -3.022         0       25.5         0   0.02748
-0.01618
       35.0      0.015    -0.3299         0         11         0 buoy flux
puff-ther
                                                             -0.002481
17.95
                                                                 jet-plume
jet-cross
                                                                 11.28
45.40
                                                                 plu-cross
jet-strat
                                                                 735.1
4.979
                                                             plu-strat
                                                                 3.307
                                                               CL conc>=
```

Ilustr. 48. Simplified Case 1 input to enable the CORMIX flow classification algorithm in PLUMES.

It is gratifying that the overall flow characterization is essentially equivalent to the one analyzed in Case 1, as it should be, i.e. it is negatively buoyant. Specific numerical differences with the previous case may be attributed to small inaccuracies in specifying the surface temperature, which was estimated graphically.

For the sake of comparison, we will attempt to correct the discrepancy between the two models by revising the assumption that the discharged chloride brine has the same equation of state as that built into the interface. To do so, make a new case, Case 3. Then move the cursor to the effluent plume density [effl den] cell, invoke the ^K command, and input the effluent density in kg/m^3 given in the CORMIX1 analysis: 998.3872. After attempting to move from the cell, the conflict resolution mode will trap the overspecification. Press <space bar> to move to the effluent salinity cell and delete its value. The interface screen should now look like

```
 Jun 24, 1992,    9: 6:50  ERL-N PROGRAM PLUMES, July 1, 1992   Case:   3 of
 3
  Title   CORMIX1 example, H&J 1992, effl den = 998.3872

   tot flow   # ports port flow   spacing  effl sal effl temp   far inc   far
dis
     0.1533         1    0.1533      1000   0.1573        20        20
  60
   port dep  port dia plume dia total vel horiz vel vertl vel asp coeff print
frq
       29.9    0.2540    0.2540     3.025     2.979    0.5254      0.10
  500
  port elev ver angle cont coef  effl den poll conc      decay  Froude #
Roberts F
        0.6        10       1.0    -1.613      3500          0     64.17
0.0006391
  hor angle red space p amb den p current    far dif    far vel K:vel/cur
Stratif #
         90    1000.0   -0.7222   0.01500  0.000453      0.015     201.7
0.02193
      depth   current   density  salinity      temp  amb conc  N (freq) red
grav.
        0.0     0.015    -3.022         0      25.5         0   0.02748
0.008750
       35.0     0.015   -0.3299         0        11         0 buoy flux
puff-ther
                                                          0.001341
22.03
                                                             jet-plume
jet-cross
                                                             15.35
45.40
                                                             plu-cross
jet-strat
                                                             397.5
4.979
                                                          plu-strat
```

Ilustr. 49. The interface screen after correction of CORMIX effluent density, with output.

that in Figure 49. The effluent salinity now indicates 0.1573 o/oo which supports the conclusion that the density of fresh water is used in the CORMIX example with chloride being treated as a noncontributing component to density. With this assumption, the flow classification now agrees with the CORMIX1 prediction -- both are S3, with no bottom attachment in the initial dilution region.

The corresponding simulation is also shown in Figure 49. The predicted dilution is now 31.53 at the end of the initial dilution zone, i.e. at maximum rise. This is almost twice large as the dilution found in Case 1 and consequently, if the density assumption were valid, which it is not, the criterion for chloride will be easily met. (Note the inverse relationship between concentrations and dilutions.) Consistent with the fact that the plume is now positively buoyant (-1.613 sigmaT < -0.7222 sigmaT), the farfield model begins at maximum rise and the advisory message about growth and aspiration entrainment may be ignored.

Density: The Linear and Nonlinear Forms of UM

At this point in the analysis you have reasons to lose confidence in the equations of state used in CORMIX and PLUMES. Obviously, the PLUMES equation of state applies strictly only to fresh and sea water and the limitations of the CORMIX method are uncertain. There is, however, another option, in UM a linear form of the equation of state. In this form, the density is assumed to be linear function, i.e. to have a constant coefficient of bulk expansion. Essentially the density is a weighted average of the densities of the plume and ambient fluids. It is a useful approximation in many cases where the non-linear form may be grossly inappropriate. However, it is totally inadequate for predicting nascent density effects.

The linear equation of state is invoked simply by entering densities instead of salinity and temperature, which are left undefined. In this mode the complex equation of state built into PLUMES is ignored in favor of the simple linear equation of state. To illustrate, create Case 4 pressing the <Page Up> key in Case 3. Then override the values in the ambient density cells. When you are done the interface and simulation should appear as in Figure 50.

In this case the differences with the previous run in Case 3 are relatively small. The predicted dilution at maximum rise for the linear form is 29.02 compared to 31.53 for the nonlinear. The differences in rise are correspondingly small: 2.63 m (29.90 - 27.27) for the linear form compared to 2.90 m (29.90 - 27.00) for the nonlinear form.

In most cases not involving suspended and dissolved solids, except for sea salt, it is best to use the nonlinear form of UM, i.e. to specify salinity and temperature rather than only density as input. It is recommended because the equation of state of water, especially fresh, cold water, is significantly nonlinear. For plumes discharged to very cold, fresh water, the linear form of the model can lead to significant errors in the predictions, in extreme cases predicting monotonically rising plumes where, in fact, real plumes will rise briefly before sinking to the bottom (Frick and Winiarski, 1978). This is the nascent density effect described in the first chapter.

To illustrate this very interesting behavior, consider the case of a highly buoyant plume discharged to fresh water near the freezing temperature. This is a common occurence in cold climates with discharges to fresh water bodies. From Case 4 press <Page down> to create Case 5. Now enter the temperatures shown in Figure 51. After you are finished run this **nonlinear** form of UM. The predicted plume reaches a trapping level at the 27.07 m level and, expending its vertical momentum, rises to a depth of 24.35 m. At this point the plume is, and remains, negatively buoyant and, therefore, descends back to the bottom.

```
 Jun 24, 1992,    9: 7:56   ERL-N PROGRAM PLUMES, July 1, 1992   Case:   4 of
4
 Title   CORMIX1 example, H&J 1992, linear equation of state

  tot flow   # ports port flow   spacing  effl sal effl temp   far inc   far dis
    0.1533          1     0.1533      1000                                 20
60
  port dep  port dia plume dia total vel horiz vel vertl vel asp coeff print frq
      29.9    0.2540    0.2540     3.025     2.979    0.5254      0.10       500
 port elev ver angle cont coef  effl den poll conc     decay  Froude # Roberts
F
       0.6        10       1.0    -1.613      3500         0     64.17 0.0006391
 hor angle red space p amb den p current   far dif   far vel K:vel/cur Stratif
#
        90    1000.0   -0.7222   0.01500  0.000453     0.015     201.7   0.02193
     depth   current   density  salinity      temp  amb conc  N (freq) red grav.
       0.0     0.015    -3.022                             0   0.02747  0.008750
      35.0     0.015   -0.3299                             0 buoy flux puff-ther
                                                            0.001341     22.03
                                                           jet-plume  jet-cross
                                                               15.34     45.40
                                                           plu-cross  jet-strat
                                                               397.5     4.979
                                                            plu-strat

                                                               2.836

                                                              CL conc>=



 CORMIX1 one port flow s3 unattached.  Use UM. (See manual)

   deg C,  deg F                                        -2.0 to 50 deg C range

 Help: F1.  Quit: <esc>.  Configuration:ATCOO.  FILE: cormixl.var;

 plume dep plume dia poll conc  dilution   CL conc   hor dis
```

Ilustr. 50. The linear equation of state mode of UM.

The reason for this behavior is due to the fact that water has its maximum density around 4C. Initially the plume is very buoyant (-7.724 sigmaT < 0.03870 sigmaT), but, as the plume ascends in the water column, it rapidly cools through entrainment and becomes more dense than the ambient fluid as the average density of the plume element approaches 4 C. At that temperature it is considerably more dense than its surroundings which has a temperature somewhere between 0 and 4C at this point. Consequently, the upward ascent of the plume is first inhibited and finally reversed due to the negative buoyancy.

```
 Jun 24, 1992,   10: 1: 5  ERL-N PROGRAM PLUMES, July 1, 1992   Case:   6 of
6
 Title   CORMIX1 example, linear mode, very cold ambient

  tot flow   # ports port flow    spacing  effl sal effl temp   far inc    far
dis
    0.1533          1    0.1533      1000                           20
60
  port dep  port dia plume dia total vel horiz vel vertl vel asp coeff print
frq
      29.9     0.2540     0.2540      3.025      2.979     0.5254      0.10
500
 port elev ver angle cont coef  effl den poll conc     decay  Froude # Roberts
F
       0.6           10          1.0      -7.724        3500            0
21.860.00007414
 hor angle red space p amb den p current   far dif   far vel K:vel/cur Stratif
#
        90      1000.0    -0.09290     0.01500  0.000453      0.015       201.7
5.706E-08
      depth    current    density  salinity       temp  amb conc   N (freq)  red
grav.
       0.0      0.015   -0.09295                                0 0.0001297
0.07542
      35.0      0.015   -0.09289                                0  buoy flux
puff-ther
                                                            0.01156
10.74
                                                                 jet-plume
jet-cross
                                                               5.227
45.40
                                                                 plu-cross
jet-strat
                                                                3426
72.47
                                                          plu-strat
```

Ilustr. 51. Discharge of a highly buoyant plume to very cold water; linear form of UM. With output.

To compare this simulation to one with the linear model, form Case 6 starting from Case 5 and override all the dependent (white) densities with equivalent independent densities. First erase any salinity or temperature values. (Or, if the conflict resolution capability is used, the ^QD command is handy for moving to the end of the cell where you can add an extra zero to the replacement string so that PLUMES knows that the densities are to become independent). When you are done the interface should look like that in Figure 52.

In this, a superficially identical case, the plume rises to the surface. Clearly it is important to be aware of these extreme differences in model behavior. They are not both right. Depending on the analysis, in one case one would conclude that benthic organisms will be affected, in the other, surface organisms. Thus, whenever the data are available and suspended and dissolved solids (excluding sea salt) are not an important factor, the nonlinear equation of state should be used.

```
 Jun 24, 1992,    9:57:22  ERL-N PROGRAM PLUMES, July 1, 1992   Case:   5 of
5
  Title   CORMIX1 example, non-linear mode, very cold ambient

   tot flow   # ports port flow   spacing  effl sal effl temp   far inc    far
dis
     0.1533         1    0.1533      1000         0        40        20
60
   port dep  port dia plume dia total vel horiz vel vertl vel asp coeff print
frq
       29.9    0.2540    0.2540     3.025     2.979    0.5254      0.10
500
  port elev ver angle cont coef  effl den poll conc    decay  Froude # Roberts
F
        0.6          10         1.0     -7.724        3500             0
21.860.00007415
  hor angle red space p amb den p current   far dif   far vel K:vel/cur Stratif
#
         90     1000.0   -0.09290   0.01500  0.000453      0.015       201.7
6.449E-08
      depth    current    density  salinity      temp   amb conc  N (freq)  red
grav.
        0.0      0.015   -0.09295         0         0          0 0.0001378
0.07542
       35.0      0.015   -0.09289         0     0.001          0 buoy  flux
puff-ther
                                                               0.01156
10.74
                                                                   jet-plume
jet-cross
                                                                 5.227
45.40
                                                                   plu-cross
jet-strat
                                                                  3426
70.29
                                                            plu-strat
```

Ilustr. 52. Discharge of a highly buoyant plume to very cold water; nonlinear form of UM. With output.

The densities of fresh water at several temperatures used in this example are compared in Figure 53.

Tabla III. PLUMES and CORMIX1 densities compared with published values (Weast, 1977).

FRESHWATER

Temperature	------- Densities (kg/m^3) -------	
(C)	UM	CORMIX1

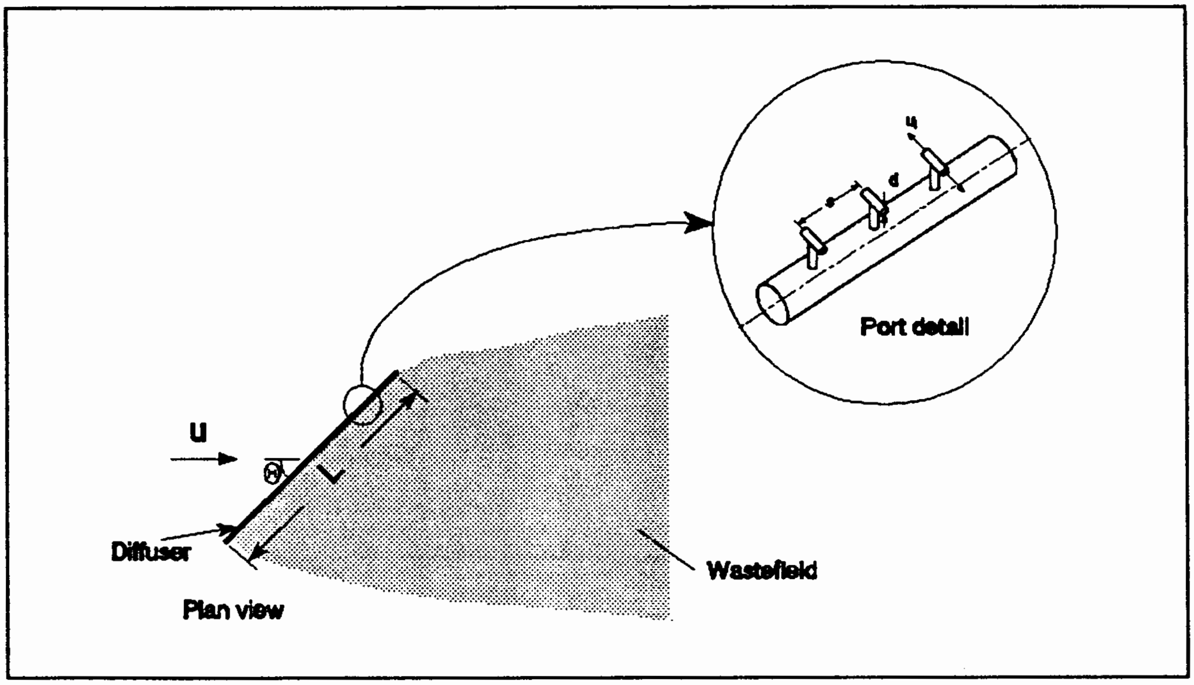
5. THE ROBERTS, SNYDER, BAUMGARTNER MODEL: RSB

THE ROBERTS, SNYDER, BAUMGARTNER MODEL: RSB

INTRODUCTION

RSB is based on the experimental studies on multiport diffusers in stratified currents described in Roberts, Snyder, and Baumgartner (1989a,b,c), which should be consulted for detailed explanations. These studies were conducted with an experimental configuration shown in Figure 53. The diffuser is straight and consists of horizontally discharging round nozzles which are uniformly spaced. The ports discharge from both sides of the diffuser, which is similar to most prototype applications. This configuration would include diffusers consisting of pipes with ports which are holes along each side or T-shaped risers each containing two ports as shown in Figure 53.

The receiving water is linearly density-stratified, and flows at a steady speed at an arbitrary angle relative to the diffuser axis. RSB is intended for stratified conditions producing a fully submerged wastefield; other models should be used for surfacing wastefields, for example ULINE (Muellenhoff et al., 1985).



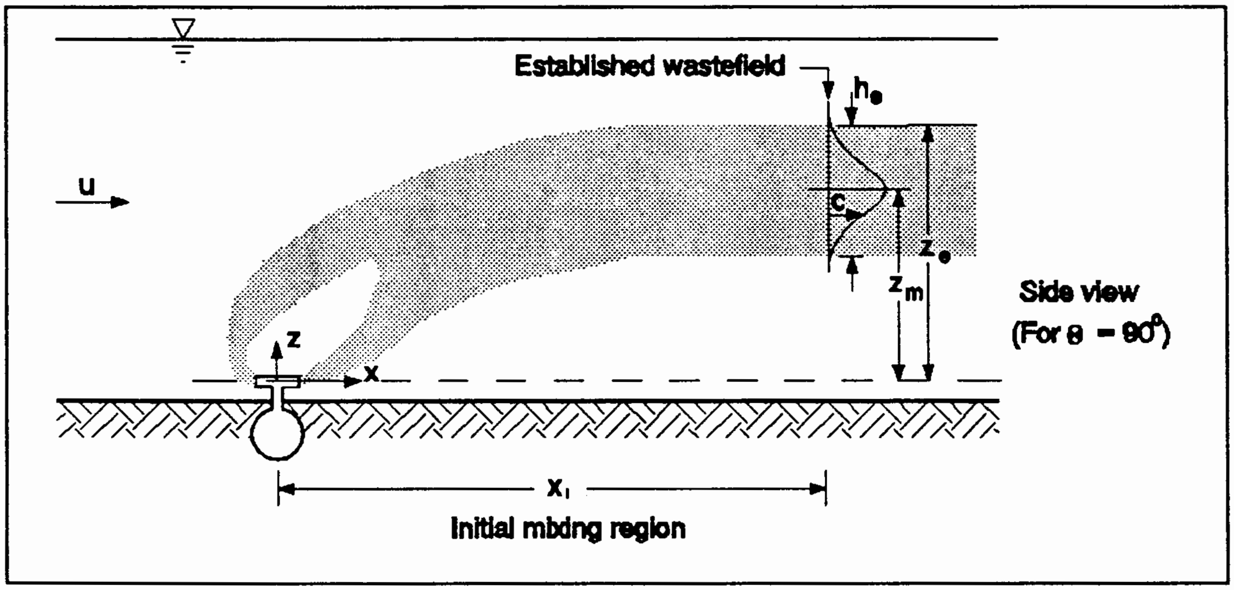
Ilustr. 53. Diffuser configuration considered by RSB.

As discussed later, RSB is also a good approximation for diffusers in which the ports are clustered in multiport risers, at least up to 8 ports per riser. The range of the experimental parameters (port spacing, port diameter, jet exit velocity, current speed, current direction, and density stratification) was chosen to be representative of highly buoyant discharges such as domestic sewage and some industrial wastes into coastal and estuarine waters. When RSB is used outside the parameter range for which these experiments were conducted, it extrapolates the results to obtain a solution and gives a warning that the answers are extrapolated.

The model can be thought of as a replacement for and a significant update of ULINE (Muellenhoff et al., 1985). Whereas ULINE was based on experiments in unstratified environments, RSB is based on experiments in stratified environments, and so is therefore more reliable in this situation. Also, ULINE applies only to single line plumes whereas RSB is based on experiments with multiport diffusers. It therefore includes the effects of port spacing and source momentum flux, and is more realistic in that it includes discharges from both sides of the diffuser.

DEFINITIONS

The definitions used in RSB in relation to the geometry of the initial mixing region are shown in Figure 54 and described below. At the end of this region the dilution is called the initial dilution and the wastefield is said to be established. The established wastefield then drifts with the ocean currents and is diffused by oceanic turbulence.



Ilustr. 54. Definition of Wastefield Geometry.

In RSB this "initial mixing region" or "hydrodynamic mixing zone" is defined to end where the self-induced turbulence collapses under the influence of the ambient stratification and initial dilution reaches its limiting value. The length of the initial mixing region is denoted by x_i. The geometrical wastefield characteristics (see Figure 53) at this point are thickness h_e height to top z_e and height to level of maximum concentration (or minimum dilution) z_m. The minimum initial dilution S_m is defined as the smallest value of dilution (corresponding to maximum concentration) observed in a vertical plane through the wastefield at the end of the initial mixing region.

MODEL BASIS

The initial mixing of wastewater discharged from a multiport diffuser depends on diffuser design and receiving water characteristics. The diffuser can be characterized by fluxes of volume, momentum, and buoyancy per unit diffuser length:

$$q = \frac{Q}{L} \qquad m = uq \qquad b = g_o' q \tag{19}$$

where Q is the total discharge, L the diffuser length, u_j the jet exit velocity, and $g_o' = g(\rho_a - \rho_o)/\rho_a$ is the reduced gravitational acceleration due, g is the acceleration due to gravity, ρ_a is the ambient density at the level of the ports and ρ_o the effluent density. A linear density stratification can be characterized by the buoyancy frequency, N, also referred to as the Brunt-Vaisala frequency, usually expressed in units of sec^{-1}:

$$N = \pm\left(\frac{g}{\rho_a}\frac{|d\rho|}{dz}\right)^{1/2} \tag{20}$$

We define three length scales:

$$l_q = \frac{q^2}{m} \qquad l_b = \frac{b^{1/3}}{N} \qquad l_m = \frac{m}{b^{2/3}} \tag{21}$$

Note that these length scales are defined based on the *total* fluxes, rather than the flux from each side of the diffuser. The geometrical characteristics defined in Figures 53 and 54 can then be expressed as:

$$z_e,\ h_e,\ z_m = f(q, b, m, s, u, N, \Theta) \tag{22}$$

Which, by means of dimensional analysis, becomes:

$$\frac{z_e}{l_b},\ \frac{h_e}{l_b},\ \frac{z_m}{l_b} = f\left[\frac{l_m}{l_b},\ \frac{s}{l_b},\ \mathrm{F},\ \Theta\right] \tag{23}$$

Where $\mathrm{F} = u^3/b$ is a dynamic variable which is a type of Froude number. In Equation 5, the effect of the source volume flux q is neglected as an independent variable. This is because l_q/l_b is usually much less than one and therefore has little dynamic effect except very near to the ports. The corresponding normalized expression for dilution is:

$$\frac{S_m q N}{b^{2/3}} = f\left[\frac{l_m}{l_b}, \frac{s}{l_b}, \mathrm{F}, \Theta\right] \qquad (24)$$

where S_m is the minimum initial dilution, as previously defined. An average dilution S_a is computed as 1.15 S_m based on hydraulic model tests by Roberts (1989).

The two length scale ratios l_m/l_b and s/l_b are diffuser parameters which characterize the significance of source momentum flux and port spacing respectively. Note that these length scale ratios encompass the jet exit velocity, port diameter, port spacing, effluent density, and ambient stratification. Based on consideration of actual operating conditions, the range of experiments was chosen to be $0.31 < s/l_b < 1.92$ and $0.078 < l_m/l_b < 0.5$. For $s/l_b < 0.3$ and $l_m/l_b < 0.2$, the discharge approximates a line plume, i.e. the individual plumes rapidly merge and the effect of source momentum flux is negligible, many ocean outfalls operate in the regime in which momentum is negligible (Roberts et al., 1989a). Therefore the range of diffuser parameters can be considered to be $s/l_b < 1.92$ and $l_m/l_b < 0.5$

A more important parameter is F, which characterizes the importance of the current speed relative to the buoyancy flux of the source. Small values of F signify little effect of current; according to Roberts et al. (1989a) the current exerts no effect on dilution if F < 0.1. Larger values of F denote situations where the plumes are rapidly swept downstream by the current; dilutions are always increased by increased current speeds, although not always at the regulatory (critical) mixing zone boundary, as shown in Figure 5. (See Figures 4 and 6 in Roberts, Snyder, and Baumgartner, 1989a for photographs of plumes at various Froude numbers, F). The tests were run at differing current speeds to obtain $F = u^3/b$ in the range 0 (zero current speed) to 100.

The effect of the current also depends on the direction of the current relative to the diffuser Θ. For a line diffuser $0 < \Theta < 90°$. Tests were run with $\Theta = 90°$ (diffuser oriented perpendicular to the current), 45°, and 0° (parallel to the current). In general, diffusers oriented perpendicular to the current result in highest initial dilutions and lowest rise heights.

MODEL DESCRIPTION

Results for wastefield geometry and initial dilution were presented graphically (Figures 8, 10-12 of Roberts et al. 1989a) in the dimensionless form of Equations 5 and 6 for line plume conditions ($s/l_b < 0.3$ and $l_m/l_b < 0.2$). Results to predict the length of the initial mixing zone x_i are in Figures 4 and 8 of Roberts et al., 1989b. For higher port spacings and higher momentum fluxes the results are given in Figures 5 and 6, and 7 and 8 of Roberts et al., 1989c.

For some of these results, semi-empirical equations are given. These equations are semi-empirical because they are physically based, but the coefficients must be obtained from the experiments. Examples are the dilution and rise height of line plumes in perpendicular currents (Equations 14 and 17 of Roberts et al., 1989a):

$$\frac{S_m q N}{b^{2/3}} = 2.19\mathrm{F}^{1/6} - 0.52 \qquad \frac{z_e}{l_b} = 2.5\mathrm{F}^{-1/6} \tag{25}$$

In other cases, for example, high momentum jets in a parallel current, only graphical solutions are available. In these cases, purely empirical equations are fitted to the curves, and the results interpolated as appropriate. RSB can therefore be thought of as a coding of the graphs and equations in the original papers. For linear stratifications, the model should give exactly the same results as obtaining the solution graphically.

For non-linear stratifications, RSB assumes that the density profile is linearized over the rise height. In RSB, the solution procedure is iterative, solving automatically for the rise height z_e.This method, which is similar to that used by Brooks (1973) is shown in Figure 55. As discussed later, this approximation works very well, even for very non-linear stratifications. In fact, this is a conservative assumption, as linear stratifications lead to less rapid spreading, thinner wastefield, less subsequent mixing, and therefore less dilution than in a wastefield at the same rise height in a non-linear stratification (Roberts, 1992).

EXAMPLES

Introduction

RSB can be run either as a standalone program or from PLUMES. When run in standalone mode, RSB uses the same UDF input file format as previous EPA models (Muellenhoff et al., 1985). This file can created using the ^YU command in PLUMES, with any ASCII text-editor, or interactively by following prompts within RSB. Note, however, that **RSB assumes discharges from both sides of the diffuser**, whereas the original EPA models implicitly assume discharge only from one side of the diffuser, so the data may be different for different models. In UM this requirement is accomodateed by running the cross-diffuser merging configuration, i.e. by specifying half spacing between ports. For example, if ports are staggered every two meters with adjacent ports on one side of the diffuser four meters apart, then the appropriate spacing is two meters. Whether the model is run standalone or from PLUMES, the solution procedure is the same, so the results should be practically identical.

Recommendations on usage are given in Appendix 3. The ambient density must be stable, i.e. density must not decrease downwards, however, under some sircumstances RSB will produce valid results if intermediate levels are specified as unstable due to the method used in RSB to calculate a linear gradient. The total number of ports n and spacing s are inputted to determine the diffuser length L which is then used to compute q and the length scales.

$$L = s\left(\frac{n}{2} - 1\right) \tag{26}$$

Seattle Example: Linear Stratification - Zero Current

The following example follows that given in Roberts et al., 1989a,b,c. The parameters are taken from the Metropolitan Seattle outfall discharging into Puget Sound (Fischer et al., 1979):

Design average flow, Q = 194 ft^3/s (5.49 m^3/s)

Number of ports = 202

Port spacing (on each side of the diffuser), s = 6 ft (1.83 m)

Port diameters, d = 4.5 to 5.75 inches (0.114 to 0.146 m)
Assume d = 5.0 inches (0.127 m)

Effluent density, ρ_o = 1.000 g/cm^3

The port depth is about 70 m, and density stratifications at nearby Alki Point vary between 0.002 and 0.025 σ_t-units per meter. Taking the strongest stratification (0.025 σ_t-units per meter) yields, for example, a density of 1.02425 g/cm^3 at the surface and 1.02600 g/cm^3 at 70 m depth. The pipe diameter is 96 inches (2.44 m) so the port elevation is 1.22 m and the total depth is set at 71.22 m.

The input and output files of the original RSB (Basic) model for zero current are shown in Figure 55. The computed length scales ratios are s/l_b = 0.14 and l_m/l_b = 0.13 which suggests no effect of the source momentum flux and port spacing so we expect the behavior of this discharge to approximate a line plume. The predicted minimum initial dilution S_m for this case is 80, and rise height z_e is 32.9 m. No farfield calculation is provided.

The corresponding PLUMES RSB and UM runs are given in Figure 56 without farfield calculations. Notice the close agreement between Basic RSB and PLUMES RSB; maximum difference are less than one percent. Also, notice the approximate agreement between the models, e.g. average dilutions of 92 and 82 for RSB and UM respectively. In the remainder of this chapter only the PLUMES RSB runs will be displayed. The corresponding UM run is given in Figure 57.

The Basic RSB program is not bundled with the plumes package.

```
Input file:

Seattle Example
     5.490       202      0.127       0.00       70.0
     0.000    90.000      1.830
      2        1.0000      0
      0.00     1.02425       0.0      0.000
     70.00     1.02600       0.0      0.000

Output file:

Input data:

 Seattle Example
  Flowrate =  5.49 m3/s
  Effluent density =  1  g/cm3
  Number of ports =  202
  Port diameter =  .127 m;      Port spacing =  1.83 m
  Discharge depth = 70 m
  Current speed =  0 m/s;    Angle of current to diffuser = 90 degrees
  Computed diffuser length = 183.0 m

    Density profile:
  Depth (m)    Density (g/cm3)
     0.0          1.02425
    70.0          1.02600

Results:

  Lengthscale ratios are:   s/lb =  0.14,  lm/lb =  0.13
  Froude number, u3/b =   0.00;   Jet Froude number, Fj = 12.1
  Rise height to top of wastefield, ze = 32.9 m
  Wastefield submergence below surface = 37.1 m
```

Ilustr. 55. Input and output of the original RSB program (Roberts, 1991).

```
Jun 28, 1992,   11:23:13  ERL-N PROGRAM PLUMES, July 1, 1992   Case:   1 of
8
 Title   Seattle Example

  tot flow   # ports port flow   spacing  effl sal effl temp   far inc   far
dis
                  202               0.9144

  port dep  port dia plume dia total vel horiz vel vertl vel asp coeff print
frq
         70     0.127    0.1270     2.145                 0.000      0.10
 500
 port elev  ver angle  cont coef   effl den  poll conc      decay   Froude #
Roberts F
          1        0.0        1.0         0        100          0      11.92
1.833E-14
 hor angle  red space  p amb den  p current    far dif    far vel  K:vel/cur
Stratif #
         90     0.9144      26.000.00001000  0.000453                  214500
0.0001221
      depth    current    density  salinity      temp   amb conc   N (freq) red
grav.
        0.0       1e-5      24.25                                    0    0.01546
0.2550
         70       1e-5      26.00                                      0 buoy flux
puff-ther
                                                                     0.006930
36.61
                                                                        jet-plume
jet-cross
                                                                        1.425
24150
                                                                        plu-cross
jet-strat
                                                                    6.930E+12
3.952
                                                                    plu-strat
                                                                        6.581
                                                                      CL  conc>=


 CORMIX1 flow category algorithm is turned off.

  5.49 m3/s, 125.3 MGD, 193.9 cfs.                               >0.0 to 100 m3/s
range
 Help: F1.  Quit: <esc>.  Configuration:ATNOO.  FILE: rsbeg.var;


                                        RSB

                   Written by Philip J. W. Roberts (12/12/89)
```

Ilustr. 56. PLUMES RSB run for Seattle example.

Seattle Example: Linear Stratification - Flowing Current

Consider now an ambient flowing current of 0.30 m/s perpendicular to the diffuser. The new input and output data files are shown in Figure 57.

The minimum dilution is now increased by the current to 181, and the rise height (to the top of the wastefield) reduced from 32.9 m to 26.5 m. This process can be continued for other current speeds to generate the results shown as Table 2 in Roberts et al., 1989a. Note that numbers may differ slightly from this table due to slightly differing interpolation procedures.

```
Jun 28, 1992,  11:27:44   ERL-NPROGRAMPLUMESJuly  1, 1992    Case:    3 of    8
Title     Seattle   Example; with  current
 tot  flow     # ports  port  flow    spacing    effl   sal  effl   temp    far  inc     far  dis
                   202    0.02718     0.9144
 port  dep   port  dia  plume dia  total   vel  horiz  vel  vertl   vel  asp  coeff  print  frq
        70      0.127     0.1270        2.145        2.145        0.000        0.10          500
port  elev  ver  angle  cont  coef    effl   den  poll  conc       decay   Froude  #  Roberts  F
         1        0.0           1.0           0          100                    11.92      0.4948
hor  angle   red  space  p  amb den  p  current     far  dif     far  vel  K:vel/cur  Stratif     #
        90    0.9144         26.00      0.3000   0.000453                    7.152  0.0001221
     depth       current       density     salinity            temp    amb conc   N (freq)    red  grav.
       0.0            0.3         24.25                                            0   0.01546      0.2550
        70            0.3         26.00                                            0  buoy  flux   puff-ther
                                                                                  0.006930       1.178
                                                                                  jet-plume    jet-cross
                                                                                      1.425      0.8049
                                                                                  plu-cross    jet-strat
                                                                                    0.2567       3.952
                                                                                  plu-strat
                                                                                      6.581
                                                                                    hor  dis>=

CORMIXflow  category   algorithm   is  turned   off.
 5.49  m3/s,  125.3  MGD,193.9  cfs.                            >0.0 to  100 m3/s range
Help:  F1.   Quit:   <esc>. Configuration:ATNO0.    FILE:  rsbeg.var;

 Case:   3:  Seattle   Example;  with   current

 Lengthscale   ratios    are:  s/lb    =    0.14  lm/lb   =    0.13
 Froude  number,  u3/b  =               3.62
 Jet  Froude  number,  Fj  =           12.1

 Rise  height   to  top  of  wastefield,     ze  =  26.5
 Wastefield   submergence  below  surface    =  43.5
 Wastefield   thickness,     he  =               21.5  m
 Height   to  level   of  cmax,  zm  =           17.4  m
 Length  of  initial     mixing  region,    xi   =  164.9  m                            AN001AAA.USSD
```

Ilustr. 57. RSB Seattle example, with current.AN001AAA.USSS[Sorry: Run WPS]

Seattle Example: Model Extrapolation

This example illustrates the effect of running RSB outside the range of values on which it is based. The port diameter is reduced to 60 mm (0.06 m); the new data files are shown in Figure 58.

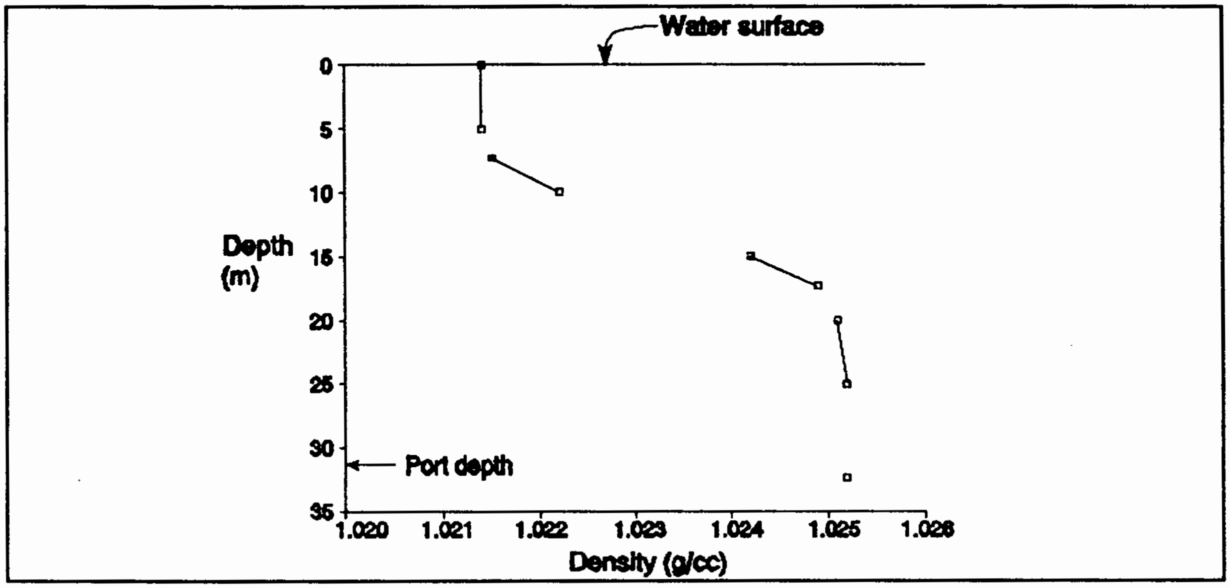
In this case the decrease in nozzle size causes an increase in nozzle exit velocity and an increase in momentum flux. The length scale ratio l_m/l_b becomes equal to 0.60, which exceeds the experimental range. Note that RSB still gives answers in these situations and gives a warning message that the predicted results are extrapolated and therefore may be unreliable; the interpretation of these results is at the discretion of the model user. The primary predicted effect of the increased momentum flux is a decrease in rise height; the dilution is unchanged. The reasons for this type of behavior are discussed in Roberts et al., 1989c.

```
 Jun 28, 1992,  11:28:14  ERL-N PROGRAM PLUMES, July 1, 1992   Case:   4 of
 8
 Title    Seattle Example; extrapolated

  tot flow   # ports port flow   spacing  effl sal effl temp   far inc   far
dis
                     202    0.02718    0.9144

  port dep  port dia plume dia total vel horiz vel vertl vel asp coeff print
frq
          70       0.06    0.06000      9.612      9.612      0.000       0.10
 500
 port elev ver angle cont coef  effl den poll conc       decay  Froude #
Roberts F
           1        0.0        1.0          0        100                 77.71
8.658E-15
 hor angle red space p amb den p current    far dif    far vel K:vel/cur
Stratif #
          90       0.9144        26.000.00001000   0.000453
9612000.00005769
      depth    current   density salinity      temp   amb conc  N (freq) red
grav.
         0.0       1e-5      24.25                                0    0.01546
0.2550
          70       1e-5       26.00                                  0 buoy flux
puff-ther
                                                                   0.006930
99.49
                                                                       jet-plume
jet-cross
                                                                       4.390
51110
                                                                       plu-cross
jet-strat
                                                                  6.930E+12
5.750
                                                                  plu-strat
                                                                       6.581
                                                                    hor dis>=
 ... RSB ...
```

Ilustr. 58. Seattle example, reduced port size, RSB model extrapolation.

Seattle Example: Non-Linear Stratification

In this example the non-linear ambient density profile shown in Figure 59 is used. The density profile is the one used in the Boston Harbor Diffuser model tests. It consists of a uniform, well-mixed surface layer about 8 m thick, followed by a sharp change in density through the thermocline, which is about 13 m thick, then a uniform density down to the bottom. The port depth in this case is 31.3 m below the water surface. The diffuser of the Seattle example is used and the new data files are given in Figure 60.



Ilustr. 58. Density Profile used in Non-Linear Example.

RSB predicts a rise height of 17.4 m, which is in the pycnocline. The solution procedure, which is transparent to the user, is to linearize the density profile over this 17.4 m.

```
 Jun 28, 1992,  11:29:16  ERL-N PROGRAM PLUMES, July 1, 1992   Case:   6 of
 8
 Title   Seattle example; Boston density profile

  tot flow   # ports port flow   spacing  effl sal effl temp   far inc   far
dis
                  202   0.02718    0.9144

  port dep  port dia plume dia total vel horiz vel vertl vel asp coeff print
frq
      31.3      0.127    0.1270     2.145     2.145     0.000      0.10
 500
 port elev  ver angle cont coef   effl den poll conc      decay   Froude #
Roberts F
         1          0         1          0        100         0      12.11
1.891E-14
 hor angle red space p amb den p current    far dif    far vel K:vel/cur
Stratif #
        90     0.9144      25.200.00001000  0.000453                  214500
0.0006118
      depth   current   density  salinity      temp  amb conc  N (freq) red
grav.
       0.0      1e-5      21.4                                  0   0.03408
0.2471
         5      1e-5      21.4                                    0 buoy flux
puff-ther
       7.3      1e-5      21.5                                  0  0.006717
36.99
        10      1e-5      22.2                                    0 jet-plume
jet-cross
        15      1e-5      24.2                                  0     1.448
24150
      17.3      1e-5      24.9                                    0 plu-cross
jet-strat
        20      1e-5      25.1                                  0 6.717E+12
2.662
        25      1e-5      25.2                                 0 plu-strat

        35      1e-5      25.2                                 0     3.609
```

Ilustr. 59. Seattle example, non-linear density profile.

Multiport Risers Example

Many outfalls with multiport risers are now operating (San Francisco and Sydney), under construction (Boston), or proposed (Hong Kong). Except for San Francisco, these are tunneled outfalls for which the cost of the risers is very high, of the order of several million dollars each. It is therefore necessary to minimize the number of risers without unduly impairing dilution. This is different from a pipe diffuser in which, for a given diffuser length, the number of ports in the pipe wall and their spacing is not a significant cost consideration.

The following example is for the Boston outfall. This is a convenient example as experimental results from the hydraulic model tests done for this diffuser are available (Roberts, 1989). The example also illustrates the effects of non-linear stratifications.

The basic assumption is that the behavior of the wastefield is the same as if the ports were uniformly distributed along both sides of the diffuser, rather than clustered in multiport risers. This was originally demonstrated by Isaacson et al. (1978, 1983) to be a good assumption for certain limited conditions. The caveat to this assumption is that entraining water must be available to the plumes. This implies that not more than 8 ports per riser be used, otherwise the flow collapses to a rising ring with reduced dilution.

The following examples are of the final design, which has 55 risers spaced a distance of 122 ft (37.2 m) apart. Each riser has 8 ports with nominal diameters of 6.2 inches (0.157 m). Tested flowrates were 390 mgd (17.08 m^3/s), 620 mgd (27.16 m^3/s), and 1270 mgd (55.63 m^3/s). If the ports were uniformly distributed along the diffuser, the port spacing s would be 122/4 = 30.5 ft (9.30 m). A typical data file for 390 mgd, zero current speed, with a density profile as shown in Figure 59 (this is referred to as the Late Summer Profile in Roberts, 1989),

Table IV. Measured and predicted wastefield characteristics for Boston Harbor Outfall.

Current speed (cm/s)	Flowrate, Q (mgd)	Minimum initial dilution, S_m		Rise height to top of wastefield, z_e (m)		Wastefield thickness, h_e (m)	
		Measured	Predicted	Measured	Predicted	Measured	Predicted
0	390	81	67	16.3	17.1	7.5	12.8
25	390	223	215	16.3	15.8	14.5	14.1
0	620	70	59	17.8	16.9	10.5	12.7
0	1270	56	46	17.8	16.9	14.5	12.7

is given in Figure 61. Table IV gives more comparisons between measured and predicted dilutions.

```
 Jun 28, 1992,  11:29:45  ERL-N PROGRAM PLUMES, July 1, 1992   Case:   7 of
8
 Title   Boston, multiport risers

  tot flow   # ports port flow   spacing  effl sal effl temp   far inc   far
dis
                      440   0.03882      4.15

  port dep  port dia plume dia total vel horiz vel vertl vel asp coeff print
frq
       31.3      0.157     0.1570      2.005      2.005      0.000       0.10
 500
 port elev ver angle cont coef   effl den poll conc       decay   Froude #
Roberts F
          1          0          1          0        100           0      10.18
1.637E-14
 hor angle red space p amb den p current    far dif     far vel K:vel/cur
Stratif #
         90      4.150      25.200.00001000  0.000453                 200500
0.0007564
      depth    current    density  salinity      temp  amb conc   N (freq) red
grav.
        0.0       1e-5       21.4                                 0   0.03408
0.2471
          5       1e-5       21.4                                   0 buoy flux
puff-ther
        7.3       1e-5       21.5                                 0  0.009593
39.82
         10       1e-5       22.2                                  0 jet-plume
jet-cross
         15       1e-5       24.2                                 0     1.505
27900
       17.3       1e-5       24.9                                  0 plu-cross
jet-strat
         20       1e-5       25.1                                0 9.593E+12
2.861
         25       1e-5       25.2                                0 plu-strat

         35       1e-5       25.2                                0     3.946

                                                                  hor dis>=

 ... RSB ...
  Case:   7: Boston, multiport risers
```

Ilustr. 60. Boston example, multiport risers; RSB and UM simulations.

It can be seen that, despite the very large difference between the conditions on which RSB is based (paired ports, linear stratification) and the Boston tests (ports clustered 8 per riser, very non-linear stratification), the predictions are very good. Dilutions are generally underestimated, i.e. the model is conservative. This is most probably due to the additional mixing which occurs in the horizontally spreading layer in the non-linear profile compared to that in the linear profile.

DESIGN APPLICATIONS

RSB is a useful tool for the design of outfall diffusers. Time can be saved when doing this by keeping in mind the following guidelines:

> The most important parameter for an ocean outfall diffuser for a fairly large flow is the length *L*. This can be chosen first, and the details, i.e. port spacing and diameter chosen later.
>
> The flow approximates a line source for $s/l_b < 0.3$. At this point the dilution is a maximum (for fixed diffuser length) and adding more ports so that the spacing is less will have no effect on dilution or rise height. Also, there is little point in making the port diameter smaller than the value which results in $l_m/l_b = 0.2$, as this will result in increased headlosses. The only constraints are internal hydraulics (which may be complex for tunneled outfalls) and that the ports flow full, i.e. $F_j > 1$.
>
> Momentum only affects dilution when $l_m/l_b > 0.2$. Therefore decreasing the port diameter to increase momentum will only affect dilution if it results in $l_m/l_b > 0.2$. Even then the primary effect on the wastefield is reduced rise height (in a linear stratification), and dilution is only slightly affected.

UM MODEL THEORY

PERSPECTIVE

UM is the latest in a line of Lagrangian models developed originally for atmospheric and freshwater applications by Winiarski and Frick (1976) and for marine applications by Teeter and Baumgartner (1979). The marine version, known as OUTPLM, became the basis of the MERGE model (Frick, 1980). Both underwent modifications to become the UOUTPLM and UMERGE models (Muellenhoff et al., 1985). Since 1985 the UMERGE model has been further generalized and enhanced; including treatments of negatively buoyant plumes and background pollution. These improvements are included in UM. Other active research focusing on the generalization to three dimensions and to geothermal applications continues (Frick, Baumgartner, and Fox, in prep.).

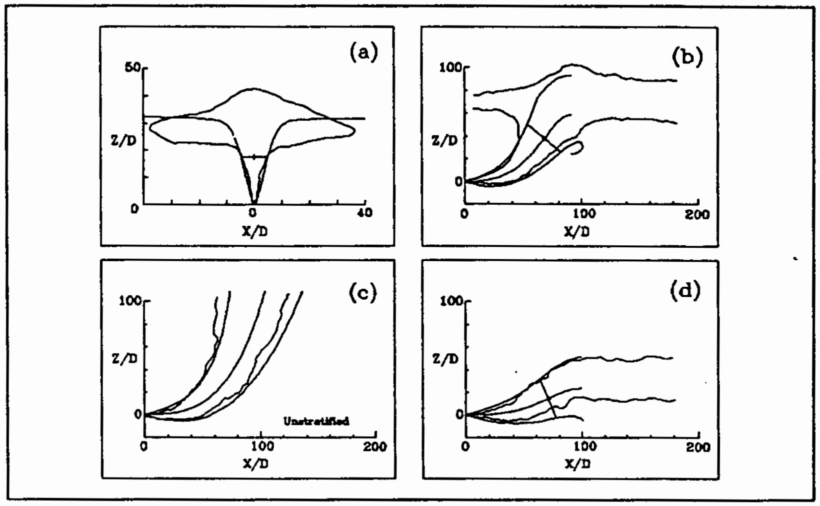
The UM line of models contains two distinguishing features: the Lagrangian formulation and the projected area entrainment (PAE) hypothesis. The Lagrangian formulation offers comparative simplicity that is useful in developing the PAE. In addition, the traditional Taylor entrainment hypothesis (Morton, Taylor, and Turner, 1956) is also used.

The projected area entrainment hypothesis is a statement of forced entrainment -- the rate at which mass is incorporated into the plume due to the presence of current. It is not original, as it was described in conceptual terms at least as early as 1960 (Rawn, Bowerman, and Brooks) and is frequently used in incomplete form (e.g. Hoult, Fay, and Forney, 1969) as demonstrated by Frick (1984).

It is not in the scope of this work to present extensive verification of the UM model, however, Figures 1 and 2 give an indication of the general quality of prediction. In Figure 2 the densimetric Froude number of the effluent is given by F_j: a measure of the ratio of momentum to buoyancy in the plume, with large Froude numbers indicating relatively high momentum and small Froude numbers indicating strong buoyancy. The ratio of efflux velocity to current is given by k; a high value indicates a relatively strong effluent velocity or low current speed.

The efficacy of the PAE hypothesis has recently been demonstrated independently by Lee and Cheung (1990) who adapt the approach to three dimensions in a model called JETLAG and show that Lagrangian plume models using PAE predict the correct asymptotic behavior in a number of limiting conditions.

The Lagrangian model and its entrainment hypotheses are described below in some detail. To understand the model it is necessary to first have an appreciation of the basic model building block -- the plume element. On that basis, the plume element dynamics, conservation principles, entrainment, and merging are more easily understood. Simultaneously, a detailed mathematical description of the model is given.



Ilustr. 61. UM centerline and boundary predictions in stagnant ambient compared to Fan (1967). (a) Jet No. 10, (b) Jet No. 16, (c) Jet No. 22, unstratified, and (d) Jet No. 32.

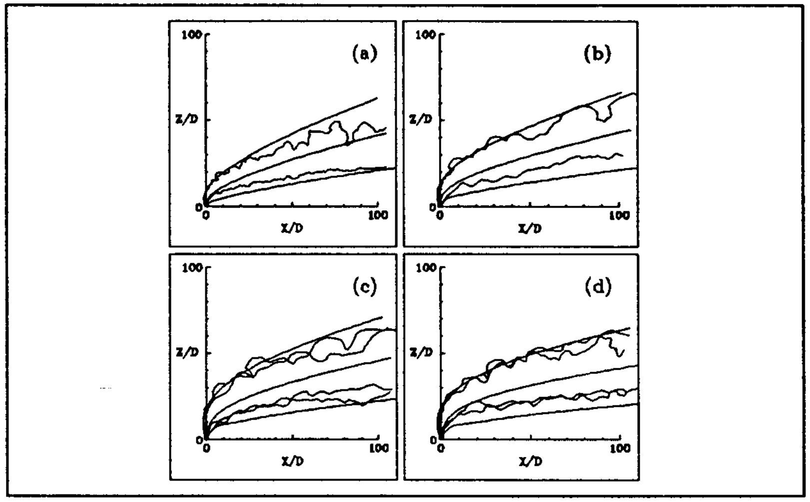
BASIC LAGRANGIAN PLUME PHYSICS

The Plume Element

The shape of the element is very important because it determines the projected area, to which forced entrainment is directly proportional, at least in the initial dilution region. In the present UM model we use a proportionality constant of 1. This entrainment and the contribution from Taylor entrainment determines the growth of the element and plays a key role in the dynamics of the element center-of-mass -- the particle.

In terms of the dynamics of the plume element, shown at three stages of development in Figure 63, simple models like the Lagrangian or Eulerian integral flux models provide only an estimate of the element trajectory, i.e., s, the path of the center-of-mass of the plume element. It is shown as a solid line passing through the centers of the elements as if all the mass of the plume element were concentrated there.

In Lagrangian and similar integral flux models, that is the only dynamic variable that is predicted by the plume model. Everything else must be inferred or assumed. The shape of the

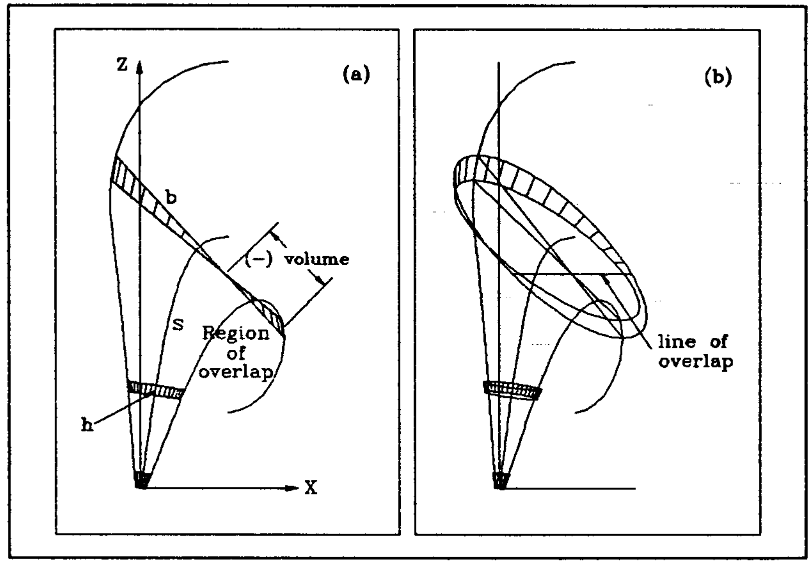


Ilustr. 62. UM predictions in flowing ambient compared to Fan (1967). (a) $F_j=10$, $k=8$, (b) $F_j=20$, $k=12$, (d) $F_j=40$, $k=16$, and (d) $F_j=80$, $k=16$.

element is established arbitrarily before the growth of the particle can be determined. In other words, the modeler determines how the shape of the plume is specified. Normally, a particular interpretation of the round plume assumption is used to establish the distribution of mass about the trajectory of the plume element; it holds that the plume element is basically cylindrical in shape.

Instead if it is assumed that the element is defined by a smooth surface on the exterior of the plume and by interior planes, or faces, that are perpendicular to the particle trajectory, and that if the plume trajectory is curved, as it generally is, then this definition results in an element that is not cylindrical but has the shape of a section of bent cone. Because the length of the element along the trajectory must be small for mathematical reasons, it is better to conceive of the element as a thin round wedge with a blunt or sharp edge. This is the element form assumed in UM. A sharp edge occurs when plume trajectory radius of curvature is smaller than the plume radius, causing the element faces to attempt to overlap, a physically impossible situation.

Secondly, the asymmetry in shape is not consistent with the general practice of symmetrically constructing plume element radii in all directions. Instead, it is recognized, though not considered here, that the plume trajectory represents the center-of-mass of the plume element which is generally not at the center of the circular cross-section.



Ilustr. 63. Plume trajectory, the element at three stages of development, and selected plume variables.

The overlap condition in other models is not identified, or even recognized, and results in the over-prediction of plume radius and entrainment, while UM issues a warning when this happens and, in its the default mode, terminates the initial dilution computation.

The plume is assumed to be in steady state. In the Lagrangian formulation that implies that successive elements follow the same trajectory. The plume envelope remains invariant while elements moving through it change their shape and position with time. However, conditions can change as long as they do so over time scales which are long compared to the time in which a discharged element reaches the end of the initial dilution phase, usually maximum rise. The steady state assumption is used to describe the length of the plume element as a function of the instantaneous average velocity, its initial length, and the initial effluent velocity.

Thus, the length of the element does not, in general, remain constant but changes with time due to the different velocities of the leading and trailing faces. It follows that the radius of the element must respond to this velocity convergence or divergence, as well as to entrainment because the fluid is practically incompressible, though incompressibility and the limiting

Boussinesq approximations (Spiegel and Veronis, 1960) are not incorporated in UM.

The exterior surface of the plume element coincides initially with the walls of the orifice from which it issues (or the vena contracta diameter). By integrating from this known initial and boundary condition the plume volume is calculated based on the entrained mass and the assumed element shape.

It is assumed that the properties of the plume at the boundary are indistinguishable from those in the adjacent ambient fluid. This has important implications, one being that drag is not an important force in plume dynamics. It also implies that mass crosses the projected area of the element at the speed of the ambient current.

Projected Area Entrainment (PAE) is a forced entrainment hypothesis that has been shown to work in moderate current without the need for a tuned coefficient. In cases of no current or light current the empirical Taylor entrainment hypothesis is needed as well. The Taylor hypothesis is retained as an additive extra entrainment term which quickly diminishes in relative importance as current increases. While the model cannot be proven to be "correct", the fact that the theory predicts observed behavior well over a range of conditions without empirical tuning is not only desirable but persuasive.

Conservation Principles

The model includes statements of conservation of mass (continuity), momenta, and energy. Conservation of mass states that the initial mass of the element and that added, or entrained, over time is conserved. In modeling terms the element mass is incremented by the amount of fluid that flows over the outside boundary of the plume element in a given amount of time. The PAE guarantees that excessive or inadequate amounts of entrainment are not inadvertently incorporated, i.e. entrained, into the plume.

Similarly, horizontal momentum is conserved. The horizontal momentum, the product of the element mass and horizontal velocity, is increased by the horizontal momentum of the entrained fluid in the same time step. Vertical momentum is not generally conserved but is altered by buoyancy, a body force arising from the density difference between the element and the ambient fluid.

Finally, energy is conserved, similarly incremented by adding an amount of energy equal to the product of a constant specific heat, the entrained mass, and the ambient temperature. An equation of state is used to obtain the densities of fresh and sea water in salinity and temperature ranges that are representative of terrestrial and coastal waters.

Entrainment and Merging

Entrainment is the process by which the plume incorporates ambient material into itself. It may be thought of as a process in which fluid flows into the plume interior through the exterior

surface. Alternatively, it may be considered to be a process of accretion followed by the redistribution of material. The former model is used here and is consistent with the projected area entrainment hypothesis.

Several mechanisms of entrainment are considered: aspirated, forced, and turbulent, or eddy, diffusion. Aspirated entrainment is shear (or Taylor) entrainment which is present even in the absence of current. It is due to the fact that high velocity regions are regions of relative low pressure which causes inflow of material into the plume. Thus the plume induces a flow field in the surrounding ambient fluid. Forced entrainment is due to the presence of current that advects mass into the plume. Diffusion is assumed to always be present but is only important beyond the zone of initial dilution. It becomes dominant after the other two entrainment mechanism die off due to the steady reduction in shear between the plume and the ambient. The transition separates the near-field from the farfield. Strictly speaking, the latter dilution is not a part of the UM theory because UM is still primarily a near-field model. Instead, farfield diffusion is parameterized by the 4/3 power law attributed to Brooks (Tetra Tech, 1982) and others.

Entrainment through the projected area of the plume is composed of three terms. The first term is proportional to the length along the trajectory (the cylinder component), the second to the growth in diameter of the plume, and the third to the curvature of the plume trajectory that opens or closes area on the element surface. All are simply mathematical parts of the overall projected area that contribute to forced entrainment. A fourth term, encompassing the entire peripheral area, accounts for aspiration entrainment.

When adjacent plumes grow sufficiently they begin to merge and entrain each other. Merging of plumes has the immediate effect of reducing entrainment by reducing the contact area between the plume and its environs. Each of the four entrainment terms is decremented to a different degree as merging proceeds. In essence, merging simply necessitates some geometric corrections. Surface and bottom effects as demonstrated by Wood (1990), or Coanda attachment (Akar and Jirka, 1990), are not modeled.

Only the merging of adjacent plumes discharging from linear diffusers (pipes) are considered here. This simplification helps to reduce the problem to two dimensions. Diffusers are assumed to be long so that end effects can be ignored and unbalanced internal diffusion is neglected.

Variations in the angle between the diffuser and the current are accommodated by mathematically reducing the spacing distance between adjacent ports by the appropriate trigonometric factor. Currents between 90 and 45 degrees may be handled in this way and lead to reductions of entrainment in agreement with measurements made by Roberts (1977). PLUMES provides a way of facilitating the appropriate conversions.

Typically diffusers are perforated on both sides. In a current the upstream plumes will then frequently merge with downstream plumes. This cross-diffuser merging is not simulated explicitly. In UM there are three ways to estimate the reduction in dilution due to cross-diffuser merging. The simplest way is to reduce the spacing between ports by a factor of two (i.e. spacing is equal to the diffuser length divided by the total number of ports). This method is justified by experience but it is not known with certainty how accurate it is. The effect may also be estimated by specifying the "background" concentration generated by the upstream plume, which results in the prediction of a reduced effective dilution. A third method involves doubling the flow per port and increasing the diameter of the port to maintain approximately the same densimetric Froude number. None of the methods account for the changes in density profile that the upstream plume effects on the downstream plume.

MATHEMATICAL DEVELOPMENT

Basic Model Theory

With respect to the foregoing discussion, it is emphasized that the element in Figure 63 is not cylindrical but is in general a section of a bent cone. The consequences of this fact cannot be overstated because the shape of the element determines the projected area which in turn determines forced entrainment, frequently the dominant source of entrainment. In general, a bent cone plume element has a projected area that differs substantially from the projected area of a simple cylinder. Thus, the growth and curvature terms are required to accurately describe the projected area of the plume element.

As has been stated, the principle of superposition allows the entrainment terms to be described separately. The projected area entrainment hypothesis states that

$$\frac{dm}{dt} = \rho_a A_p u \qquad (27)$$

where dm is the incremental amount of mass entrained in the time increment dt, A_p is the projected area, u is the ambient current speed normal to the projected area, and ρ_a is the local ambient density. This hypothesis, neglecting Taylor entrainment for a moment, makes it possible to explain observed plume behavior in simple terms.

Equation 27 can be written in vector terms

$$\frac{dm}{dt} = -\rho_a \underline{A}_p \cdot \underline{U} \qquad (28)$$

where the underline notation is used to indicate a vector. $\underline{A}_p$ is a vector in a vertical plane containing the current vector but pointing generally upstream (i.e. out of the element) and equal in magnitude to the projected area. $\underline{U}$ is the average velocity of the ambient flow through the projected area. The minus sign is due to the fact that $\underline{A}_p$ and $\underline{U}$ point in opposite directions so that their dot product is intrinsically negative.

To estimate the projected area it is necessary to express mathematically how the length of the element, h, changes in response to changes in other plume properties. The reason h changes is due to the differences in velocities of the leading and trailing faces of the element which causes the faces to converge or diverge with time. Just how much depends on whether the local current velocity is less than or greater than the element velocity. Changes in h result in changes in the radius because mass is conserved. The effect is confirmed by dilution and radii data tabulated by Fan, 1967.

Referring to Figure 64, $\Delta|\underline{V}|$ is seen to be the difference in velocity at two opposing faces of the semi-infinitesimal element. (The velocity vectors are proportional to the displacement vectors shown. Also, in both Lagrangian and Eulerian formulations the element is infinitesimal only along the trajectory. Thus it is a hybrid integrating volume which must be treated differently from truly infinitesimal volume elements.) Since the Lagrangian formulation deals with material elements and it is assumed the velocity is uniform, the faces separate or converge in time, proportionally to $\Delta|\underline{V}|$, i.e.,

$$\Delta h = \Delta|\underline{V}|\,\delta t \qquad (29)$$

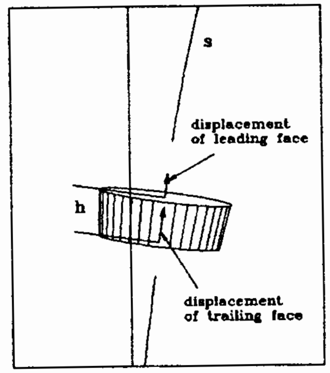


Figure 64. Convergence of element faces due to differences in face velocities.

where δt is an arbitrary, but constant, time increment. Integrating Equation 29 and noting that the corresponding speeds and lengths are $\Delta|\underline{V}_o|$ and h_o, and, $\Delta|\underline{V}|$ and h respectively yields

$$\int_{h_o}^{h} dh = \delta t \int_{u_{so}}^{u_s} du_s \qquad (30)$$

where $u_s = |\underline{V}|$ and $u_{so} = |\underline{V}_o|$. Equation 30 can be integrated to yield

$$h - h_o = (u_s - u_{so})\delta t \qquad (31)$$

Finally, since δt can be chosen to be h_o/u_{so}

$$\frac{h}{h_o} = \frac{u_s}{u_{so}} \qquad (32)$$

and $|\underline{V}|$ and h change proportionally.

Plume Dynamics

It is convenient to begin a discussion of the Lagrangian plume equations with the equation of continuity, in other words, the entrainment equation. Equations 27 or 28 is a partial expression for entrainment; it states that the amount of mass added to the element in time dt is equal to the total mass flux through the element surface. The complete entrainment equation is a sum of the forced and Taylor entrainment terms

$$\frac{dm}{dt} = -\rho \underline{A}_p \cdot \underline{U} + \rho A_T v_t \qquad (33)$$

where A_T is the area of the plume element in contact with the ambient fluid and v_T is the Taylor aspiration speed. Since A_T wraps completely around the element it is not expressed as a vector. v_T is often related to an average plume velocity through a proportionality coefficient, α:

$$v_T = \alpha |\underline{V}| \qquad (34)$$

where $|\underline{V}|$ is the average, or top hat, plume element velocity (but in other formulations it could be the centerline velocity with α scaled accordingly).

For plumes (jets with buoyancy) adequately described by a Gaussian profile (see a subsequent section entitled "Average and Centerline Plume Properties") a value of 0.082 is often attached to α. However, this is based on a nominal plume boundary which encompasses only the central portion of the plume. The corresponding value for jets in stagnant ambient is 0.057. However, Frick (1984) makes arguments for a constant α. The conversion from nominal Gaussian plumes to a "top hat", or average, description of the plume element yields corresponding values of 0.116 and 0.081. According to Frick, the latter is underestimated so that an average value for α of 0.1 is thought to be slightly conservative in terms of describing aspiration entrainment.

Strictly speaking, the areas are infinitesimal areas which might be indicated with the differential d prefix. This is because h is an infinitesimal distance. However, the model equations are approximations in which small algebraic values substitute for infinitesimal ones.

Both entrainment areas need further elaboration. The Taylor aspiration area in the absence of merging, dynamic collapse, and element facial overlap (sharp trajectory curvature) is simply

$$A_T = 2\pi b h \qquad (35)$$

where b is the element radius. The reduction in this area due to merging is described in a later section. Dynamic collapse (Frick et al., 1990) is not included in UM.

Deriving the projected area is more difficult than deriving the Taylor entrainment area. A derivation is presented, one that applies to three-dimensional flow.

Since the current, $\underline{U}$, is a vector field it may be transformed into a useful coordinate system

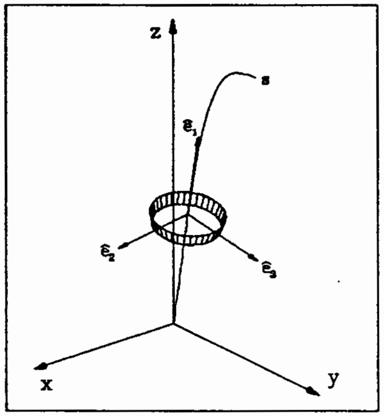


Figure 65. The local coordinate system.

by well known rules of vector rotation. A particularly useful coordinate system is the local coordinate system shown in Figure 65. The ambient velocity vector, i.e., the current, can be expressed as the sum of components in each of the local coordinate system directions

$$\underline{U} = u_1\hat{e}_1 + u_2\hat{e}_2 + u_3\hat{e}_3 \qquad (36)$$

where $\hat{e}_1$, $\hat{e}_2$, and $\hat{e}_3$ are the unit vectors in the direction of the trajectory, the horizontal normal to the trajectory, and in a vertical plane respectively. The vector $\hat{e}_3$ can be expressed in terms of the cross-product of $\hat{e}_1$ and $\hat{e}_2$:

$$\hat{e}_3 = \hat{e}_1 \times \hat{e}_2 \qquad (37)$$

The unit vectors are derived by constructing a rotation matrix that transforms between the coordinate systems.

As far as each velocity component is concerned the corresponding projected areas are particularly simple, see Figure 66. Again ignoring merging, collapse, and overlap, the projected area associated with u_1, i.e., A_1, is simply an annulus that wraps around the plume

$$A_1 = \pi b \, \Delta b \tag{38}$$

where Δb is the difference between the radius of the leading and trailing faces of the plume element. This is the "growth" contribution to the projected area (see Figure 66a). The assumption is made that only the upstream portion of the area, half the circumference, has flow going through it. The flow in the wake is altered and is assumed to flow parallel to the plume surface.

The difference in radius over the length of the element is

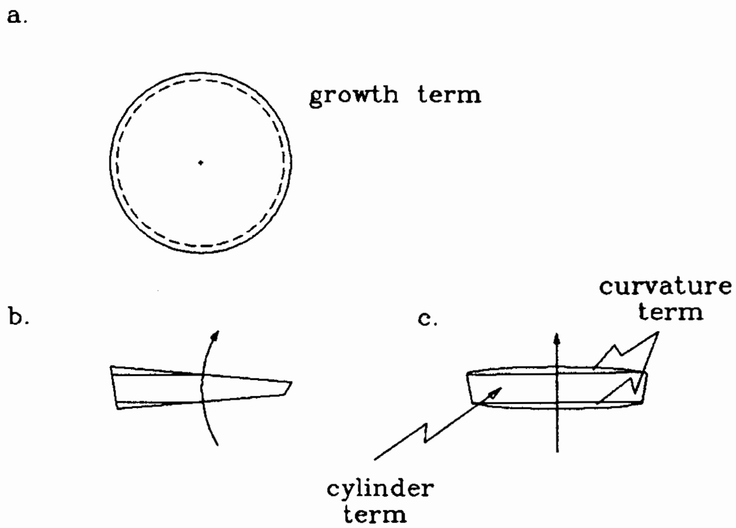


Figure 66. The projected area entrainment components: a) the growth area, b) side view of the element, and c) the cylinder and curvature area.

$$\Delta b = \frac{\partial b}{\partial s} h \tag{39}$$

where s is the distance along the centerline. The derivative is estimated from the difference in radius in successive program steps divided by the distance traversed.

Each one of the velocity components u_2 and u_3 has two projected area terms associated with it, one which is due to the curvature of the plume trajectory, the other simply being the projection of a cylinder (see Figure 66b and 66c respectively). Since only the two-dimensional problem is considered the u_3 component is ignored; its cylinder and curvature contributions are due to current flowing into the side of the plume element caused by directional changes with depth in the ambient flow.

The cylinder projected area is simply

$$A_{cyl} = 2bh \tag{40}$$

The change in direction of the average plume element velocity, $\underline{V}$, which is parallel to $\hat{e}_1$, over the length of the plume element h, in other words the curvature of the centerline s, produces the "curvature" component to the projected area. Since the faces defining the element are normal to s, in regions of strong trajectory curvature the element is deformed into a wedge shape. A depiction is given in Figure 67.

The curvature component of the projected area is

$$A_{cur} = -\frac{\pi}{2} b^2 \frac{\partial \theta}{\partial s} h \tag{41}$$

where θ is the elevation angle of s. This area can be positive or negative depending of the sign of $\partial\theta/\partial s$ which is determined with reference to successive values of $\underline{U}$. A negative curvature and area has the effect of reducing the total projected area.

Historically the growth and curvature terms have either not been recognized or have been thought to be small compared to the cylinder term (Schatzmann, 1979). However, in general, it can be shown that all three contributions to the total projected area are important. Any earlier perceived inadequacies in the projected area entrainment hypothesis can be attributed to the omission of the growth and curvature terms. Further details on this subject are available elsewhere (Frick, Baumgartner, and Fox, in prep; Frick 1984).

Conservation of momentum is given by

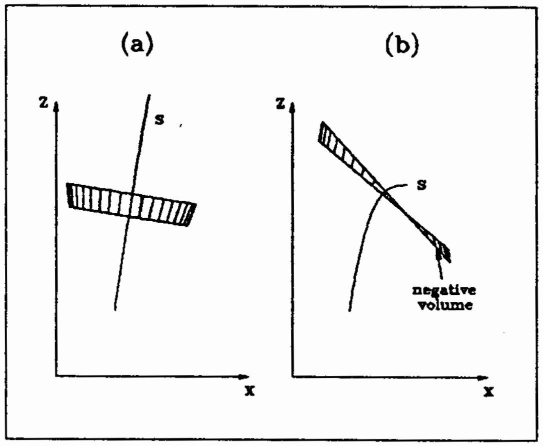


Figure 67. a) The plume element in a region of weak trajectory curvature and b) strong trajectory curvature (showing overlap).

$$\frac{dm\underline{V}}{dt} = \underline{U}\frac{dm}{dt} - m\frac{(\rho_a - \rho)}{\rho}\underline{g} \qquad (42)$$

where m is the mass of the plume element ($m = \rho\pi b^2 h$), ρ_a and ρ are the ambient and average element densities respectively, and $\underline{g}$ is the gravity vector. $\underline{U}$ represents the average ambient velocity over the exposed plume surface. This point is worth emphasizing since the surface area is infinitesimal only along the centerline and can be extensive in the two dimensions orthogonal to the centerline, over which, therefore, the ambient velocity can vary significantly. In UM it is equal to the ambient velocity at the level of the particle, i.e., the center of the cross-section.

Equation 42 states that the change in momentum in the element is due to the amount of momentum possessed by the entrained mass dm and the change in vertical momentum generated by the buoyant force. The implicit assumption is that drag effects are absent. This is consistent with the conception of the element having the same properties as the ambient on the outside surfaces of the element. Effectively, there are no shears that can generate drag.

Interactions with solid surfaces and the free interface are not included, in the model,

however, the user is warned by UM about interactions with the surface and the bottom, assumed to be flat. In Muellenhoff et al. (1985) predicted dilutions were reduced by 10% when the sea surface was encountered. Generally, plumes rise in a matter of minutes so that the Coriolis force is safely ignored. However, very large plumes, like so-called hydrothermal mega-plumes, are likely to be affected significantly by earth's rotation.

To evaluate the buoyancy term in the conservation of momentum equation, it is necessary to define the conservation of energy equation, approximated by

$$\frac{dmc_p(T-T_{ref})}{dt} = c_p(T_a - T_{ref})\frac{dm}{dt} \qquad (43)$$

where c_p is the specific heat at constant pressure. T, T_a, and T_{ref} are the average element temperature, the ambient temperature, and an arbitrary reference temperature, respectively. More correctly, the terms in Equation 43 should be represented by integrals. However, it is assumed that c_p is constant over the range of interest permitting Equation 45 to be simplified,

$$\frac{dmT}{dt} = T_a\frac{dm}{dt} \qquad (44)$$

In Equation 44 there are no source terms since radiation, conduction, and diffusion are assumed to be small. Like salinity, temperature is assumed to be a conservative property.

Several other relationships are necessary: Conservation of salinity

$$\frac{dmS}{dt} = S_a\frac{dm}{dt} \qquad (45)$$

where S and S_a are the average element salinity and the ambient salinity respectively. The symbol for ambient salinity should not be confused with average dilution of the plume.

Conservative pollutants would be expressed similarly, however, since important pollutants, such as coliform, are subject to decay, a first order decay term is included.

$$\frac{dm\chi}{dt} = \chi_a\frac{dm}{dt} - km\chi \qquad (46)$$

where χ and χ_a are the concentrations of the species of interest in the element and ambient respectively and k is a first order decay constant.

The momentum equation includes the reduced gravity, $((\rho_a - \rho)/\rho)\ \underline{g}$, which must be determined. Densities are derived from the equation of state (SigmaT function) used by Teeter and Baumgartner (1979). It is independent of pressure, limiting UM to shallow, by deep ocean standards, water. It is also limited to ordinary temperatures.

Boundary Conditions and Other Pertinent Relationships

To complete the model, the boundary and initial conditions must also be specified. The main boundary condition is the location of the source from which the subsequent position of the element may be determined by integrating the trivial relationship

$$\frac{d\underline{R}}{dt} = \underline{V} \tag{47}$$

where $\underline{R}$ is the radius vector of the particle, i.e., the center of the element. To give an example of how the equations are solved in a finite difference model, the new $\underline{R}$ is

$$\underline{R}_{t+dt} = \underline{R}_t + \underline{V}\, dt \tag{48}$$

Another boundary condition is the initial plume radius. Initial conditions include the efflux velocity, the effluent temperature, etc..

Several other auxiliary equations are necessary. They include linear interpolations that determine ambient conditions at the level of the particle. Also, because the Lagrangian plume equations require a very small time step initially, but not later in the simulation, a method of varying the size of the time step is used to control the relative amount of mass that is entrained during any one single step. This is done in the interest of computational efficiency.

The general computational procedure followed in the model is: 1) a time step is provided (guessed), 2) the entrainment equations are then used to determine the amount of mass that will be added given this time step, 3) this increase is then compared with the target mass increase and the appropriate adjustments are made to the time steps and the entrainment components to meet the appropriate doubling criterion, 4) the equations of motion and other model equations are solved, and 5) the new time step is established and the cycle is repeated.

It is important to recognize that some of the above equations are not always solved for the quantity on the left hand side of the equal sign. In other words, the dependent variable may be some other variable besides the one on the left hand side of the equal sign. For example consider Equation 49 which expresses the mass of the element in terms of its dimensions and the density:

$$m = \rho \pi b^2 h \tag{49}$$

For modeling purposes the radius, b, is not an independent variable, rather it is a dependent variable. Since mass is computed by integrating from its initial value using the entrainment, or continuity, equation, it is effectively an independent variable in Equation 49. In other words, Equation 49 must be inverted to solve for the radius. Thus

$$b_{t+dt} = \sqrt{\frac{m_{t+dt}}{\pi \rho_{t+dt} h_{t+dt}}} \tag{50}$$

Merging

The basic approach to handling plume merging is to 1) reduce the entrainment areas, both Taylor and forced, to account for the loss of exposed surface area that occurs when neighboring plumes intersect each other, and, 2) to confine the plume mass from each plume to the space between them that is known to be available from symmetry considerations.

Considering Taylor entrainment first, the conditions of merging are depicted in Figure 68.

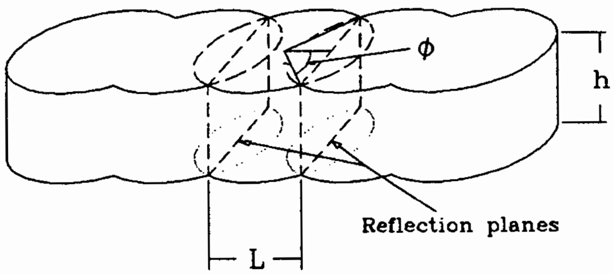


Figure 68. Merging geometry and reflection planes.

It is seen that the uncorrected Taylor entrainment area can be multiplied by a factor equal to the ratio of the exposed circumference to the total circumference to reduce it to the actual exposed area. The side of the plume element that is longer and larger in area due to trajectory curvature nearly compensates for the opposite side that is shorter and smaller.

The appropriate ratio of correction is

$$a_T = \frac{\pi - 2\phi}{\pi} \tag{51}$$

where

$$\phi = \arctan\sqrt{\frac{4b^2 - L^2}{L^2}} \tag{52}$$

where ϕ is defined in Figure 68 and L is the spacing between adjacent ports. The same correction factor applies to the growth entrainment term.

While it is assumed that the current is perpendicular to the diffuser axis, the method may be used for angles between 45 and 135 degrees (90 degrees being equivalent to a current perpendicular to the diffuser) by multiplying L by the factor sin ψ where ψ is the angle between $\underline{U}$ and the diffuser axis. This method is justified by measurements of dilution of merging plumes (Roberts, 1977).

The correction factor for the cylinder projected area is simply

$$a_{cyl} = \frac{L}{2b} \tag{53}$$

Finally, the correction term for the curvature projected area entrainment contribution is

$$a_{cur} = 1 - \frac{2\phi}{\pi} + \frac{\sin 2\phi}{\pi} \tag{54}$$

Equations 49 and 50 must also be modified when merging occurs. As was pointed out in the previous section, the mass of the plume element is obtained by knowing the initial mass and integrating the entrainment equation. Given that the mass, average plume density, and element length are known, the element volume can be determined. Upon merging, the transverse dimension of the plume element (i.e. along $\hat{e}_2$) is assumed to be limited to a maximum length of L, the spacing distance. Effectively, a vertical plane half way between the ports acts as a wall or reflecting plane. This technique is common in air pollution modeling (Turner, 1970) where a fictitious mirror source is used to estimated dispersion in the presence of an actual physical barrier. With plume merging the sources are real.

Thus, the volume of the plume element can be thought to be the product of h and the area of a rounded rectangle, see Figure 69. This area is the quotient of the element volume and the length which, after simplification, becomes

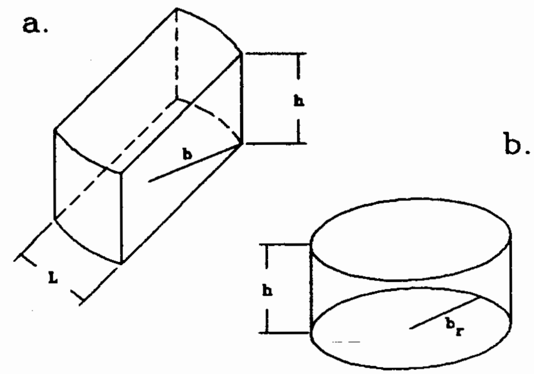


Figure 69. Derivation of dimensions under merging: a) the merged element with volume confined between reflection planes, and b) the corresponding unmerged element of equal volume.

$$\pi b_r^2 = \pi b^2\left(1 - \frac{2\phi}{\pi}\right) + 2b^2 \sin\phi\cos\phi \tag{55}$$

where b_r is the unmerged round element radius and b is now the radius of the element in the vertical plane. In other words, b describes the plume element parallel to $\hat{e}_3$. Solving for b

$$b = \frac{\pi b_r}{\pi - 2\phi + 2\sin\phi\cos\phi} \tag{56}$$

the subscript $t+\delta t$ has been left off for simplicity. Since φ is larger than $\sin\varphi\cos\varphi$, b is larger than b_r.

Average and Centerline Plume Properties

The previous discussion is in terms of average plume properties because average plume properties are physically compatible with the average motion of the plume element. We do not expect that centerline buoyancy can accurately describe, via vertical acceleration, the plume trajectory traced by the plume element as a whole. After all, the element is an entity which stretches from one boundary with the ambient flow to the other, with widely varying properties in between.

On the other hand, centerline concentrations are of greatest concern to environmentalists because they have the greatest potential for causing adverse impacts to living organisms. Fortunately, plumes are often found to possess predictable patterns of cross-sectional properties. For example, plumes discharge into quiescent fluid tend to display the Gaussian profile, very dilute at the edges and concentrated at the center. However, the Gaussian profile is not very compatible with the plume element described above because it extends to infinity whereas we have described an element with definite boundaries. Consequently, another profile, the 3/2 power profile (Kannberg and Davis, 1976), which closely matches the Gaussian profile over the concentrated portion of its range, is used to determine the centerline concentration as a function of the average concentration, or dilution, that UM predicts.

The 3/2 power profile is expressed as

$$\Phi = \left(1 - \left(\frac{r}{b}\right)^{\frac{3}{2}}\right)^2 \qquad (57)$$

where Φ is instantaneous scaling factor relating differences between the plume and the ambient of an appropriate property, such as the concentration of some pollutant or velocity, b is the plume radius, and r is the distance from the center of the plume to the point within the plume at which Φ is measured.

The peak-to-mean ratio is simply the ratio of the centerline to the average concentration, it is obtained from a flux integral. We start with the relationship for the average concentration

$$C_{avg} = \frac{\int_A C v dA}{\int_A v dA} \qquad (58)$$

where C_{avg} is equivalent to the average concentration obtained from UM, C and v are the instantaneous concentration and velocity in the plume element, A is the cross-sectional area, and dA is the corresponding infinitesimal area. The peak-to-mean ratio is defined to be C_{max}/C_{avg}

$$\frac{C_{max}}{C_{avg}} = \frac{C_{max}\int_A v dA}{\int_A C v dA} \qquad (59)$$

where C_{max} is the centerline concentration. This integrals in this quotient are not easy to solve analytically and, therefore, are estimated numerically in UM.

It is illuminating to define limiting values of the coefficient. When dilutions and currents are large a simplification is possible. In this case the velocity can be considered constant and can be factored from the integrals, giving

$$\frac{C_{max}}{C_{avg}} = \frac{C_{max}\int_A dA}{\int_A C dA} \qquad (60)$$

Using this approximation and assuming the 3/2 power profile a peak-to-mean ratio of 3.89 is found for round plumes. The corresponding ratio for a fully merged line plume is 2.22. However, the ratios vary and in much of the plume the peak-to-mean ratios are considerably smaller than these limiting values, in fact, near the source they often approach 1.0, depending on the uniformity of the source.

Experimental Justification of the Projected Area Entrainment Hypothesis

In 1989, Roberts, Snyder, and Baumgartner published three papers in ASCE (1989a,b,c) which measure and record the behavior of merging laboratory plumes in flowing, stratified environments. Although they did not set out to do so, their findings directly corroborate PAE, as shown below:

Start with Equation 13a of Roberts, Snyder, and Baumgartner (1989a)

$$\frac{S_m q N}{b^{2/3}} = 1.08 F^{1/6} \qquad (61)$$

where S_m is the centerline dilution in the plume, q is the diffuser volume flux per unit length, b is the buoyancy flux per unit length (i.e. the product of the reduced gravitational acceleration and the volume flux per unit length), F is a type of Froude number (u^3/b, where u is the current speed), and N is the buoyancy (Brunt-Vaisala) frequency

$$N = \left(-\frac{g}{\rho_a}\frac{d\rho}{dz}\right)^{1/2} \qquad (62)$$

and $d\rho/dz$ is the ambient density gradient. Then add Equation 13b of same reference which states

$$\frac{z_e}{l_b} = 1.85F^{-1/6} \tag{63}$$

where z_e is the rise above the port datum of the top of the fully merged wastefield and l_b is a buoyant length scale defined by RSBa Equation 4

$$l_b = \frac{b^{1/3}}{N} \tag{64}$$

Combining, noting that $q = Q/L$, where L is the length of the diffuser and Q is the diffuser total volume flux, and making the appropriate substitutions yields

$$S_m = \frac{1.08}{1.86}\frac{Lz_e u}{Q} \tag{65}$$

The quantity $Lz_e u$ is, of course, just the flux through the projected area, which is the integrated form of PAE! The coefficient is within the general range described in the previous section.

This derivation proves, at least in an overall sense, that, given sufficiently great current, initial dilution is given simply by the quotient of the flux through the projected area of the wastefield divided by the source flux, multiplied by a constant factor. In lieu of convincing evidence to the contrary, it is eminently reasonable to assume that such an integrated outcome is the result of adding the individual projected area fluxes throughout the plume trajectory. In other words, it is not reasonable to assume, a priori, that the plume entrains differentially over its projected area, perhaps at twice the rate at one point and half the rate at another. Any such deviations are thought to be due to the aspiration effect of the Taylor entrainment coefficient which can be treated separately. In other words, the two entrainment mechanisms act independently, are mathematically linear, and may be added.

7. FARFIELD ALGORITHM

6. UM MODEL THEORY

7. FARFIELD ALGORITHM

FARFIELD ALGORITHM

Farfield dispersion is estimated using the method of Brooks (Fischer, 1979). This method is well established and relatively simple to implement. Two equations are used, one for estimating concentrations in open coastal waters, large estuaries, and other large water bodies, and another for channels, streams, and other confined systems.

The open water equation is given by,

$$S = \frac{S_i}{erf\left(\sqrt{\frac{1.5}{(1 + 8Kw^{\frac{4}{3}}\frac{t}{w^2})^3 - 1}}\right)} \tag{33}$$

where *erf* is the error function, S is the centerline dilution in the farfield plume, S_i is the initial dilution (at maximum rise, overlap, or other special condition), K is a dispersion coefficient, w is the width of the plume field at the end of initial dilution, and t is the time of travel from the point of the end of initial dilution to the point of interest.

The confined water (riverine, small lakes) equation is given by

$$S = \frac{S_i}{erf\left(\sqrt{\frac{w^2}{16K\,w^{\frac{4}{3}}t}}\right)} \tag{35}$$

where the same definitions apply, but the dispersion coefficient may take on a different value.

In both Equations 66 and 67 the width, w, is the horizontal width of the wastefield measured perpendicular to the current:

$$w = (N - 1)s_{ef} + 2b \tag{36}$$

where N is the number of ports, s_{ef} is the effective spacing (spacing multiplied by $\sin\psi$), and b is the radius of the plume at the end of initial dilution. Equation 68 is simply the physical projection of the diffuser plus the additional growth of the plumes outside of this region. It is an approximation which does not account for the "attraction" of the plumes to each other or other mechanisms which can affect the width of the wastefield, including upstream intrusion.

Equations 66 and 67 only provide estimates of volume dilution, which is appropriate for conservative pollutants (decay = 0) and unpolluted ambient receiving water. UM uses additional

equations to estimate the effect of first order decay and ambient background concentrations. The sequence in each time step is as follows.

First a distance (path), presumed to be along ambient streamlines, is established. It is computed by adding the value in the [far inc] cell to the distance of the element in the present time step. If the sum is greater than the value found in the [far dis] cell then it is set to that value, and eventually the program is terminated. The time elapsed in traversing the distance between succesive values is found by solving the distance is equal to rate times time formula. The total time is also incremented and Equations 66 and 67 may be solved. The incremental mass gained by the element during the time step is determined by

$$\Delta m = (S_{t+\Delta t} - S_t)m_o \tag{37}$$

where Δm is the mass entrained during the time step and m_o is the plume element mass at the port. The total pollutant in the element is given by

$$m_{pt+\Delta t} = m_{pt}e^{-k\Delta t} + \Delta m\,\chi_a e^{-\frac{k}{2}t} \tag{38}$$

where m_p is the total mass of pollutant in the plume element, k is the first order decay constant, χ_a is the local ambient pollutant concentration.

If the decay rate, k, is equal to zero then the exponentials in the above equation are unity. In this case the ambient concentration may be constant. However, the pollutant in question is not conservative but is present in the ambient water, then it is also subject to decay. Equation 70 states that the ambient concentration follows the same decay law as that in the plume. These assumptions could impact the analysis of species such as coliform bacteria.

The final farfield calculation made during each time step determines the local average pollutant concentration in the plume element:

$$\bar{\chi} = \frac{\chi_o m_p}{S m_o} \tag{39}$$

where χ with the bar is the average pollutant concentration in the element and χ_o is the pollutant concentration in the effluent.

The farfield algorithm is much simpler than the initial dilution part of UM. The quality of the estimates should not, in general, be expected to be as high as the initial dilution model. Consequently, if better methods for estimating the farfield concentration are available they should be considered.

REFERENCES

Akar P.J. and G.H. Jirka, July 1990. CORMIX2: An expert system for hydrodynamic mixing zone analysis of conventional and toxic multiport diffuser discharges. Draft, ERL, Office of Research and Development, USEPA, Athens, GA 30613.

Albertson, M.L., Y.B. Dai, R.A. Jensen, and H. Rouse, 1948. Diffusion of submerged jets. Transactions of the American Society of Civil Engineers, pp 1571-1596.

Anon., 1982. Code of Federal Regulations. Parts 122 and 125. Modifications of secondary treatment requirements for discharge into marine waters. Federal Register. Vol 47, No. 228. pp 53666-85. (November 26, 1982).

Anon., 1983. Clean water act amendments of 1983. Report of the committee on environment and public works, United States Senate. Report No. 98-233. September 21, 1983. U. S. Government Printing Office. Washington.

Anon., 1987. Water quality act of 1987, Public Law 100-4, February 4, 1987. Congress of the United States. U. S. Government Printing Office. Washington.

APHA, 1975. Standard methods for the examination of water and wastewater. 14th Edition. American Public Health Association. Washington. 1193 pp.

Baker, E.T., J.W. Lavelle, R.A. Feely, G.J. Massoth, and S.L. Walker, 1989. Episodic venting of hydrothermal fluids from the Juan de Fuca Ridge. Journal of Geophysical Research.

Baumgartner, D.J., and D.S. Trent, 1970. Ocean outfall design Part I, literature review and theoretical development. NTIS No. PB 203-749. (April 1970).

Baumgartner, D.J., W.E. Frick, W.P. Muellenhoff, and A.M. Soldate, Jr., 1986. Coastal outfall modeling: status and needs. Proceedings Water Pollution Control Federation 59th Annual Conference. Los Angeles, CA. (October 7, 1986).

Behlke, C.E. and F.J. Burgess, 1964. Comprehensive study on ocean outfall diffusers. Oregon State University, Engineering Experiment Station, Department of Civil Engineering. 26 pp. May 1, 1964.

Bodeen, C.A., T.J. Hendricks, W.E. Frick, D.J. Baumgartner, J.E. Yerxa, and A. Steele, 1989. User's guide for SEDDEP: a program for computing seabed deposition rates of outfall particulates in coastal marine environments. EPA Report 109-ERL-N. Environmental Protection Agency, Newport, OR 97365. 79 pp.

Brooks, N.H., 1956. Methods of analysis of the performance of ocean outfall diffusers with application to the proposed Hyperion outfall. Report to Hyperion Engineers, Los Angeles California (April 5, 1956).

Brooks, N.H., 1960. Diffusion of sewage effluent in an ocean current. pp 246-267. Proceedings of the First Conference on Waste Disposal in the Marine Environment. Ed. E.A. Pearson. Pergamon Press. New York. 569 pp.

Brooks, N.H., 1973. Dispersion in hydrologic and coastal environments. EPA-660/3-73-010. (August 1973).

Carhart, R.A., A.J. Policastro, S. Ziemer, S. Haake, and W. Dunn, 1981. Studies of mathematical models for characterizing plume and drift behavior from cooling towers, Vol. 2: mathematical model for single-source (single-tower) cooling tower plume dispersion. Electric Power Research Institute, CS-1683, Vol. 2, Research Project 906-01.

Carhart, R.A., A.J. Policastro and S. Ziemer, 1982. Evaluation of mathematical models for natural-draft cooling-tower plume dispersion. Atmospheric Environment, Vol 16, pp. 67-83.

Carr, V.E., W.D. Watkins, and J.F. Musselman, 1985. Ocean Outfall Study, Morro Bay California. Report to Region IX Shellfish Specialist. Northeast Technical Services Unit. Davisville, R.I. U.S. Department of Health and Human Services. 74 pp.

Callaway, R.J. 1971. Application of some numerical models to Pacific Northwest estuaries. pp 29-97. Proceedings Technical Conference on Estuaries in the Pacific Northwest. Oregon State University, Engineering Experiment Station Circular 42. (March 19, 1971).

Doneker, R.L. and G.H. Jirka, 1990. Expert system for hydrodynamic mixing zone analysis of conventional and toxic submerged single port discharges (CORMIX1). EPA/600/3-90/012, ERL, Office of Research and Development, USEPA, Athens, GA 30613.

Fan L.N., 1967. Turbulent buoyant jets into stratified or flowing ambient fluids. Report No. KH-R-15, W.M. Keck Lab. of Hydraulics and Water Resources, California Institute of Technology, Pasadena, California.

Fischer, H.B., E.J. List, R.C.Y. Koh, J. Imberger, and N. H. Brooks, 1979. Mixing in inland and coastal waters. Academic Press. New York. 483 pp.

Frick, W.E. and L.D. Winiarski, 1975. Comments on "The rise of moist buoyant plumes". J. of Applied Meteorology, Vol. 14, p. 421.

Frick, W.E. and L.D. Winiarski, 1978. Why Froude number replication does not necessarily ensure modeling similarity. Proceedings of the Second Conference on Waste Heat Management and Utilization. Dept. of Mechanical Engrg., Univ. of Miami (December 4-6, 1978).

Frick, W.E., 1981. A theory and users' guide for the plume model MERGE, revised, Tetra Tech Inc., Environmental Research Laboratory, Corvallis, OR.

Frick, W.E., 1984. Non-empirical closure of the plume equations. Atmospheric Environment, Vol. 18, No. 4, pp. 653-662.

Frick, W.E., C.A. Bodeen, D.J. Baumgartner, and C.G. Fox, 1990. Empirical energy transfer function for dynamically collapsing plumes. Proceedings of International Conference on Physical Modeling of Transport and Dispersion, MIT, (August 7-10, 1990).

Frick, W.E., C.G. Fox, and D.J. Baumgartner, 1991. Plume definition in regions of strong bending. Proceedings of the International Symposium of Environmental Hydraulics (December 16-18, 1991).

Frick, W.E., D.J. Baumgartner, and C.G. Fox, 1992. Elements and control volumes in plume modeling. In prep, 1992.

Grace, R.A., 1978. Marine Outfall Systems. Prentice-Hall. Englewood Cliffs. 600 pp.

Gremse, F., 1980. Transmittal of the DPHYDR program. Personal communication.

Hendricks, T.J., 1983. Numerical model of sediment quality near an ocean outfall. NOAA Final Report on Grant # NA8ORAD00041. Seattle, WA.

Hendricks, T., 1982. An advanced sediment quality model. Biennial Report for the years 1981-82. SCCWRP. Long Beach, CA. pp 247-257.

Hinton, S.W. and G. Jirka, 1992. User's guide for the Cornell mixing zone expert system (CORMIX). National Council of the Paper Industry for Air and Stream Improvement, Inc.. Technical Bulletin No. 624. February, 1992.

Hoult D.P., J.A. Fay, and L.J. Forney, 1969. A theory of plume rise compared with field observations. J. of Air Pollution Control Association, Vol 19, pp. 585-589.

Hunt, J., 1990. Particle Removal by coagulation and settling from a waste plume. Oceanic Processes in Marine Pollution, Vol. 6. Physical and Chemical Processes: Transport and Transformation. Eds. D.J. Baumgartner and I.W. Duedall. Krieger Publishing Co. Malabar Florida. 248 pp.

Isaacson, M.S., R.C.Y. Koh, and N.H. Brooks, 1978. Sectional hydraulic modeling study of plume behavior: San Francisco Southwest Ocean Outfall Project. W.M. Keck Laboratory of Hydraulics and Water Resources, California Institute of Technology, Technical Memorandum 78-2.

Isaacson, M.S., R.C.Y. Koh, and N.H. Brooks, 1983. Plume dilution for diffusers with multiple risers. Journal of Hydraulic Engineering, ASCE, Vol. 109, No. 2, pp 199-220.

Jones, G.R., 1990. "CORMIX3: An expert system for the analysis and prediction of buoyant surface discharges" Masters Thesis, Cornell University.

Kannberg, L.D. and L.R. Davis, 1976. An experimental/analytical investigation of deep submerged multiple buoyant jets. USEPA Ecological Research Series, EPA-600/3-76-101, USEPA, Corvallis, OR.

Lee J.H.W. and V. Cheung, 1990. Generalized Lagrangian model for buoyant jets in current. ASCE J. of Environmental Engineering, Vol. 116, No. 6, pp. 1085-1106.

Menzie, C. A. and Associates, 1986. Technical Information and Research needs to Support A National Estuarine Research Strategy. Battelle Contract No. 68-01-6986 Final Report to EPA. Various Paging. (January 1986).

Morton, B.R., G.I. Taylor, and J.S. Turner, 1956. Turbulent gravitational convection from maintained and instantaneous sources. Proceedings of the Royal Society of London. A234: pp 1-23.

Morton, B.R., 1959. Forced plumes. Journal of Fluid Mechanics. 5: pp 151-197.

Muellenhoff, W.P., A.M. Soldate, Jr., D.J. Baumgartner, M.D. Schuldt, L.R. Davis, and W.E. Frick, 1985. Initial mixing characteristics of municipal ocean outfall discharges: Volume 1. Procedures and Applications. EPA/600/3-85/073a. (November 1985).

Ozretich, R.J. and D.J. Baumgartner, 1990. The utility of buoyant plume models in predicting the initial dilution of drilling fluids. Oceanic Processes in Marine Pollution, Vol. 6. Physical and Chemical Processes: Transport and Transformation. Eds. D. J. Baumgartner and I.W. Duedall. Krieger Publishing Co. Malabar Florida. 248 pp.

Policastro, A.J., R.A. Carhart, S.E. Ziemer, and K. Haake, 1980. Evaluation of mathematical models for characterizing plume behavior from cooling towers, dispersion from single and multiple source draft cooling towers. U.S. Nuclear Regulatory Commission Report NUREG/CR-1581 (Vol. 1).

Pomeroy, R., 1960. The empirical approach for determining the required length of an ocean outfall. pp 268-278. Proceedings of the First Conference on Waste Disposal in the Marine Environment. Ed. E. A. Pearson. Pergamon Press. New York. 569 pp.

Rawn, A.M., F.R. Bowerman, and N.H. Brooks, 1960. Diffusers for disposal of sewage in sea water. Proceedings of the American Society of Civil Engineers, Journal of the Sanitary Engineering Division. 86: pp 65-105.

Roberts, P.J.W., 1977. Dispersion of buoyant waste water discharged from outfall diffusers of finite length. W. M. Keck Laboratory of Hydraulics and Water Resources, California Institute of Technology. Pasadena CA. (Report # KH-R-35).

Roberts, P.J.W., 1989. Dilution Hydraulic Model Study of the Boston Wastewater Outfall. Report Number SCEGIT 89-101, School of Civil Engineering, Georgia Institute of Technology.

Roberts, P.J.W., W.H. Snyder, and D.J. Baumgartner, 1989 a. Ocean outfalls I: submerged wastefield formation. ASCE Journal of Hydraulic Engineering. 115. No. 1. pp 1-25.

Roberts, P.J.W., W.H. Snyder, and D.J. Baumgartner, 1989 b. Ocean outfalls II: spatial evolution of submerged wastefield. ASCE Journal of Hydraulic Engineering. 115. No. 1. pp 26-48.

Roberts, P.J.W., W.H. Snyder, and D.J. Baumgartner, 1989 c. Ocean outfalls III: effect of diffuser design on submerged wastefield. ASCE Journal of the Hydraulic Engineering. 115. No. 1. pp 49-70.

Roberts, P.J.W., 1990. Outfall design considerations. The Sea. Ocean Engineering Science. Vol 9. Eds. B. LeMehaute and D. M. Hanes. Wiley and Sons. New York. pp 661-689.

Roberts, P.J.W., 1991. Basic language RSB program. Personal communication.

Roberts, P.J.W., 1992. "Hydraulic Model Study for the Boston Outfall. I: Riser Configuration," To be submitted to Journal of Hydraulic Engineering.

Schatzmann M., 1979. An integral model of plume rise. Atmospheric Environment, Vol. 13, pp. 721-731.

Spiegel, E.A. and G. Veronis, 1960. On the Boussinesq approximation for a compressible fluid. Astrophys. J., 131, pp 442-447.

State Water Resources Control Board, 1988. Water Quality Control Plan for Ocean Waters of California, California Ocean Planeastacramento. (September 22, 1988).

Teeter, A.M. and D.J. Baumgartner, 1979. Prediction of initial mixing for municipal ocean discharges. CERL Publ. 043, 90 pp. U. S. Environmental Protection Agency Environmental Research Laboratory, Corvallis, Oregon.

Tetra Tech, 1980. Technical evaluation of Sand Island wastewater treatment plant section 301(h) application for modification of secondary treatment requirements for discharge into marine waters. Prepared for U.S. EPA, Washington, D.C..

Tetra Tech, 1982. Revised Section 301(h) Technical Support Document. Prepared for U. S. Environmental Protection Agency. EPA 430/9-82-011. (November 1982).

Tetra Tech, 1984. Technical review of the Sand Island wastewater treatment plant section 301(h) application for modification of secondary treatment requirements for discharge into marine waters. Prepared by Tetra Tech, Inc.

Tetra Tech, 1987. A simplified deposition calculation (DECAL) for organic accumulation near marine outfalls. Prepared for USEPA. Washington, D.C.

Turner D.B., 1970. Workbook of atmospheric dispersion estimates. Office of Air Programs Publication No. AP-26. USEPA, Research Triangle Park, North Carolina.

U. S. Environmental Protection Agency, 1982. Revised Section 301(h) Technical Support Document. EPA 430/9-82-011. (November 1982)

U. S. Environmental Protection Agency, 1985. Technical Support Document for Water Quality-based Toxics Control. EPA-400/4-85-032. (September 1985).

U. S. Environmental Protection Agency, 1986. Quality Criteria for Water, 1986. EPA 400/ (May, 1986).

Ward, G.H. Jr., and W.H. Espey Jr., Eds., 1971. Estuarine Modeling: An Assessment. Capabilities and Limitations for Resource Management and Pollution Control. EPA Water Pollution Control Research Series. 16070 DZV 02/71. 497 pp. February, 1971.

Weast, R.C., 1978. CRC Handbook of Chemistry and Physics. CRC Press, Inc., Cleveland, OH 44128.

Weil J.C., 1974. The rise of moist buoyant plumes. Journal of Applied Meteorology, Vol. 13, No. 4.

Winiarski, L.D. and W.E. Frick, 1976. Cooling tower plume model. USEPA Ecological Research Series, EPA-600/3-76-100, USEPA, Corvallis, Oregon.

Winiarski, L.D. and W.E. Frick, 1978. Methods of improving plume models. Presented at Cooling Tower Environment -- 1978. University of Maryland. (May 2-4 1978).

Wood, I.R. and M.J. Davidson, 1990. The merging of buoyant jets in a current. Proceedings of International Conference on Physical Modeling of Transport and Dispersion, MIT, (August 7-10, 1990).

Wright, S.J., 1984. Buoyant jets in density-stratified crossflow. J. of Hydraulic Engineeringg., ASCE, 110(5), pp 643-656.

8. APPENDIX 1: MODEL RECOMENDATIONS
APPENDIX 2: SUPPORT FOR TABLES I AND II (CHAPTER 1)

8. APPENDIX 1: MODEL RECOMENDATIONS
APPENDIX 2: SUPPORT FOR TABLES I AND II (CHAPTER 1)

APPENDIX 1: MODEL RECOMMENDATIONS

JUSTIFICATION FOR USES OF PLUMES MODELS IN FRESH WATER

The title of this work "Dilution models for effluent discharges" signifies that this report encompasses a broader scope than Muellenhoff et al. (1985) which addressed primarily ocean discharges. The reasons are many but most importantly, users of Muellenhoff et al. (1985) often applied the plume models to freshwater outfalls because experience showed that some of the models, UMERGE included, worked well in that setting.

However, since 1985 the CORnell MIXing zone models (Hinton and Jirka, 1992), CORMIX, have been developed, supported in part by EPA, for the express purpose of addressing the problem of discharges to comparatively shallow, terrestrial freshwater systems. CORMIX uses a classification scheme based on length scales, linking to it a number of formulae and methods appropriate for each sub-category, and linking together separate plume behaviors into an estimate of overall behavior, much like PLUMES links RSB and UM to a farfield algorithm. This is done for a broad range of conditions, including single ports, merging plumes, and surface discharges, covering many conditions encountered in practice.

In addition to this practical reason for addressing the freshwater uses of our models, there are valid reasons for occasionally recommending them, even for those categories for which CORMIX was expressly developed. Speed of analysis is one reason. Suppose, for example, that it is to be established what percentage of time annually a plume surfaces. Say that this estimate is to be based on available hourly data collected during a monitoring study. This may require hundreds of simulations. As presently constituted, it may be unreasonable to expect an applicant to do the analysis using CORMIX.

MODEL RECOMMENDATION TABLES

General Considerations

Recommendations for use of the models UM and RSB are based on the experience of the authors who have contributed to the formulation of the models and the interface, PLUMES, and have gained experience with the models in a large number of design and analysis applications. Our experience with CORMIX is not as extensive, and we have not contributed directly to its formulation. Furthermore, CORMIX is only recently available for multiport discharges and we have seen few the results of its application to actual cases. Our recommendations for use of CORMIX are somewhat tenuous and depend in part on the capabilities claimed by Dr. Gerhard Jirka, one of the principal authors of the CORMIX system. In addition to the CORMIX references we cited we have benefitted from his personal communications, although we are responsible for the way we have chosen to use or not to use his comments, some of which are given at the end of this appendix.

given at the end of this appendix.

The basic responsibility for choice of a model lies with the user, especially in relation to application for regulatory permits, which may carry important legal implications in addition to professional responsibility. There are many models and other approaches than can be used to estimate initial dilution that may be acceptable to regulatory agencies. By presenting the following recommendations we do not claim that any others should not be used. In fact we include ULINE (Muellenhoff et al., 1985) in the tables which follow (indicated by "L") based on our experience and its close connection to RSB. We do not provide recommendations for UPLUME and UOUTPLM because wherever they may have been used appropriately in the past we now believe UM is used more effectively, even in the case where the regulatory agency requires use of zero ambient current. We do not include UDKHDEN (Muellenhoff et al., 1985) in our recommendations because we have not followed its use since 1985 and we believe Dr. Lorin Davis has made improvements to his original model from which UDKHDEN was adapted.

In general we believe RSB (indicated by "R" or "r" in the following tables) is applicable to at least any case that matches closely with the experimental conditions used for its development. See Figure 2 of Roberts et al., 1989a as a guide: a complete list of experimental parameters is included as Appendix 1 (Table 5) of Roberts et al., 1989c. Other cases in which the density gradient over the height of rise can be represented by a linear gradient may be effectively modeled by RSB. Submerged pipelines with fairly closely spaced multiport risers may be modeled by RSB (Roberts, 1989). Additional classifications for environmental and discharge situations based on the scheme developed for CORMIX are shown in the tables of recommendations. Lower case "r" is used in several types of situations to indicate the RSB model might be useful in analyzing these cases.

The model UM (designated "U" in the following tables) is likewise useful for the typical domestic sewage discharge conditions in coastal waters for which RSB was created, recognizing that for UM the port spacing must be specified as a half that used for RSB. Multiport risers may also be modeled with UM if the risers are close enough so that there is merging between riser plumes. If not, UM may be used to analyze the merged plumes generated from each riser. UM may also be used for freshwater discharges, providing unique capability in cases where the maximum density temperature is encountered, owing to a robust and rigorously defined equation of state. Vertical non-uniformities in current speed and direction, as well as non-uniform density and ambient contaminant concentrations are handled directly by UM, although approximate corrections can be made to RSB dilution predictions for vertically uniform ambient concentrations of contaminants. UM is well suited for very dense liquid or slurry discharges because the model is not constrained by the Boussinesq approximations and in additions can handle negatively buoyant flows. While not frequently encountered, UM is appropriate for analysis of diffusers with ports only along one side. A lower case "u" is used to indicate where UM is less useful, such as in the case of parallel currents and in shallow water discharges.

Caveats

The recommendations given in the following tables are intended for general guidance purposes and to emphasize the complementary capabilities of the RSB and UM models and the CORMIX expert system. No attempt is made to define a rigorous classification system as defined in CORMIX, which, between CORMIX 1, 2, and 3, classifies perhaps 90% of common plume problems. The CORMIX classification system is made possible by adopting assumptions which, while making it possible to analyze the majority of freshwater outfall problems objectively, unfortunately either misanalyzes or excludes the remainder. Some of the latter are important in certain regions of the country and/or under special circumstances. Hence, a more flexible structure is needed, albeit one which must appeal to the user for help to assure that the models are appropriately implemented. The user must be the ultimate judge of the applicability of any given model under the circumstances at hand.

Description and Usage

Table V specifies the applicability of the CORMIX1 (single port CORMIX) and UM models to single port submerged discharge problems. Similarly, Table VI addresses multiport submerged diffusers. General applicability is indicated by the placement in alphabetical order of either a *C* for CORMIX1 or R, U, u, or r for RSB or UM. Because we are more knowledgeable with our own models than with CORMIX, we indicate a general quality of our models with an upper case letter, e.g. U, signifying that we think the model generally performs well in this category, or lower case letter, e.g. u, suggesting that the user may wish, depending on the sensitivity of the project and other considerations, to seek other models, like CORMIX, if they applies.

An italicized C, i.e. *C*, for CORMIX conveys the fact that we are not experts in CORMIX usage and do not feel justified in assigning a measure of quality it. We simply include it to indicate the general domain of applicability of the CORMIX models, bearing in mind that the importance of a particular category is not necessarily represented by the relative size of the box. In its domain CORMIX can always be used in analysis and be accepted by the authors and regulators, in review situations, providing that some special circumstances, some of which are identified below, do not preempt such usage.

Each table classifies conditions and effluent types in an array in which the categories are not exclusive, but rather assimilative. Guidance is derived from the tables by identifying the appropriate effluent type and then examining the applicability ratings in that row. The row can be likened to a chain in which each condition relevant to the problem is a link. The weakest link determines the strength of the chain.

For example, with respect to Table V, if we have a deeply submerged outfall (i.e. boundary conditions, BCs, are unimportant), discharging effluent which is moderately buoyant, into a lake

which is stratified into two layers, with co-flowing current (directed in the same general direction as the effluent), and no background pollution, decay, or upstream intrusion (the presences of which would be indicated by UM with an overlap message), then both CORMIX and UM would
be applicable. In this case, the chain would consist of the 1,2 stratification and 2-D current links which show U's in both instances and there are no weak links.

If the current were not co-flowing but directionally stratified, implying need for the 3-D current link, then the UM link would be relatively weak, and, assuming as we do that all CORMIX simulation modules use formulae and coefficients of uniformly high quality, CORMIX would be the model of choice. On the other hand, going back to the original case, if background pollution is present then the CORMIX chain contains a weak link.

It should be noted that CORMIX does not explicitly include background or decay in its simulations accounting for the absence of a *C* in the corresponding table boxes. However, calculations could be made separately to estimate the consequences of these effects on predictions.

The meaning of the table columns and rows and other comments are given in the following sections.

Single Port Diffuser Model Recommendations: Table V

<u>Table V: Columns</u>

Table V sub-divides the stratification column into three, one each for unstratified, singly or doubly stratified, or multiply stratified water bodies. Length scale analysis may be used, as it is in CORMIX, to define these categories more precisely. Whether layering is important depends on the strength of stratification as well as the buoyancy flux of the source, however, an unstratified system is one in which truly buoyant (non-nascent density) discharges reach the surface, which can be established quickly simply by running UM. In stratified systems the density varies with depth and the plume will trap (come to equilibrium) at some intermediate depth.

For current, the 2-D sub-column is restricted to effluents and conditions where the current is either substantially co-flowing or counter-flowing, or, the current is sufficiently weak not to affect trajectory plume direction significantly in the initial dilution region, i.e. before maximum rise, overlap, or trapping. The latter condition, i.e. weak current, justifies the use of UM in the example given in the CORMIX1 Comparison chapter (the fact that the analysis was conservative further justifying its use). Three dimensional current (3-D) means there is a

Tabla V. Single port discharge model recommendations.

Conditions / Effluent Types	Stratification: none	Stratification: 1,2	Stratification: 3+	Current: 2-D	Current: 3-D	Other sources, decay	BCs	Intrusion	VSW
Buoyant discharges: sewage, industrial waste especially to saline waters	*C* U	*C* U	U	*C* U	*C* u	U	*C* u	*C* u	u
Slightly buoyant discharges, signif. momentum: thermal discharges	*C* U	*C* U	U	*C* U	*C* u	U	*C* u	*C* u	u
Dense discharges: light brine, R.O. discharge, industrial waste	*C* U	*C* U	U	*C* U	*C* u	U	*C* u	*C* U	u
Discharges with nascent or non-linear density effects: thermal discharge to cold water	U	U	U	U	u	U	u	u	u

significant component of current perpendicular to the flow of the effluent or the current direction varies with depth and significantly affects the trajectory.

The other sources, decay column indicates that there are significant levels of uniform horizontally distributed background pollution (concentration) in the water body, or that there is a nearby source which creates a localized background pollution field in the vicinity of the outfall, and/or the pollutant in the effluent is subject to first order decay. Note, while the effect of uniform horizontally distributed background is well simulated by UM, nearby sources may create fields with large horizontal gradients which may make farfield estimates, especially, less robust. For example, can the user establish that spatially separated plumes actually interact? Also note, that UM assumes background fluid is entrained at the level of the center-of-mass of the plume element so that pollution profiles may need to be adjusted to compensate for the effect of this assumption. In other words, given a body of water stratified with high pollution near the surface and low pollution near the bottom. Future versions of PLUMES will provide a method for better estimating this effect.

The boundary conditions (BCs) column indicates that boundaries, bottom, surface, and/or sides, play an important role in the plume problem. The concern here is that the models appropriately limit entrainment due to the interference of the boundary. If side boundaries are important then, assuming there are no missing links, CORMIX should be used exclusively. However, if surface boundaries are important, then UM can generally be used up to the point where it indicates the surface is hit. In general, the indication in UM that the bottom is hit is less important because the bottom is hit by the weakly entraining side of the plume. However, for negatively buoyant plumes, the bottom boundary condition is as important as the surface boundary condition is to truly buoyant plumes.

The intrusion column indicates that portions of the plume will flow upstream and form either stable or unstable upstream protrusions. If an estimate of the length of the effect is wanted, CORMIX is the only model to use, again assuming there are no missing links. However, in estimating the dilution in the wastefield, UM will provide estimates, at the point where the surface is hit, which are consistent with the amount of dilution water available for entrainment due to current or aspiration and can be considered to be reliable. As in Muellenhoff et al. (1985), the dilution could be reduced by ten percent to assure the analysis is conservative.

The final column, VSW or very shallow water, that is, water less than three plume diameters deep, was built into UM to take advantage of its merging algorithm (reflection technique) to estimate initial dilution in cases in which CORMIX provides no estimates, an excluded category brought to our attention by one of our reviewers. It includes cases in which the depth of water depth is less than three times the diameter of the port. UM can be applied using the <shallow/surface Z> command. (Run the READ1st.exe file for the latests developments on this topic.) In such cases the surface or bottom are encountered almost immediately and no criterion is known to establish an appropriate beginning of the farfield. As a result, widely varying estimates of plume spreading are given, depending on where the farfield zone is initiated using the Pause cell capability in the Configuration menu for the farfield start. Our recommendation is that the VSW capability be used only for screening purposes. If it needs to be established that a migration path exists for various fish, then the solution giving the greatest spread might be used as a conservative indicator of wastefield width. If maximum concentration at a mixing zone are of concern, the solution giving the highest concentration might be used.

Table V: Rows

The first three rows in Table V are self-explanatory. For additional background, CORMIX manuals (Doneker and Jirka, 1990; Hinton and Jirka, 1992) are recommended and may be consulted.

The nascent density row is important, even though the effect is not well known. At low ambient temperatures, including freezing temperatures, the non-linearities in the equation of state for fresh or low salinity water, particular in the 0 to approximately 10 C range, cause buoyant plumes to become negatively buoyant as they cool by mixing. The effect, described in the first chapter, is important in cold climate regions. As explained in the CORMIX example chapter, existing versions of CORMIX do not address the problem, in fact, they misanalyze it.

As was pointed out in the CORMIX Comparison chapter, the problem causes some models

to fail completely (one could say catastrophically), by predicting that the effluent will rise to the surface instead of sinking to the bottom. The ramifications could be serious, causing, for example, a monitoring program to be designed to study healthy surface biota whereas the benthic communtity is actually affected.

Multiport Outfall Model Recommendations: Table VI

Tabla VI. Model recommendations for multiport diffusers.

Conditions / Effluent Types	Stratification			Current		Merging		Other sources & decay	BCs	Intrusion	Stage	VSW
	no	1,2	3+	cross	par'l	part	full					
Buoyant discharges: sewage, industrial waste especially to saline waters	C L U	C R U	R U	C R U	C R	R U	C R U	U	C u	C R u	C	u
Slightly buoyant discharges, signif. momentum: thermal discharges	C l U	C r U	r U	C R U	C R	C R U	C R U	U	C u	C R u	C	u
Dense discharges: light brine, R.O. discharge, industrial waste	C U	C U	U	C r U	C r	C R U	C R U	U	C u	C R U	C	u
Discharges with nascent or non-linear density effects: thermal discharge to cold water	U	U	U	U	u	U	U	U	u	u		u

Table VI: Columns and Rows

The multiport discharge model recommendations are given in Table VI. In general, the same comments applying to Table V apply to Table VI as well. **Notable differences are the addition of the models RSB (denoted by R or r), ULINE (denoted by L or l), and columns for degree of merging and staged diffusers.**

The current sub-columns have been changed to indicate the importance of diffuser alignment on plume behavior. Generally, cross-diffuser flow is from perpendicular to 45 degrees off perpendicular, other cases falling in the parallel sub-column.

The merging column indicates the degree of merging, either partial or full. It is worth noting that RSB is considered to be particulary appropriate to tunneled outfalls with multiport risers.

With respect to the intrusion column, only CORMIX provides an estimate of the length of of penetration upstream. However, in estimating the dilution in the wastefield, RSB and UM will provide estimates. RSB, while not predicting the depth of penetration, is considered to be especially applicable for making dilution estimates and provides other information lacking with CORMIX. Again, UM predictions should be interpreted at the point where the surface is hit, that dilution being consistent with the amount of dilution water available for entrainment due to current or aspiration. Also, as in Muellenhoff et al. (1985), the dilution could be reduced by ten percent to assure the analysis is conservative.

The "stage" column refers to staged diffusers, diffuser pipes with ports not perpendicular to the diffuser axis. Such diffusers are staged to use the momentum in the effluent to carry effluent farther from shore. Of the models under consideration, only CORMIX applies to this diffuser configuration.

The Very Shallow Water (VSW) model is included for completeness. In this case the UM model is only recommended for providing very conservative estimates of dilution, i.e. between discharge and the plume hitting the bottom or the surface. The usefulness is very limited, being useful only for closely spaced ports.

SURFACE DISCHARGES

Except for the very shallow water (VSW) category identified in Table V, CORMIX (CORMIX3) is recommended for modeling surface discharges.

OTHER VIEWPOINTS AND RECOMMENDATIONS

Some of the review comments of the draft manuscript dissent from our interpretation of modeling usage and warrant inclusion here. Gerhard Jirka, upon being solicited for his opinion, indicated that UM was applicable to the following CORMIX flow classes:

Single ports: S1, S2, S3, S4, S5, V1, V2, V3, V5, H1, H2, H3, H4, NV1, NV2, NH1, NH2, and NH4, provided they are not associated with an attachment suffix (A..).

Multipport diffusers: MS1, MS2, MS3, MS4, MS5, MS6, MS7, MS8, MU1V, MU1H, and

MNU2.

APPENDIX 2: SUPPORT FOR TABLES I AND II (CHAPTER 1)

TABLE 1

Input and Output for Case 1

```
 Apr  3, 1992,  10:20:10  ERL-N Model PLUMES, Feb 19, 1992        Case:   1 of
20
 Title    EXAMPLE 1, 5 CM/SEC

  tot flow   # ports port flow  spacing  salinity      temp   far inc   far dis
     4.381      100  0.04381       10         0          15       30       100
 plume dep  port dia plume dia total vel horiz vel vertl vel asp coeff print frq
     29.87    0.1524   0.1446    2.669     2.669     0.000      0.10       150
 port elev ver angle cont coef plume den poll conc     decay  Froude # Roberts
F
    0.6096      0.0       0.9   -0.8363       100 1.157E-08     14.13  0.001673
 hor angle eff spacg   amb den   current  far dif   far vel K:vel/cur Stratif
#
        90    10.000     24.28   0.05000  0.000453     0.05     53.370.00003593
     depth   current   density  salinity      temp  amb conc  N (freq) red grav.
       0.0      0.05     24.10        32        13       1.6  0.007731    0.2466
     30.48      0.05     24.29        32        12       1.6 buoy flux puff-ther
                                                                 0.01080     2.936
                                                             jet-plume  jet-cross
                                                                 1.924     6.838
                                                             plu-cross  jet-strat
                                                                 86.41     6.650
                                                               plu-strat

                                                                 12.36

                                                               hor dis>=
```

```
CORMIX-1 (single port) flow: h3; no bottom attachment.  Use CORMIX (DJB)

  deg C,   deg F                                       -2.0 to 50 deg C range
Help: F1.   Quit: <esc>.  Configuration:ATCMO.  FILE: djbmanul.var;
```

CL conc	plume dep m	plume dia m	dilution	avg conc	hor dis m
100.0	29.87	0.1446	1.000	100.0	0.000
70.50	29.86	0.3974	2.783	36.98	0.6407
26.64	29.59	1.051	7.828	14.21	2.386
10.78	27.14	2.188	22.03	6.104	5.809
5.877	23.17	3.489	47.32	3.719	8.418
4.105	19.49	4.783	78.28	2.897	10.28
3.260	16.17	6.069	113.3	2.509	11.85
2.794	13.17	7.359	151.3	2.290	13.26
2.510	10.44	8.670	191.9	2.153	14.58
2.318	7.997	10.01	234.4	2.060	15.83< merging
2.315	7.981	10.02	234.6	2.059	15.84

2.146	5.746	11.61	274.5	1.998	17.06
2.062	3.696	13.45	311.2	1.956	18.25
2.056	3.526	13.62	314.3	1.953	18.35< surface hit

```
Far-field calculations based on Brooks (1960), see guide for details:
Far-field dispersion based on wastefield width of         1004m
                     TSD -----Eq. VI.20-----      -----Eq. VI.21-----
      Time       distance     avg conc  dilution    avg conc  dilution
    sec    hrs          m
  232.9    0.1      30.00        1.952     315.1       1.952     315.1
  832.9    0.2      60.00        1.953     314.7       1.952     315.4
   1433    0.4      90.00        1.953     314.5       1.952     315.6
   1633    0.5      100.0        1.953     314.5       1.951     315.8
==================================================
```

SELECTED INPUT

```
 Apr  3, 1992,  10:20:42  ERL-N Model PLUMES, Feb 19, 1992        Case:   2 of
20
 Title    EXAMPLE 1, 10 CM/SEC

  tot flow   # ports port flow   spacing  salinity      temp   far inc    far dis
     4.381       100   0.04381       10         0         15        30        100
 plume dep  port dia plume dia total vel horiz vel vertl vel asp coeff print frq
     29.87    0.1524    0.1446     2.669     2.669     0.000      0.10        150
 port elev ver angle cont coef plume den poll conc     decay  Froude # Roberts
F
    0.6096       0.0       0.9   -0.8363       100 1.157E-08     14.13   0.01339
 hor angle eff spacg   amb den   current   far dif   far vel K:vel/cur Stratif
#
        90    10.000     24.28    0.1000  0.000453       0.1     26.690.00003593
     depth   current   density  salinity      temp  amb conc  N (freq) red grav.
       0.0       0.1     24.10        32        13       1.6  0.007731    0.2466
     30.48       0.1     24.29        32        12       1.6 buoy flux puff-ther
                                                                0.01080    2.330
                                                             jet-plume  jet-cross
                                                                 1.924     3.419
                                                             plu-cross  jet-strat
                                                                 10.80     6.650
                                                               plu-strat

                                                                 12.36

                                                               hor dis>=


CORMIX-1 (single port) flow: h1; no bottom attachment.  Use EPA models (DJB)

  deg C,   deg F                                          -2.0 to 50 deg C range
Help: F1.  Quit: <esc>.  Configuration:ATCM0.  FILE: djbmanul.var;



 Apr  3, 1992,  10:20:55  ERL-N Model PLUMES, Feb 19, 1992        Case:   3 of
20
 Title    EXAMPLE 1, 20 CM/SEC

  tot flow   # ports port flow   spacing  salinity      temp   far inc    far dis
     4.381       100   0.04381       10         0         15        30        100
 plume dep  port dia plume dia total vel horiz vel vertl vel asp coeff print frq
     29.87    0.1524    0.1446     2.669     2.669     0.000      0.10        150
 port elev ver angle cont coef plume den poll conc     decay  Froude # Roberts
F
    0.6096         0       0.9   -0.8363       100 1.157E-08     14.13    0.1071
 hor angle eff spacg   amb den   current   far dif   far vel K:vel/cur Stratif
#
        90    10.000     24.28    0.2000  0.000453       0.2     13.340.00003593
     depth   current   density  salinity      temp  amb conc  N (freq) red grav.
```



```
       0.0       0.2     24.10        32        13       1.6  0.007731    0.2466
     30.48       0.2     24.29        32        12       1.6 buoy flux puff-ther
                                                                0.01080    1.850
                                                             jet-plume  jet-cross
                                                                 1.924     1.710
                                                             plu-cross  jet-strat
                                                                 1.350     6.650
                                                               plu-strat

                                                                 12.36

                                                               hor dis>=


CORMIX-1 (single port) flow: s4; attachment: a1.  Use EPA models (DJB)

  deg C,   deg F                                          -2.0 to 50 deg C range
Help: F1.  Quit: <esc>.  Configuration:ATCM0.  FILE: djbmanul.var;
```

```
Apr  3, 1992,  10:21: 2  ERL-N Model PLUMES, Feb 19, 1992       Case:   4 of
20
 Title   EXAMPLE 1, 40 CM/SEC

  tot flow   # ports port flow   spacing  salinity      temp   far inc    far dis
     4.381      100   0.04381        10         0         15        30        100
 plume dep  port dia plume dia total vel horiz vel vertl vel asp coeff print frq
     29.87    0.1524    0.1446     2.669     2.669     0.000      0.10        150
 port elev ver angle cont coef plume den poll conc     decay  Froude # Roberts
F
    0.6096       0.0       0.9   -0.8363       100 1.157E-08     14.13     0.8567
 hor angle eff spacg   amb den   current   far dif   far vel K:vel/cur Stratif
#
        90    10.000     24.28    0.4000  0.000453       0.4    6.6710.00003593
     depth   current   density  salinity      temp  amb conc  N (freq) red grav.
       0.0       0.4     24.10        32        13       1.6  0.007731     0.2466
     30.48       0.4     24.29        32        12       1.6 buoy flux puff-ther
                                                              0.01080     1.468
                                                            jet-plume  jet-cross
                                                                1.924     0.8548
                                                            plu-cross  jet-strat
                                                               0.1688     6.650
                                                              plu-strat

                                                                12.36

                                                             hor dis>=


CORMIX-1 (single port) flow: s4; attachment: a2.  Use EPA models (DJB)

  deg C,   deg F                                          -2.0 to 50 deg C range
Help: F1.  Quit: <esc>.  Configuration:ATCM0.  FILE: djbmanul.var;



Apr  3, 1992,  10:21: 6  ERL-N Model PLUMES, Feb 19, 1992       Case:   5 of
20
 Title   EXAMPLE 1, 80 CM/SEC

  tot flow   # ports port flow   spacing  salinity      temp   far inc    far dis
     4.381      100   0.04381        10         0         15        30        100
 plume dep  port dia plume dia total vel horiz vel vertl vel asp coeff print frq
     29.87    0.1524    0.1446     2.669     2.669     0.000      0.10        150
 port elev ver angle cont coef plume den poll conc     decay  Froude # Roberts
F
    0.6096       0.0       0.9   -0.8363       100 1.157E-08     14.13      6.853
 hor angle eff spacg   amb den   current   far dif   far vel K:vel/cur Stratif
#
        90    10.000     24.28    0.8000  0.000453       0.8    3.3360.00003593
     depth   current   density  salinity      temp  amb conc  N (freq) red grav.
       0.0       0.8     24.10        32        13       1.6  0.007731     0.2466
     30.48       0.8     24.29        32        12       1.6 buoy flux puff-ther
                                                              0.01080     1.165
                                                            jet-plume  jet-cross
```



```
                                                                1.924     0.4274
                                                            plu-cross  jet-strat
                                                              0.02110     6.650
                                                              plu-strat

                                                                12.36

                                                             hor dis>=


CORMIX-1 (single port) flow: s4; attachment: a2.  Use EPA models (DJB)

  deg C,   deg F                                          -2.0 to 50 deg C range
Help: F1.  Quit: <esc>.  Configuration:ATCM0.  FILE: djbmanul.var;
```

SELECTED INPUT FOR TABLE II

```
 Apr  3, 1992,  10:21:44  ERL-N Model PLUMES, Feb 19, 1992        Case:  17 of
20
 Title    EXAMPLE FOR EFFLUENT FLOW = 6 CM/S

  tot flow   # ports port flow   spacing  salinity      temp   far inc   far dis
         6       100   0.06000        10         0        15        30       100
 plume dep  port dia plume dia total vel horiz vel vertl vel asp coeff print frq
     29.87    0.1524    0.1446     3.655     3.655     0.000      0.10       150
 port elev ver angle cont coef plume den poll conc     decay  Froude # Roberts
F
    0.6096       0.0       0.9   -0.8363       100 1.157E-08     19.36  0.001222
 hor angle eff spacg   amb den   current   far dif   far vel K:vel/cur Stratif
#
        90    10.000     24.28   0.05000  0.000453      0.05     73.090.00003593
     depth   current   density  salinity      temp  amb conc  N (freq) red grav.
       0.0      0.05     24.10        32        13       1.6  0.007731    0.2466
     30.48      0.05     24.29        32        12       1.6 buoy flux puff-ther
                                                              0.01479     4.021
                                                           jet-plume  jet-cross
                                                                2.635     9.365
                                                           plu-cross  jet-strat
                                                                118.3     7.783
                                                             plu-strat

                                                                13.38

                                                             hor dis>=


CORMIX-1 (single port) flow: h3; no bottom attachment.  Use CORMIX (DJB)

  deg C,   deg F                                      -2.0 to 50 deg C range
Help: F1.   Quit: <esc>.  Configuration:ATCMO.   FILE: djbmanul.var;



 Apr  3, 1992,  10:21:54  ERL-N Model PLUMES, Feb 19, 1992        Case:  18 of
20
 Title    EXAMPLE FOR EFFLUENT FLOW = 8 CM/S

  tot flow   # ports port flow   spacing  salinity      temp   far inc   far dis
         8       100   0.08000        10         0        15        30       100
 plume dep  port dia plume dia total vel horiz vel vertl vel asp coeff print frq
     29.87    0.1524    0.1446     4.873     4.873     0.000      0.10       150
 port elev ver angle cont coef plume den poll conc     decay  Froude # Roberts
F
    0.6096       0.0       0.9   -0.8363       100 1.157E-08     25.81 0.0009163
 hor angle eff spacg   amb den   current   far dif   far vel K:vel/cur Stratif
#
        90    10.000     24.28   0.05000  0.000453      0.05     97.460.00003593
     depth   current   density  salinity      temp  amb conc  N (freq) red grav.
       0.0      0.05     24.10        32        13       1.6  0.007731    0.2466
```



```
     30.48      0.05     24.29        32        12       1.6 buoy flux puff-ther
                                                              0.01972     5.361
                                                           jet-plume  jet-cross
                                                                3.513     12.49
                                                           plu-cross  jet-strat
                                                                157.8     8.986
                                                             plu-strat

                                                                14.37

                                                             hor dis>=


CORMIX-1 (single port) flow: h3; attachment: a4.  Use CORMIX (DJB)

  deg C,   deg F                                      -2.0 to 50 deg C range
Help: F1.   Quit: <esc>.  Configuration:ATCMO.  FILE: djbmanul.var;
```

```
 Apr  3, 1992,   10:22:25  ERL-N Model PLUMES, Feb 19, 1992         Case:  19 of
20
 Title    EXAMPLE FOR EFFLUENT FLOW = 10 CM/S

  tot flow   # ports port flow   spacing  salinity      temp   far inc    far dis
        10       100    0.1000        10         0        15        30        100
 plume dep  port dia plume dia total vel horiz vel vertl vel asp coeff print frq
     29.87    0.1524    0.1446     6.091     6.091     0.000      0.10       150
 port elev ver angle cont coef plume den poll conc     decay  Froude # Roberts
F
    0.6096       0.0       0.9   -0.8363       100 1.157E-08     32.26 0.0007330
 hor angle eff spacg   amb den   current   far dif   far vel K:vel/cur Stratif
#
        90    10.000     24.28   0.05000  0.000453      0.05    121.80.00003593
     depth   current   density  salinity      temp  amb conc  N (freq) red grav.
       0.0      0.05     24.10        32        13       1.6  0.007731    0.2466
     30.48      0.05     24.29        32        12       1.6 buoy flux puff-ther
                                                               0.02466     6.702
                                                            jet-plume  jet-cross
                                                                4.391      15.61
                                                            plu-cross  jet-strat
                                                                197.2      10.05
                                                              plu-strat

                                                                15.20

                                                              hor dis>=


CORMIX-1 (single port) flow: h3; attachment: a4.  Use CORMIX (DJB)

  deg C,   deg F                                        -2.0 to 50 deg C range
Help: F1.   Quit: <esc>.  Configuration:ATCMO.   FILE: djbmanul.var;



 Apr  3, 1992,   10:22:29  ERL-N Model PLUMES, Feb 19, 1992         Case:  20 of
0
 Title    EXAMPLE FOR EFFLUENT FLOW = 12 CM/S

  tot flow   # ports port flow   spacing  salinity      temp   far inc    far dis
        12       100    0.1200        10         0        15        30        100
 plume dep  port dia plume dia total vel horiz vel vertl vel asp coeff print frq
     29.87    0.1524    0.1446     7.309     7.309     0.000      0.10       150
 port elev ver angle cont coef plume den poll conc     decay  Froude # Roberts
F
    0.6096       0.0       0.9   -0.8363       100 1.157E-08     38.71 0.0006108
 hor angle eff spacg   amb den   current   far dif   far vel K:vel/cur Stratif
#
        90    10.000     24.28   0.05000  0.000453      0.05    146.20.00003593
     depth   current   density  salinity      temp  amb conc  N (freq) red grav.
       0.0      0.05     24.10        32        13       1.6  0.007731    0.2466
     30.48      0.05     24.29        32        12       1.6 buoy flux puff-ther
                                                               0.02959     8.042
```



```
                                                            jet-plume  jet-cross
                                                                5.269      18.73
                                                            plu-cross  jet-strat
                                                                236.7      11.01
                                                              plu-strat

                                                                15.91

                                                              hor dis>=


CORMIX-1 (single port) flow: h3; attachment: a4.  Use CORMIX (DJB)

  deg C,   deg F                                        -2.0 to 50 deg C range
Help: F1.   Quit: <esc>.  Configuration:ATCMO.   FILE: djbmanul.var;
```

APPENDIX 3: THE DIFFUSER HYDRAULICS MODEL PLUMEHYD

MODEL DESCRIPTION

The model PLUMEHYD is based on the hydraulics model DPHYDR used by Tetra Tech in the early 1980's to help assess 301(h) applications (Gremse, 1980), and, based on a limited number of trials, gives approximately identical results. It is appropriate for use with multiport diffusers with bell shaped or sharp-edged ports. It also considers multi-segmented diffusers of varying diameter. The program uses metric (SI) units and works in batch mode.

A description of diffuser hydraulics is available in Grace (1978).

MODEL USAGE

At this time PLUMEHYD.exe works only in the batch mode, which means you must construct the input file in an ASCII editor, like the built-in Turbo Pascal editor. Sample input is shown in Figure 78.

```
 Honouliuli    diffuser
hydraulics
 74 4 0.0267
 bell
  1  1 1.22  7.315 0.0 0.215
  2 22 1.22  7.315 0.0 0.134
 23 47 1.677 7.325 0.0 0.129
```

Ilustr. 1. PLUMEHYD batch input file.

The first line of input is a title. It is followed by a line containing the number of ports, number of diffuser sections, and the ratio of the density difference between the ambient and effluent fluids to the effluent density, $(\rho_a - \rho_e)/\rho_e$. The individual values must be separated by blanks.

The third line should contain the words "bell" or "sharp", which is more informative than the original version input.

There follow a variable number of lines defined by the number of diffuser sections on the second line of input, in this case, 4. Each line, starting from the end of the diffuser, specifies the number of the first port in the section, the last port, the pipe diameter, the port spacing, the rise, and finally the port diameter. Note that in this case the diffuser has a large port at the end of the diffuser probably put there for flushing purposes.

The last line of input specifies the Mannings number and the total flow rate.

PLUMEHYD COMPUTER LISTINGS

Pascal Version of PLUMEHYD

```
{$r+}
{
 Program PLUMEHYD.pas
 Metric system (SI) units assumed
}
const
 g = 9.807;
 criterion = 1e-6;
type
 porttype = (bell,sharp);
 st80 = string[80];
var
 piped,dxpipe,dzpipe,ff,portd: array[1..20] of real;
 fin,fxn,title: st80;
 nf,nl: array[1..20] of integer;
 qq,ee: array[1..50] of real;
 e,cd,pipev,portfn,portv,q: array[1..400] of real;
  ab,al,a1,cdc,dr,dx,dz,error,eorg,e0,f,fnf,gprime,hlf,hlz,
  mann,pd,pid4,pod,qc,qorg,qsum,qt,q0,v,vnew,vorg,zman: real;
 i,iter,np,ns,ans: integer;
 ptype: porttype;
 fi,fx: text;
{
 dr = drho/rho
 dxpipe = horizontal length of the section
 dzpipe = vertical rise of the section
 mann = Manning's n
 nf = number of the first port in a given section
 nl = number of the last port in a given section
 np = number of ports
 ns = number of diffuser sections
 piped = pipe diameter of the section
 portd = port diameter
 ptype = port type, bell or sharp
 qt = total discharge
}
```

```
function pwr(a,b: real):real; var sign: integer;
{ an exponentiation function }
begin
if a < 0 then begin sign:=-1; a:=-a; end else sign:=1;
a:=exp(b*ln(a)); if sign = -1 then pwr:=-a else pwr:=a; end;

function strip(s:st80): st80;
{ strips blanks out of a string of characters }
begin while s[1] = ' ' do delete(s,1,1); strip:=s; end;

procedure cvnew(var enew,vold,cd,vnew: real);
{ sets up PLUMEHYD for analyzing diffusers with bell or sharp-edged ports }
var dv,f1,f2,v,v2: real;
begin
v:=0;
f1:=0.5/g/enew;
f2:=al/ab*sqrt(2*g*enew);
  if ptype = bell then begin
  v:=vold;
    repeat
    v:=vnew;
    v2:=sqr(v);
    cd:=0.975*pwr((1-v2*f1),0.375);
    vnew:=vold+cd*f2;
    dv:=v-vnew;
    v:=vnew;
    until abs(dv) - criterion < 0;
  end
  else
  begin { sharp }
  v:=vold;
    repeat
    v2:=sqr(v);
    cd:=0.63-0.58*v2*f1;
    vnew:=vold+cd*f2;
    dv:=v-vnew;
    v:=vnew;
    until abs(dv) - criterion < 0;
  end;
end;

procedure loop; var j,k,n1,n2: integer;
```



```
{ main program element }
begin
vorg:=0; eorg:=e0; k:=0; qsum:=0;
  for j:=1 to ns do begin
  pd:=piped[j];
  ab:=pid4*sqr(pd);
  dx:=dxpipe[j];
  dz:=dzpipe[j];
  f:=ff[j];
  pod:=portd[j];
  al:=pid4*sqr(pod);
  fnf:=1/al/sqrt(gprime*pod);
  n1:=nf[j];
  n2:=nl[j];
  hlz:=dz*dr;
  hlf:=f*dx/pd/2/g;
    for i:=n1 to n2 do begin
    cvnew(eorg,vorg,cdc,vnew);
    k:=k+1;
    e[k]:=eorg;
    qc:=(vnew-vorg)*ab;
    q[k]:=qc;
    cd[k]:=cdc;
    pipev[k]:=vnew;
    portv[k]:=qc/al;
    portfn[k]:=qc*fnf;
    eorg:=hlz+eorg+vnew*vnew*hlf;
    qorg:=qc;
    qsum:=qsum+qc;
    vorg:=vnew;
    end;
  {}if j-ns < 0 then begin
    v:=vorg*sqr(piped[j]/piped[j+1]);
    eorg:=eorg+0.7*sqr(v-vorg)/2/g;
    vorg:=v;
    end;
  end;
iter:=iter+1; ee[iter]:=e0; qq[iter]:=qt-qsum; end;

procedure input; var portst: st80;
begin
write('Input file (CR for default name of "HYD.IN": '); readln(fin);
```

```
if fin = '' then fin:='hyd.in';
assign(fi,fin); reset(fi);
write('Output file (CR for default name of "HYD.EX": '); readln(fxn);
if fxn = '' then fxn:='hyd.ex';
assign(fx,fxn); rewrite(fx);
readln(fi,title); readln(fi,np,ns,dr);
readln(fi,portst); portst:=strip(portst);
if upcase(portst[1]) = 'B' then ptype:=bell else ptype:=sharp;
  for i:= 1 to ns do
  readln(fi,nf[i],nl[i],piped[i],dxpipe[i],dzpipe[i],portd[i]);
{ write('Input Mannings n, q (m^3/sec)'); } readln(fi,mann,qt);
end;

procedure initialize;
{ initializes program variables }
begin
error:=0.001; pid4:=pi/4;
zman:=124.58*mann*mann;
for i:=1 to ns do ff[i]:=zman/pwr(piped[i],0.33333);
q0:=qt/np;
a1:=pid4*sqr(portd[1]);
eorg:=sqr(q0/a1)/2/g;
ee[1]:=eorg; e0:=eorg;
iter:=0; gprime:=dr*g; end;

procedure outputit; var j,k: integer; begin
writeln(fx,title); writeln(fx);
writeln(fx,'Number of ports      = ',np:4);
writeln(fx,'drho/rho             = ',dr:9:4);
writeln(fx,'Number of sections   = ',ns:4);
  if ptype = bell then writeln(fx,'bell')
  else
  writeln(fx,'sharp');
writeln(fx);
writeln(fx,'Mannings N           = ',mann:9:4);
writeln(fx,'Desired Q            = ',qt:9:4);
writeln(fx,'Calculated Q         = ',qc:9:4); writeln(fx);
  for k:= 1 to ns do begin
    writeln(fx,
    'Friction factor F    = ',ff[k]:9:4,' ':9,
    'Pipe diameter      =',piped[k]:9:4);
    writeln(fx,
```



```
    'Length between ports = ',dxpipe[k]:9:4,' ':9,
    'dz between ports   =',dzpipe[k]:9:4);
    writeln(fx,'Port diameter        = ',portd[k]:9:4);
  writeln(fx);
    writeln(fx,
    ' Port   Specific   Coeff    Pipe     Port     Port      Port');
    writeln(fx,
    'number   energy      cd    velocity velocity discharge  Froude #');
    writeln(fx,
    '           (m)               (m/sec)   (m/sec)  (m^3/sec)');
  writeln(fx);
    for j:=nf[k] to nl[k] do
    writeln(fx,j:6,e[j]:10:4,cd[j]:10:4,pipev[j]:10:4,
    portv[j]:10:4,q[j]:10:4,portfn[j]:10:4);
  writeln(fx); end;
end;

{ main program element }
begin
input; initialize;
  repeat
  loop;
    if iter = 1 then
    e0:=ee[1]*sqr(qt/qsum)
    else
    e0:=(ee[iter-1]*qq[iter]-ee[iter]*qq[iter-1])/(qq[iter]-qq[iter-1]);
  until abs(qq[iter]) < error;
qc:=qsum;
outputit;
close(fi); close(fx); end.
```

Sample Input File

```
Honouliuli diffuser hydraulics
74 4 0.0267
bell
 1  1 1.22   7.315 0.0 0.215
 2 22 1.22   7.315 0.0 0.134
23 47 1.677 7.325 0.0 0.129
48 74 1.982 7.315 0.0 0.123
0.014 0.1818
```

Sample Output File

```
Honouliuli diffuser hydraulics

Number of ports      =   74
drho/rho             =    0.0267
Number of sections   =    4
bell

Mannings N           =    0.0140
Desired Q            =    0.1818
Calculated Q         =    0.1818

Friction factor F    =    0.0229        Pipe diameter     =   1.2200
Length between ports =    7.3150        dz between ports  =   0.0000
Port diameter        =    0.2150
```

Port number	Specific energy (m)	Coeff cd	Pipe velocity (m/sec)	Port velocity (m/sec)	Port discharge (m^3/sec)	Port Froude #
1	0.0017	0.9747	0.0055	0.1763	0.0064	0.7429

```
Friction factor F    =    0.0229        Pipe diameter     =   1.2200
Length between ports =    7.3150        dz between ports  =   0.0000
Port diameter        =    0.1340
```

Port number	Specific energy (m)	Coeff cd	Pipe velocity (m/sec)	Port velocity (m/sec)	Port discharge (m^3/sec)	Port Froude #
2	0.0017	0.9744	0.0076	0.1762	0.0025	0.9408
3	0.0017	0.9739	0.0097	0.1762	0.0025	0.9405
4	0.0017	0.9734	0.0119	0.1761	0.0025	0.9402
5	0.0017	0.9728	0.0140	0.1761	0.0025	0.9399
6	0.0017	0.9721	0.0161	0.1760	0.0025	0.9396
7	0.0017	0.9713	0.0182	0.1759	0.0025	0.9393
8	0.0017	0.9704	0.0203	0.1759	0.0025	0.9391
9	0.0017	0.9694	0.0225	0.1759	0.0025	0.9389
10	0.0017	0.9683	0.0246	0.1759	0.0025	0.9388
11	0.0017	0.9671	0.0267	0.1759	0.0025	0.9388
12	0.0017	0.9658	0.0288	0.1759	0.0025	0.9389
13	0.0017	0.9644	0.0310	0.1759	0.0025	0.9392
14	0.0017	0.9629	0.0331	0.1760	0.0025	0.9396
15	0.0017	0.9613	0.0352	0.1761	0.0025	0.9402
16	0.0017	0.9597	0.0373	0.1763	0.0025	0.9410
17	0.0017	0.9580	0.0395	0.1764	0.0025	0.9419
18	0.0017	0.9562	0.0416	0.1767	0.0025	0.9431
19	0.0018	0.9543	0.0437	0.1769	0.0025	0.9445
20	0.0018	0.9524	0.0459	0.1772	0.0025	0.9462
21	0.0018	0.9504	0.0480	0.1776	0.0025	0.9481
22	0.0018	0.9483	0.0501	0.1780	0.0025	0.9503

```
Friction factor F    =    0.0206        Pipe diameter     =   1.6770
Length between ports =    7.3250        dz between ports  =   0.0000
Port diameter        =    0.1290
```


Port number	Specific energy (m)	Coeff cd	Pipe velocity (m/sec)	Port velocity (m/sec)	Port discharge (m^3/sec)	Port Froude #
23	0.0018	0.9672	0.0276	0.1834	0.0024	0.9981
24	0.0018	0.9666	0.0287	0.1835	0.0024	0.9984
25	0.0018	0.9659	0.0298	0.1836	0.0024	0.9988
26	0.0018	0.9653	0.0309	0.1836	0.0024	0.9992
27	0.0018	0.9646	0.0320	0.1837	0.0024	0.9997
28	0.0019	0.9639	0.0331	0.1838	0.0024	1.0002
29	0.0019	0.9632	0.0341	0.1839	0.0024	1.0008
30	0.0019	0.9625	0.0352	0.1841	0.0024	1.0015
31	0.0019	0.9617	0.0363	0.1842	0.0024	1.0022
32	0.0019	0.9609	0.0374	0.1843	0.0024	1.0030
33	0.0019	0.9601	0.0385	0.1845	0.0024	1.0039
34	0.0019	0.9593	0.0396	0.1847	0.0024	1.0049
35	0.0019	0.9585	0.0407	0.1849	0.0024	1.0059
36	0.0019	0.9576	0.0418	0.1851	0.0024	1.0070
37	0.0019	0.9568	0.0429	0.1853	0.0024	1.0082
38	0.0019	0.9559	0.0440	0.1855	0.0024	1.0095
39	0.0019	0.9550	0.0451	0.1858	0.0024	1.0109
40	0.0019	0.9541	0.0462	0.1861	0.0024	1.0124
41	0.0019	0.9532	0.0473	0.1864	0.0024	1.0140
42	0.0020	0.9523	0.0484	0.1867	0.0024	1.0156
43	0.0020	0.9513	0.0495	0.1870	0.0024	1.0174
44	0.0020	0.9504	0.0506	0.1873	0.0024	1.0193
45	0.0020	0.9494	0.0517	0.1877	0.0025	1.0213
46	0.0020	0.9484	0.0528	0.1881	0.0025	1.0233
47	0.0020	0.9475	0.0539	0.1885	0.0025	1.0255

```
Friction factor F    =    0.0194        Pipe diameter     =   1.9820
Length between ports =    7.3150        dz between ports  =   0.0000
Port diameter        =    0.1230
```

Port number	Specific energy (m)	Coeff cd	Pipe velocity (m/sec)	Port velocity (m/sec)	Port discharge (m^3/sec)	Port Froude #
48	0.0020	0.9607	0.0394	0.1921	0.0023	1.0706
49	0.0020	0.9602	0.0401	0.1923	0.0023	1.0715
50	0.0021	0.9596	0.0408	0.1925	0.0023	1.0725
...						
59	0.0021	0.9547	0.0475	0.1944	0.0023	1.0833
60	0.0021	0.9541	0.0483	0.1947	0.0023	1.0848
61	0.0021	0.9536	0.0491	0.1950	0.0023	1.0863
62	0.0021	0.9530	0.0498	0.1952	0.0023	1.0879
63	0.0021	0.9524	0.0506	0.1955	0.0023	1.0895
64	0.0022	0.9518	0.0513	0.1958	0.0023	1.0912
65	0.0022	0.9512	0.0521	0.1961	0.0023	1.0929
66	0.0022	0.9506	0.0528	0.1965	0.0023	1.0948
67	0.0022	0.9500	0.0536	0.1968	0.0023	1.0966
68	0.0022	0.9494	0.0543	0.1972	0.0023	1.0986
69	0.0022	0.9488	0.0551	0.1975	0.0023	1.1005
70	0.0022	0.9482	0.0559	0.1979	0.0024	1.1026
71	0.0022	0.9476	0.0566	0.1983	0.0024	1.1047
72	0.0022	0.9470	0.0574	0.1986	0.0024	1.1069
73	0.0023	0.9464	0.0582	0.1991	0.0024	1.1091
74	0.0023	0.9457	0.0589	0.1995	0.0024	1.1115

APPENDIX 4: PLUMES WINDOW MESSAGES AND INTERPRETATIONS

CORMIX WINDOW RECOMMENDATIONS

For a long time work culminating in this manuscript and corresponding software and the EPA sponsored work on CORMIX proceeded independently. Since about 1990, efforts have been made to integrate the two approaches to take advantage of their complementary capabilities, as explained in Appendix 1. For example, a CORMIX work element exists to in some way include the traditional EPA models within its framework. The CORMIX window, currently restricted to CORMIX1, is our attempt to do the same.

Providing there are no limitations on its use as described in Tables V and attending text, CORMIX1 is considered to be an appropriate solution to the plume problem under consideration in the PLUMES interface. It is assumed that the Configuration menu has been used to turn the CORMIX1 algorithm on.

Note, since RSB is exclusively designed for merging plumes, only CORMIX1 and UM are applicable to this discussion. Also, in all questionable cases, at least a few runs using CORMIX1 are recommended for the sake of comparison and mutual validation.

Single: use CORMIX1; merging: UM ok

Displayed in cases in which PLUMES predicts flow categories v4 and v6: The use of CORMIX is definitely recommended for single plumes, but only in cases in which nascent density effects are absent and other weak links in the CORMIX chain (see Appendix 1) do not exist. Excluded cases must be handled on a case-by-case basis.

To the extent that the CORMIX1 flow classification scheme can also be used in a limited way to estimate CORMIX2 (merging plume) classification schemes, the use of UM and RSB is appropriate for merging plumes. Mutual validation with CORMIX and the use of the more conservative analysis is recommended in questionable cases.

Use CORMIX

Displayed in cases in which PLUMES predicts flow categories h4-90, h5-90, nv5, nh3: The use of CORMIX1 is definitely recommended, but only in cases in which nascent density effects are absent and other weak links in the CORMIX chain (see Appendix 1) do not exist. Excluded cases must be handled on a case-by-case basis.

Use CORMIX or UM to surface hit

Displayed in cases in which PLUMES predicts flow categories nv3, nv4, and nh5: It is appropriate to continue the analysis with UM until the surface is hit. The use of CORMIX is appropriate and possibly preferred, but only in cases in which nascent density effects are absent and other weak links in the CORMIX chain (see Appendix 1)

do not exist. Mutual validation with CORMIX and the use of the more conservative analysis is recommended in questionable cases.

Use UM

Displayed in cases in which PLUMES predicts no CORMIX1 category or flow categories s1, s3, s4: It is appropriate to continue the analysis with UM. The use of CORMIX is appropriate, but only in cases in which nascent density effects are absent and other weak links in the CORMIX chain (see Appendix 1) do not exist.

Use UM to bottom hit

Displayed in cases in which PLUMES predicts flow categories nv1, nv2, nh1, nh2, and nh4: It is appropriate to continue the analysis with UM until the bottom is hit. The use of CORMIX is appropriate, but only in cases in which nascent density effects are absent and other weak links in the CORMIX chain (see Appendix 1) do not exist.

Because two of the entrainment terms are disabled after plume vertical directional reversal, the UM analysis is thought to be conservative. Mutual validation with CORMIX and the use of the more conservative analysis is recommended.

Use UM to overlap point

Displayed in cases in which PLUMES predicts flow categories s2, s5, h4-180, h5-180: UM is considered appropriate to the point of overlap, with the farfield model being initiated at that point. The use of CORMIX is appropriate, but only in cases in which nascent density effects are absent and other weak links in the CORMIX chain (see Appendix 1) do not exist.

Use UM until near surface

Displayed in cases in which PLUMES predicts flow categories v3, v5, h3, h40: UM is weaker and CORMIX is correspondingly stronger in these categories. The ten percent prohibition suggested by Muellenhoff et al. (1985) may be appropriate and can be implemented using the Pause criterion in the Farfield configuration of PLUMES. The use of CORMIX is appropriate, but only in cases in which nascent density effects are absent and other weak links in the CORMIX chain (see Appendix 1) do not exist.

Use UM until surface hit

Displayed in cases in which PLUMES predicts flow categories v1, v2, h1, h2, and h5-0: UM is considered appropriate to the point of the surface being hit, with the farfield model being initiated at that point. The use of CORMIX is appropriate, but only in cases in which nascent density effects are absent and other weak links in the CORMIX chain (see Appendix 1) do not exist.

DIALOGUE WINDOW MESSAGES

The following messages are more or less frequently displayed by the PLUMES interface. The are listed here in alphabetical order. A few of them begin with terms, denoted here by square brackets [], which depend on the context of the message, are listed first. The rest follow in alphabetically order.

[message] at [variable]?. BEWARE of inconsistencies! <space>
Appears when a data inconsistency is detected. This can be automatic or happen when the <cHeck consistency> command is used. While efforts should be made to resolve inconsistencies, they do not always indicate incompatible input data.

["new" case file name] exists, file must be new
Appears after issuing the <make New file> command if the specified file name already exists. You are asked to provide another case file name. An empty name, i.e. a simple carriage return, may be used to cancel the command at this point.

[string] not a number, correction attempted.
You tried to input non-numerical information in a numerical cell. PLUMES removes the non-numeric characters from the input data and tries to convert the remaining string to numeric data. Other conditions, such as multiple decimal points, will also cause this message to be issued. The value should be checked and corrected if necessary.

A descriptive title.
Used to describe the title cell.

At [variable] Change sign or <key> to accept [default]
This message usually indicates that PLUMES is trying to define the identified cell from an equation involving a square root for which both positive and negative roots are valid. You have to make the appropriate choice.

Back, Inequalities, Output, Variables(space), or <esc>
Used to manipulate data in the Pause cell.

Bad file name, old or default file restored
Indicates a non-existant case file, such as PLMSTUFF.VAR, was specified for opening. Usually this happens when you have forgotten the name of the case files and inadvertantly specify a non-existing file name. Exit to DOS and use the DIR command to refresh yourself on the appropriate names.

Default table, or New table?
Asks you whether to include the default output variables when running UM or to clear

the table (New) for the addition of variables of your choosing using the <add to Output> command.

Discharge in Middle or Surface/bottom of water column?
Appears when the <shallow/surface Z> command is used. You must choose <M> or <S> (or <esc>) to specify your choice, which establishes the proper spacing for the reflection surfaces and other parameters.

Error detected in case range
Appears after invoking the <Beget new cases> (Miscellany Menu) to indicate that an error in specifying the number of cases to which to copy the current cell has been made.

From this case on, run how many cases?
Appears after issuing the ^B (RSB) or ^U (UM) commands. You specify how many cases, starting at the case number shown at the upper right corner of the screen, to run. The default is always 1 which may be selected by simply pressing the spacebar.

Go to case (<space> for default): [default case number]
Used to specify how many cases to translate into Universal Data File (UDF) format necessary to run the 1985 plume models (Muellenhoff et al., 1985). All cases in the range from the present case to the specified case will be translated.

Go to case (<space> to accept default): [default case number]
Appears after issuing the ^C command to ask you the case number to which to move. The next case is always the default value and may be selected by pressing the spacebar. A value greater than the number of cases currently in the file causes a new case to be appended to the end of the file and movement to that case (the data in the present case are automatically loaded into the new case).

Hit hilited letter or arrow keys and <CR>; use control sequences for speed
Issued when accessing the main menu to remind you that the control key sequence for issuing commands is faster than using the menus.

Inconsistency at [variable name 1]: [value 1] vs. [variable name 2]: [value 2]
These messages may appear when using the <cHeck consistency> command if tolerances are not met. In other words, if two different equations of the same dependent variable yield values which differ by more than 1 part per thousand, then this message is issued.

Input file name:
Requests you to enter the name of the case file, i.e. the non-ASCII file used to store the input screen data, such as PLMSTUFF.VAR. These files cannot be edited by an ASCII

editor.

Input starting longitudinal coordinate:
When the Brooks equation width input toggle in the Configuration String is set to "user", PLUMES prompts for the initial width of the wastefield and the initial starting distance, thus allowing for the override of these two parameters. This allows runs of the Brooks equation which are essentially independent of the initial dilution estimates.

Input wastefield width:
See related message, "Input starting longitudinal coordinate:", above.

<key> for far field prediction
RSB output is displayed on two screens, the near field output and the far field output.

<key> once again to start PLUMES
While using the <shallow/surface Z> command, some condition needing your attention in the initialization phase has been identified. Make tot flow, spacing, plume dep, port dia, port eleve cells independent, and, a non-surface independent ambient depth cell must be defined, which must satisfy: ambient depth >= plume dep. A message appears on three separate dialogue windows when some or all cell values needed to complete the <shallow/surface Z> command are missing.

No changes made
Appears if a choice other than Middle or Surface/bottom, i.e. no choice, is made after issuing the <shallow/surface Z> command.

No direct independents to hilite for [variable], remove others.
Issued when the problem is overspecified and a confict arises. This happens when a dependent (white) value is replaced by an independent (yellow) value but no immediate independent values for the cell can be identified, i.e. the cell is totally defined by other dependent (white) values. YOU SHOULD IMMEDIATELY REMOVE THE LAST VALUE YOU INPUT OR FIND OTHER INDEPENDENT VALUES TO REMOVE. USE THE <cHeck consistency> COMMAND TO ASSURE CONSISTENCY.

NO GO, incomplete effluent/ambient blocs.
Advises you that the data necessary for running UM are not complete. Return to the input screen and check for missing cells.

Only for adding hidden variables to the table. <key>
Back, Inequalities, Output, Variables(space), <esc>
Used to manipulate data in the Pause cell. Moves the pause cell pointer through the list of available variables.

Out of range?
Indicates a bad case number was issued under the ^C command.

Plumes not merged, Brooks method may be invalid.
The Brooks equations are based on a continuous wastefield, an assumption which is not valid when the plumes are not merged. However, the equations are probably valid if the unmerged distance is small.

Probable corrupted data file, check SETUP, and files.
SETUP should be deleted; program to terminate!
An error has been identified in the case file. Possibly you asked for a file that is not in the binary case file format, you have moved your files to some new directory and PLUMES is unable to find the files, or some other terminal condition exists. Check the SETUP file for clues, delete it, and start over (or shift attention to other case files).

Quit, all others to continue
Message appears when execution of UM has been interrupted. <Q> will cause the current run to be abandoned.

Replicate this cell to case (<space> to accept default):
A value in a particular cell in a particular case may be copied to the corresponding cell in a specified number of additional cases starting with the next case.

See guidance material for explanation
Appears when the Miscellany Menu is accessed. Guidance may be found in the section entitled "User's guide to the model interface, PLUMES" in the manual.

See users' guide for details
Appears when the Configuration Menu is accessed.

Specify max reversals; 0: PLUMES chooses (see manual: configuration):
You are asked how many vertical velocity reversals UM should use before giving control over to the far field model. Reversals occur in stable ambient at the top of rise or when the plume sinks to a maximum depth (fall). If the trajectory is plotted out, these points are the crests and troughs of the resulting waveform.

Start far-field at Max-rise, Overlap, or Pause criterion?
Issued when invoking <Farfield start> on the Configuration Menu for control of the UM model. You are to specify at which point the initial dilution model should end and the far field model begin. The overlap condition is recommended.

Sure you want to zap variables? (y/n):

Reminder after issuing the <Zap most variables> command on the Miscellany Menu, that all variables except the aspiration coefficient, output frequency, decay, far field dispersion coefficient, and surface ambient depth cell will be blanked out.

Temperature A) [temperature 1] or <key> [temperature 2]?

This message appears when temperature is the dependent variable (defined by density and salinity). In this case an approximation technique is used to solve the density function for temperature. This choice is presented when two solutions, starting at different initial guesses, of the problem converge on separate values.

To use command, number of ports must = 1

Reminder that the <shallow/surface Z> command can only be used for single port outfalls.

To use Fill ambient cells: <key> for usage:
cursor must be on a full cell below cells to be interpolated

Instructs you how to fill embedded empty cells in the ambient block. You must move the cursor to a filled cell below the embedded empty cells. The corresponding surface cell must also contain a value. The cells in between will be interpolated on the values of the depths in the depth column.

UM running, key to interrupt

Please wait message. UM can be interrupted and stopped at anytime.

Use control key sequences or see the Guide for better movement and control

Appears when the Movement menu is accessed, reminds you that better movement controls are available by consulting the manual.

With regard to [variable name] resolve conflicts:

Issued when the problem is overspecified and a confict arises. This happens when a dependent variable is replaced by an independent variable, i.e. one you input. You are forced to move between the highlighted cells until you delete one of them, by pressing <D> or the <delete> on the flashing (chosen) cell.

Write to ("prn" for printer, "console", or disk file name): [default name]

Appears after specifying the number of cases to run after issuing the ^B or ^U commands (see "From this case on..."). You are asked to specify the output device which can be the printer (type in the letters prn), monitor (type in console), or disk file (any legal DOS file name). The spacebar may be used to accept the default value.

xx = current variable, x2 = 1st argument in PRECEDING ns.

Provides definitions of xx and x2.

9. PROGRAM UDKHDEN

9. PROGRAM UDKHDEN

PROGRAM UDKHDEN - MAY 1994 VERSION 2.7

THIS IS AN UPDATED VERSION OF THE DKHPLM PROGRAM DEVELOPED BY DR. LORIN R. DAVIS FROM THE ORIGINAL SINGLE PORT MODEL DEVELOPED BY ERIC HIRST. THE MODEL WAS TUNED TO EXPERIMENTAL DATA BY DR. LANDIS KANNBERG.

IT CALCULATES THE CHARACTERISTICS OF A LINE OF EQUALLY SPACED BUOYANT DISCHARGES INTO A FLOWING STRATIFIED AMBIENT THE METHOD OF SOLUTION INVOLVES 7 ORDINARY DIFFERENTIAL EQUATIONS WHICH ARE 1. CONSERVATION OF MASS 2. CONSERVATIONS OF ENERGY 3. CONSERVATION OF CONCENTRATION 4. DENSITY DEFICIENCY 5.6.7. MOMENTUM EQNS IN THE Z (AXIAL), K (VERTICAL) J (HORIZONTAL AND PARALLEL TO AMBIENT CURRENT DIRECTION) EQN 4 IS USED ONLY IN ALGEBRAIC FORM. THE SIX OTHER EQUATIONS ARE WRITTEN FOR A CONTROL VOLUME WHICH IS FINITE IN A DIRECTION PERPENDICULAR TO THE JET AND INFINITESIMAL IN THE DIRECTION OF THE JET AXIS. OUTPUT IN THE ZONE OF FLOW ESTABLISHMENT HAS BEEN SUPPRESSED.

A SINGLE PORT DISCHARGE CAN BE MODELED BY SPACING THE PORTS WIDE APART SO MERGING DOES NOT OCCUR.

THIS VERSION IS INTERACTIVE WITH THE TERMINAL ALLOWING FOR MULTIPLE RUNS TO BE RUN WHILE ONLY CHANGING ONE VARIABLE. THIS IS MADE POSSIBLE BY SETTING THE FIRST OPTION VARIABLE ON THE SECOND LINE OF INPUT TO A NUMBER DIFFERENT FROM ZERO

THE INPUT TO UDKHDEN IS IN THE FORMAT OF A "UNIVERSAL DATA FILE" THAT IS GENERATED BY THE EPA "PLUMES" INFERFACE VERSION 3
TWO COPIES OF THE PROGRAM ARE PROVIDED. IF ADDITIONAL DISKS ARE REQUIRED, CONTACT DR. LORIN R. DAVIS,
440 NW. 34th, CORVALLIS, OREGON 97330 , (503) 737-7017

TO RUN THE PROGRAM, GENERATE AN INPUT FILE FOR ONE OR MORE RUNS AS OUTLINED BELOW AND STORE IT ON A DISK WITH PLENTY OF FREE SPACE.
TYPE UDKH2_7 (OR UDKH2_6) <RETURN>. THE SYSTEM WITH RESPOND WITH "ENTER FILENAME FOR INPUT>". TYPE X:FILENAME.EXT <RETURN> WHERE FILENAME.EXT IS THE NAME WITH EXTENSION OF YOUR INPUT FILE . WHERE X IS THE DRIVE WHERE YOUR FILE IS SAVED. THE SYSTEM WILL RESPOND WITH "ENTER FILENAME FOR OUTPUT>". IF YOU WISH THE OUTPUT TO BE DIRECTED TO THE TERMINAL SCREEN, TYPE CON.

IF YOU WANT TO SAVE THE OUTPUT ON A DISK FILE, TYPE X:FILENAME2.EXT WHERE FILENAME2.EXT IS THE NAME AND EXTENSION YOU WISH YOUR OUTPUT FILE TO HAVE. FILENAME2 MUST BE A NEW FILE AND

CAN NOT ALREADY EXIST ON THE DISK. IF IT DOES, YOU WILL GET AN ERROR MESSAGE.

***** INPUT REQUIREMENTS *****

INPUT IS GENERATED BY USING THE EPA "PLUMES" INTERFACE VERSION 3 AND CREATING A udf.in FILE. THIS IS DONE IN "PLUMES" BY PRESSING ^Y (CONTROL Y) AND SELECTING THE u OPTION, CREATE UDF FILE. THEN ANSWER WRITE WHEN YOU GET A PROMPT. NOTE THAT PLUMES APPENDS THE NEW CASE TO ANY EXISTING CASES ALREADY IN THE udf.in FILE. THIS ALLOWS YOU TO STACK SEVERAL RUNS BACK TO BACK AND RUN THEM ALL AT ONCE WITH udkh. OTHERWISE YOU MUST DELETE ALL PREVIOUS CASES BEFORE RUNNING. I USUALLY RENAME MY udf FILE EACH TIME TO THE SITE CASE BEING STUDIED SO THAT THE udf FILE ONLY CONTAINS THE LAST CASE.

YOU CAN MAKE MULTIPLE RUNS USING THE SAME AMBIENT INPUT TABLE BY CHANGING THE FIRST OPTION FLAG ON LINE TWO OF THE INPUT TO A NUMBER DIFFERENT FROM ZERO. IF YOU DO, THE PROGRAM WILL ASK YOU AT THE END OF A GIVEN CASE IF YOU WANT TO CHANGE ANY OF THE DISCHARGE VARIABLES. YOU CAN SELECT WHICH ONE YOU WANT TO CHANGE, CHANGE IT, AND GIVE THE CASE A NEW TITLE, THEN RE-RUN.

A SUBROUTINE CHECKS LIMITS OF PORT DEPTH (GREATER THAN 0. AND LESS THAN OR EQUAL TO PROFILE DEPTH), DISCHARGE ANGLE (EQUAL TO OR GREATER THAN -5 BUT LESS THAN OR EQUAL TO 130 DEG (COMPUTATIONAL PROBLEMS WOULD OCCUR WITH LARGE DISCHARGE ANGLES, I.E. SQUARE ROOT OF A NEGATIVE NUMBER, AND PROGRAM WOULD EXIT) .
ANY CURRENT ANGLE IN THE RANGE OF 0 TO 180 IS OK, BUT FOR ANGLES GREATER THAN 90 DEG THE PROGRAM SETS THE CURRENT ANGLE (HANG) EQUAL TO THE SUPPLEMENTARY ANGLE. FOR MULTI PORT DIFFUSERS HANG, BETWEEN 45 AND 135 IS RECOMMENDED FOR ACCURACY OF PREDICTIONS.

IT CHECKS TO SEE THAT THE EFFLUENT DENSITY IS EQUAL TO OR LESS THAN THE AMBIENT DENSITY AND THAT THERE ARE AMBIENT PROFILE VALUES FOR THE SURFACE, I.E. DP(1)=0. IF INTER=1, THE PORT DEPTH, ANGLES AND EFFLUENT DENSITY CAN BE CORRECTED INTERACTIVELY BUT SURFACE DATA CORRECTION MUST BE MADE TO THE DATA SET(S) AND REENTERED. IF INTER=0, ALL CORRECTION MUST BE MADE TO THE DATA SET(S) AND REENTERED.

****

***** OUTPUT FORMAT ******

THERE ARE TWO VERSIONS OF OUTPUT. UDKH2_7 HAS OUTPUT FORMATTED SO IT WILL FIT ON A 8-1/2 INCH PAGE BUT IT DOES NOT PRINT OUT THE LOCAL FLOW ANGLES AND EXPOSURE TIME. UDKH2_6 GIVES COMPLETE OUTPUT AS DESCRIBED BELOW BUT IT WILL NOT FIT ON A 8-1/2 PAGE UNLESS A SMALL FONT IS USED. I USUALLY USE A COURIER 9 PT FONT OR COMPRESSED MODE DEPENDING ON THE PRINTER.

THE PROGRAM ECHOS THE INPUT VARIABLES THEN CALCULATES THE DISCHARGE FROUDE NUMBER AND PORT TO DIAMETER SPACING. IT THEN GOES INTO A SOPHISTICATED ROUTINE THAT CALCULATES THE CHARACTERISTICS OF THE PLUME IN THE ZONE OF FLOW ESTABLISHMENT NEAR THE SOURCE AND PRINTS OUT THE LENGTH OF THE DEVELOPMENT ZONE OR STARTING LENGTH. THIS USUALLY TAKES SOME TIME DEPENDING ON THE COMPUTER. THEN FOR EACH INTEGRATION STEP IT PRINTS OUT
THE FOLLOWING:

X - CROSS CURRENT DISTANCE
Y - DOWN CURRENT DISTANCE
Z - VERTICAL DISTANCE FROM SOURCE

THESE THREE COORDINATES GIVE THE THREE DIMENSIONAL TRAJECTORY OF THE CENTERLINE OF THE REFERENCE PLUME

TH1 - IS THE LOCAL PLUME FLOW DIRECTION RELATIVE TO THE CURRENT WITH 90 DEGREES BEING IN THE DIRECTION OF THE CURRENT.

TH2 - IS THE LOCAL PLUME FLOW DIRECTION RELATIVE TO THE HORIZONTAL WITH 90 DEGREES BEING VERTICAL.

WIDTH OR DIAM- IS THE DIAMETER OF A SINGLE PLUME OR SIDE WIDTH OF MERGED PLUMES (NOT LENGTH SINCE THAT DEPENDS ON THE LENGTH OF THE DIFFUSER)

DUCL, DRHO, DCCL, AND DTCL ALL REFER TO THE RATIO OF THE LOCAL DIFFERENCE BETWEEN THE LOCAL CENTERLINE VALUE AND AMBIENT VALUE TO THAT SAME DIFFERENCE AT DISCHARGE FOR VELOCITY, DENSITY, CONCENTRATION, AND TEMPERATURE RESPECTIVELY. FOR EXAMPLE DUCL = (Ucl - Ua) AT ELEVATION Z DIVIDED BY (Ujet - Ua) AT DISCHARGE ELEVATION WHERE U IS VELOCITY, cl IS CENTERLINE, a IS AMBIENT, AND jet IS THE DISCHARGE VALUE.

TIME - IS THE TIME OF TRAVEL FROM THE DISCHARGE TO THIS POINT.

DILUTION - IS THE LOCAL AVERAGE PLUME DILUTION, 1.0 BEING NO

DILUTION AT ALL.

THE PROGRAM PRINTS OUT WHEN THE PLUMES JUST BEGIN TO MERGE, WHEN AND IF THE PLUME REACHES AN EQUILIBRIUM HEIGHT IN A STRATIFIED AMBIENT (DRHO BECOMES NEGATIVE AT THIS POINT), AND IF THE PLUME REACHES THE SURFACE. THE PROGRAM STOPS ONCE THE PLUME REACHES THE SURFACE SINCE THE MODEL IS NOT GOOD BEYOND THAT POINT.

IF THE PLUME REACHED A TRAPPING LEVEL, IT IS PRINTED OUT ALONG WITH THE DILUTION AT THAT POINT.

IF YOU INDICATED IN THE INPUT FILE THAT YOU WANTED AN INTERACTIVE RUN, THE PROGRAM WILL PROMPT YOU FOR NEW INPUT VALUES AND RUN TITLES. IF NOT, THE PROGRAM WILL GO TO THE NEXT DATA SET IN THE INPUT FILE IF THERE IS ONE.

10. AN ANALYSIS OF DEEP SUBMERGED MULTIPLE-PORT BUOYANT DISCHARGES

10. AN ANALYSIS OF DEEP SUBMERGED MULTIPLE-PORT BUOYANT DISCHARGES

L. D. Kannberg
Battelle—Northwest,
Richland, Wash.
Assoc. Mem. ASME

L. R. Davis
Oregon State University,
Corvallis, Ore.
Mem. ASME

An Analysis of Deep Submerged Multiple-Port Buoyant Discharges

The results of an experimental study of deep submerged multiple-port thermal discharges are compared to the predictions of a theory treating the dilution of merging multiple-port buoyant jets discharge from a row of equally spaced ports. The paper summarizes the considerable alteration of the Hirst [11] model necessary to adequately treat merging multiple jets. The essential features of the analysis are: (1) the gradual transition of the profiles from simple axisymmetric profiles to merging profiles and finally to fully merged, pseudo-slot, two-dimensional profiles, and (2) an entrainment based on the available entrainment surface. Results indicate that the overprediction of plume characteristics associated with certain other models as compared to experimental data may be overcome using such an analysis and that suitable prediction may be obtained.

Introduction

It is predicted that the future energy needs of this country will be met primarily by nuclear- and fossil-fueled electric generating stations. Since these plants are for the most part 35–40 percent thermodynamically efficient, a considerable amount of the energy will be released as waste heat. With this magnitude of waste heat, the effects of large thermal discharges to the environment become a legitimate concern. Regulating agencies have developed specifications concerning ambient temperature and dilution requirements for the regions near the waste-heat discharges. It is, therefore, necessary for the utility (or other appropriate industries) to design and construct discharge systems which meet dilution requirements and are cost effective. One design which has been advanced to meet these goals is the multiport diffuser. This paper presents a multiport discharge model recently advanced and compares the predictions of this method to those of other models and to experimental data.

The plumes from submerged multiport discharges can be divided into distinct flow regions characterized by their velocity and temperature profiles. Near the source is the zone of flow establishment where the "top hat" profiles of each discharge change to gaussian-shaped profiles. In the zone of established flow, these profiles remain similar, changing only in magnitude. In the merging zone, these single plumes slowly merge into one another forming a long pseudo-slot plume. This merging region can occur anywhere along the trajectory of the plume, depending upon ambient conditions and the initial port spacing. If the plumes reach the free surface, there is an additional surface transition zone where they change to drifting surface plumes. Since it was desired to investigate the effects of merging, in this paper the plume is considered unconfined and attention is focused on the zone of established flow and the merging zone.

Historical Background

Most of the analysis performed to date has dealt with either the single round jet or the single slot jet, both of which were successfully treated by two-dimensional integral analyses with the assistance of similarity assumptions. Only recently have attempts been made to treat discharges in which similarity is not possible using numerical methods (Trent and Welty [1],[1] Shetz [2]).

Efforts to extend beyond the single port outfall or the slot jet have not been directed toward modeling the merging process of several jets but rather to adapting the solutions of the single round jet and the slot jet to give predictions for the multiple jet case. The two notable works in this regard are those of Koh and Fan [3] and Jirka and Harleman [4]. Koh and Fan simulated the multiple buoyant discharges as single jets until certain merging criteria were met and then shifted simulation to the slot analysis. Transition occurred when the entrainments for both cases were equal or when the width of the round jets equaled the spacing between the jets. Jirka and Harleman suggested that the multiple round jet discharges could be treated as a slot jet provided certain adjustments were made in the parameters describing the plume. For their "equivalent slot" method, the same discharge per unit diffuser length and the same momentum flux per unit length as the multiport discharge is required. This results in a theoretical slot of width, $B = D^2\pi/4L$ where D and L are the actual port diameter and spacing between ports.

Experimental studies have been performed by Jirka and Harleman [4], Koh, et al. [6], Argue [7], Liseth [8], Iwasa and Yatsuzuka [9], and Kannberg and Davis [5]. With the exception of Kannberg and Davis [5] and Kannberg [10], all of these studies involved discharges that were either into confined ambients or from diffusers that did not constitute a single row of discharge ports. Thus, little work has been done on the effects of merging on dilution and trajectory.

Contributed by the Heat Transfer Division and presented at the Winter Annual Meeting, New York, N. Y., December 5–10, 1976. Manuscript received by the Heat Transfer Division February 18, 1977. Paper No. 76-WA/HT-19.

[1] Numbers in brackets designate References at end of paper.

The Merging Jet Model

Hirst [11, 12] presented an excellent analysis of the single port discharge. He developed the axisymmetric equations of motion and energy from the Navier-Stokes and energy transport equations. He employed the Boussinesq assumption, boundary-layer approximations, low Eckert number, fully turbulent, steady, and incompressible flow considerations to make the equations tractable. By assuming axisymmetry and integrating in the radial direction the equations may be reduced to the following form:

Continuity

$$\frac{d}{ds}\int_0^\infty \bar{u} r dr = -\lim_{r\to\infty}(r\bar{v}) = E \quad (1)$$

Energy

$$\frac{d}{ds}\int_0^\infty \bar{u}(\bar{T}-\bar{T}_\infty) r dr = -\frac{d\bar{T}_\infty}{ds}\int_0^\infty \bar{u} r dr - \lim_{r\to\infty}\overline{(rv'T')} \quad (2)$$

s-momentum

$$\frac{d}{ds}\int_0^\infty \bar{u}^2 r dr = \bar{U}_\infty E \sin\theta_1 \cos\theta_2 + \int_0^\infty g\beta(\bar{T}-\bar{T}_\infty) r dr \sin\theta_2 - \lim_{r\to\infty}\overline{(ru'v')} \quad (3)$$

Curvature equations

$$\frac{d\theta_2}{ds} = g\int_0^\infty [\beta(\bar{T}-\bar{T}_\infty) r dr \cos\theta_2 - EU_\infty \sin\theta_1 \sin\theta_2]/\bar{q}$$

and

$$\frac{d\theta_1}{ds} = \kappa_1 = \frac{EU_\infty \cos\theta_1}{\bar{q}\cos\theta_2} \quad (5)$$

where

$$\bar{q} = \int_0^\infty \bar{u}^2 r dr - \frac{E^2}{4} - \lim_{r\to\infty}\overline{(r^2v'^2)} \quad (6)$$

The various terms in these equations are defined in the Nomenclature.

The equations have been divided by 2π and the specific heat where appropriate. The curvature equations are formed from appropriate combinations of the component momentum equations. Fig. 1 illustrates the coordinate system used and defines θ_1 and θ_2. The integrals in these equations can be easily evaluated once suitable velocity and temperature profiles are assumed. Most researchers employ gaussian profiles; however, for this work the 3/2 power profiles successfully employed by Stolzenbach [13] were found to be more convenient since they have a defined edge.

In the zone of established flow, axisymmetric profiles of temperature and velocity are assumed to be

$$u = \Delta u + U_\infty \cos\Theta_2 \sin\Theta_1$$

where

$$\Delta u = \Delta u_c\left(1-\left(\frac{r}{b}\right)^{3/2}\right)^2$$

and

$$\Delta T = \Delta T_c\left(1-\left(\frac{r}{b}\right)^{3/2}\right)^2$$

where b is the full half-width of the plume. These profiles are very

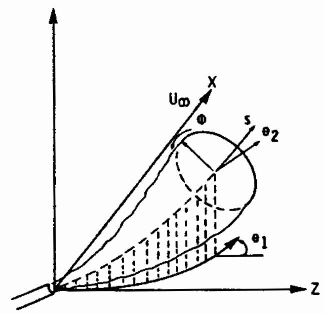


Fig. 1 The coordinate system employed by Hirst [11]

Nomenclature

A = area
a = entrainment coefficient
$a_{0,1,2,...}$ = entrainment coefficients
B = slot plume discharge point width
b = 3/2 power profile plume half width
C_D = drag coefficient
D = port diameter
E = entrainment
F = Froude number = $U_0/(\Delta\rho/\rho_0 gD)^{1/2}$
F_D = drag force
F_L = local Froude number
$\bar{f}$ = time averaged quantity
f' = fluctuating quantity
$G_{1,2,3,4}$ = flux quantities in the zone of single plume flow
$\bar{g}$ = gravitational force (without bar—the gravitational constant)
$H_{1,2,3,4}$ = flux quantities in the merging zone
$h_{1,2,3}$ = incomplete integrals defined in the merging zone
L = distance between ports
$N_{1,2,3}$ = normal terms, employed in the drag force relation
$\bar{q}$ = defined in the text
R = towing ratio = U_∞/U_0
r = plume radius and radial coordinate
S,s = distance along center line and center-line coordinate
T = temperature
T_∞ = ambient temperature
T_c = center-line temperature
ΔT_c = $(T_c - T_\infty)$
U_0 = port discharge velocity
U_∞ = ambient velocity
u = velocity in the S-direction
u_c = center-line velocity in the S-direction
Δu_c = $(u_c - U_\infty \cos\Theta_2 \sin\Theta_1)$
v = velocity in radial, r-direction
w = plume width
X = horizontal downstream distance and coordinate
Y = vertical coordinate and height above ports
Z = transverse coordinate along line of ports
α = measure of merging = L/b
β = coefficient of thermal expansion
ζ = merging coordinate along line of jet center lines
η = merging coordinate perpendicular to the S-ζ plane
Θ_2 and Θ = angle of plume center line to the X-Z plane
Θ_1 = angle of projection of the plume center line on the plume center line on the X-Z plane from the Z-axis
ρ = density
$\Delta\rho$ = $(\rho - \rho_\infty)$
ρ_0 = discharge density
ρ_∞ = ambient density
ϕ = circumferential plume coordinate

similar in shape to Gaussian profiles, but their defined edge and the assumption that velocity and temperature are the same give a definite point for merging calculations to start. With these relations the integrals may be evaluated and the following differential equations obtained:

Continuity of mass

$$\frac{d}{ds} G_1 = E \quad (7)$$

Energy

$$\frac{d}{ds} G_2 = -\frac{dT_\infty}{ds} G_1 \quad (8)$$

s-momentum

$$\frac{d}{ds} G_3 = EU_\infty \sin\theta_1 \cos\theta_2 + G_4 g \sin\theta_2 \quad (9)$$

Curvature equations

$$\frac{d\theta_1}{ds} = EU_\infty \cos\theta_1 \Big/ \left[\left(G_3 - \frac{E^2}{4}\right)\cos\theta_2\right] \quad (10)$$

and

$$\frac{d\theta_2}{ds} = (-EU_\infty \sin\theta_2 \sin\theta_1 + g\cos\theta_2 G_4) \Big/ \left(G_3 - \frac{E^2}{4}\right) \quad (11)$$

where G_i become the independent variables, defined as

$$G_1 = 0.12857\Delta u_c b^2 + \frac{b^2}{2} U_\infty \cos\theta_2 \sin\theta_1 \quad (12)$$

$$G_2 = 0.066758\Delta u_c \Delta T_c b^2 - 0.12857 b^2 \Delta T_c U_\infty \cos\theta_2 \sin\theta_1 \quad (13)$$

$$G_3 = 0.066758\Delta u_c{}^2 b^2 + 0.25714 b^2 \Delta u_c U_\infty \cos\theta_2 \sin\theta_1 + \frac{b^2}{2} U_\infty{}^2 \cos^2\theta_2 \sin^2\theta_1 \quad (14)$$

$$G_4 = 0.12857 b^2 \beta \Delta T_c \quad (15)$$

The ambient turbulence terms have been omitted from this presentation for brevity, but they are included in the computer program for use when information is available. The quantities G_i ($i = 1,2,3,4$) are the local mass flux, energy flux, momentum flux, and density deficiency, respectively, divided by 2π and specific heat where appropriate. The values of G_i can be obtained by stepwise integration of equations (7)–(11) using a Runge-Kutta or Hamming Predictor-Corrector scheme once a proper entrainment function, E, is obtained. The subsequent calculation of the center-line temperature and velocity and the plume width is an exercise in solving three equations in three unknowns. The initial conditions for these calculations can be obtained from zone of flow establishment calculations similar to those presented by Hirst [12] using the appropriate 3/2 power profiles.

The calculation continues in the streamwise (s) direction until the width of the plume is equal to the spacing between the jets. At this point the jets begin to merge and the profiles are no longer axisymmetric. For the purposes of smooth transition and a continuous solution of the differential equations between the zone of established flow and the zone of merging, the profiles employed for the merging region should have the following qualities:

1 The profiles should be smooth in all directions.

2 The slopes should be zero at $\zeta = 0$, $\eta = 0$, and $\zeta = L/2$, $\eta = 0$ (see Fig. 2).

3 When the plumes just begin to merge they should retain their single-plume profiles.

4 The profiles should be the superposition of the single-plume profiles (where applicable) with no point allowed to exceed center-line properties.

5 The profiles should maintain the characteristics of similar profiles in s.

With the foregoing considerations, the following profiles are assumed in the zone of merging:

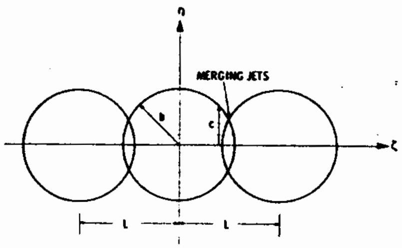


Fig. 2 The coordinate system for the merging plume analysis

$$u = \Delta u + U_\infty \cos\theta_2 \sin\theta_1$$

$$\Delta u = \Delta u_\zeta \left[1 - \left(\frac{\eta}{c}\right)^{3/2}\right]^2$$

$$\Delta u_\zeta = \Delta u_c \left[1 - \left(\frac{\zeta}{b}\right)^{3/2}\right]^2 \quad \text{for} \quad 0 \le \zeta \le L - b$$

$$\Delta u_\zeta = \Delta u_c \left[\left(1 - \left(\frac{\zeta}{b}\right)^{3/2}\right)^2 + \left(1 - \left(\frac{L-\zeta}{b}\right)^{3/2}\right)^2\right] \quad \text{for} \quad L - b \le \zeta \le L/2$$

$$\Delta T = \Delta T_\zeta \left[1 - \left(\frac{\eta}{c}\right)^{3/2}\right]^2$$

$$\Delta T_\zeta = \Delta T_c \left[1 - \left(\frac{\zeta}{b}\right)^{3/2}\right]^2 \quad \text{for} \quad 0 \le \zeta \le L - b$$

$$\Delta T_\zeta = \Delta T_c \left[\left(1 - \left(\frac{\zeta}{b}\right)^{3/2}\right)^2 + \left(1 - \left(\frac{L-\zeta}{b}\right)^{3/2}\right)^2\right] \quad \text{for} \quad L - b \le \zeta \le L/2$$

where

$$c^2 = b^2 - \zeta^2$$

After $\Delta u_\zeta = \Delta u_c$ at $\zeta = L/2$, then it is assumed that $\Delta u_\zeta = \Delta u_c$ for all ζ. The same is assumed for temperature. Theoretically, before $\Delta u_\zeta = \Delta u_c$ at $\zeta = L/2$, $b > L$ implying that there are three plumes merging instead of two. The contribution of nonadjacent plumes is considered minor and is neglected in the analysis. This is consistent with other fluid dynamic assumptions.

With the profiles described, the governing relations may be written as follows.

The continuity equation becomes

$$\frac{d}{ds}\left[\frac{1}{2\pi}\int\int_A u\,dA\right] = \frac{d}{ds}\left[\frac{2}{\pi}\int_0^{L/2}\int_0^c (\Delta u + U_\infty \cos\theta_2 \sin\theta_1)\,d\eta\,d\zeta\right] = \frac{d}{ds} H_1 = E \quad (16)$$

where

$$H_1 = \frac{0.90}{\pi} b^2 \Delta u_c h_1(\alpha) + \frac{b^2}{\pi} U_\infty \cos\theta_2 \sin\theta_1 h_2(\alpha) \quad (17)$$

$$\alpha = L/b$$

$$h_1(\alpha) = \int_0^{\alpha/2} \sqrt{1-\chi^2}\,(1 - \chi^{3/2})\,d\chi + \int_{\alpha-1}^{\alpha/2} \sqrt{1-\chi^2}\,(1 - (\alpha - \chi)^{3/2})^2\,d\chi \quad (18)$$

and

$$h_2(\alpha) = 2\int_0^{\alpha/2} \sqrt{1-\chi^2}\,d\chi^2 = \frac{\alpha}{2}\sqrt{1 - \left(\frac{\alpha}{2}\right)^2} + \sin^{-1}\left(\frac{\alpha}{2}\right) \quad (19)$$

After $\Delta u_\zeta = \Delta u_c$ at $\zeta = L/2$, one obtains,

$$H_1 = \frac{b^2}{\pi} 0.45\Delta u_c + U_\infty \cos\theta_2 \sin\theta_1\, h_2(\alpha) \quad (20)$$

For the energy equation,

$$\frac{d}{ds}\left[\frac{1}{2\pi}\int\int_A u\Delta T\,dA\right] = \frac{d}{ds}\left[\frac{2}{\pi}\int_0^{L/2}\int_0^c (\Delta u + U_\infty \cos\theta_2 \sin\theta_1)\Delta T_c\,d\eta\,d\zeta\right] = \frac{d}{ds} H_2 = -\frac{dT_\infty}{ds} H_1 \quad (21)$$

where,

$$H_2 = \frac{0.63116}{\pi} b^2 \Delta T_c \Delta u_c h_3(\alpha) + \frac{0.90}{\pi} b^2 \Delta T_c U_\infty \cos\theta_2 \sin\theta_1 h_1(\alpha) \quad (22)$$

$$h_3(\alpha) = \int_0^{\alpha/2} \sqrt{1-\chi^2}\,(1 - \chi^{3/2})^4\,d\chi + \int_{\alpha-1}^{\alpha/2} \sqrt{1-\chi^2} \cdot [(1 - \chi^{3/2})^2 (1 - (\alpha - \chi)^{3/2})^2 + (1 - (\alpha - \chi)^{3/2})^4]\,d\chi \quad (23)$$

With the given profile description, Δu_ζ approaches Δu_c at $\zeta = L/2$ to the same degree that ΔT_ζ approaches ΔT_c. Hence, when $\Delta T_\zeta = \Delta T_c$, $\Delta u_\zeta = \Delta u_c$, and one obtains

$$H_2 = \frac{b^2}{\pi} \Delta T_c\,(0.31558\Delta u_c + 0.45 U_\infty \cos\theta_2 \sin\theta_1) h_2(\alpha) \quad (24)$$

The s-momentum equation takes the following form for the zone of merging:

$$\frac{d}{ds}\left[\frac{1}{2\pi}\int\int_A u^2\,dA\right] = \frac{d}{ds}\left[\frac{2}{\pi}\int_0^{L/2}\int_0^c (\Delta u + U_\infty \cos\theta_2 \sin\theta_1)^2\,d\eta\,d\zeta\right] = \frac{d}{ds} H_3 = EU_\infty \cos\theta_2 \sin\theta_1 + H_4 g \sin\theta_2$$

where

$$H_3 = \frac{0.63116}{\pi} b^2 \Delta u_c h_3(\alpha) + \frac{1.8}{\pi} b^2 \Delta u_c U_\infty \cos\theta_2 \sin\theta_1 h_1(\alpha) + \frac{1}{\pi} b^2 U_\infty{}^2 \cos^2\theta_2 \sin^2\theta_1 h_2(\alpha) \quad (26)$$

and

$$H_4 = \frac{0.9}{\pi} b^2 \beta \Delta T_c h_1(\alpha) \quad (27)$$

After $\Delta u_\zeta = \Delta u_c$,

$$H_3 = \frac{b^2 h_2(\alpha)}{\pi}(0.31558\Delta u_c{}^2 + 0.9\Delta u_c U_\infty \cos\theta_2 \sin\theta_1 + U^2_\infty \cos^2\theta_2 \sin^2\theta_1) \quad (28)$$

and

$$H_4 = \frac{0.45 b^2}{\pi} \beta \Delta T_c h_2(\alpha) \quad (29)$$

For the zone of merging the curvature equations are

$$\frac{d\theta_1}{ds} = \frac{EU_\infty \cos\theta_1}{\left(H_3 - \frac{E^2}{4}\right)\cos\theta_2} \quad (30)$$

and

$$\frac{d\theta_2}{ds} = \frac{-EU_\infty \sin\theta_2 \sin\theta_1 + H_4 g \cos\theta_2}{H_3 - \frac{E^2}{4}} \quad (31)$$

In these equations the term $\alpha = L/b$ represents the degree of merging since for $\alpha = 2$, the plumes are just beginning to merge and for $\alpha = 1$, the plumes are nearly merged. The functions $h_1(\alpha)$, $h_2(\alpha)$, and $h_3(\alpha)$ are incomplete integrals in α. These integrals are not solved in closed form at present but are readily solved numerically.

The quantities H_i ($i = 1, 2, 3, 4$) are the same as G_i in the previous section. They represent the local mass flux, energy flux, momentum flux, and density deficiency. As in the previous section, the quantities Δu_c, ΔT_c, and b may be obtained from these relations by solving three equations simultaneously. The initial conditions for the zone of merging are the final plume conditions of the zone of established flow.

The proper governing differential equations have now been developed and the initial and boundary conditions specified. The only element remaining for closure is the entrainment.

The Entrainment Function

Taylor [14] was the first to suggest the formulation of an entrainment function, and the use of characteristic values of the plume to describe this function. Morton, et al. [15] have employed this concept to develop the expression

$$E = ab\Delta u_c$$

which forms the basis for all the entrainment functions developed since. Numerous other researchers have modified this relation to obtain better agreement with the widely varying discharge and ambient conditions that affect the plume. Some have used the first moment of the momentum equation to determine the form of the entrainment function. Any alterations to the entrainment function should be based on the probable changes in the mechanics of jet mixing at different discharge orientations and ambient conditions.

The entrainment function employed for this study is based on the one proposed by Hirst with modifications for merging effects.

For the Zone of Established Flow (before merging), the entrainment (for $\theta_1 = 90°$) is expressed as

$$E = \left(a_1 + \frac{a_2}{F_L}\right) \times \left[b|u_c - U_\infty \cos\theta_2|\left(1 - \frac{a_4 b}{L}\right) + a_3 U_\infty b \sin\theta_2\right] \quad (32)$$

and for Zone of Merging, it is expressed as

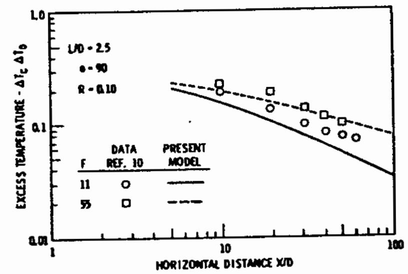


Fig. 3 Comparison of experimental and model predicted excess temperature for $L/D = 2.5$, $R = 0.10$, and $\theta = 90$

$$E = \left(a_1 + \frac{a_2}{F_L}\right)\left[b|u_c - U_\infty \cos\theta_2|\left(1 - \frac{a_4}{2}\right)\right.$$
$$\left.\times\left(1 - \frac{2}{\pi}\cos^{-1}\frac{L}{2b}\right) + a_3 U_\infty \frac{L}{2}\sin\theta_2\right] \quad (33)$$

The coefficients, a_1, a_2, and a_3 are similar to those used by Hirst. a_4 accounts for reduced entrainment due to plume competition. The term $(1 - 2/\pi \cos^{-1} L/2b)$ is the entraining surface during merging.

Results

The computer code employed was originally that of the Hirst model. Extensive revision of the code was performed in order to accommodate the different profiles and the merging process. The code was then used in the present study by tuning the entrainment coefficients so that predictions best matched experimental data. The calculations were carried out on an IBM 370/158 computer operated by Optimum Systems, Inc., Bethesda, Maryland. The coefficients were determined by successively simulating more complex discharge conditions starting with single momentum jet data without current, progressing to multiple buoyant merging plumes with current.

Data for this tuning process were taken from [10, 15–17]. In tuning the model for single buoyant plumes, it was found that the best overall fit could be obtained with $a_1 = 0.05$, $a_2 = 0.0$, and $a_3 = 11.5$. Better fit to a particular case could be obtained by adjusting these coefficients for each type of flow, but this procedure reduces the generality of the model. In trying to model the multiple-port data for ambient flow of Kannberg [10], it was found that good agreement could not be obtained for both trajectory and dilution by adjusting a_3 alone, particularly for closer port spacings. It was found that a drag force term had to be included in the momentum equation before satisfactory agreement could be obtained. The drag force was assumed to be a function of plume size, spacing, and ambient current dynamic head. For the unmerged plume this gave

$$F_D = C_D \frac{b}{\pi}\left(\frac{b}{L}\right)\frac{U_\infty^2}{2} N$$

for the merging plume

$$F_D = C_D \frac{L}{\pi}\frac{U_\infty^2}{2} N$$

where

$$N = \sqrt{N_1^2 + N_2^2 + N_3^2}$$

and

$$N_1 = \cos^2\theta_2 \sin\theta_1 \cos\theta_1$$
$$N_2 = \sin^2\theta_2 + \cos^2\theta_2 \cos^2\theta_1$$
$$N_3 = \sin\theta_2 \cos\theta_2 \sin\theta_1$$

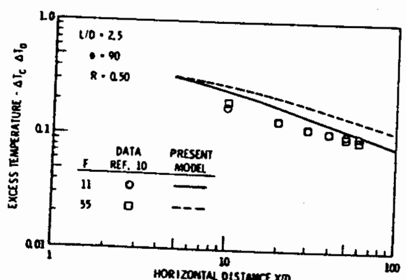


Fig. 4 Comparison of experimental and model predicted excess temperature for $L/D = 2.5$, $R = 0.5$, and $\theta = 90$

This gave a zero drag force for single plumes and a maximum drag force for closely spaced multiple plumes.

With this drag force included the best overall agreement was obtained with $a_4 = 0.2$ and $C_D = 2$. Although the effect was minor, better agreement was obtained for $R = U_\infty/U_0$ values near 0.1 if C_D was increased to 3.0, and reduced to 1.0 or below for $R = 0.5$.

The following figures show the final model predictions as compared to multiport data using the values of coefficients giving the best overall agreement. Fig. 3 is a plot of excess temperature ratio versus horizontal distance for vertical discharge into a cross current with $R = 0.1$.

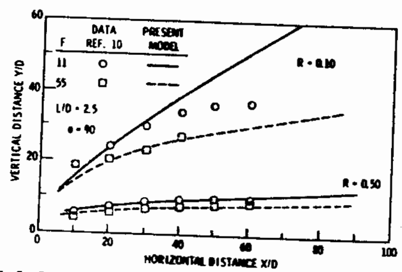


Fig. 5 Comparison of model predicted trajectories with data for $L/D = 2.5$, $F = 11$ and 55, $R = 0.10$ and 0.50

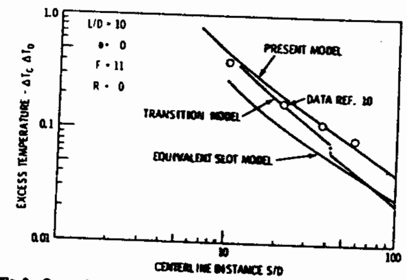


Fig. 6 Comparison of excess temperature predicted by several models and experimental data for $L/D = 10$, $R = 0.0$, and $F = 11$

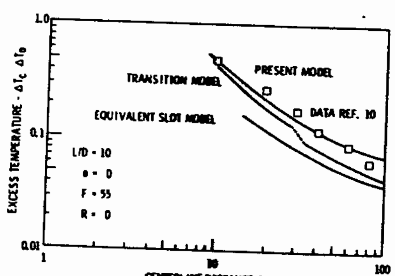


Fig. 7 Comparison of excess temperature predicted by several models and experimental data for $L/D = 10$, $R = 0.0$, and $F = 55$

The port spacing is 2.5 dia; values for discharge Froude numbers of the order of 10 and 60 are shown. The data points represent the mean of the experimental multiport data presented in reference [10]. The lines are predicted values from the model described in this paper. Fig. 4 presents similar information for $R = 0.5$. The agreement is seen to be quite good. Fig. 5 is a plot of plume trajectory for the foregoing cases. Again both experimental values and model predictions are shown. The agreement is very good for the high-current cases but not as good for the low-current cases.

Excess temperature decay along the plume center line, s/D, are shown on Fig. 6 and 7 for horizontal discharge into a stagnant ambient, for discharge Froude numbers of about 10 and 60. The port spacing in this case is 10 dia. For comparison, the experimental values of reference [10] are plotted along with predictions from the Koh and Fan transition model, the Jirka and Harleman equivalent slot model, and the merging model described here. The Koh and Fan and Jirka and Harleman models rely on entrainment coefficients determined for single round and slot jets, respectively. While it is possible to change the entrainment coefficients of these models, it would not be consistent with the nature of those models, nor would reliable predictions be obtained for extreme port spacings. Therefore the entrainment coefficients suggested by the respective authors were used in the other models. It is seen that the transition model and the equivalent slot model, although giving reasonable values, both overpredict the measured dilution, whereas the merging model does much better. The final two figures, Fig. 8 and 9 show trajectory and width comparisons for horizontal discharge into stagnant ambients, where the spacing is again 10 dia. The agreement between experimental data and merging model predictions is seen to be very good for both trajectory and width within the experimental Froude number ranges of reference [10].

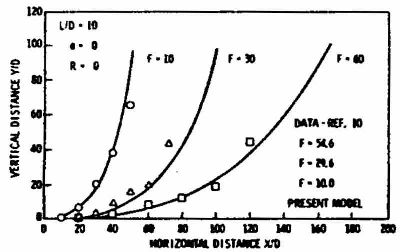


Fig. 8 Comparison of model predicted trajectories with experimental data for $L/D = 10$, $R = 0.0$, and $\theta = 0.0$ for several Froude numbers

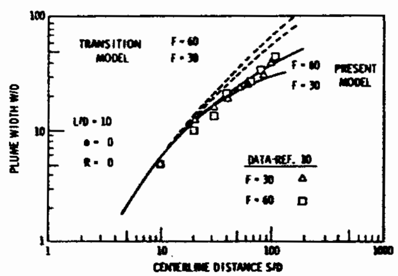


Fig. 9 Comparison of width predictions of the Koh and Fan transition model and present model with experimental data

Conclusions

The model presented here successfully treats the merging of adjacent round jets from a row of equally spaced ports into a pseudo-slot jet in a manner that is more continuous than other methods advanced in recent years. Use of the model gives better agreement with experimental data than other models considered. The model requires additional terms in the entrainment function to account for competition of entraining fluid and for the reduction in the entrainment area of the jet as merging progresses. Unfortunately, agreement with cross-flow data could not be obtained for multiple jets without the addition of a drag term and a corresponding additional coefficient. The values of these coefficients were easily determined by comparing data from progressively more complex flow cases with model predictions.

Acknowledgments

The work reported herein was performed under U. S. Environmental Protection Agency grant R800818. This support is gratefully acknowledged.

References

1 Trent, D. S., and Wehy, J. R., "Numerical Thermal Plume Model for Vertical Outfalls in Shallow Water," Environmental Protection Agency Technology Series, EPA-R2-73-162, Mar. 1973.

2 Shetz, J. A., Chien, C. J., and Sill, B. L., "Heat Transfer and Fluid Mechanics of the Thermal Pollution Problem," Thermal Pollution Analysis, Vol. 36 c.f. *Progress in Astronautics and Aeronautics*, J. A. Shetz, ed., American Institute of Astronautics and Aeronautics, 1975.

3 Koh, R. C., and Fan, L. N., "Mathematical Models for the Prediction of Temperature Distributions Resulting From the Discharge of Heated Water in Large Bodies of Water," Environmental Protection Agency Water Pollution Control Research Series Report No. 1613DW/1/70, Oct. 1970.

4 Jirka, G., and Harleman, D. R. F., "The Mechanics of Submerged Multiport Diffusers for Buoyant Discharges in Shallow Water," Massachusetts Institute of Technology, Ralph M. Parsons Laboratory for Water Resources and Hydro-dynamics, Report No. 169, Mar. 1973.

5 Kannberg, L. D., and Davis, L. R., "Experimental Investigation of Deep Submerged Multiple Buoyant Jets Into Stagnant and Co-Moving Ambients," Paper presented at Thermal Pollution Analysis Conference held at Virginia Polytechnic Institute and State University, Blacksburg, Va., May 1974.

6 Koh, R. C., Brooks, N. H., Wolanski, E. H., and List, E. J., "Basin Model Studies of Diffusers," SCE Report No. 4, W. M. Keck Hydraulics Laboratory, California Institute of Technology, May 1973.

7 Argue, J., "The Mixing Characteristics of Submerged Multiple Port Diffusers for Heated Effluents in Open Channel Flow," MA thesis, University of Iowa, May, 1973.

8 Liseth, P., "Mixing of Merging Buoyant Jets From a Manifold in Stagnant Receiving Water of Uniform Density," Hydraulic Engineering Laboratory Report HEL 231, University of California, Berkeley, Nov. 1970.

9 Iwasa, Y., and Yatsuzuka, M., "Spread of Heated Waters From Multiport Diffuser," *Proceedings of the U. S.-Japan Joint Seminar on Engineering and Environmental Aspects of Waste Heat Disposal*, Paper No. 9, Tokyo, Japan, Apr. 15, 1974.

10 Kannberg, L. D., "An Experimental and Analytical Investigation of Deep Submerged Multiple Buoyant Jets," PhD thesis, Oregon State University, 1976.

11 Hirst, E. A., "Analysis of Round Turbulent, Buoyant Jets Discharged to Flowing Stratified Ambients," Oak Ridge, Oak Ridge National Laboratory, Report No. ORNL-4685, 1971, p. 36.

12 Hirst, E. A., "Analysis of Buoyant Jets Within the Zone of Flow Establishment," Oak Ridge National Laboratory, Report No. ORNL-TM-3470, Aug. 1971.

13 Stolzenbach, K. D., and Harleman, D. R. F. "An Analytical and Experimental Investigation of Heated Water," Ralph M. Parsons Laboratory for Water Resources and Hydrodynamics, Report No. 135, Cambridge, Massachusetts Institute of Technology, 1971, p. 212.

14 Taylor, Sir Geoffrey, "Dynamics of a Mass of Hot Gas Rising in Air," United States Atomic Energy Commission, MDDC 919, LADC 276, 1945.

15 Morton, B. R., Taylor, Sir Geoffrey, Turner, J. S., "Turbulent Gravitational Convection From Maintained and Instantaneous Sources," *Proceedings of the Royal Society of London*, Series A, Vol. 234; pp. 1–23.

16 Fan, L-N., "Turbulent Buoyant Jets Into Stratified or Flowing Ambient Fluids," Keck Laboratory of Hydraulic and Water Resources, California Institute of Technology, Report No. KH-R-15, June 1967.

17 Anwar, H. O., "Behavior of Buoyant Jets in Calm Fluid," *Journal of the Hydraulics Division*, ASCE, Vol. 95, No. HY4, 1969, pp. 1289–1303.

18 Fox, D. G., "Forced Plume in a Stratified Fluid," *Journal of Geophysical*

Research, Vol. 75, No. 33, 1970, pp. 6818–35.
19 Abraham, G., "Horizontal Jets in Stagnant Fluid of Other Density," *Journal of the Hydraulics Division*, ASCE, Vol. 91, No. HY4, 1965, pp. 139–153.
20 Platten, J. L., and Keffer, J. F., "Entrainment in Deflected Axisymmetric Jets at Various Angles to the Free Stream," Mechanical Engineering Department, University of Toronto, UTME-TP-6808, 1968.
21 Hoult, D. P., Fay, J. A., and Forney, L. J., "A Theory of Plume Rise Compared with Field Observations," *Journal of the Air Pollution Control Association*, Vol. 19, No. 9, 1969, pp. 585–90.

DISCUSSION

G. H. Jirka[1]

Kannberg and Davis address two important points in the design and analysis of multiple-port diffusers: (1) What is the mechanism of jet merging and how to model it? (2) What is the effect of jet merging on the mixing capacity of such diffusers?

(1) The lateral merging of the individual jets emanating from the diffuser nozzles appears to be a complicated process. As the jet boundaries approach each other, the space for the entrainment flow is limited between the jets leading to low pressure zones and dynamic attachment effects. The turbulent diffusion characteristics change from an axisymmetric shearing zone to a two-dimensional one. Clearly, jet integral techniques, with their assumptions of jet similarity and hydrostatic pressure distribution, can describe this transition in an approximate fashion only. Kannberg and Davis' profile specification in the transition region represents a reasonable approach in this direction and has advantages over Koh and Fan's earlier model, inasmuch as it avoids abrupt changes in the center line temperature prediction. Questions, however, remain as to the choice of the entrainment coefficient (two- or three-dimensional value) and the assumptions of a turbulent Prandtl number of unity, which is in contradiction with experimental observations on jets and plumes. The pressure effect on the merging process caused by ambient currents is even more difficult. The form drag assumption, equation (34), proposed by the authors seems a tenuous one in view of the complicated geometry. Such model deficiencies, however, could be corrected through the best-fit choice of the drag coefficients, at least within the parameter range of the author's experiments.

(2) The basic argument of the two-dimensional slot diffuser concept, as used in the model by Jirka and Harleman [4], is that beyond the line of merging the flow field of the actual multiple-port diffuser is equal to that of a slot discharge. Or in other words, the initial three-dimensional details have no distinguishable effects at longer distances. This concept is physically reasonable, as the equivalent slot diffuser has the same kinetic energy input per unit length, which is ultimately expanded in turbulent jet mixing. This behavior of laterally limited jets is in analogy with the nozzle shape effects on free turbulent jets in which, after a certain characteristic distance, the initial shape effects are no longer felt and all jets approach an axisymmetric shape (Sforza, et al. [22][2], experiments by Yevdjevich reported in [21]). The equivalent slot concept is useful for diffuser classification as the number of governing parameters is reduced. It has been found of sufficient accuracy for multiple-port diffusers by Liseth [8] and more recently by Buhler [23]. The lack of agreement which is indicated by Kannberg and Davis' experiments may be due to equipment effects, since the horizontal momentum input of the jets is likely to cause some circulation and possibly re-entrainment in a finite size laboratory facility. This may account for the lower centerline temperatures which were observed.

Finally, some caution has to be expressed regarding Kannberg and Davis' introductory claim that their analysis is applicable for the predictions of condenser heat discharges from electric power generation. In fact, the low buoyancy of such heated discharges in combination with the limiting water depth at usual disposal sites gives rise to dynamic instabilities in form of a recirculating eddy extending over the entire water depth. This discharge condition cannot be analysed by means of simple buoyant jet models as proposed by the authors. Stability criteria, which depend on relative water depth, discharge Froude number and angle, have been developed by Jirka and Harleman [4, 24] and are plotted in Fig. 10. Multiple-port diffuser designs for condenser heat discharge generally fall into the unstable discharge range. The practical application of the stable range, in which the authors' model is applicable, relates to sewage and blowdown diffusers, usually with higher buoyancy and lower flowrates.

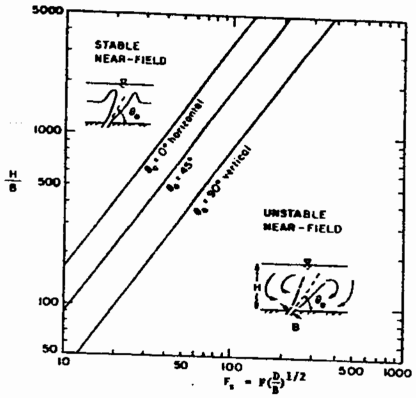


Fig. 10 Stability of the flow field for multiport diffusers discharging into shallow water as a function of slot densimetric Froude number F_s, relative water depth H/B, and discharge angle, θ_o ($B = D^2\pi/4L$ is the equivalent slot width); after Jirka and Harleman [24]

Additional References

22 Sforza, P. M. Steiger, M. H., and Trentacoste, N., "Studies on Three-Dimensional Viscous Jets," *AIAA Journal*, Vol. 4, 1966.
23 Buhler, J., "Model Studies on Multiport Outfalls in Unstratified, Stagnant or Flowing Ambient Water," PhD thesis, Department of Civil Engineering, University of California, Berkeley, 1974.
24 Jirka, G. H., and Harleman, D. R. F., "Buoyant Jets in Confined Surroundings," in *Thermal pollution Analysis*, J. A. Schetz, ed., Vol. 36 of Progress in Astronautics and Aeronautics, AIAA, New York, 1975.

[1] Massachusetts Institute of Technology, Cambridge, Mass.
[2] Numbers in brackets designate Additional References at end of discussion.

Authors' Closure

We appreciate Dr. Jirka's taking time to comment on the analysis and results of this paper. We agree with much of what he has said, but feel a few points need to be further clarified.

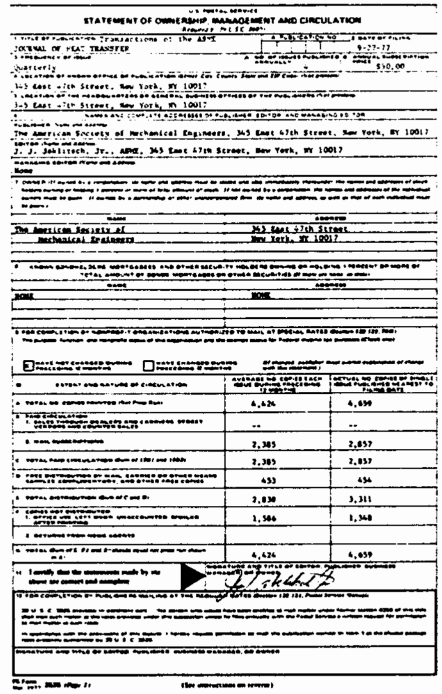
A unity turbulent spreading parameter was introduced into the analysis to make the smooth merging of both temperature and velocity easier. The authors realize that this may not be exactly the case but it is felt that the errors introduced are minor in light of the complexity of the problem.

The equivalent slot concept is a useful approach in analyzing the characteristics of certain multiport discharges, but as with most simplified analyses, the limits of use should be understood. The fundamental premise behind the equivalent slot is that beyond the point of merging the plumes from a row of equally spaced jets become the same as the plume from a slot jet having the same mass and momentum flux per unit length as the multiple jets. This presumes that up to the point of merging, the mixing mechanisms of the two are the same, and of course they are not. The effect of this difference on the ultimate dilution would be minor if the major portion of the dilution occured after merging. This would either be for large distances away from the point of discharge or for closely spaced discharge ports. Most researchers realize this. For example, Liseth [8] suggested the use of an equivalent slot for closely spaced ports and only attempted to compare results at $y/L = 80$. For an $L/D = 10$, this at a $y/D = 800$. At this point the dilution is so great that the excess temperature, regardless of how it is calculated, is usually no longer of interest.

In the near and intermediate field where most of the data reported in this paper were taken, the story is different. One of the causes of this difference is the development length. To illustrate, consider a diffuser with 1-ft dia ports spaced 10-ft apart discharging into stagnant water. The jets for this system merge about 30 ft from discharge and have a development length where the temperatures begin to decay of about 6 ft. The equivalent slot for this system has a width, B, of 0.08 ft. Therefore, at the end of the actual single jet development zone where the excess temperature is of the order of 1.0–0.8, the equivalent slot is at an s/B of 75 where it predicts an excess temperature of the order of 0.2.

In addition to the development zone problem, entrainment is also different for the two systems. Entrainment has been suggested as $2\pi(0.082)bu_c$ and $2(0.16)Lu_c$ for the round and slot jets, respectively [3]. For the example case considered in the foregoing, the slot jet entrainment is $3.2u_c$ while the round jet entrainment varies due to plum growth from $0.26u_c$ at discharge to $2.6u_c$ at merging. Thus, even though a multiport discharge merges into essentially a two-dimensional plume where the individual character of the original jets no longer exists, its properties are different from those of a plume that originated from a slot. As a result, the authors feel that the indiscriminate use of the equivalent slot to model multiple port diffusers could lead to gross errors.

We agree with Dr. Jirka that the present model should not be used in shallow stagnant receiving water where surface effects and instabilities are present. In flowing ambients, however, even at low velocities these effects are reduced and in some cases eliminated. In many cases the receiving water is sufficiently deep to use the model to predict the major portion of the plume's characteristics.


STATEMENT OF OWNERSHIP, MANAGEMENT AND CIRCULATION

Journal of Heat Transfer — Quarterly — \$30.00

345 East 47th Street, New York, NY 10017

The American Society of Mechanical Engineers, 345 East 47th Street, New York, NY 10017

J. J. Jaklitsch, Jr., ASME, 345 East 47th Street, New York, NY 10017

None

The American Society of Mechanical Engineers — 345 East 47th Street, New York, NY 10017

None — None

	Average no. copies each issue during preceding 12 months	Actual no. copies of single issue published nearest to filing date
A. Total no. copies printed (net press run)	4,424	4,659
B. Paid circulation 1.	--	--
2. Mail subscriptions	2,385	2,857
C. Total paid circulation	2,385	2,857
D. Free distribution	453	454
E. Total distribution	2,838	3,311
F. Copies not distributed 1.	1,586	1,348
2. Returns from news agents		
G. Total	4,424	4,659

11. THEORETICAL ANALYSIS OF THE PDS PROGRAM

11. THEORETICAL ANALYSIS OF THE PDS PROGRAM

THEORETICAL ANALYSIS OF THE PDS PROGRAM

The theoretical analysis used to develop the three-dimensional surface plume program (PDS) is based on an earlier model by Prych. It was modified to its present form by Davis and was tuned to data by Shirazi, hence PDS (see "Workbook of Thermal Plume Prediction-Vol.2-Surface Discharge," Environmental Protection Agency Report EPA-R2-72-005b, May 1984 by M.A. Shirazi and L.R. Davis).

In order to make the results of the mathematical model more general and independent of any particular system of units, the governing equations and all calculated plume characteristics are made dimensionless internal to the program by dividing all length characteristics by H_o, the discharge depth; velocities by U_o, the discharge velocity; momentum, pressure, shear and drag forces by $U_o^2H_o^2$; the flow rate by $U_oH_o^2$; diffusivity by U_oH_o; excess temperature by (T_o-T_a), the initial discharge to ambient temperature difference; and the kinematic heat transfer coefficient by U_o. Input variables are entered in metric units. They are then non-dimensionalized as specified above. All calculations are made. Output is then converted back to metric units. Different units can be used by changing the non-dimensinalization units.

The method of analysis is an integral approach which assumes similarity of temperature and velocity profiles and the principle of entrainment The profiles assumed are Gaussian such that:

$$T_r = T\exp(-\eta^2/B^2)\cdot\exp(-Z^2/H^2) \qquad (1)$$

$$U_r = U\exp(-\eta^2/B^2)\cdot\exp(-Z^2/H^2) + V\cos\theta \qquad (2)$$

where η and Z are distances perpendicular to the plume centerline in the lateral and vertical directions, respectively, T and U are the Centerline temperature and Velocity respectively and V is the ambient velocity as shown on Fig. 1.

With the temperature and velocity profiles assumed, the energy, volume and momentum fluxes can be integrated across the plume at any cross section leaving them in terms of centerline values and plume characteristics width, B, and Depth, H. Accordingly, the volume flux becomes

$$Q = \iint_A (U_r)\, d\eta\, dZ = \pi HB\left(\frac{U}{2} + V\cos\theta\right) \qquad (3)$$

where the limits of integration for V cos θ are taken as the bottom half of the region:

Solving (3) for U, yields

$$\left(\frac{\eta}{\sqrt{2}B}\right)^2 + \left(\frac{Z}{\sqrt{2}H}\right)^2 \le 1 \qquad (4)$$

$$U = 2\left(\frac{Q}{\pi HB} - V\cos\theta\right) \qquad (5)$$

The heat flux, J, is:

$$J = \iint_A U_r T_r \, d\eta \, dZ = \frac{\pi}{2} TBH\left(\frac{U}{2} + V\cos\theta\right) = \frac{QT}{2} \qquad (6)$$

The momentum flux, M, is:

$$M = \iint_A U_r^2 \, d\eta \, dZ = \pi BH\left(\frac{U}{2} + V\cos\theta\right)^2 = \frac{Q^2}{\pi BH} \qquad (7)$$

The quantities dQ/ds, dT/ds, and dM/ds are calculated from conservation equations. dQ/ds is assumed to be due to contributions of jet entrainment and ambient turbulent mixing, thus:

$$\frac{dQ}{ds} = \frac{dQ}{ds}\bigg|_j + \frac{dQ}{ds}\bigg|_a \qquad (8)$$

The jet and ambient contributions are both divided into vertical and horizontal components. The horizontal jet entrainment fluid is:

$$\frac{dQ}{ds}\bigg|_{j,h} = 2E_0 \int_{-\sqrt{2}H}^{0} \Delta U \, dZ \qquad (9)$$

where

$$\Delta U = (U^2 + V^2 \sin^2\theta)^{1/2} \exp(-Z^2/H^2) \qquad (10)$$

and E_o is an entrainment coefficient. By inserting (10) into (9) we obtain:

$$\frac{dQ}{ds}\bigg|_{j,h} = \sqrt{\pi} H E_o (U^2 + V^2 \sin\theta)^{1/2} \qquad (11)$$

The vertical jet entrained fluid is:

$$\frac{dQ}{ds}\bigg|_{j,v} = 2 \int_0^{\sqrt{2}B} E \Delta U_v \, d\eta \qquad (12)$$

where $E = E_o f(Ri)$ and Ri is the local Richardson number given by:

$$R_i = \frac{\sqrt{2}}{F_o^2} \frac{HT(s,\eta,o)}{\Delta U_v^2} \qquad (13)$$

The function $f(R_i)$ is a curve fit to data. It is:
The velocity difference ΔU_v is given by:

$$f = [\exp(-5R_i) - 0.0183]/0.982 \qquad (14)$$

$$\Delta U_v = [U^2 \exp(-2\eta^2/B^2) + V^2 \sin^2\theta]^{1/2} \qquad (15)$$

The term T is the surface excess temperature at a distance η from the plume centerline. The value of the integral (12) is determined numerically in the program.

The effective entrainment due to ambient turbulent mixing is calculated as follows:

$$\frac{dQ}{ds}\bigg|_{a,h} = 11.0 \frac{H}{B} \frac{\epsilon_h}{U_o H_o} \qquad (16)$$

$$\frac{dQ}{ds}\bigg|_{a,v} = 11.0 \frac{B}{H} \frac{\epsilon_v}{U_o H_o} f(R_i') \qquad (17)$$

where ϵ_h and ϵ_v are the horizontal and vertical turbulent diffusion coefficients, respectively.

The change in heat flux along the plume due to heat exchange with the atmosphere is expressed as:

$$\frac{dJ}{ds} = -2 \int_0^{\sqrt{2}B} K T_r \, d\eta = \sqrt{\pi} KTB \qquad (18)$$

where K is a dimensionless heat exchange coefficient. Substituting (18) into (10) yields:

$$\frac{dT}{ds} = -\frac{T}{Q}\left(2\sqrt{\pi}KB + \frac{dQ}{ds}\right) \qquad (19)$$

The conservation of momentum is applied in the s-direction and then divided into X and Y components. The net forces on the plume are balanced by the change in momentum flux. The forces considered important are (a) the internal pressure forces due to buoyancy, (b) form drag due to ambient current and (c) interfacial shear forces.

The pressure forces are found by determining the excess pressure due to buoyancy as a function of depth and then integrating the pressure over the vertical cross section of the plume. Thus, the normalized pressure force is:

$$P = \frac{1}{F_o^2}\iint_A\left(\int_{-\infty}^{z} T_r dz\right)dA = \frac{\sqrt{\pi}TH^2B}{2F_o^2} \qquad (20)$$

The form drag acting normal to the plume centerline is assumed similar to the drag on a solid body such that

$$F_D = \frac{1}{2}\sqrt{2}\,C_D H V|V|\sin^2\theta \qquad (21)$$

where C_D is a drag coefficient.

The interfacial shear forces are assumed to be similar to turbulent flow over a flat surface with a boundary layer thickness of $(2)^{1/2}$H and a velocity equal to the vector velocity difference between the plume and ambient current. Accordingly, the X and Y components of this shear force reduce to:

$$SF_X = C_F\left(\frac{1}{R_eH}\right)^{1/4}\int_0^{\sqrt{2}B}\Delta U_v^{3/4}[V\sin^2\theta - U\cos\theta\exp(-\eta^2/B^2)]\,d\eta \qquad (22)$$

$$SF_Y = -C_F\left(\frac{1}{R_eH}\right)^{1/4}\int_0^{\sqrt{2}B}\Delta U_v^{3/4}[V\cos\theta - U\exp(-\eta^2/B^2)]\,d\eta \qquad (23)$$

where C_F is a friction coefficient and R_e is the jet discharge Reynolds number. The value of C_F is determined by experiment.

The change in momentum flux includes the effects of the momentum of the entrained ambient fluid, V(dQ/ds), which acts in the X-direction. Equating the forces to the change in momentum flux in the X and Y directions yields:

$$\frac{d}{ds}[(M+P)\cos\theta] = SF_X + F_D\sin\theta + V\,dQ/ds \qquad (24)$$

$$\frac{d}{ds}[(M+P)\sin\theta] = SF_Y - F_D\cos\theta \qquad (25)$$

Using equations (7) and (20) for M and P, multiplying (24) by -sin θ, (25) by cos θ and combining yields an expression for the change in flow direction,

$$\frac{d\theta}{ds} = \frac{SF_Y\cos\theta - SF_X\sin\theta - F_DV\sin\theta\,(dQ/ds)}{\dfrac{Q^2}{\pi BH} + \dfrac{\sqrt{\pi}}{2F_o^2}TH^2B} \qquad (26)$$

Differentiating M and P, multiplying (24) by cos θ and (25) by sin θ and combining yields equation (27).

$$\frac{dH}{ds} = [SF_Y\sin\theta + SF_X\cos\theta + (V\cos\theta - 2Q/\pi BH)(dQ/ds) - (\sqrt{\pi}BH^2/2F_o^2)(dT/ds) + (Q^2/\pi BH - \sqrt{\pi}H^2/2F_o^2)(dB/ds)]\,[\sqrt{\pi}THB/2F_o^2 - Q^2/\pi BH^2]^{-1} \qquad (27)$$

It is noted that this expression for change in depth is undefined when the denominator is zero. Hence, results beyond this singularity are questionable.

Momentum in the lateral direction is included only indirectly through lateral spreading. It is assumed that the contributions to spreading by non buoyant horizontal jet mixing and buoyancy are independent of one another such that

$$\frac{dB}{ds} = \left(\frac{dB}{ds}\right)_{nb} + \left(\frac{dB}{ds}\right)_b \qquad (28)$$

where the subscripts b and nb denote buoyant and non-buoyant terms. the non-buoyant spreading is found by writing equation (27) without the buoyancy terms and assuming that

$$\left(\frac{dB/ds}{dH/ds}\right)_{nb} = (B/H)\,(dQ/ds)_h/(dQ/ds)_v \qquad (29)$$

where $(dQ/ds)_h$ and $(dQ/ds)_v$ are the horizontal and vertical entrainment rates.

$$\left(\frac{dB}{ds}\right)_{nb} = \frac{SF_Y\sin\theta + SF_X\cos\theta + \left(V\cos\theta - \dfrac{2Q}{\pi BH}\right)\dfrac{dQ}{ds}}{-(Q^2/\pi BH)\,[(dQ/ds)_v/(dQ/ds)_h + 1]} \qquad (30)$$

The spreading due to buoyancy is assumed to be a function of the local excess density ratio, plume depth and aspect ratio such that

$$\left(\frac{dB}{ds}\right)_b = \sqrt{\frac{2}{\dfrac{B}{H}F^2 - 1}} \qquad (31)$$

It is noted that this also has a singularity. Due to B/H usually being large, this singularity is not encountered in most problems.

The preceding equations are sufficient to perform a step-wise integration along the plume. From the local conditions of the plume, dQ/ds is calculated. When this is known dT/ds, dθ/ds and dB/ds are calculated. With these known, dH/ds can be calculated. These derivatives are integrated step-wise along the plume trajectory to give local values of X, Y, T, H, B, θ, and Q.

In order to start the integration within the developed zone where the above analysis is valid, starting conditions must be calculated. These are determined by a simplified analysis of the development zone and assuming that the development length is given by the equation

$$S_i = 5.4\left(\frac{A^2}{F_o}\right)^{1/3} \qquad (32)$$

where A is the discharge flow area and F_o is the discharge densimentric Froude number.

COMPUTER PROGRAM

The PDS program is written in FORTRAN 77 and consists of a main program entitled PDS and five subroutines, KHPCG, AREA, FCT, RED, and OUTP.

The main program, PDS, prompts the user for input variables, initializes constants, non dimensionalizes the variables and calls subroutine KHPCG which performs the actual calculation. Subroutine KHPCG is a standard scientific subroutine which performs the stepwise integration of differential equations by the Hamming Predictor-Corrector Method.

Subroutine SIGMAT is a subroutine that calculates the density of the water as a function of temperature and salinity. It is an empirical curve fit to data.

Subroutine AREA is a step-wise integration of the area enclosed by surface isotherms. Subroutine FCT calculates the derivatives of the program variables which are used in KHPCG. Subroutine RED calculates the reduction in the vertical entrainment coefficient as a function of local Richardson's number.

Subroutine OUTP converts the desired variables to dimensional quantities and prints them out at each integration step.

INPUT AND OUTPUT

Input is interactive with the terminal. The user answers prompts. After a run, the user is given a chance to change any of the input variables and repeat a run. The input variables required are:

1) A run title (one line)

2) The discharge flow rate in m^3/s

3) The discharge channel width, m

4) The discharge channel depth (assumed rectangular), m

5) The ambient current, m/s

6) Discharge angle relative to the current, degrees (zero is downstream and 90 is perpendicular to current)

7) Discharge water temperature, C

8) Ambient water temperature, C

9) Salinity, ppt (both discharge and ambient assumed the same). If fresh water, use a small salinity such as 0.01.

10) Surface heat transfer rate (you are given three choices, mild, average, or severe. This has little effect on temperatures or dilution in the near field)

11) The distance downstream where simulations are to stop. (The program stops when this value is reached or you have 20 pages of output or when the excess temperature is .005 of the initial excess temperature.)

The output consists of the following:

1) X, distance downstream in the direction of current, m.

2) Y, distance perpendicular to current, m

3) Centerline excess temperature (temperature above ambient), C.

4) Time of travel to this point along centerline, sec.

5) Q/Q0, average plume dilution, total flow in plume divided by discharge flow.

6) QM/Q0, minimum dilution at plume centerline.

7) DEPTH, plume depth, m ($2^{1/2}$ standard deviations of Gaussian curve)

8) WIDTH, plume with, m ($2x2^{1/2}$ standard deviations of a Gaussian curve)

9) AREA, surface area within specified excess temperature isotherms, m^2. These are given for each 1/2 degree of excess temperature. (If the simulation is terminated before a particular isotherm is reached on the plume centerline, only a partial area is given)

PLEASE NOTE: The program does not keep track of receiving water boundaries, i.e., shore or bottom. The user must check to see that the plume does not attach itself to either shore or bottom by following the trajectory, plume width and depth. Simulations beyond these attachment points are in error due to changes in entrainment.

The program has been tuned to a wide variety of data and agrees with the average of these data as outlined in the reference given at the beginning of this paper. It may not agree exactly to each specific case but should give reasonable answers if boundaries are not encountered.

The program checks its own accuracy at each integration step. If the accuracy is poor, it repeats the calculations for that step with the step size cut in half. If the accuracy is still poor, it continues to half the step size until it reaches satisfactory accuracy. If it cuts the step size in half 11 (eleven) times and still cannot achieve specified accuracy the program stops. This is usually when the user has input bad data. It also occasionally occurs for certain combinations of current, density difference, and discharge direction. When it does, try changing input variables a small amount.

RUNNING THE PROGRAM

1. Copy the PDS.EXE file on one of the source disks to one of your formatted working disks or to your hard drive.

2. Store the source disks in a safe place.

3. Put the working disk in your computer's A: drive if you are working from floppies and type A:PDS.

   If you are working from your hard drive, change your directory to the directory where you copied PDS.EXE and type PDS.

4. Answer all questions regarding for your particular discharge and ambient conditions.

5. When the program prompts you for an output file, type in the name of an <u>non existing</u> file where you want the output to be stored. If you do not want the output stored and just want to see the results, type CON or TTY. This sends the output to the screen. You can also send the screen output simultaneously to the printer by pressing CTRL and PRINT SCREEN at the same time before running the program.

6. At the end of the run, the program will prompt you for further input if you want to change any of the existing input variables. If so, make the changes and it will re-run the previous case with your new changes.

7. If you want a hard copy of your output when you are finished and have not done so while running, type <u>COPY *filename* PRN</u> where filename is the name of the output file you specified.

12. A COMPUTATIONAL MODEL FOR SHORE-ATTACHED PLUMES IN MEANDERING RIVERS

32

12. A COMPUTATIONAL MODEL FOR SHORE-ATTACHED PLUMES IN MEANDERING RIVERS

A Computational Model for Shore-Attached Plumes in Meandering Rivers

by

Jacques DE LA TORRE

A PROJECT
submitted to
Oregon State University

in partial fulfillment of
the requirements for the
degree of
Master of Science
Completed February 2, 1990

ACKNOWLEDGMENT

I wish to express my sincere appreciation to my major professor, Dr. Lorin R. Davis, for his continued advise, consul, support and encouragement during the course of this work.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF SELECTED SYMBOLS

a = initial dilution factor;
A = area of river flow cross section;
B = width of river flow cross section;
C = concentration, by weight, of dispersing sudstance;
$\bar{C}$ = average concentration;
C' = deviation from average concentration;
C_i = excess concentration after initial dilution;
C_o = initial excess concentration;
d = local depth flow;
$\bar{d}$ = average depth of river flow cross section;
D = transverse diffusion factor;
E_x = longitudinal mixing coefficient;
E_z = overall transverse mixing coefficient;
g = acceleration due to gravity;
$K_{x,z}$ = convective dispersion coefficients;
$m_{x,y,z}$ = metric coefficients for orthogonal curvilinear coordinate system;
M = rate of change of mass of a tracer mixed with water;
n = Manning's coefficient;
p = normalized cumulative discharge;
P = fraction of river flow occupied by plume;
q = discharge per unit width;
$\bar{q}$ = width-averaged discharge per unit width;
q_c = cumulative discharge from near bank;
Q_o = volumetric discharge rate of thermal effluent;
Q_r = river flow rate;
r_c = radius of curvature of river bank;
S = river slope;

$x_{1,2,3}$ = length coordinates in a rectangular coordinate system;
x,y,z = length coordinates in longitudinal, vertical and transverse directions, respectively;
x_o = position of virtual source;
u,v,w = local velocities in x, y and z directions, respectively;
u',v',w' = deviation from average values of u, v and w;
$\bar{u}$ = cross-sectional average value of u;
u^* = shear velocity;
Y,Y_s,Y_b = values of vertical distance y at flow boundaries;
α = dimensionless transverse mixing coefficient;
$\varepsilon_{1,2,3}$ = local turbulent mass transfer coefficients in a rectangular coordinate system, also called turbulent diffusion coefficients;
$\varepsilon_{x,y,z}$ = turbulent diffusion coefficients in x, y and z direction, respectively, in a orthogonal curvilinear coordinate system;
ρ = mass density of fluid;
σ_p = variance of the normal probability density function, Eq. 30;

I. INTRODUCTION

The water pollution problem, created by continuing population growth and industrial development, menaces the balance of the ecosystem Earth. To maintain a liveable environment, man must apply effective measures of water pollution control and should try to predict the ecological consequences of the discharges of waste heat into various bodies of water.

Rivers and streams are mostly used as conveyance channels for the disposal of industrial, agricultural and domestic wastes. Waterways have traditionally performed this function for many centuries. If pollution is not controlled, however, the availability of water for other more important uses may be sharply reduced. In order to control pollution, the release of potentially harmful contaminants and heated effluents into waterways must be regulated so as not to exceed the capacity of the stream to maintain the concentrations of contaminants and overall average water body temperature rise within permissible limits. Water pollution is a very complicated problem that involves many fields in science and engineering. In this study, the lateral mixing of a neutrally-buoyant pollutant in an idealized fixed-boundary meandering channel flow is investigated for the purpose to improving the understanding of the hydrodynamic behavior of polluants in natural streams.

Natural mixing due to turbulent and convective transfer play an important role in river ecology. The ability to predict lateral mixing is important to water pollution control. It directly affects the location and design of effluent systems. Furthermore, longitudinal dispersion depends on the combined effects of longitudinal convection and lateral mixing. However, most of the past studies have been limited to the idealized geometries of straight channels and long continuous bends, and are therefore inadequate for describing the lateral mixing phenomenon in natural streams.

In this study, a computer program, capable of taking the lateral mixing phenomenon in any channel shape into account, will be presented. It can be used to determine the maximum shoreline concentration, the contaminated surface area and the maximum length of a mixing zone given within specified isopleths discharge conditions.

II. REVIEW OF FORMULATION

1. Comment of Formulation

The formulation of the equations for thermal and mass diffusion are the same, except heat can be dissipated at a free surface into air by convection. This difference will be discussed later. In the following derivations we shall use the concept of dispersion to formulate a plume pattern in a river without the consideration of convective heat dissipation into air.

2. Convection-Diffusion Equation

To derive the dispersion equation, one of the most direct theoretical ways is based on the principle of conservation of mass. Consider an arbitrary volume as illustrated on Fig. 1 in a flow field of which dV is a volume element, dS is an element of surface area, where $\vec{n}$ = unit normal vector directed outward from surface of control volume, and $\vec{u}$ defines the velocity field. The rate of change of mass M of a tracer mixed with water in the control volume is

$$\frac{dM}{dt} = \frac{\text{Rate of change of mass}}{\text{due to convection}} + \frac{\text{Rate of change of mass}}{\text{due to diffusion}}$$

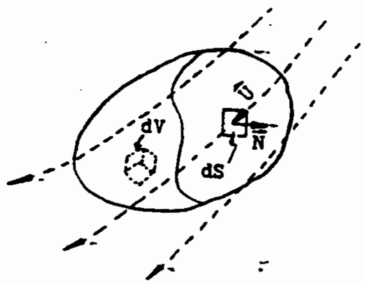


Fig. 1. **Control Volume of A Flow Field**

Where **M** is the mass of a conservative trace. Expressed in integral form

$$\frac{dM}{dt} = -\rho \int_S C\,\vec{u}\cdot\vec{n}\,dS + \rho\int_S \varepsilon_m \nabla C\cdot\vec{n}\,dS \qquad (1)$$

where ρ = mass density of dispersing substance, C = its concentration by weight, and ε_m = molecular diffusion coefficient. Noting that

$$\frac{dM}{dt} = \rho\frac{d}{dt}\int_V C\,dV = \rho\int_V \frac{\partial C}{\partial t}\,dV, \qquad (2)$$

and applying the divergence theorem whereby

$$\int_S C\,\vec{u}\cdot\vec{n}\,dS = \int_V \nabla\cdot(C\,\vec{u})\,dV \qquad (3)$$

and

$$\int_S \varepsilon_m \nabla C\cdot\vec{n}\,dS = \int_V \varepsilon_m \nabla^2 C\,dV \qquad (4)$$

Then the equation for the conservation of mass of the dispersant becomes

$$\frac{\partial C}{\partial t} + \nabla.\,C\,\vec{u} = \varepsilon_m \nabla^2 C \quad . \qquad (5)$$

But, if incompressible flow is assumed $\nabla\vec{u} = 0$, so the above equation further reduces to

$$\frac{\partial C}{\partial t} + \vec{u}.\nabla C = \varepsilon_m \nabla^2 C \qquad (6)$$

For turbulent flows, it is customary to resolve the instantaneous concentration and velocity into sums of time averaged and fluctuating components, employing the Reynolds averaging procedure

$$C = \overline{C} + C' \qquad (7)$$
$$u_i = \overline{u_i} + u_i'$$

where the subscript i denotes the i'th coordinate direction. The averaging period is sufficiently long to permit the averages of C' and u_i' to converge to zero, but not so long as to significantly damp the variation of $\overline{C}$ with respect to time. Also in turbulent flows, the contribution of molecular diffusion is usually negligible to the mixing process. So that it is customary to discard the molecular diffusion term. Eq. 6 for turbulent flows then becomes, in Cartesien tenser form,

$$\frac{\partial \overline{C}}{\partial t} + \overline{u}_i \frac{\partial \overline{C}}{\partial x_i} = -\frac{\partial}{\partial x_i} \overline{C' u_i'} \qquad (8)$$

where x_i represents distance and $i = 1, 2, 3$, indicates the direction in a rectangular coordinate system, respectively. By analogy with the Boussinesq eddy viscosity concept, the concentration velocity covariance is usually represented as a gradient-type transfer term

$$\overline{C' u_i'} = -\varepsilon_{ij} \frac{\partial \overline{C}}{\partial x_j} \qquad (9)$$

If it is assumed that the principal axes of the turbulent diffusion tensor ε_{ij} coincides with the axes of the coordinate system, Eq. 9 can be simplified to

$$\overline{C' u_i'} = -\varepsilon_i \frac{\partial \overline{C}}{\partial x_i} \qquad (9\text{-}a)$$

where ε_i is a scalar quantity, interpreted as a local coefficient for the turbulent transfer of mass in the i'th direction.

Substitution of Eq. 9-a into Eq. 8 , expansion to conventional rectangular Cartesien notation, and dropping the averaging bars, results in the usual form of the general convection-diffusion equation for a conservative substance in a turbulent open-channel flow

$$\frac{\partial C}{\partial t} + u_i \frac{\partial C}{\partial x_i} = -\frac{\partial}{\partial x_i}\left(\varepsilon_i \frac{\partial C}{\partial x_i}\right) \qquad (10)$$

Now, consider the dispersion process for a neutrally buoyant dispersant under conditions of uniform flow in a straight channel of constant cross section. Choosing a coordinate system such that the indices $i = 1, 2, 3$ indicate respectively the direction of flow, the direction normal to the channel bed and the transverse direction as show on Fig. 2

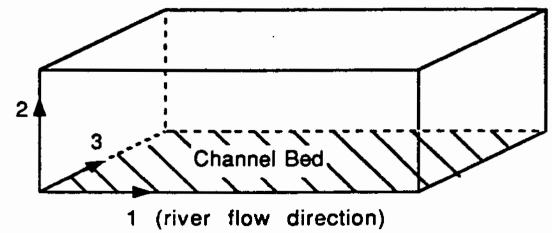


Fig. 2 Indications of a Rectangular Coordinate System

For a uniform flow in a straight channel of constant rectangular cross section, the basic flow and turbulent structure do not change with respect to the direction of the flow, and the velocity components u_2 and u_3 are of secondary importance. Furthermore, it is to be expected that ε_1 is independent of x_1, although it may vary with x_2 and x_3.

Finally, the dispersion equation can be rewritten as

$$\frac{\partial C}{\partial t} + u_1 \frac{\partial C}{\partial x_1} = \varepsilon_1 \frac{\partial^2 C}{\partial x_1^2} + \frac{\partial}{x_2}\left(\varepsilon_2 \frac{\partial C}{\partial x_2}\right) + \frac{\partial}{\partial x_3}\left(\varepsilon_3 \frac{\partial C}{\partial x_3}\right) \qquad (11)$$

The next step is to change the Convection-Diffusion equation by using orthogonal curvilinear coordinate instead of the Cartesian Coordinate system.

3. Orthogonal Curvilinear Coordinate System

A schematic of this system shown in Fig. 3, is set up by three mutually orthogonal sets of coordinate planes which were called longitudinal, transverse, and horizontal coordinate surfaces by Yotsukura And Sayre [9].

For a natural river channel, the longitudinal and transverse coordinate surfaces are vertical, typically curved, and nonparallel. But the horizontal coordinate surfaces are all parallel horizontal planes. For the best advantage, it is suggested to align the longitudinal coordinate surfaces , at least approximatly, in the direction of the depth-averaged total velocity vectors.

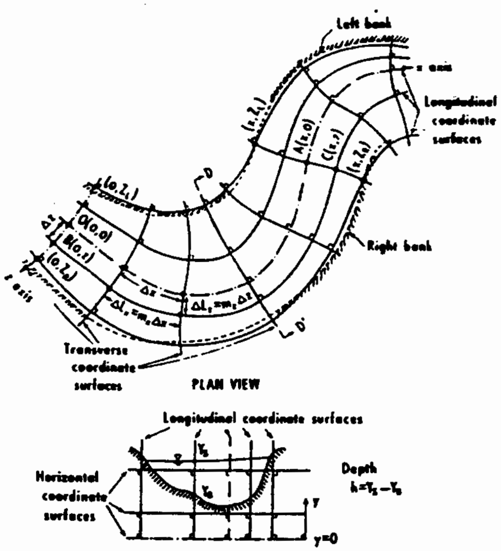


Fig. 3 Orthogonal Curvilinear Coordinate System for a Natural Channel. [9]

The origin 0, is located at the intersection point of three specially selected coordinate surfaces as already shown in the Fig. 3. The X-axis is defined as the intersection of the longitudinal and horizontal coordinate surfaces (positive in the downstream direction). The Z-axis is the intersection of the transverse and horizontal coordinate surfaces (positive to the right bank), and the Y-axis is the intersection of the transverse and longitudinal surfaces (positive in the upward direction). The x and z coordinates of a point C in the Fig. 3 are defined as the horizontal distances L_{oa} along the X-axis, and L_{ob} along the Z-axis, respectively. Therefore all points on a transverse coordinate surface have a common coordinate value x, and similarly, all points on a longitudinal coordinate surface have a common coordinate value z. The coordinate value for y is measured vertically upward from the X-Z plane.

Because of curvature in the channel alignment and/or variations in width along the channel, horizontal distances, measured along different longitudinal (or transverse) coordinate surfaces from one transverse (or longitudinal) coordinate surface to another, are in general not equal. In order to take care of this situation, we may introduce the metric coefficients m_x, and m_z to correct the differences between distances along the respective axes. As illustrated in Fig. 3, the horizontal distance along the longitudinal coordinate surface from B to C is obtained by

$$L_{bc} = \int_0^x m_x \, dx$$

and the one along the transverse coordinate surface from A to C by

$$L_{ac} = \int_0^z m_z \, dz \quad .$$

The values of m_x and m_z may vary from point to point and are in general function of both x and z, except that $m_x = 1$ on the X-axis and $m_z = 1$ on the Z-axis. Other values of m_x and m_z can be estimated from a sketch like the upper one in Fig. 3 by the relationships $m_x = (\Delta L_x)/\Delta X$ and $m_z = (\Delta L_z)/\Delta Z$, where ΔX and ΔZ are distances along the respective axes.

In the curvilinear coordinate system for a meandering channel the metric coefficients are

$$\left.\begin{aligned} m_x &= 1 + z/r_c && \text{for a bend curving to the left} \\ &= 1 && \text{for a straight reach} \\ &= 1 - z/r_c && \text{for a bend curving to the right} \end{aligned}\right\} \quad (13)$$

$$m_y = m_z = 1$$

where r_c is the radius of curvature of the bend at the channel axis and the longitudinal coordinate surfaces are circular cylinders concentric with the X-axis. The positive sign in Eq. 13 is for bends curving to the left looking downstream and the negative one is for bends curving to the right.

Experiments show that the metric coefficient m_z in a channel with converging or diverging banks has an approximate relationship which is expressed as

$$m_z = B/B_o$$

where B is the channel width measured along a transverse coordinate surface of interest and Bo is the width along the Z-axis, wherein the longitudinal coordinate surfaces are more or less equally spaced. Furthermore, if the angle of convergence or divergence between two adjacent longitudinal coordinate surfaces is less than 30º, the distance between these surfaces along a curved transverse coordinate surfaces is closely approximated by the linear distance between the intersection points.

In Eq. 13 it is interesting to observe that when r_c goes to infinity, that is in a straight channel, m_z reduced to 1, which means that the coordinate system reduces to a rectangular cartesien coordinate system.

When we switch the diffusion-dispersion equation from the Cartesian Coordinate system to the Orthogonal Curvilinear Coordinate system, we should note that integration or differentiation of a function along a curved coordinate surface is always related to an infinitesimal distance; in this case , dL_x, dL_y, dL_z are equal to $m_x dx$, $m_y dy$, $m_z dz$, respectively. We may see the main advantage of the orthogonal Curvilinear Coordinate system over the Cartesian Coordinate system is that the diffusion-dispersion equation for natural channels can be expressed in a simpler and more rigorous representation. So after switching the coordinate, the diffusion-dispersion equation is given by

$$m_x m_z \frac{\partial C}{\partial t} + \frac{\partial}{\partial x}(m_z u' C) + m_x m_z \frac{\partial}{\partial y}(v' C) + \frac{\partial}{\partial z}(m_x w' C)$$
$$= \frac{\partial}{\partial x}\left(\frac{m_z}{m_x}\varepsilon_x \frac{\partial C}{\partial x}\right) + m_x m_z \frac{\partial}{\partial y}\left(\varepsilon_y \frac{\partial C}{\partial y}\right) + \frac{\partial}{\partial z}\left(\frac{m_x}{m_z}\varepsilon_z \frac{\partial C}{\partial z}\right) \quad (14)$$

the variables u', v', w' are local velocity components in the curvilinear coordinate system; ε_x, ε_y, ε_z are local turbulent mass diffusivities, where the subscripts x, y, z indicate alignment with longitudinal coordinate surfaces, vertical direction and transverse coordinate surfaces, respectively.

4. Two Dimensional (Depth-Average) Equation

After integrating Eq. 13 over the depth of flow from the bed $Y_b(x,z,t)$ to the water surface Y_s, the two dimensional convection-diffusion equation can be obtained. The process of this integration was carried out by Leenderste [3] and Holley [2] by using Leibnitz's rule. Neglecting the detail of the integrations, the resulting depth-integrated equation is

$$m_x m_z \frac{\partial}{\partial t}(\overline{d}\,\overline{C}) + \frac{\partial}{\partial x}(m_z \overline{d}\,\overline{uC}) + \frac{\partial}{\partial z}(m_x \overline{d}\,\overline{wC}) = \frac{\partial}{\partial x}\left(\frac{m_z}{m_x}\overline{d}\,E_x\frac{\partial \overline{C}}{\partial x}\right) + \frac{\partial}{\partial z}\left(\frac{m_x}{m_z}\overline{d}\,E_z\frac{\partial \overline{C}}{\partial z}\right) \quad (15)$$

In the previous equation $\overline{d}$ is the averaged-depth across the channel and E_x may be designated as the longitudinal mixing coefficient, which includes the combined effect of depth-averaged turbulent diffusion and convective dispersion, i.e., $E_x = \varepsilon_x + K_x$. E_z is defined in the same way. Like before, all terms with a bar are designed as the depth-averaged quantities.

Before we introduce the cumulative discharge system, we would like to eliminate some terms from Eq. 15.

1. For most discharge effluent cases, the operations are usually under steady state conditions. In this paper we are not interested in involving complicated unsteady state cases. So the time derivative terms can be dropped from Eq. 15.
2. It is well known that the dispersive transport, $(E_x/m_x)dC/dx$, is relatively small compare to the convective transport vC in an unidirectional flow. Furthermore, Sayre and Chang [6] proved analytically that longitudinal dispersion has very little influence on the steady state mixing process in a straight, rectangular channel, except near the source.

Under steady state conditions, the longitudinal mixing term and the time derivative term both can be dropped out from Eq. 15, yielding: .

$$\frac{\partial}{\partial x}(m_z \overline{d}\,\overline{uC}) + \frac{\partial}{\partial z}(m_x \overline{d}\,\overline{wC}) = \frac{\partial}{\partial z}\left(\frac{m_x}{m_z}\overline{d}\,E_z\frac{\partial \overline{C}}{\partial z}\right) \quad (16)$$

Obviously, $\overline{u}, \overline{w}, \overline{C}$, become not only depth-averaged but time-averaged quantities. And now let us introduce the space cumulative discharge system to simplify Eq. 16 to a more condense form.

5. Space Cumulative Discharge System

A solution of the equations given in previous sections is given in P. P. Pailey and W. W. Sayre [4]. It is repeated here with few modifications for completeness.

It is very helpful to simplify the dispersive convective equation if cumulative discharge system were used. The space cumulative discharge is defined as

$$q_c = \int_{z_1}^{z} m_z \overline{d}\,\overline{u}\,dz \quad (17)$$

where $m_z \overline{d}\,\overline{u}$ represents the unit width local discharge which was suggested by Yotsukura and Sayre [9]. The integrated steady-state depth-averaged continuity equation can be expressed as

$$\frac{\partial}{\partial x}(m_z \overline{d}\,\overline{u}) + \frac{\partial}{\partial z}(m_x \overline{d}\,\overline{w}) = 0 \quad (18)$$

First integrate the continuity equation with respect to z from the left bank z_1 to z to obtain

$$m_x \overline{d}\,\overline{w} = -\frac{\partial}{\partial x}\int_{z_1}^{z} m_z \overline{d}\,\overline{u}\,dz = -\frac{\partial q_c}{\partial x} \quad (19)$$

by making use of the Leibnitz's rule and the conditions that $\overline{u} = \overline{w} = 0$ at z_1 with z independent of x. Next substituting the redefined integrated continuity equation and cumulative discharge system equation into Eq. 16 we get

$$m_z \overline{d}\,\overline{u} \frac{\partial \overline{C}}{\partial x} - \frac{\partial q_c}{\partial x} \frac{\partial \overline{C}}{\partial z} = \frac{\partial}{\partial z}\left(\frac{m_x}{m_z} \overline{d}\, E_z \frac{\partial \overline{C}}{\partial z} \right) \qquad (20)$$

Finally, let $\overline{C}(x,z)$ be replaced by $\overline{C}(x,q_c)$ by using the chain rule of partial derivatives so that Eq. 20 reduces to

$$\frac{\partial \overline{C}}{\partial x} = \frac{\partial}{\partial q_c}\left(m_x \overline{d}^{\,2} \overline{u} E_z \frac{\partial \overline{C}}{\partial q_c} \right) \qquad (21)$$

The above equation can be normalized by substituting $p = q_c/Q_r$ into Eq. 21. Thus a normalized diffusion equation can be obtained as

$$\frac{\partial \overline{C}}{\partial x} = \frac{\partial}{\partial p}\left(\frac{\overline{d}^{\,2} \overline{u} E_z}{Q_r^2} \frac{\partial \overline{C}}{\partial p} \right) \qquad (22)$$

where Q_r is the total river flow rate at the location. If we assume that

$$\overline{d}^{\,2} \overline{u} E_z = Q_r^2 D$$

in which D = transverse diffusion factor, a constant to be discussed in the following section, the dispersion-convection equation can be more simplify to the form of a simple one-dimensional diffusion equation

$$\frac{\partial C}{\partial x} = D \frac{\partial^2 C}{\partial p^2} \qquad (23)$$

6. Numerical Prediction of Concentration Distribution

The profiles for velocity and concentration distribution are assumed to be described by the Gaussian function

$$f_r(p-\zeta\,;x) = \frac{1}{\sqrt{2\pi}} e^{-(p-\zeta)^2} \qquad (24)$$

The general form of the solution for a constant effluent discharge from a relative source strength distribution function $C_i(\zeta,0)$ at $x = 0$ is given by

$$C(p,x) = \int_{p=0}^{p=1} C_i(\zeta,0)\, f_r(p-\zeta\,;x)\, d\zeta \qquad (25)$$

in which $f_r(p-\zeta\,;x)$ is given in Eq. 24 for a continuous source of unit strength concentrated at $p = \zeta$, $x = 0$. The initial dilution factor for nearfield mixing is defined as :

$$a = \frac{C_0}{C_i} \qquad (26)$$

where C_0 is the initial concentration of the discharge plume at the origin of coordinates, $x = 0$, $p = 0$, at the discharge bank of the river, this is just far enough downstream from the outfall so that at the origin the initial nearfield mixing has taken place. The fraction of the total river flow rate initially mixed with the discharge effluent at the origin is

$$P = a \frac{Q_0}{Q_r} \qquad (27)$$

in which Q_0 = volumetric rate of effluent discharge, Q_r = total river flow rate. Refering to Fig. 4 in which the variable ζ represents the displacement from the origin in the p direction (obviously $p = 0$ at the outfall discharge bank, $p = 1$ at the opposite bank),

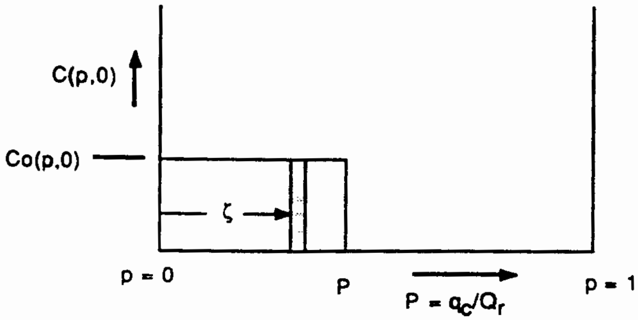


Fig. 4 Definition Sketch for Initial Conditions

then the initial transverse concentration distribution at the origin $x = 0$, is

$$C_i(\zeta,0) = \frac{C_0}{a} \quad ; \quad 0 < \zeta \le P$$
$$C_i(\zeta,0) = 0 \quad ; \quad P < \zeta \le 1 \tag{28}$$

Inserting the initial conditions specified by Eq. 28 into Eq. 25, yields:

$$C(p,x) = \int_0^P f_r(p-\zeta\,;\,x)\,d\zeta \tag{29}$$

Since the plume is assumed shore attached, the momentum of the plume may be reflected from the banks, then the term inside the

integral of Eq. 28 expressed in normal standardized form as

$$f_r(p-\zeta\,;\,x) = \frac{1}{\sigma_p}\left(\frac{1}{\sqrt{2\pi}}\exp\left(-\frac{s^2}{2}\right)\right) \tag{30}$$

in which $s = (p-\zeta)/\sigma_p$ = the standardized normal variable, and $\sigma_p = \sqrt{2Dx}$ = the standard deviation in the p-domain. Secondly, if reflections from the banks are taken into account, the method of images may be used to account for it. Considering infinite image sources and taking account of the ambient conditions, the general form for such solution is

$$C(p,x) - C_a = \frac{(C_0-C_a)}{a}\left(\left(F_r\left(\frac{p+P}{\sigma_p}\right) - F_r\left(\frac{p-P}{\sigma_p}\right)\right) + \sum_{n=1}^{\infty}\left(F_r\left(\frac{2n+(p-P)}{\sigma_p}\right) - F_r\left(\frac{2n-(p-P)}{\sigma_p}\right)\right) + \sum_{n=1}^{\infty}\left(F_r\left(\frac{2n-(p+P)}{\sigma_p}\right) - F_r\left(\frac{2n+(p+P)}{\sigma_p}\right)\right)\right) \tag{31}$$

in which C_a = ambient concentration and $F_r(\ )$ = the standardized cumulative normal distribution function corresponding to the probability function $f_r(\ )$.

A fully exact solution for finite width requires an infinite number of image sources spaced at normalized distance $p = 2$. But all these image (reflection) sources are not believed necessary, however. As a matter of fact, the terms in the infinite series representing the bank reflections are quite insignificant if $n > 4$ or 5.

7. Maximum Center Line Concentration

The maximum center line concentration Cmax(p,x), for a shore attached-plume occurs along the near shore, i.e., p = 0. The solution for ($C_{max}(0,x)$ - C_a) from Eq. 31 is

$$C_{max}(0,x) - C_a = \frac{(C_0 - C_a)}{a}\left(\left(2F_r\left(\frac{P}{\sigma_p}\right) + 2\sum_{n=1}^{\infty}\left(F_r\left(\frac{2n+P}{\sigma_p}\right) - F_r\left(\frac{2n-P}{\sigma_p}\right)\right) - 1\right) \quad (32)$$

8. Transverse Diffusion Factor D

We assumed previously the transverse diffusion factor D appearing in Eq. 22 to be a constant to simplify the analysis as well as the formulation. In fact, Yotsukura and Cobb [8] have shown that variations in transverse distribution of D do not affect solutions of Eq. 22 as long as its average value

$$D = \frac{E_z}{Q_r^2}\int_0^1 d^2\, u\, dp \quad (33)$$

remains fairly constant. The approximation of the integral in Eq. 33 is

$$\int_0^1 d^2\, u\, dp = \bar{d}^2\, \bar{u} = \bar{d}\, \bar{q} \quad (34)$$

in which as before $\bar{d} = A_C/B$ = average depth of flow section, A_C and B = area and width of flow section respectively, $\bar{u} = Q_r/A_C$ = average

river flow velocity and $q = Q_r/B$ = width-averaged value of local unit discharge, q.

The final form of the relation for transverse diffusion factor can be written as

$$D = \frac{\alpha}{.063}\frac{n\sqrt{g}}{B^2}(\bar{d})^{5/6} \quad (35)$$

This was done using the Manning's formula for the river discharge, Q_r, and an empirical relationship for the transverse mixing coefficient, E_z, which are expressed as

$$Q_r = \frac{.063}{n}\bar{B}\,\bar{d}\,(\bar{d})^{2/3}(S)^{1/2} \quad (36)$$

and

$$E_z = \alpha\,\bar{d}\,u^* \quad (37)$$

respectively. In the foregoing equations, n = Manning's coefficient, S = slope of energy gradient, α = dimensionless transverse mixing coefficient, $u^* = (g\bar{d}S)^{1/2}$ = shear velocity and g = acceleration due to gravity. The value of D determined by Eq. 32 is less accurate compared to the one by Eq. 31, mainly because of the approximation used in Eq. 32. It is empirical work to evaluate the transverse mixing coefficient E_z or α, for a natural river. Experiments have shown that α depends upon the characteristics of the river channel and the ambient flow conditions, however there are as yet no reliable predictive relationship between them. The most common and reliable method for determining the values of E_z for natural channel is by laboratory simulation or direct field measurement followed by the use of empirical equations to match the measuring data. Several investigators have found values of α by the method just mentioned above. For example, Fischer [1] regressed the Missouri River data and got the following equation for the curve section

$$\alpha = 0.4\,(B/\bar{d})^2\,(\bar{u}/u^*)^2\,(\bar{d}/r_c)^2 \qquad (38)$$

If $r_c \rightarrow \infty$, which is the case of a straight section, we have $\alpha \rightarrow 0$. To overcome this problem it is necessary to have a limiting value of the radius of curvature. Hence, if $r_c \geqslant 25$ B, we set $r_c = 25$ B.

We should not forget that the value of the numerical coefficient in Eq. 38 may be different for other rivers.

When the total length of the river reach affected by the effluent discharge is large, it is necessary to divide it into several convenient subreaches to consider the change in value of D in the longitudinal direction. In order to have a continuous solution of C(p,x), the following relationship should be maintained

$$D_j\,(x_{j-1} - x_{oj}) = D_{j-1}\,(x_{j-1} - x_{oj-1}) \qquad (40)$$

in which D_j = value of D for the j'th subreach, and x_{oj} = longitudinal coordinate of the virtual source for the j'th subreach. The values of the x_{oj} for all subreaches are determined from Eq. 40 starting with $x_{oj} = 0$ for the first subreach, and using the known values of D for all subreaches. The values of D_j and x_{oj} for the j'th subreach are used to determined the standard deviation for that subreach by the relation

$$\sigma_p = \sqrt{2\,D_j\,(x_{j-1} - x_{oj})} \qquad (41)$$

which in turn is used in Eq. 31 to compute C(p,x) for the j'th subreach.

The space-dependance of the model should be noted in the formulation of the standard deviation, $\sigma_p \sim x^{1/2}$.

9. Initial Dilution Factor a

The value of the initial dilution factor a, is related to the design and orientation of the discharge outfall structure, the initial plume characteristics and the ambient flow properties. For no initial dilution, a is equal to 1, but can reach values of 4 in the case of an effluent not discharged parallel to the river flow and for an initial excess concentration at the outfall discharge C_o greater than the excess concentration after initial dilution C_i.

10. Isopleth (or Isotherm) Surface Area

The mixing zone is defined as the region enclosed by a line of specified concentration (isopleth) or temperature (isotherm) as shown in Fig. 5.

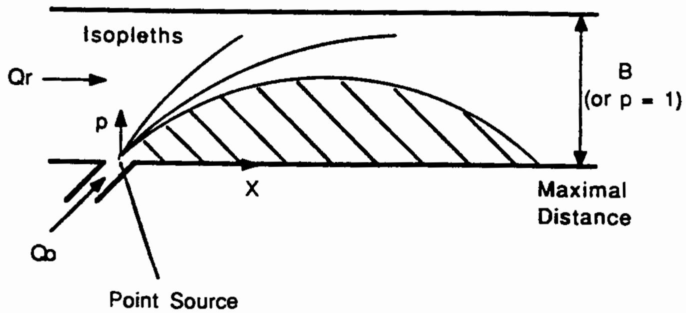


Fig. 5 Schematic view of effluent discharge and mixing zone along river bank

The computer program PSR listed in the Appendix II is based on Eq. 31 and 32. It can be used to find the concentration distribution in the shallow flow field. In addition, the program will determine the surfaces within areas of different specified dimensionless concentration contours contaminated by the effluent discharge. The computer program PSR is described in the following section.

III. COMPUTER PROGRAMMING OF THIS MODEL

1. Description

The computer program used to perform the computations of Eq. 31 and 32, is listed in the Appendix II. The language used is FORTRAN 77.

This computer program consists of the MAIN program and three subroutine subprograms, DIST, GEQUAT, ERFC. A brief description of the program and the relationships between the different parts of the program is given in Fig. 6, 7, 8 and 9.

Starting at 0.1 meter, the dimensional iterative increment ΔX, doubles until $x = 50$. Beyond that point it is fixed to 50 meters. For the last 50 meters and only during the determination of the Isopleth Surface Area, an increment of 5 meters is taken to find the final values, values for which the lateral distance of the plume reachs zero.

2. Inputs

To start the program two ways to enter the input parameters are offered :

1. interactive
2. using an input file.

The description of the input variables and the read-in format for the input file are the following :

1. Interactive :

   Start the program and follow the directives.

2. By Input File :

Input for PSR is arranged in three major groups as follows :

- Group 1. Title
- Group 2. Data
- Group 3. Characteristics of the effluent for each different subreach.

Each group must be separated by a blank line and arranged in the above order. The blank line signals the end of the input for the particular group.

The Input File must be available under the name "INPUT", before execution can begin.

TITLE *First group* *80-Characters*

Qo
Qr
Co *Second group*
Ca
A
NSUB

RAD_i B_i $DEPTH_i$ $SLOPE_i$ XL_i *Third group*

TITLE : The TITLE must consist of 80 characters or less.

Qo : Effluent Flow Rate (m^3/sec.)
Qr : River Flow Rate (m^3/sec.)
Co : Initial Excess Concentration
Ca : Ambient Concentration
A : Dilution Factor
NSUB : Number of Subreaches

RAD_i : Radius of Curvature of the i'th subreach (m)
B_i : Width of the ith subreach (m)
$DEPTH_i$: Averaged depth of the i'th subreach (m)
$SLOPE_i$: River Slope of the i'th subreach
XL_i : Downstream end of the i'th subreach (m)

The Second group contains 6 parameters, each on a different line. In the case of the Third group, the parameters for a specific subreach must appear on the same line and must be separated by at least one blank character. If the number or subreaches is j, the Third group will be constituted by j lines.

See next page for an example of Input file.

3. Example of Input File

Study of the McKenzie River *First group*

0.68
53.83
1.0 *Second group*
.0
1.0
9

0.	18.8	1.5	.001	20.9	
0.	12.5	1.5	.001	64.7	
0.	13.6	1.5	.001	114.8	
0.	16.7	1.5	.001	195.2	
300	18.8	1.5	.001	277.7	*Third group*
300	11.5	1.5	.001	294.5	
300	10.4	1.5	.001	325.0	
110	10.4	1.5	.001	328.9	
15	5.0	1.5	.001	329.3	

A zero in the input file, correspond to an infinit value of the radius of curvature.

4. Flow charts

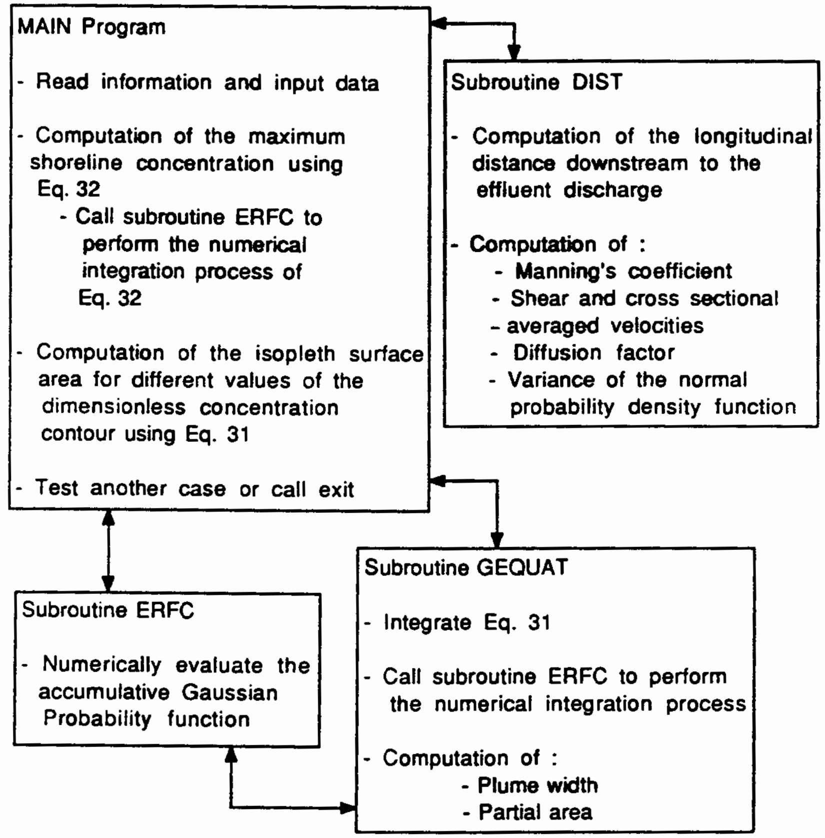
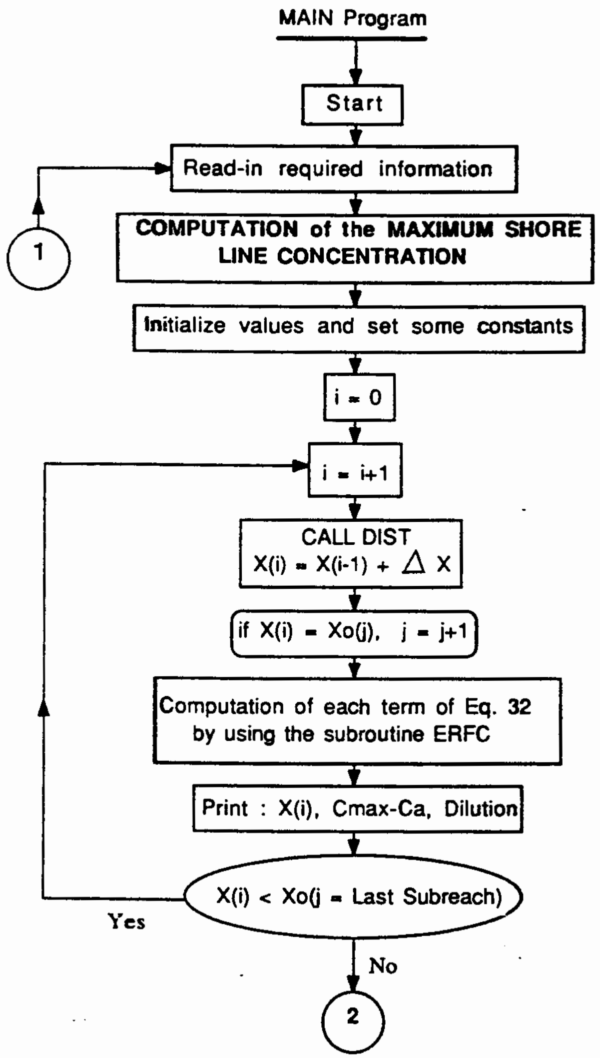
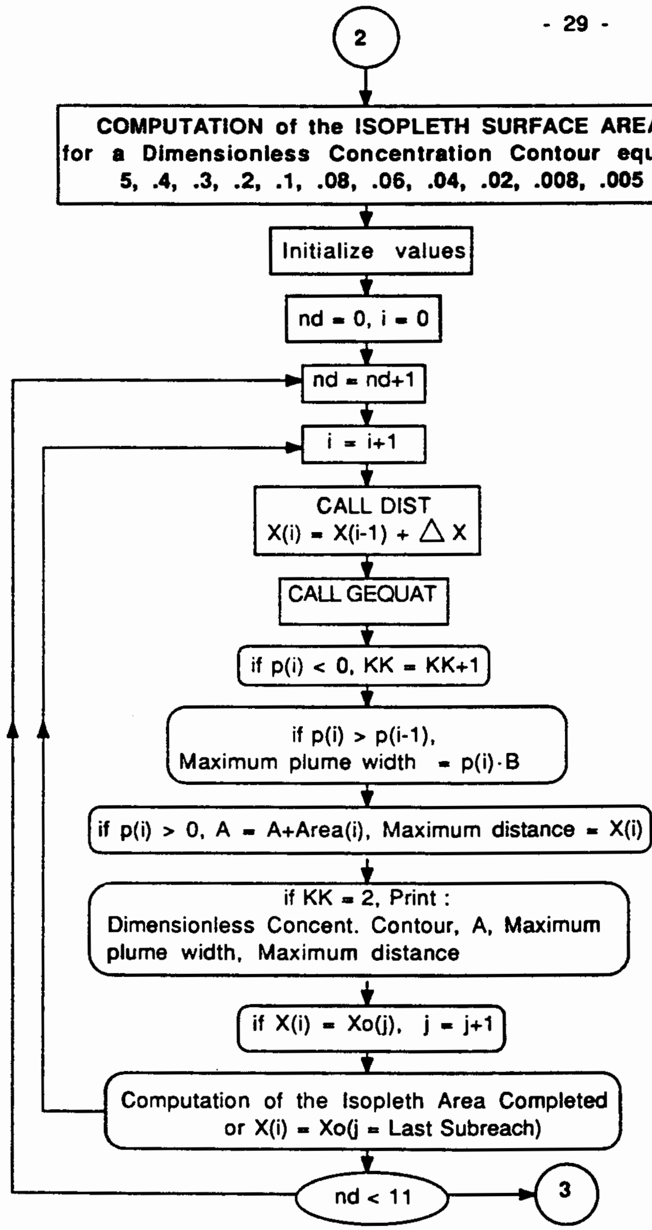


Fig. 6 General flow chart


MAIN Program
Start
Read-in required information
1
COMPUTATION of the MAXIMUM SHORE LINE CONCENTRATION
Initialize values and set some constants
i = 0
i = i+1
CALL DIST
X(i) = X(i-1) + △ X
if X(i) = Xo(j), j = j+1
Computation of each term of Eq. 32 by using the subroutine ERFC
Print : X(i), Cmax-Ca, Dilution
X(i) < Xo(j = Last Subreach)
Yes
No
2


2
COMPUTATION of the ISOPLETH SURFACE AREA for a Dimensionless Concentration Contour equal 5, .4, .3, .2, .1, .08, .06, .04, .02, .008, .005
Initialize values
nd = 0, i = 0
nd = nd+1
i = i+1
CALL DIST
X(i) = X(i-1) + △ X
CALL GEQUAT
if p(i) < 0, KK = KK+1
if p(i) > p(i-1),
Maximum plume width = p(i)·B
if p(i) > 0, A = A+Area(i), Maximum distance = X(i)
if KK = 2, Print :
Dimensionless Concent. Contour, A, Maximum plume width, Maximum distance
if X(i) = Xo(j), j = j+1
Computation of the Isopleth Area Completed or X(i) = Xo(j = Last Subreach)
nd < 11
3

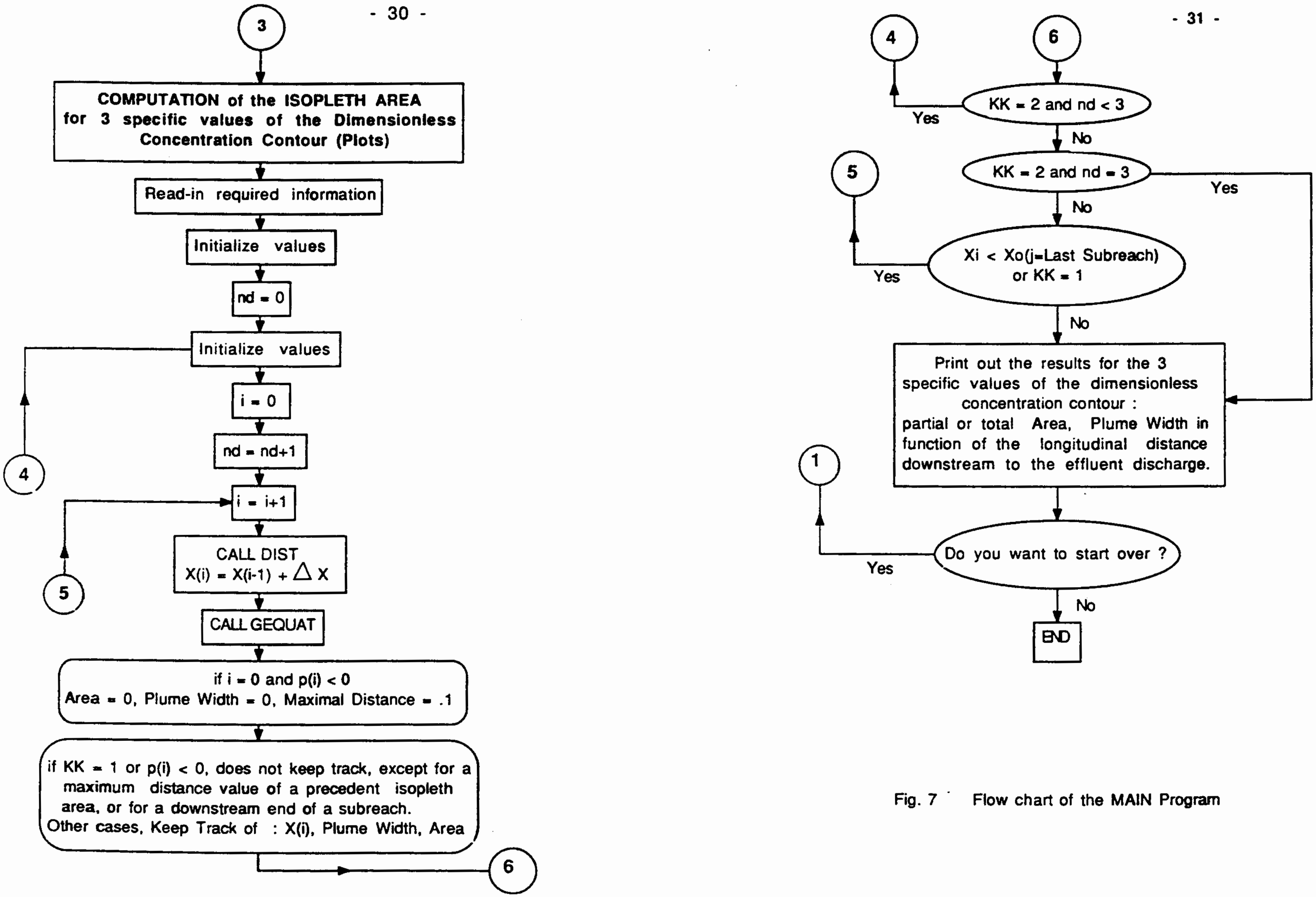


Fig. 7 Flow chart of the MAIN Program

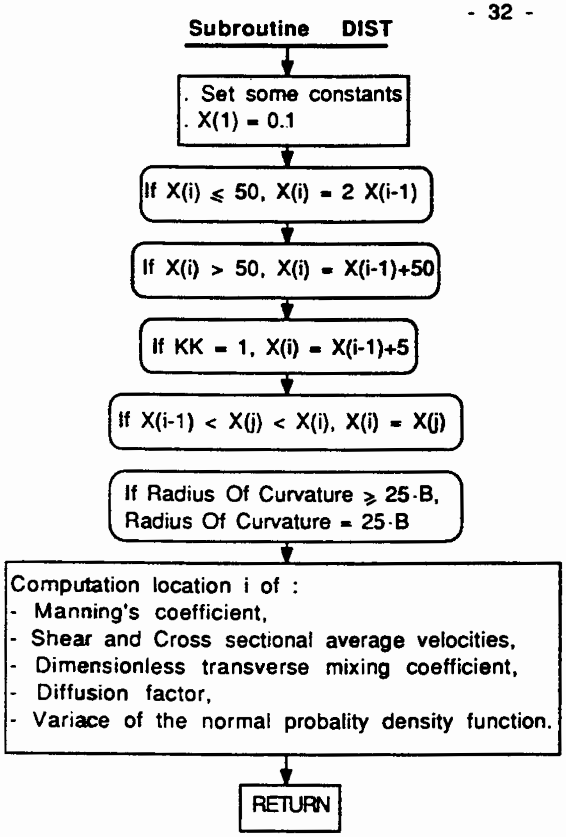


Fig. 8 Flow chart of the subroutine DIST

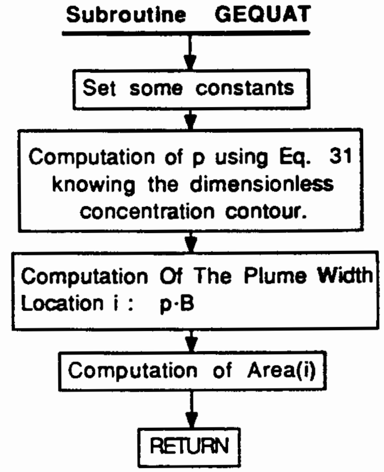


Fig. 9 Flow Chart of The Subroutine GEQUAT

IV. APPLICATION OF THIS MODEL

1. Procedures

The application of the present model is suggested using the following procedures :

(a) Measure the flow rates for both of the ambient river and the discharge effluent,

(b) Divide the flow field into several sections according to the geometric variations. The data of the radius of curvature, the width, the average depth and the slope of the river channel are needed at each location to take into account the effects of the geometric variations on dilution factor D,

2. Case study

We now analyse the Weyco Discharge in the McKenzie River near Springfield, Oregon, using the computer code. The slope of the river and the average depth are approximatively 1/1000 and 1.5 respectively. The radius of curvature, the width of the river and the distance from the origin to the downstream end of each subreach are shown on Table I. The computation for this case was performed for a 340 meter stretch of the river which was divided into 9 subreaches downstream from the discharge port as shown in the insert map in Fig. 10, supplied by R. H. Thut [7].

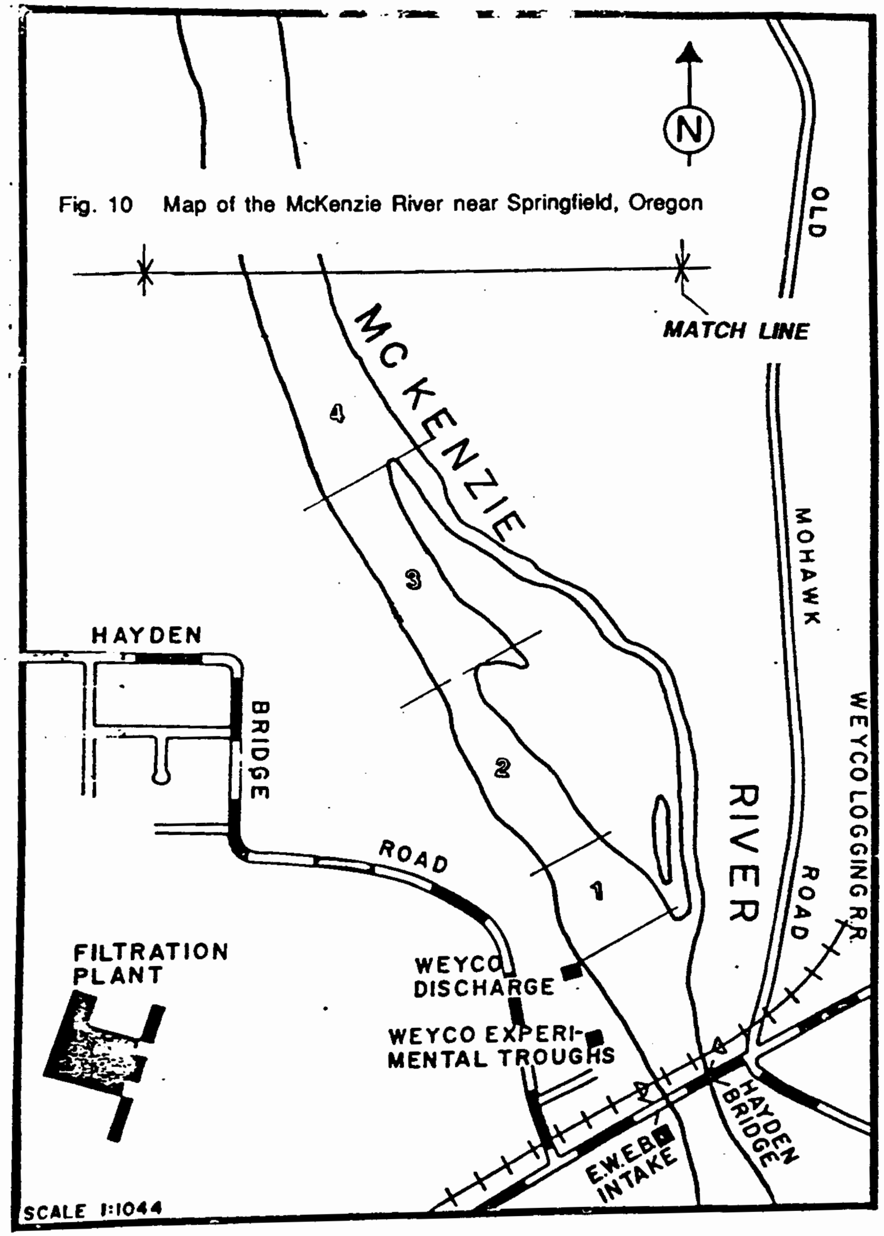


Fig. 10 Map of the McKenzie River near Springfield, Oregon

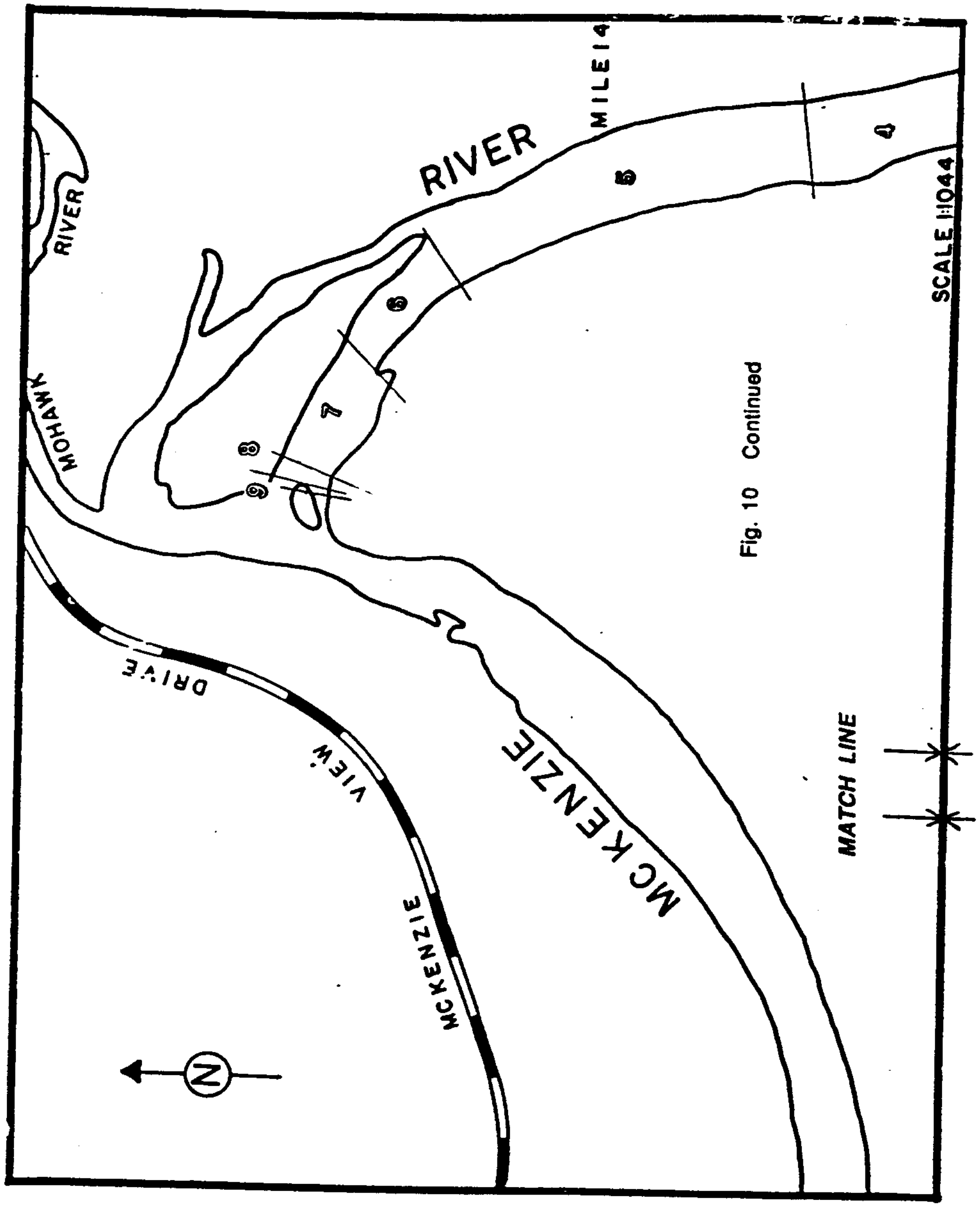


Fig. 10 Continued

Subreach Nr.	Radius curvature (Meter)	Width (Meter)	Downstream end (Meter)
1	0.	18.8	20.9
2	0.	12.5	64.7
3	0.	13.6	114.8
4	0.	16.7	195.2
5	300	18.8	277.7
6	300	11.5	294.5
7	300	10.4	325.0
8	110	10.4	328.9
9	15	5.0	329.3

Table 1 Measurement data for the McKenzie River case study

The object is to find the shoreline dilution as a function of distance downstream and the surface areas for different specified dimensionless concentration contours which are contaminated by the effluent discharge.

This case study is a typical illustration of the use of the model. The next section will discuss how well this model works and how accurately this model can predict plumes with shore attached center line discharge into shallow river channels.

V. RESULTS

1. Comparison of the predicted values with the field measured data

One major capability of this model is that it allows the inclusion of the effect of the cross channel flow induced by channel curvature. Even though the river flow bends weighted toward one bank or the other due to the geometric structure of the channel, the present model still can predict the expansions and contractions of the discharge plume. This feature is depicted in Table II and Fig. 11 which show the measured and predicted minimum dilution (inverse of concentration) along the near shore as a function of the longitudinal distance, of the Kraft Mill Effluent in the McKenzie river near Springfield, Oregon, December 9, 1976 [7]. The effluent flow rate and the river flow rate were 0.68 m^3/sec. and 53.83 m^3/sec. respectively, on this specific day.

By comparing the minimum dilutions along the shoreline with the field data, we find that the predicted model is fairly good approximation.

Distance downstream from source (Meter)	Minimum Dilution	
	Measured	Predicted
4.2	2	2.0
32.4	6	7.2
66.8	12	12.2
76.2	13	13.0
150.3	15	17.2
162.9	17	17.7
326.8	20	28.0
327.8	24	28.5
328.9	30	29.0

Table 2 . Comparison of measured and predicted minimum shoreline dilution.

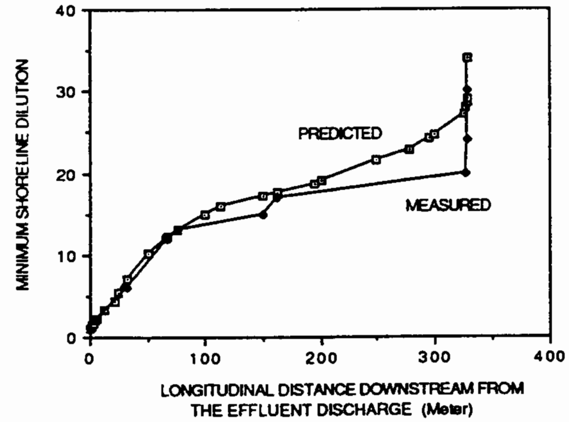


Fig. 11 Comparison of Measured and predicted minimum Shoreline Dilution.

Using the flow rate given and the characteristics of the channel shape, the output results generated by PSR program are listed in Appendix III.

Next we would like to show the isopleth curves on a strainghtend coordinate river channel plane as in Fig. 12. The contaminated surface areas can be used as a regulation for plant operations, either by changing the discharge flow rate into the ambient river or by chemical treatment which will attenuate the concentration of the discharge before it is dumped into the river. On the other hand, the ambient river flow rates are different from time to time, so the plant discharge may need to be shut down somewhat during the flow periods.

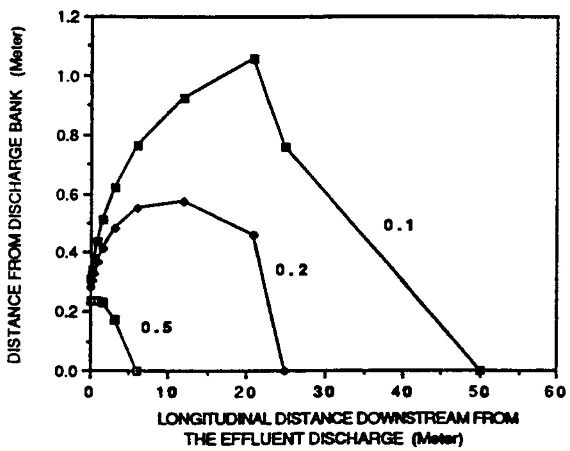


Fig. 12 Predicted Isopleth Areas for the McKenzie River near Springfield, Oregon, December 9,1976.

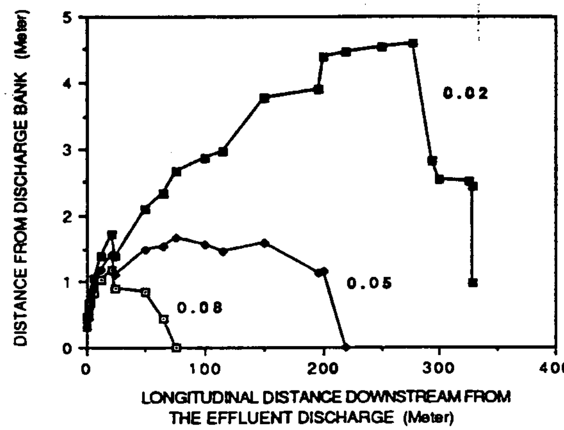


Fig. 12 Predicted Isopleth Areas for the McKenzie River near Springfield, Oregon, December 9,1976.

2. <u>Consideration of heat transfer from the stream surface</u>

Up to this point, we have not discussed the heat transfer between the river surface and the atmospheric air. For heated rivers resulting from thermal discharge from power plants and other artificial thermal sources, once complete mixing between the heated effluent and the cooler ambient river flow is achieved, the surface heat exchange becomes the dominant factor which controls the temperature distributions. A fully physical consideration of heat exchange at the interface of water-air is shown on Fig. 13.

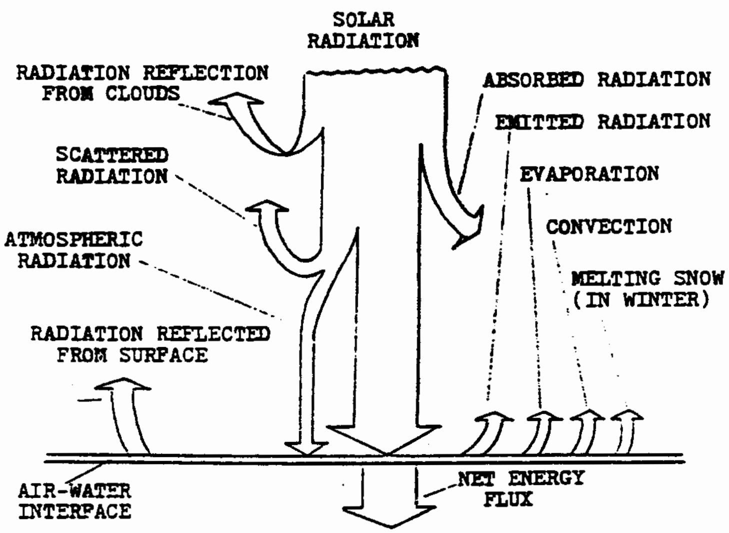


Fig. 13 Heat Transfer at the Interface of Water and Air

Heat exchange at the interface of water and air is obviously quite complex for actual physicai model consideration.

A reasonable solution of temperature distribution for heat flux through the water-air interface is

$$\Delta T'(p,x) = \Delta T(p,x)\ \exp\left(-\frac{Kx}{\rho c u}\right) \qquad (42)$$

in which $\Delta T(p,x)$ = Temperature Distribution for complete mixing of heated effluent and the ambient river flow given by Eq. 31, K = the Surface Heat Transfer Coefficient assumed independent of x, ρ = Density of Water at Equilibrium Bulk Temperature of the Water Body,

c = Heat capacity of Water Body at Equilibrium Bulk Temperature.

Studies have shown that the reduction in temperatures due to the surface heat exchange is less than 10% for each length up to 10 miles. So in most situations the surface heat exchange effects can be neglected without loss of accuracy.

VI. CONCLUSION

A computational model, based on a diffusion equation, has been developed for predicting the maximum concentration, the maximum lengths and the enclosed surface areas of mixing zone with specified concentration ratio for shore-attached effluent discharge plumes in rivers. From the predicted results, appropriate discharge rates and discharge concentrations (or temperatures) can be extracted for a given river flow to meet the requirement of water quality control.

Comparison of the model results with some field measurements indicates good agreement. The limitations of application of this model are restricted to :

(1) Wide and shallow streamway,

(2) nonbuoyant, low momentum discharge,

(3) non-stratified, parallel flow between the ambient river and the effluent discharge.

Further refinements of this model are necessary to include the effects of some real world geometries, for example, a branch stream meeting some other branches to become a main stream, at locations downstream of the discharge port. How to make these adjustments is yet to be determined. Most of the thermal plumes are stratified and buoyant because of the temperature differences. Depth and width averaged approximations may not be a good assumption. The refinement to take into account the stratification and buoyancy effects to this model will improve its capability in predicting temperature distribution of the mixing zone. If this refinement were successfully done, this model could be extended to deep river channels.

In general, this model is a good prediction for shore-attached plumes and can be used to assess the environmental pollution of power thermal plant discharges and artificial effluent discharges in natural rivers. This predictive model is needed in the preparation of environmental impact statements for pre-operational site studies and also useful in post operational studies. It is suggested that a considerable amount of time and expense could be saved by making use of the predicting models in planning field surveys and plant operations.

BIBLIOGRAPHY

1. Fischer, H. B. , " The Effect of Bends on Dispersion in Streams", Water Resources Research, Vol. 5, No. 2, April 1969, pp. 496-506.

2. Holly, E. R., "Transverse Mixing in Rivers," Report No. S 132, Deftt Hydraulics Laboratory, Delft, The Netherlands.

3. Leendertse, J. J., "A Water-Quality Simulation Model For Well Mixed Estuaries and Coastal Seas, 1, Principle of Computation," Memo RM-6230-RC, Rand Corporation, Santa Monica, California, 1970.

4. Paily P. P., and Sayre W. W., "Modeling for Shore-Attached Thermal Plumes in Rivers", Journal of the Hydraulics Division, ASCE, Vol. 104, No. HY5, May 1978, pp. 709

5. Sayre W. W., "Natural Mixing Processes in Rivers", Environmental Impact on Rivers, Shen H. W., ed. and publisher, Ft. Collins, Colo., 1973, Chap. 6.

6. Sayre, W. W., and Chang, F. M., "A Laboratory investigation of the Open Channel Dispersion Process for Dissolved, Suspended and Floating Dispersants," U. S. Geological Survey Professional Paper 433-E, 1968.

7. Thut R. H., Personal Communication, Weyherhaueser Company, Tacoma, Washington, Nov., 1979.

8. Yotsukura, N., and Cobb, E. D., "Transverse Diffusion of Solutions in Natural Streams," United States Geological Survey Profeesional Paper 582-C, 1972.

9. Yotsukura, N., and Sayre, W. W., "Transverse Mixing in Natural Channels", Water Resources Research, Vol. 12, No. 4, August 1976, pp. 695-704

APPENDIX III

OUTPUT LISTING OF THE MCKENZIE RIVER CASE STUDY

McKenzie River

SUBREACH NR.	RADIUS CURVATURE (Meter)	WIDTH (Meter)	AVERAGE DEPTH (Meter)	SLOPE	DOWNSTREAM END (Meter)
1	0.0	18.8	1.5	0.00100	20.9
2	0.0	12.5	1.5	0.00100	64.7
3	0.0	13.6	1.5	0.00100	114.8
4	0.0	16.7	1.5	0.00100	195.2
5	300.0	18.8	1.5	0.00100	277.7
6	300.0	11.5	1.5	0.00100	294.5
7	300.0	10.4	1.5	0.00100	325.0
8	110.0	10.4	1.5	0.00100	328.9
9	15.0	5.0	1.5	0.00100	329.3

Volumetric discharge rate of thermal
effluent (m3/s) : Qo .... > 0.68
River discharge (m3/s) : Qr .... > 53.83
Initial excess concentration : Co .... > 1.000
Ambient concentration : Ca .... > 0.000
Initial dilution factor : a .... > 1.00
Number of subreaches .... > 9

AXIMUM SHORELINE CONCENTRATION

Distance : Meter
Dilution : (Co-Ca)/(Cmax-Ca)

DISTANCE	Cmax-Ca	DILUTION
0.1	0.999984E+00	1.000
0.2	0.997750E+00	1.002
0.4	0.969251E+00	1.032
0.8	0.873379E+00	1.145
1.5	0.735396E+00	1.360
3.0	0.569789E+00	1.755
6.0	0.423010E+00	2.364
12.0	0.306725E+00	3.260
20.9	0.234955E+00	4.256
25.0	0.183033E+00	5.464
50.0	0.992998E-01	10.071
64.7	0.834794E-01	11.979
100.0	0.671218E-01	14.898
114.8	0.626402E-01	15.964
150.0	0.580149E-01	17.237
195.2	0.533476E-01	18.745
200.0	0.526106E-01	19.008
250.0	0.463962E-01	21.553
277.7	0.437788E-01	22.842
294.5	0.413972E-01	24.156
300.0	0.404602E-01	24.716
325.0	0.368824E-01	27.113
328.9	0.344331E-01	29.042
329.3	0.218988E-01	45.665

OWNSTREAM END OF THE LAST SUBREACH

ISOPLETH SURFACE AREAS

Maximum Distance : Meter
Maximum Width :
Meter from discharge bank
Area : Meter**2
Dimensionless Concentration
Contour : (C-Ca)/(Co-Ca)

DIMENSIONLESS ONCENTRATION CONTOUR	AREA	MAXIMUM WIDTH	MAXIMUM DISTANCE
0.500	0.1214E+01	0.237E+00	6.0
0.400	0.2591E+01	0.288E+00	12.0
0.300	0.5574E+01	0.378E+00	20.9
0.200	0.1270E+02	0.571E+00	25.0
0.100	0.3823E+02	0.105E+01	50.0
0.080	0.6194E+02	0.118E+01	75.0
0.060	0.1512E+03	0.133E+01	135.0
0.040	0.6396E+03	0.267E+01	305.0
0.020	0.1245E+04	0.542E+01	329.3
0.008	0.1751E+04	0.798E+01	329.3
0.005	0.1960E+04	0.902E+01	329.3

:OPLETH SURFACE AREAS
(PLOTS)

Distance From Effluent Discharge : Meter
Lateral distance (Plume Width) :
Meter from discharge bank
Area : Meter**2
Dimensionless Concentration Contour :
(C-Ca)/(Co-Ca)

:STANCE	DIMENSIONLESS CONCENTRATION CONTOUR					
	0.100		0.200		0.500	
	PLUME WIDTH	AREA	PLUME WIDTH	AREA	PLUME WIDTH	AREA
0.1	0.3079E+00	0.308E-01	0.2837E+00	0.284E-01	0.2375E+00	0.237E-01
0.2	0.3371E+00	0.645E-01	0.3029E+00	0.587E-01	0.2375E+00	0.475E-01
0.4	0.3784E+00	0.140E+00	0.3300E+00	0.125E+00	0.2375E+00	0.950E-01
0.8	0.4367E+00	0.315E+00	0.3683E+00	0.272E+00	0.2370E+00	0.190E+00
1.5	0.5100E+00	0.672E+00	0.4158E+00	0.563E+00	0.2300E+00	0.356E+00
3.0	0.6196E+00	0.160E+01	0.4819E+00	0.129E+01	0.1710E+00	0.701E+00
6.0	0.7608E+00	0.388E+01	0.5487E+00	0.293E+01	0.0000E+00	0.121E+01
12.0	0.9250E+00	0.943E+01	0.5715E+00	0.636E+01		
20.9	0.1054E+01	0.188E+02	0.4578E+00	0.114E+02		
25.0	0.7569E+00	0.219E+02	0.0000E+00	0.127E+02		
50.0	0.0000E+00	0.382E+02				

DISTANCE	DIMENSIONLESS CONCENTRATION CONTOUR					
	0.020		0.050		0.080	
	PLUME WIDTH	AREA	PLUME WIDTH	AREA	PLUME WIDTH	AREA
0.1	0.3504E+00	0.350E-01	0.3279E+00	0.328E-01	0.3147E+00	0.315E-01
0.2	0.3971E+00	0.748E-01	0.3653E+00	0.693E-01	0.3467E+00	0.661E-01
0.4	0.4633E+00	0.167E+00	0.4183E+00	0.153E+00	0.3920E+00	0.145E+00
0.8	0.5568E+00	0.390E+00	0.4932E+00	0.350E+00	0.4559E+00	0.327E+00
1.5	0.6747E+00	0.862E+00	0.5875E+00	0.762E+00	0.5364E+00	0.702E+00
3.0	0.8549E+00	0.214E+01	0.7308E+00	0.186E+01	0.6575E+00	0.169E+01
6.0	0.1105E+01	0.546E+01	0.9251E+00	0.463E+01	0.8173E+00	0.414E+01
12.0	0.1443E+01	0.141E+02	0.1177E+01	0.117E+02	0.1013E+01	0.102E+02
20.9	0.1790E+01	0.300E+02	0.1419E+01	0.243E+02	0.1184E+01	0.208E+02
25.0	0.1448E+01	0.360E+02	0.1109E+01	0.289E+02	0.8856E+00	0.244E+02
50.0	0.2271E+01	0.928E+02	0.1486E+01	0.660E+02	0.8342E+00	0.465E+02
64.7	0.2551E+01	0.130E+03	0.1528E+01	0.885E+02	0.4404E+00	0.588E+02
75.0	0.2911E+01	0.160E+03	0.1656E+01	0.106E+03	0.0000E+00	0.619E+02
100.0	0.3178E+01	0.240E+03	0.1567E+01	0.147E+03		
114.8	0.3307E+01	0.289E+03	0.1469E+01	0.170E+03		
150.0	0.4234E+01	0.438E+03	0.1582E+01	0.234E+03		
195.2	0.4420E+01	0.637E+03	0.1136E+01	0.305E+03		
200.0	0.5009E+01	0.662E+03	0.1149E+01	0.311E+03		
220.0	0.5138E+01	0.764E+03	0.0000E+00	0.329E+03		
250.0	0.5298E+01	0.923E+03				
277.7	0.5418E+01	0.107E+04				
294.5	0.3377E+01	0.113E+04				
300.0	0.3075E+01	0.115E+04				
325.0	0.3144E+01	0.123E+04				
328.9	0.3173E+01	0.124E+04				
329.3	0.9823E+00	0.124E+04				

-OWNSTREAM END OF THE LAST SUBREACH

13. CORMIX - USER'S GUIDE FOR THE CORNELL MIXING ZONE EXPERT SYSTEM

CORMIX1
CORMIX2
CORMIX3

ncasi technical bulletin

NATIONAL COUNCIL OF THE PAPER INDUSTRY FOR AIR AND STREAM IMPROVEMENT, INC., 260 MADISON AVENUE, NEW YORK, N.Y. 10016

USER'S GUIDE FOR THE CORNELL MIXING ZONE EXPERT SYSTEM (CORMIX)

TECHNICAL BULLETIN NO. 624

FEBRUARY 1992

NATIONAL COUNCIL OF THE PAPER INDUSTRY FOR AIR AND STREAM IMPROVEMENT, INC.
260 MADISON AVE. NEW YORK, N.Y. 10016 (212) 532-9000 FAX: (212) 779-2849

February 24, 1992

Dr. Isaiah Gellman
President
(212) 532-9000

TECHNICAL BULLETIN NO. 624

USER'S GUIDE FOR THE CORNELL MIXING ZONE EXPERT SYSTEM (CORMIX)

To evaluate the potential impacts of effluent discharges into receiving waters, environmental engineers must make estimates of plume geometry and dilution characteristics. Previous National Council investigations have reviewed: (a) the use of aerial photogrammetry and fluorescent spectroscopy to evaluate the dispersion of kraft effluents from ocean outfalls, (b) simple estuarine analysis techniques to estimate average dilution rates, and (c) some outfall dispersions models. These subjects are the contents of NCASI Technical Bulletins No. 231, 236 and 486.

The recently reissued "TSD for Water-quality Based Toxics Control" has accentuated the need to understand the projectory and dilution characteristics of effluent discharges, particularly near outfall structures. To meet this need, EPA funded the development of the Cornell Mixing Zone Expert System (CORMIX) which is a series of software subsystems for the analysis, prediction, and design of aqueous toxic or conventional pollutant discharges into diverse water bodies. Although its major emphasis is on the prediction of plume geometry and dilution characteristics within a receiving water's initial mixing zone so that compliance with regulatory constraints may be judged, the system also predicts the behavior of the discharge plume at larger distances. The highly user-interactive CORMIX system (a) is implemented on microcomputers (IBM-PC, or compatible), (b) is being promoted and distributed free of charge by EPA's Center for Exposure Assessment Modeling, and (c) allows users with little or no training in hydrodynamic modeling to evaluate the dilution characteristics of existing effluent discharge structures, and make recommendations on design changes to meet the chronic and acute concentration goals stated in the Toxics TSD.

Continuing in its program to review state-of-the-art approaches for (a) determining the extent of effluent dilution by receiving waters and (b) providing guidance for the appropriate use of such approaches, NCASI has worked cooperatively with CORMIX's developer, Professor Gerhard H. Jirka of Cornell

University, to develop a user's manual which will (a) acquaint industry personnel with CORMIX's features and limitations and (b) assist inexperienced and infrequent users with input development and output interpretation. This report is the result of that effort.

Due to the apparent depth of preparation which went into the development of the CORMIX systems and the fact that they have been presented in the peer-reviewed literature, NCASI staff have not independently evaluated the modeling assumptions used or verified the computational accuracy of the CORMIX software. NCASI is monitoring field applications of the CORMIX systems and will undertake a more detailed review of them if the need arises. In addition to the routine inspection of simulation results for reasonableness, potential users should exercise extra caution when using simulation results for regulatory or design purposes, until such time that the programs have been demonstrated to be error free.

This report was co-authored by Dr. Jirka, Professor and Director of the DeFrees Hydraulics Laboratory within the School of Civil and Environmental Engineering at Cornell University and Dr. Steven W. Hinton, Research Engineer at the NCASI Northeast Regional Center. Dr. Hinton managed this project and edited the report into its present form with assistance from Nancy Bartlett also of the Northeast Regional Center staff. Questions or comments on the contents of this report are welcome and should be directed to Dr. Hinton at the NCASI Northeast Regional Center, Tufts University, Department of Civil Engineering, 001 Anderson Hall, Medford, Massachusetts 02155, telephone (617) 627-3254.

Very truly yours,

Dr. Isaiah Gellman
President

TABLE OF CONTENTS

APPENDICES

USER'S GUIDE FOR THE CORNELL
MIXING ZONE EXPERT SYSTEM (CORMIX)

TECHNICAL BULLETIN NO. 624
FEBRUARY, 1992

ABSTRACT: A user's guide for the Cornell Mixing Zone Expert System (CORMIX) is presented. CORMIX is a series of software subsystems for the analysis, prediction, and design of aqueous toxic or conventional pollutant discharges into diverse water bodies. Its major emphasis is on the prediction of plume geometry and dilution characteristics within a receiving water's initial mixing zone so that compliance with regulatory constraints may be judged. The system also predicts the behavior of the discharge plume at larger distances. The highly user-interactive CORMIX system is implemented on microcomputers (IBM-PC, or compatible), and consists of three subsystems:
--- CORMIX1 for submerged single port discharges,
--- CORMIX2 for submerged multiport diffuser discharges,
--- CORMIX3 for buoyant surface discharges.
The user's guide gives a comprehensive and uniform description of all three CORMIX subsystems; it provides guidance for assembly and preparation of required input data; it delineates ranges of applicability of the three subsystems; it provides guidance for interpretation and graphical display of system output; and it illustrates practical system application through three case studies.

KEYWORDS: CORMIX, Cornell Mixing Zone Expert System, mixing zones, effluent dilution, discharge structures, initial mixing, surface water quality models.

RELATED TECHNICAL PUBLICATIONS:

(1) "The Use of Aerial Photogrammetry and Fluorescent Spectroscopy to Evaluate Ocean Outfall Dispersion of Kraft Effluent" NCASI Technical Bulletin No. 231. (December 1969)

(2) "An Introduction to Determination of Estuarine Assimilative Capacity" NCASI Technical Bulletin No. 236. (April 1970)

(3) "Initial Review of Some Outfall Dispersion Models and an Asymptotic Jet Dispersion Model for Use in River Systems" NCASI Technical Bulletin No. 486. (March 1986)

I INTRODUCTION

The Cornell Mixing Zone Expert System (CORMIX) is a series of software subsystems for the analysis, prediction, and design of aqueous toxic or conventional pollutant discharges into diverse water bodies. It was developed under cooperative funding agreements (1) for U.S. EPA and is the recommended analysis tool in key guidance documents (2,3,4) on the permitting of industry discharges to receiving waters. Although the system's major emphasis is on predicting the geometry and dilution characteristics of the initial mixing zone so that compliance with acute and chronic regulatory constraints may be judged, the system also predicts the behavior of the discharge plume at larger distances.

The highly user-interactive CORMIX system is implemented on IBM-PC/XT/AT compatible microcomputers, utilizes an 'expert' systems approach to data input and processing, and consists of three subsystems. These are: (a) CORMIX1 for the analysis of submerged single port discharges, (b) CORMIX2 for the analysis of submerged multiport diffuser discharges and (c) CORMIX3 for the analysis of buoyant surface discharges. Without specialized training in hydrodynamics, users can make detailed predictions of mixing zone conditions, check compliance with regulations and readily investigate the performance of alternative outfall designs.

Several factors provided the original impetus for system development including: (a) the considerable complexity of mixing processes in the aquatic environment, resulting from the great diversity of discharge and site conditions and requiring advanced knowledge in a specialized field of hydrodynamics; (b) the failure of previously existing models (e.g. the U.S. EPA plume models (5) originally developed for municipal discharges in deep coastal waters) to adequately predict often routine discharge situations, especially for more shallow inland sites; (c) the issuance by the U.S. EPA of additional guidelines (6) for the permitting of toxic aqueous discharges, placing yet another burden on both applicants and regulators in delineating special zones for the initial mixing of these substances; and (d) the availability of new computer methods, so-called expert systems, for making accessible to the user, within a simple personal computing environment, the expert's knowledge and experience in dealing with complex engineering problems.

Three separate publications (7,8,9) describe the scientific basis for the CORMIX system, demonstrate comparison and validation with field and laboratory data, and serve as user's

manuals for model application. The results of these works are summarized in the peer-reviewed literature (10,11,12). The CORMIX system approach and its performance relative to the earlier U.S. EPA plume models in the context of estuarine applications is also described in EPA's technical guidance manual for performing waste load allocations in estuaries (13). NCASI has not independently evaluated the modeling assumptions used or verified the computational accuracy of the CORMIX software.

EPA's long range goal is to make the CORMIX system freely available to all potential users through its modeling software distribution facility at the U.S. EPA Center for Environmental Assessment Modeling (CEAM) in Athens, Georgia. Some of the CORMIX subsystems have been available to the industrial and regulatory user communities since December 1989 when distribution of CORMIX1 was commenced by Cornell University for the purpose of identifying subtle programming errors through application to actual mixing zone analysis problems by a controlled users group. After this testing was deemed complete, CEAM commenced the distribution of CORMIX1 in November 1990. A similar approach is being used to introduce CORMIX2 which is scheduled to begin CEAM distribution in October 1991 and to introduce CORMIX3 which is currently undergoing initial user testing. This Technical Bulletin covers versions available as of August 1991 including: CORMIX1, Version 1.30, August 1991; CORMIX2, Version 1.20, August 1991; and CORMIX3, Version 1.10, August 1991.

The objectives of this user's guide are as follows: (a) to provide a comprehensive and uniform description of all three CORMIX subsystems; (b) to provide guidance for assembly and preparation of required input data; (c) to delineate ranges of applicability of the three subsystems; (d) to provide guidance for the interpretation and graphical display of system output; and (e) to illustrate practical system application through several case studies.

This bulletin is organized to meet the informational needs of two distinctly different groups of readers; personnel in environmental management positions desiring an overview of the CORMIX systems capabilities and technical staff needing assistance in actual applications. Section II provides a summary of the physical processes of effluent mixing, as well as an overview of the regulatory background and practice on mixing zone applications. The general features of the CORMIX system are explained in Section III including summaries of: (a) predictive capabilities and limitations, (b) overall system structure and method of processing information, (c) user interaction, and (d) individual calculational elements. Detailed guidance on the preparation and entry of input data, as required by the three CORMIX subsystems, is given in Section IV. Section V provides a description of system output, containing both descriptive and quantitative numerical information on the predicted effluent flow. The closing remarks in Section VI contain information on system availability and user support, and on future system developments and enhancements.

Appendices to this guide present three case studies on the application of all three CORMIX subsystems. These are adapted from actual situations and illustrate the complete input requirements and output capabilities of the systems. In addition, some of the assumptions on data schematization, problem simplification, output interpretation, and construction of graphical displays are discussed in a context typical of many mixing zone model applications.

II BACKGROUND: MIXING PROCESSES AND MIXING ZONE REGULATIONS

When performing design work and predictive studies on effluent discharge problems, it is important to clearly distinguish between the physical aspects of hydrodynamic mixing processes that determine the effluent fate and distribution, and the administrative construct of mixing zone regulations that intend to prevent any harmful impact of the effluent on the aquatic environment and associated uses.

A. Hydrodynamic Mixing Processes

The mixing behavior of any wastewater discharge is governed by the interplay of ambient conditions in the receiving water body and by the discharge characteristics.

The ambient conditions in the receiving water body, be it stream, river, lake, reservoir, estuary or coastal waters, are described by the water body's geometric and dynamic characteristics. Important geometric parameters include plan shape, vertical cross-sections, and bathymetry, especially in the discharge vicinity. Dynamic characteristics are given by the velocity and density distribution in the water body, again primarily in the discharge vicinity. In many cases, these conditions can be taken as steady-state with little variation because the time scale for the mixing processes is usually of the order of minutes up to perhaps one hour. In some cases, notably tidally influenced flows, the ambient conditions can be highly transient and the assumption of steady-state conditions may be inappropriate.

The discharge conditions relate to the geometric and flux characteristics of the submerged outfall installation. For a single port discharge the port diameter, its elevation above the bottom and its orientation provide the geometry; for multiport diffuser installations the arrangement of the individual ports along the diffuser line, the orientation of the diffuser line, and construction details represent additional geometric features; and for surface discharges the cross-section and orientation of the flow entering the ambient watercourse are important. The flux characteristics are given by the effluent discharge flow rate, by its momentum flux and by its buoyancy flux. The buoyancy flux represents the effect of the relative density difference between the effluent discharge and ambient conditions

in combination with gravitational acceleration. It is a measure of the tendency for the effluent flow to rise (i.e. positive buoyancy) or to fall (i.e. negative buoyancy).

The hydrodynamics of an effluent continuously discharging into a receiving water body can be conceptualized as a mixing process occurring in two separate regions. In the first region, the initial jet characteristics of momentum flux, buoyancy flux, and outfall geometry influence the jet trajectory and mixing. This region will be referred to as the "near-field", and encompasses the buoyant jet flow and any surface, bottom or terminal layer interaction. In this near-field region, outfall designers can usually affect the initial mixing characteristics through appropriate manipulation of design variables.

As the turbulent plume travels further away from the source, the source characteristics become less important. Conditions existing in the ambient environment will control trajectory and dilution of the turbulent plume through buoyant spreading motions and passive diffusion due to ambient turbulence. This region will be referred to here as the "far-field". It is stressed at this point that the distinction between near-field and far-field is made purely on hydrodynamic grounds. It is unrelated to any regulatory mixing zone definitions.

(1) Near-Field Processes- Three important types of near-field processes are submerged buoyant jet mixing, boundary interactions and surface buoyant jet mixing as described in the following paragraphs.

Submerged Buoyant Jet Mixing: The effluent flow from a submerged discharge port provides a velocity discontinuity between the discharged fluid and the ambient fluid causing an intense shearing action. The shearing flow breaks rapidly down into a turbulent motion. The width of the zone of high turbulence intensity increases in the direction of the flow by incorporating ("entraining") more of the outside, less turbulent fluid into this zone. In this manner, any internal concentrations (e.g. fluid momentum or pollutants) of the discharge flow become diluted by the entrainment of ambient water. Inversely, one can speak of the fact that both fluid momentum and pollutants become gradually diffused into the ambient field.

The initial velocity discontinuity may arise in different fashions. In a "pure jet" (also called "momentum jet" or "non-buoyant jet"), the initial momentum flux in the form of a high-velocity injection causes the turbulent mixing. In a "pure plume," the initial buoyancy flux leads to local vertical accelerations which then lead to turbulent mixing. In the general case of a "buoyant jet" (also called a "forced plume"), a combination of initial momentum flux and buoyancy flux is responsible for turbulent mixing.

Thus, buoyant jets are characterized by a narrow turbulent fluid zone in which vigorous mixing takes place. Furthermore, depending on discharge orientation and direction of buoyant acceleration, curved trajectories are generally established in a stagnant uniform-density environment as illustrated in Figure 1a.

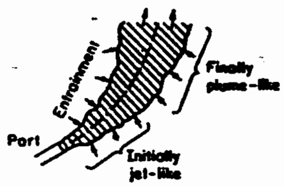
Buoyant jet mixing is further affected by ambient currents and density stratification. The role of ambient currents is to gradually deflect the buoyant jet into the current direction as illustrated in Figure 1b and thereby induce additional mixing. The role of ambient density stratification is to counteract the vertical acceleration within the buoyant jet leading ultimately to trapping of the flow at a certain level. Figure 1c shows a typical buoyant jet shape at the trapping or terminal level.

Finally, in case of multiport diffusers, the individual round buoyant jets behave independently until they interact, or merge, with each other at a certain distance from the efflux ports. After merging, a two-dimensional buoyant jet plane is formed as illustrated in Figure 1d. Such plane buoyant jets resulting from a multiport diffuser discharge in deep water can be further affected by ambient currents and by density stratification as discussed in the preceding paragraph.

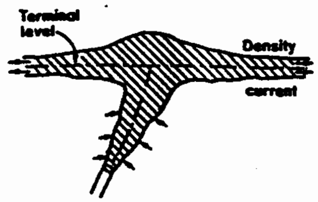
Boundary Interaction Processes and Near-Field Stability: Ambient water bodies always have vertical boundaries. These include the water surface and the bottom, but in addition, "internal boundaries" may exist at pycnoclines. Pycnoclines are layers of rapid density change. Depending on the dynamic and geometric characteristics of the discharge flow, a variety of interaction phenomena can occur at such boundaries, particularly where flow trapping may occur.

In essence, boundary interaction processes provide a transition between the buoyant jet mixing process in the near-field, and buoyant spreading and passive diffusion in the far-field. They can be gradual and mild, or abrupt leading to vigorous transition and mixing processes. They also can significantly influence the stability of the effluent discharge conditions.

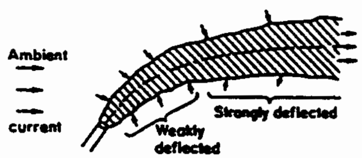
The assessment of near-field stability, i.e. the distinction of stable or unstable conditions, is a key aspect of effluent dilution analyses. It is especially important for understanding the behavior of the two-dimensional plumes resulting from multiport diffusers, as shown by some examples in Figure 2. "Stable discharge" conditions, usually occurring for a combination of strong buoyancy, weak momentum and deep water, are often referred to as "deep water" conditions (Figures 2a,c). "Unstable discharge" conditions, on the other hand, may be considered synonymous to "shallow water" conditions (Figure 2b,d). Technical discussions on discharge stability are presented elsewhere (14,15).



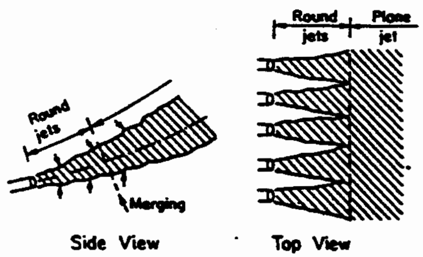
a) Buoyant Jet in Stagnant Uniform Ambient



c) Buoyant Jet in Stagnant Stratified Ambient

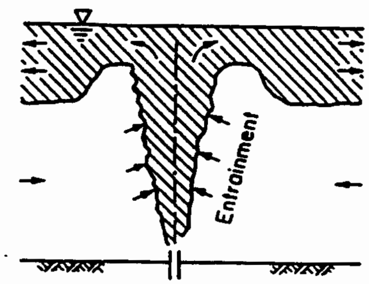


b) Buoyant Jet in Uniform Ambient Cross-Current

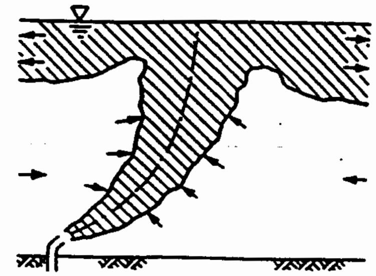


d) Jet Merging for Unidirectional Multiport Diffuser Forming Plane Buoyant Jet

FIGURE 1 TYPICAL BUOYANT JET MIXING FLOW PATTERNS

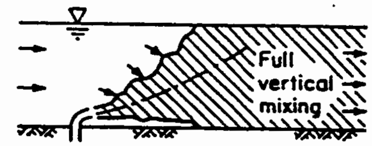


a) Deep Water, High Buoyancy, Vertical: Stable Near-Field



c) Deep Water, High Buoyancy, Near-Horizontal: Stable Near-Field

b) Shallow Water, Low Buoyancy, Vertical: Unstable Near-Field with Local Mixing and Restratification



d) Shallow Water, Low Buoyancy, Near-Horizontal: Unstable Near-Field with Full Vertical Mixing

FIGURE 2 EXAMPLES OF NEAR-FIELD STABILITY CONDITIONS

A few important examples of boundary interaction for a single round buoyant jet are illustrated in Figure 3. If a buoyant jet is bent-over by a cross-flow, it will gradually approach the surface, bottom or terminal level and will undergo a smooth transition with little additional mixing (Figure 3a). However, a jet impinging normally, or near-normally, on a boundary will rapidly spread in all directions. Mixing conditions at this impingement point can take on one of the following forms: (a) If the flow has sufficient buoyancy it will ultimately form a stable layer at the surface (Figure 3b). In the presence of weak ambient flow this will lead to an upstream intrusion against the ambient current. (b) If the buoyancy of the effluent flow is weak or its momentum very high, unstable recirculation phenomena can occur in the discharge vicinity (Figure 3c). This local recirculation leads to re-entrainment of already mixed water back into the buoyant jet region. (c) In the intermediate case, a combination of localized vertical mixing and upstream spreading may result (Figure 3d).

Another type of interaction process concerns submerged buoyant jets discharging in the vicinity of the water bottom into a stagnant or flowing ambient. Two types of dynamic interaction processes can occur that lead to rapid attachment of the effluent plume to the water bottom as illustrated in Figure 4. These are wake attachment forced by the receiving water's crossflow or Coanda attachment forced by the entrainment demand of the effluent jet itself. The latter is due to low pressure effects as the jet periphery is close to the water bottom.

Surface Buoyant Jet Mixing: Positively buoyant jets discharged horizontally along the water surface from a laterally entering channel or pipe (Figure 5) bear some similarities to the more classical submerged buoyant jet. For a relatively short initial distance, the effluent behaves like a momentum jet spreading both laterally and vertically due to turbulent mixing. After this stage, vertical entrainment becomes inhibited due to buoyant damping of the turbulent motions, and the jet experiences strong lateral spreading. During stagnant ambient conditions, ultimately a reasonably thin layer may be formed at the surface of the receiving water; that layer can undergo the transient buoyant spreading motions depicted in Figure 5a.

In the presence of ambient crossflow, buoyant surface jets may exhibit any one of following three types of flow features: They may form a weakly deflected jet that does not interact with the shoreline (Figure 5b). When the crossflow is strong, they may attach to the downstream boundary forming a shore-hugging plume (Figure 5c). When a high discharge buoyancy flux combines with a weak crossflow, the buoyant spreading effects can be so strong that an upstream intruding plume is formed that also stays close to the shoreline (Figure 5d).

Intermediate-Field Effects for Multiport Diffuser Discharges: Some multiport diffuser installations induce flows in shallow

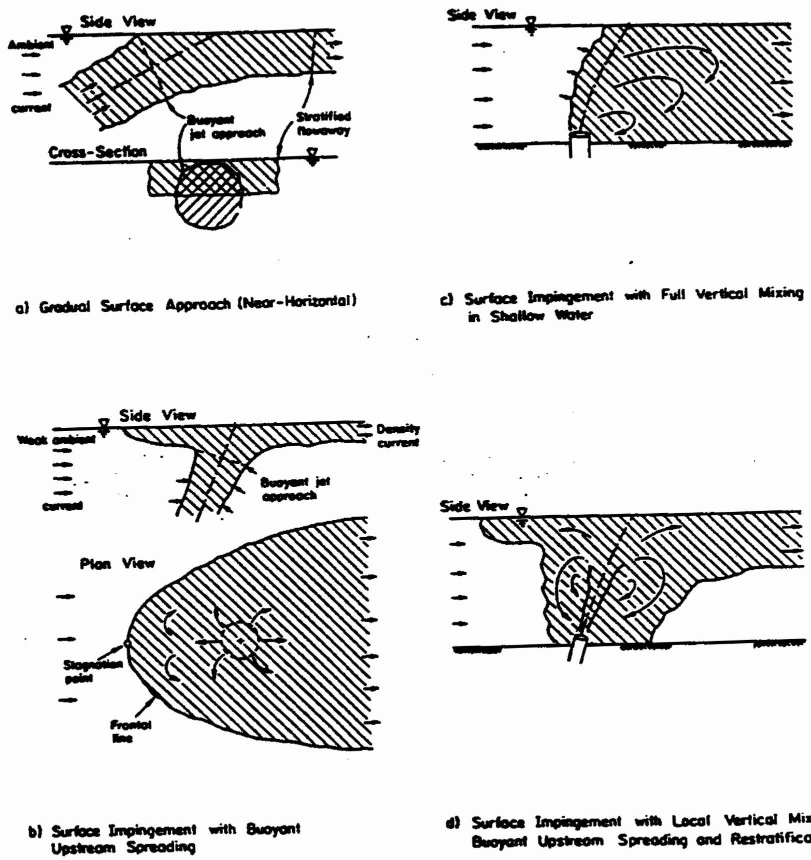
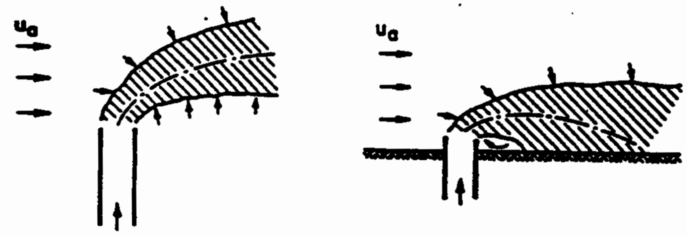


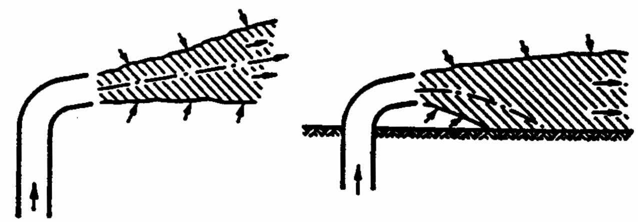
FIGURE 3 EXAMPLES OF BOUNDARY INTERACTIONS FOR A SINGLE ROUND JET



i) Free Deflected Jet/Plume in Cross-flow

ii) Wake Attachment of Jet/Plume

a) Wake Attachment



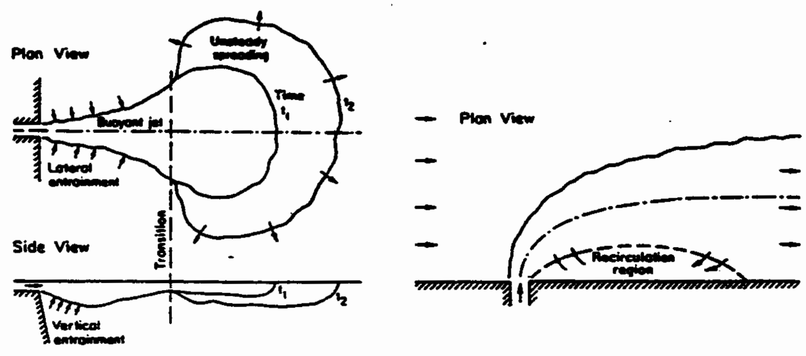
i) Free Jet

ii) Attached Jet

b) Coanda Attachment

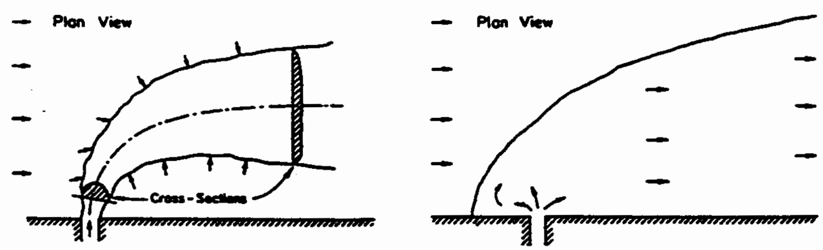
Note: Flow Patterns for single port discharges (i) away from and (ii) in close proximity to bottom surface.

FIGURE 4 EXAMPLE OF WAKE AND COANDA ATTACHMENT CONDITIONS



a) Buoyant Surface Jet in Stagnant Ambient

c) Shoreline-Attached Surface Jet in Strong Ambient Crossflow



b) Buoyant Surface Jet in Ambient Crossflow

d) Upstream Intruding Plume in Weak Ambient Crossflow

FIGURE 5 TYPICAL BUOYANT SURFACE JET MIXING FLOW PATTERNS

water which extend beyond the strict near-field region. The resulting plumes are sometimes referred to as the "intermediate-field" (14) because they interact with the receiving water at distances that are substantially greater than the water depth; the order of magnitude of the water depth is typically used to define the dimensions of the near-field region. Intermediate fields may occur when a multiport diffuser represents a large source of momentum with a relatively weak buoyancy effect. Such a diffuser will have an unstable near-field with shallow water conditions. For certain diffuser geometries (e.g. unidirectional & staged diffuser types; see Section V) strong motions can be induced in the shallow water environment in the form of vertically mixed currents that laterally entrain ambient water and may extend over long distances before they re-stratify or dissipate their momentum.

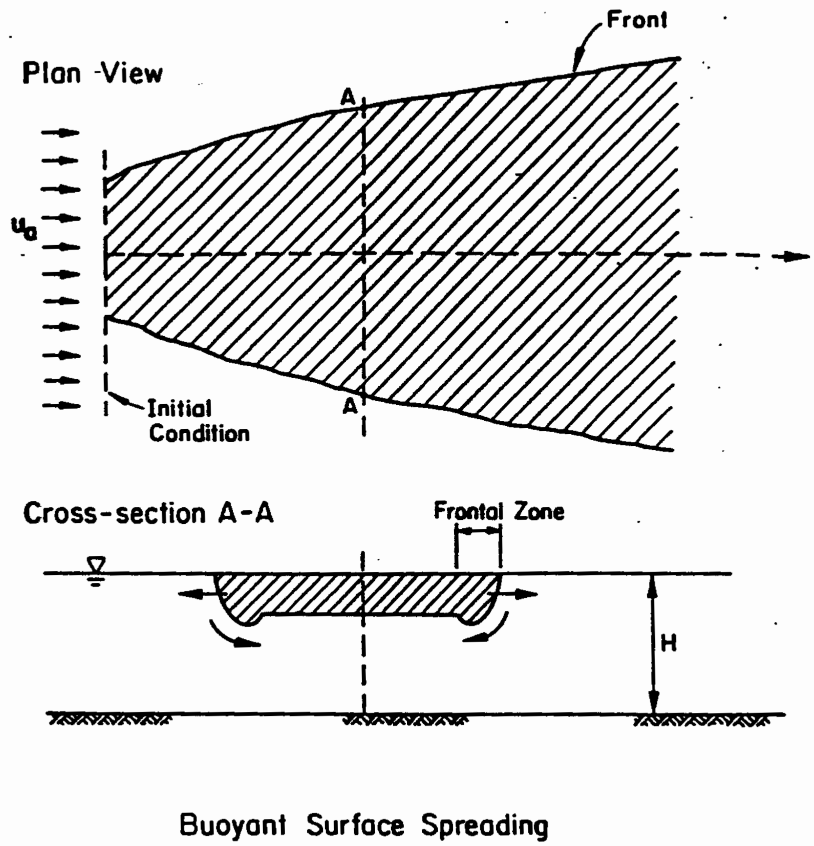
(2) Far-Field Processes- Far-field mixing processes are characterized by the longitudinal advection of the mixed effluent by the ambient current velocity.

Buoyant Spreading Processes: These are defined as the horizontally transverse spreading of the mixed effluent flow while it is being advected downstream by the ambient current. Such spreading processes arise due to the buoyant forces caused by the density difference of the mixed flow relative to the ambient density. They can be effective transport mechanisms that can quickly spread a mixed effluent laterally over large distances in the transverse direction, particularly in cases of strong ambient stratification. In this situation, effluent of considerable vertical thickness at the terminal level can collapse into a thin but very wide layer unless this is prevented by lateral boundaries. If the discharge is non-buoyant, or weakly buoyant, and the ambient is unstratified, there is no buoyant spreading region in the far-field, only a passive diffusion region.

Depending on the type of near-field flow and ambient stratification, several types of buoyant spreading may occur. These include: (a) spreading at the water surface, (b) spreading at the bottom, (c) spreading at a sharp internal interface (pycnocline) with a density jump, or (d) spreading at the terminal level in continuously stratified ambient fluid.

As an example, the definition diagram and structure of surface buoyant spreading processes somewhat downstream of the discharge in unstratified crossflow is shown in Figure 6. The laterally spreading flow behaves like a density current and entrains some ambient fluid in the "head region" of the current. During this phase, the mixing rate is usually relatively small, the layer thickness may decrease, and a subsequent interaction with a shoreline or bank can impact the spreading and mixing processes.

Passive Ambient Diffusion Processes: The existing turbulence in the ambient environment becomes the dominating mixing mechanism



Note: Possible shoreline interaction not depicted.

FIGURE 6 DEFINITION DIAGRAM AND STRUCTURE OF SURFACE BUOYANT SPREADING PROCESSES DOWNSTREAM OF THE DISCHARGE POINT IN UNSTRATIFIED CROSSFLOW at sufficiently large distances from the discharge point. In general, the passively diffusing flow grows in width and in thickness until it interacts with the channel bottom and/or banks as illustrated in Figure 7.

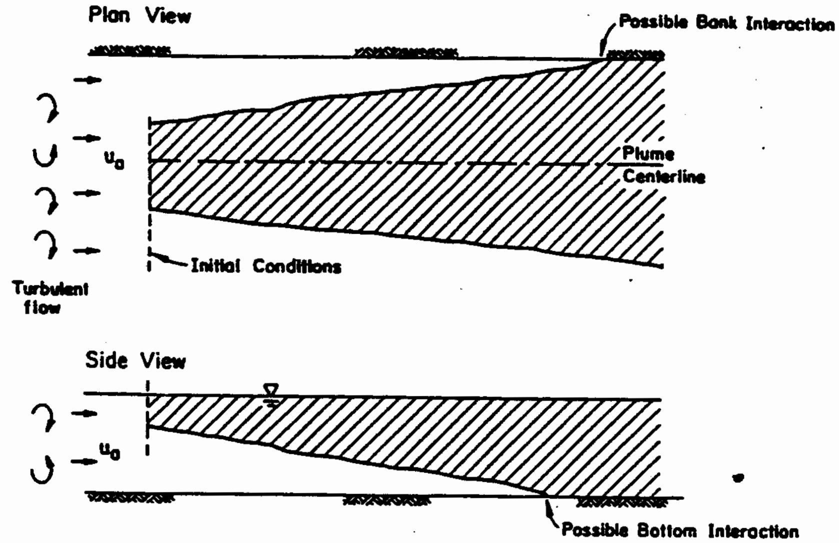
The strength of the ambient diffusion mechanism depends on a number of factors relating mainly to the geometry of the ambient shear flow and the amount of ambient stratification. In the context of classical diffusion theory (16), gradient diffusion processes in the bounded flows of rivers or narrow estuaries can be described by constant diffusivities in the vertical and horizontal direction that depend on turbulent intensity and on channel depth or width as the length scales. In contrast, wide "unbounded" channels or open coastal areas are characterized by plume size dependent diffusivities leading to accelerating plume growth described, for example, by the "4/3 law" of diffusion. In the presence of a stable ambient stratification, the vertical diffusive mixing is generally strongly damped.

B. Mixing Zone Regulations

The discharge of waste water into a water body can be considered from two vantage points regarding its impact on ambient water quality. On a larger scale, seen over the entire receiving water body, care must be taken that water quality conditions that protect designated beneficial uses are achieved. This is the realm of the general waste load allocation (WLA) procedures and models.

On a local scale, or in the immediate discharge vicinity, additional precautions must be taken to insure that high initial pollutant concentrations are minimized and constrained to small zones, areas, or volumes. The generic definition of these zones, commonly referred to as "mixing zones", is embodied in federal water quality regulations and often cited in the regulations of permit granting authorities. As stated previously, mixing zones are administrative constructs that are independent of hydrodynamic mixing processes.

(1) Legal Background- The Clean Water Act of 1977 defines five general categories of pollutants. These are: (a) conventional, (b) nonconventional, (c) toxics, (d) heat, and (e) dredge and fill spoil. The Act distinguishes between new and existing sources for setting effluent standards. Pollutants designated as "conventional" would be "generally those pollutants that are naturally occurring, biodegradable, oxygen demanding materials and solids. In addition, compounds which are not toxic and which are similar in characteristics to naturally occurring, biodegradable substances are to be designated as conventional pollutants for the purposes of the provision." Examples of conventional pollutants are: biochemical oxygen demand (BOD), total suspended solids, and fecal coliform bacteria. Pollutants designated as "nonconventional" would be "those which are not toxic or conventional", and some examples are: chemical oxygen



Passive Diffusion Process

FIGURE 7 EXAMPLE OF AMBIENT DIFFUSION FLOW PATTERNS

demand (COD), fluoride, and ammonia. "Toxic" pollutants are those that cause harmful effects, either acute or chronic, at very low concentrations; examples of some designated toxic substances are: nickel, chloroform, or benzidine.

(2) Mixing Zone Definitions- The mixing zone is defined as an "allocated impact zone" where numeric water quality criteria can be exceeded as long as acutely toxic conditions are prevented. A mixing zone can be thought of as a limited area or volume where the initial dilution of a discharge occurs (17). Water quality standards apply at the boundary of the mixing zone, not within the mixing zone itself. The U.S. EPA and its predecessor agencies have published numerous documents giving guidance for determining mixing zones. Guidance published by U.S. EPA in Water Quality Standards Handbook (1984) supersedes these sources.

In setting requirements for mixing zones, U.S. EPA (18) requires that "the area or volume of an individual zone or group of zones be limited to an area or volume as small as practicable that will not interfere with the designated uses or with the established community of aquatic life in the segment for which the uses are designated," and the shape be "a simple configuration that is easy to locate in the body of water and avoids impingement on biologically important areas," and "shore hugging plumes should be avoided."

The U.S. EPA rules for mixing zones recognize the State has discretion whether or not to adopt a mixing zone and to specify its dimensions. The U.S. EPA allows the use of a mixing zone in permit applications except where one is prohibited in State regulations. A previous review (7) of individual State mixing zone policies (18,6) found that 48 out of 50 States make use of a mixing zone in some form; the exceptions are Arizona and Pennsylvania. State regulations dealing with streams or rivers generally limit mixing zone widths or cross-sectional areas, and allow lengths to be determined on a case-by-case basis.

In the case of lakes, estuaries and coastal waters, some states specify the surface area that can be affected by the discharge. The surface area limitation usually applies to the underlying water column and benthic area. In the absence of specific mixing zone dimensions, the actual shape and size is determined on a case-by-case basis.

Special mixing zone definitions have been developed for the discharge of municipal wastewater into the coastal ocean, as regulated under Section 301(h) of the Clean Water Act (19). Often these same definitions are used also for industrial and other discharges into coastal waters or large lakes, resulting in a plurality of terminology. For those discharges, the mixing zone was labeled as the "zone of initial dilution" in which rapid mixing of the waste stream (usually the rising buoyant fresh water plume within the ambient saline water) takes place. EPA requires that the "zone of initial dilution" be a regularly shaped area (e.g. circular or rectangular) surrounding the discharge structure (e.g. submerged pipe or diffuser line) that encompasses the regions of high (exceeding standards) pollutant concentrations under design conditions (19). In practice, limiting boundaries defined by dimensions equal to the water depth measured horizontally from any point of the discharge structure are accepted by the EPA provided they do not violate other mixing zone restrictions (19).

(3) <u>Special Mixing Zone Requirements for Toxic Substances-</u> The U.S. EPA maintains two water quality criteria for the allowable concentration of toxic substances: a criterion maximum concentration (CMC) to protect against acute or lethal effects; and a criterion continuous concentration (CCC) to protect against chronic effects (2). The CMC value is greater than or equal to the CCC value and is usually the more difficult of the two criterion to satisfy. The CCC must be met at the edge of the same regulatory mixing zone specified for conventional and nonconventional discharges.

In order to prevent lethal concentrations of toxics in the regulatory mixing zone, the CMC criterion must be met within a short distance from the outfall or in the pipe itself. If dilution of the toxic discharge in the ambient environment is allowed, this requirement, which is commonly called the toxic dilution zone (TDZ), is usually more restrictive than the legal mixing zone for conventional and nonconventional pollutants. In the 1985 Toxics TSD document (6), a minimum exit velocity of 3 meters per second (10 feet per second) was required in order to provide sufficiently rapid mixing that would minimize organism exposure time to toxic material. The revised 1991 Toxics TSD document (2) makes the minimum exit velocity simply a recommendation for new discharges, recognizing that the restrictions listed in the following paragraph can in many instances also be met by other designs, especially if the ambient velocity is large.

The outfall design must meet all of the following geometric restrictions for a TDZ:

- The CMC must be met within 10% of the distance from the edge of the outfall structure to the edge of the regulatory mixing zone in any spatial direction.
- The CMC must be met within a distance of 50 times the discharge length scale in any spatial direction. The discharge length scale is defined as the square-root of the cross-sectional area of any discharge outlet. This restriction is intended to ensure a dilution factor of at least 10 within this distance under all possible circumstances, including situations of severe bottom or surface interaction.
- The CMC must be met within a distance of 5 times the local water depth in any horizontal direction. The local water depth is defined as the natural water depth (existing prior to the installation of the discharge outlet) prevailing under mixing zone design condition (e.g. low flow for rivers). This restriction will prevent locating the discharge in very shallow environments or very close to shore, which would result in significant surface and bottom concentrations (2).

(4) <u>Current Permitting Practice on Mixing Zones-</u> It is difficult to generalize the actual practice in implementing the mixing zone regulations, given the large number and diverse types of jurisdictions and permit-granting authorities involved. By and large, however, current procedure falls into one of the following approaches, or may involve a combination thereof.

(i) The mixing zone is defined by some numerical dimension, as discussed above. The applicant must then demonstrate that the existing or proposed discharge meets all applicable standards for conventional pollutants or for the CCC of toxic pollutants at the edge of the specified mixing zone.

(ii) No numerical definition for a mixing zone may apply. In this case a mixing zone dimension may be proposed by the applicant. To do so the applicant generally uses actual concentration measurements for an existing discharges, dye dispersion tests or model predictions to show at what plume distance, width, or region, the applicable standard will be met. The applicant may then use further ecological or water use-oriented arguments to demonstrate that the size of that predicted region provides reasonable protection. The permitting authority may evaluate that proposal, or sometimes pursue its own independent proposal for a mixing zone. This approach resembles a negotiating process with the objective of providing optimal protection of the aquatic environment consistent with other uses.

As regards the acute, or CMC, criterion for toxic pollutants, the spatial restrictions embodied in the Toxics TSD document (2) call for very specific demonstrations of how the CMC criterion is met at the edge of the "toxic dilution zone". Again, field tests for existing discharges or predictive models may be used.

(5) <u>Relationship Between Actual Hydrodynamic Processes and Mixing Zone Dimensions-</u> The spatial requirements in mixing zone regulations are not always correlated with the actual hydrodynamic processes of mixing. With few exceptions, the toxic dilution criteria apply to the near-field of most discharges since the TDZ criteria (2) are spatially highly restrictive. The regular mixing zone boundaries, however, may be located in the near-field or the far-field of the actual effluent discharge flow since they are administratively determined by the permit-granting authority. Thus, the analyst must have tools at his disposal with the capability to address both the near and far-field situations.

III GENERAL FEATURES OF THE CORMIX SYSTEM

This section provides a general description of common features of all three CORMIX subsystems. The subsystems are CORMIX1, CORMIX2 AND CORMIX3 for the analysis of submerged single port, submerged multiport and buoyant surface outfall configurations, respectively. The following two sections give a detailed user's guide for developing the required input data and for understanding program output. Reference is made throughout this bulletin to the subsystem versions dated August 1990; other versions may differ somewhat.

A. Overview

The CORMIX system represents a robust and versatile computerized methodology for predicting both the qualitative features (e.g. flow classification) and the quantitative aspects (e.g. dilution ratio, plume projectory) of the hydrodynamic

mixing processes resulting from different discharge configurations and in all types of ambient water bodies, including small streams, large rivers, lakes, reservoirs, estuaries, and coastal waters. The methodology: (a) has been extensively verified by the developers through comparison of simulation results to available field and laboratory data on mixing processes (7,8,9), (b) has undergone independent peer review (10,11,12) and (c) is equally applicable to a wide range of problems from a simple single submerged pipe discharge into a small stream with rapid cross-sectional mixing to a complicated multiport diffuser installation in a deeply stratified coastal water. System experience suggests that CORMIX1 applies to better than 95% of submerged single-port designs, CORMIX2 to better than 80% of multiport diffusers, and CORMIX3 to better than 90% of surface discharges.

The methodology provides answers to questions that typically arise during the application of mixing zone regulations for both conventional and toxic discharges. More importantly, this is accomplished by utilizing the customary approaches often used in evaluating and implementing mixing zones, thereby providing a common framework for both applicants and regulatory personnel to arrive at a consensus view of the available dilution and plume trajectory for the site and effluent discharge characteristics.

The methodology also provides a way for personnel with little or no training in hydrodynamics to investigate improved design solutions for aquatic discharge structures. To limit misuse, the system contains limits of applicability that prevent the simulation of situations for which no safe predictive methodology exists, or for discharge geometries that are undesirable from a hydrodynamic viewpoint. Furthermore, warning labels, data screening mechanisms, and alternative design recommendations are furnished by the system. The system is not fool proof, however, and final results should always be examined for reasonableness.

Finally, CORMIX is an educational tool that intends to make the user more knowledgeable and appreciative about effluent discharge and mixing processes. The system is not simply a black box that produces a final numerical or graphical output, but contains a interactive menu of user guidance, help options, and explanatory material of the relevant physical processes. These assist users in understanding model predictions and exploring the sensitivity of model predictions to assumptions.

B. Capabilities and Major Assumptions of the Three Subsystems

CORMIX1 predicts the geometry and dilution characteristics of the effluent flow resulting from a submerged single port diffuser discharge, of arbitrary density (positively, neutrally, or negatively buoyant) and arbitrary location and geometry, into an ambient receiving water body that may be stagnant or flowing and have ambient density stratification of different types.

CORMIX2 applies to three commonly used types of submerged multiport diffuser discharges under the same general effluent and ambient conditions as CORMIX1. It analyzes unidirectional, staged, and alternating designs of multiport diffusers and allows for arbitrary alignment of the diffuser structure within the ambient water body, and for arbitrary arrangement and orientation of the individual ports. CORMIX2 uses the "equivalent slot diffuser" concept and thus neglects the details of the individual jets issuing from each diffuser port and their merging process, but rather assumes that the flow arises from a long slot discharge with equivalent dynamic characteristics. Hence, if details of the effluent flow behavior in the immediate diffuser vicinity are needed, an additional CORMIX1 simulation for an equivalent partial effluent flow may be recommended.

CORMIX3 analyzes buoyant surface discharges that result when an effluent enters a larger water body laterally, through a canal, channel, or near-surface pipe. In contrast to CORMIX1 and 2, it is limited to positively or neutrally buoyant effluents and assumes a uniform (i.e. unstratified) ambient density. The latter is not a major limitation because even in actual density-stratified environments, only the ambient density in the near-surface layer is dynamically important for these discharges and it can readily be assumed as uniform. Different discharge geometries and orientations can be analyzed including flush or protruding channel mouths, and orientations normal, oblique, or parallel to the bank.

Additional major assumptions include the following. All subsystems require that the actual cross-section of the water body be described as a rectangular channel that may be bounded laterally or unbounded. The ambient velocity is assumed to be uniform within that cross-section. In addition to a uniform ambient density possibility, CORMIX1 and 2 allow for four types of ambient stratification profiles to be used for the approximation of the actual vertical density distribution (see Section IV). All CORMIX subsystems are steady-state models and assume a conservative pollutant without decay or growth processes. To incorporate such mechanisms into an analysis, the user can modify model predictions with additional calculations.

C. System Processing Sequence and Structure

All three CORMIX subsystems have an identical system configuration. Figure 8 shows the overall structure and the execution sequence of the program elements of each CORMIXn subsystem where "n" stands for 1, 2 or 3, respectively. During program execution, the elements are loaded automatically and sequentially by the system. Each element provide user interaction and prompting in response to displayed information. This may somewhat extend the total time required for a single CORMIXn session, but has offsetting benefit of allowing the user to gain process knowledge and insight on design sensitivity.

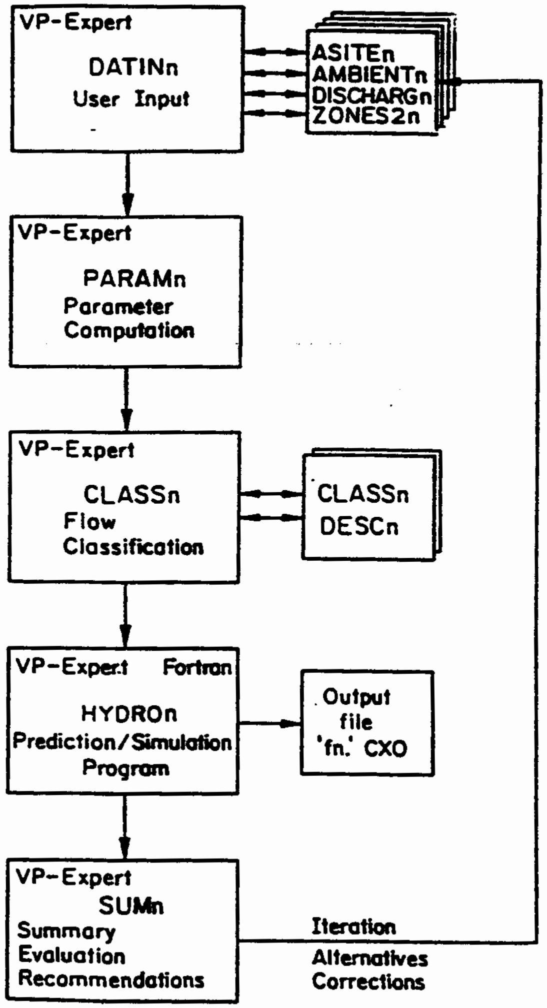


FIGURE 8 CORMIXn PROGRAM ELEMENTS AND PROCESSING SEQUENCE

The program elements of CORMIXn are composed of DATINn, PARAMn, CLASSn, HYDROn, and SUMn (Figure 8). DATINn is the program element for the entry of data and initialization of other program elements. PARAMn uses the input data to compute a number of important physical parameters and length scales, as precursor to CLASSn which performs the hydrodynamic classification of the given discharge/ambient situation into one of many possible generic flow configurations. HYDROn performs the actual detailed numerical prediction of the effluent plume characteristics. Finally, SUMn summarizes the results from the classification and prediction, interprets them as regards mixing zone regulations, suggests design alternatives, and allows sensitivity analysis to be conveniently conducted using the current input data.

Due to its diverse programming requirements, CORMIX is written in two programming languages: VP-Expert, an "expert systems shell", and FORTRAN. The former is powerful in knowledge representation and logical reasoning, while the latter is adept at mathematical computations. Program elements DATINn, PARAMn, CLASSn, and SUMn are written exclusively in VP-Expert. HYDROn is written in VP-Expert and FORTRAN.

D. Data Input Features

All data are entered interactively in response to the CORMIX system prompts generated by the data input program element DATINn. DATINn queries the user for a complete specification of the physical environment of the discharge, as well as the applicable regulatory considerations for the situation undergoing analysis. A CORMIX1 or 2 session commences with questions on four topics which are asked sequentially in this order: site/case descriptions, ambient conditions, discharge characteristics, and level of output detail and regulatory definitions. These questions are generated by program segments ASITEn, AMBIENTn, DISCHn, and ZONESn, respectively. Data entry is entirely guided by the system and the available advice menu options provide expanded descriptions of the questions, if clarification is needed. Section IV provides complete details on input specification for the three CORMIXn subsystems.

E. Logic Elements of CORMIXn: Flow Classification

To make predictions of an effluent discharge's dilution and plume projectory, CORMIXn typically combines the solutions of several simple flow patterns to provide a complete analysis from the efflux location all the way into the far-field. The logic processing elements of CORMIXn identify which solutions should be combined to provide the complete analysis. This process, called flow classification, develops a generic qualitative description of the discharge flow and is based on known relationships between flow patterns and certain calculated physical parameters.

PARAMn is the progra element that computes relevant physical parameters including: the various length scales, fluxes, and

other values needed for the execution of other program elements. Length scales are calculated measures of the length of dynamic influence of various physical processes (see Section IV & V).

At the heart of CORMIXn is a flow classification system contained in the program element CLASSn. It provides a rigorous and robust expert knowledge base that carefully distinguishes among the many hydrodynamic flow patterns that a discharge may exhibit. As examples, these possibilities include discharge plumes attaching to the bottom, plumes vertically mixing due to instabilities in shallow water, plumes becoming trapped internally due to density stratification, and plumes intruding upstream against the ambient current due to buoyancy, and many others. Theoretically based hydrodynamic criteria using length scale analysis and empirical knowledge from laboratory and field experimentation, are applied in a systematic fashion to identify the most appropriate flow classification for a particular analysis situation. For all three subsystems, a total of about 80 generic flow configurations or classes can be distinguished.

The classification procedure of CORMIXn is based on technical principles and has been verified by the developer through repeated testing and data comparison. It has also undergone independent peer review (10,11,12). The three documentation manuals (7,8,9) give the detailed scientific background for the classification scheme, in form of a number of criteria. The actual criteria constants are listed in the technical reports with comments on their sources and degree of reliability; they also can be inspected in the files \CMXn\DATAn\CONSTn. Experienced users, especially those involved in research applications, may want to inspect these files and occasionally vary some constant values within certain limits in order to examine improved prediction fits with available high-quality data. Extreme caution must be exercised when doing that as some values are interdependent; furthermore, if changes are made, they should be carefully documented.

When CLASSn has executed, a description of the particular flow class is available to the user in the form of on-screen or hardcopy computer output; these description are also contained in the documentation manuals (7,8,9). It is recommended that the novice or intermediate user review these to gain an appreciation of the involved hydrodynamic mixing processes.

F. Simulation Elements of CORMIXn: Flow Prediction

Once a flow has been classified, CORMIXn assembles and executes a sequence of appropriate hydrodynamic simulation modules in the program element HYDROn. HYDROn consists of: (a) control programs or "protocols" for each hydrodynamic flow classification and (b) a large number of subroutines or "simulation modules" corresponding to the particular flow processes, and their associated spatial regions, that occur within a given flow classification. The simulation modules are based on buoyant jet similarity theory, ambient diffusion theory, and stratified flow theory, and on simple dimensional analysis, as described elsewhere (7,8,9). The basic tenet of the simulation methodology is to arrange a sequence of relatively simple simulation modules which, when executed together, predict the trajectory and dilution characteristics of a complex flow. Each of the simulation models uses the final values of the previous module as "initial conditions".

G. System Output Features: Design Summary and Iterations

SUMn is the final program element that summarizes the hydrodynamic simulation results for the case under consideration. The output of SUMn is arranged in four groups:

(1) Site summary gives the site identifier information, discharge and ambient environment data, and discharge length scales.

(2) Hydrodynamic simulation and mixing zone summary lists conditions at the end of the hydrodynamic mixing zone (HMZ), regulatory mixing zone (RMZ) conditions, toxic dilution zone (TDZ) conditions, region of interest (ROI) conditions, upstream intrusion information, bank attachment locations, and a passive diffusion mixing summary. Users should be cognizant of the four major zone definitions, and associated acronyms, introduced above and defined as follows:

Hydrodynamic Mixing Zone (HMZ): The HMZ is simply the zone of strong initial mixing, corresponding to the "near-field" processes discussed in Section II. It has no regulatory implication whatsoever. However, the information on size and mixing conditions at the edge of the HMZ is given as a useful guide to the discharge designer because mixing in the HMZ is usually sensitive to design conditions, and therefore somewhat controllable. A notable exception is the effluent discharge into very shallow flow-limited streams where the actual discharge port design detail may have little bearing on instream concentrations.

Regulatory Mixing Zone (RMZ): The RMZ corresponds to either: (1) the applicable mixing zone regulation with specified size dimensions, or (2) a preliminary proposal for a mixing zone (see Subsection IIB(3)). In CORMIX1 and 2 versions prior to August 1991, and in the some reports, this was labeled inconsistently as the Legal Mixing Zone (LMZ).

Toxic Dilution Zone (TDZ): The TDZ corresponds to the EPA's definition of where toxic chemical concentrations may exceed the CMC value (see Subsection IIB(3)).

Region of Interest (ROI): The ROI is a user defined region of the receiving waterbody where mixing conditions are to be analyzed. It is specified as the maximum analysis distance in the direction of mixed effluent flow and is particularly

important when legal mixing zone restrictions do not exist or when information over a larger area is of interest.

(3) Data analysis section presents further details on toxic dilution zone criteria, regulatory mixing zone criteria, stagnant ambient environment information, and region of interest criteria.

(4) Design recommendations section contains design suggestions in three general areas for improving initial dilution. These include: (a) geometry variations in discharge port design, (b) sensitivity to ambient conditions, and (c) process variations in discharge flow characteristics. The user is given guidance on the potential changes in mixing conditions from varying parameter values within these groups.

The user can choose among the following printed output options, all in text form: (a) a summary of the entire CORMIX session, (b) the summary contained in the SUMn element as described above, (c) printouts of the qualitative flow class description, and (d) a detailed file giving the numerical values of the hydrodynamic simulation generated in program element HYDROn. At present, CORMIX does not contain any graphical output display.

Finally, SUMn is also used as an interactive loop to guide the user back to DATINn to alter design variables and perform sensitivity studies. Different options for iteration exist depending on what input data changes are to be made. The importance of performing an ample number of CORMIX iterations cannot be sufficiently stressed. To obtain a design that adequately meets water quality and engineering construction objectives, it is necessary to get a feel for the physical situation and its sensitivity to design changes through repeated system use.

H. Equipment Requirements, System Installation and Run Times

The minimum hardware configuration required for CORMIX is an IBM-PC/XT compatible microcomputer with: (a) a minimum of 582Kb of available RAM memory, (b) a math co-processor, (c) approximately 1Mb of hard disk space per installed subsystem, and (d) DOS 3.3 or higher operating system. Computers with more advanced main processors (e.g. 80386) are preferable because they substantially reduce the time to complete an analysis.

The RAM memory requirement of CORMIX may present an obstacle to many users because the configuration requirements of many commercial applications packages and the installation of memory resident software frequently reduce available RAM memory to less than 582kB. The amount of available RAM memory can be determined with the DOS command CHKDSK. Although there are numerous approaches for increasing the size of a computer's available RAM memory, the simplest way is "boot" the computer from a floppy "system" disk that contains no AUTOEXEC.BAT file and a CONFIG.SYS file which contains only the statements "FILES=20" and "BUFFERS=20" on separate lines. This should be done just prior to beginning an analysis session since it will temporary disable programs that consume RAM memory. At the completion of the analysis session, the computer should be "booted" from the hard drive to restore normal operations. A bootable floppy system disk can be created with the DOS command FORMAT a:/S.

The CORMIXn subsystems must be installed on a hard disk drive. The directory structure of each subsystem CORMIXn (<u>Table 1</u>) is fixed; it gets set up during the installation process; and it consists of a subsystem root directory, called "CMXn", and eight daughter directories, all ending with the suffix "n". Complete installation instructions are available with CORMIXn distribution diskettes.

Depending on computer configuration, a typical CORMIX2 session for one discharge/ambient condition may take about 5 minutes for an advanced 80386-based computer to about 20 minutes for an 80286-based computer if all necessary input data is at hand.

IV DATA INPUT

A. General Aspects of Interactive Data Input

The three subsystems have similar data input features and requirements. Data input occurs interactively in response to CORMIX system prompts and is entirely guided by the system. The user is automatically prompted for a complete specification of: site/case descriptions, ambient conditions, discharge characteristics, and level of output detail desired and regulatory definitions. The data for each of these four topical areas are called input data sequences herein. Questions are asked in plain English. Advice menu options within the program are available to provide help on how to prepare and enter data values when clarification of the system prompts is needed. The contents these are also available in the documentation manuals (7,8,9).

Data can be entered in an open format without concern for letter case or decimal placement. The only constraint is that <u>the following characters may not be entered</u> in response to any question:

+ = { } , < > ' " / \ ;

The system checks data entries for consistency with question type (e.g. a alpha character for water depth), obvious physical errors (e.g. a negative length), possible inconsistencies with

TABLE 1 DIRECTORY STRUCTURE FOR SUBSYSTEM CORMIXn

Directory Name	Comments
pd\CMXn	system root directory; contains VP-Expert system files and the knowledge base program CORMIXn (system driver)
pd\CMXn\ADVICEn	contains all user-requested advice files
pd\CMXn\BATn	contains DOS batch files for program execution, data file manipulation, and program control
pd\CMXn\CACHEn	contains cache "fact" files exported from knowledge base programs
pd\CMXn\DATAn	contains constants used in flow classification and other knowledge base programs
pd\CMXn\DESCn	contains flow descriptions for each flow class
pd\CMXn\KBSn	contains all knowledge base programs
pd\CMXn\PGMSn	contains Fortran hydrodynamic simulation program and file manipulation programs
pd\CMXn\SIMn	contains simulation results

Note: n = 1, 2, or 3
pd = parent directory
Examples for pd:
pd = d:\CORMIX (recommended)
pd = d: (if installation directly on hard disk drive)
pd = d:\MIXING\CORMIX (where "MIXING" may be a major directory containing all effluent related work)
d: = valid hard disk drive

previous entries (e.g. an angular value implying that a port points directly back to the shoreline) and situations outside the ranges of model applicability. Inconsistency with question type and obvious physical errors require immediate re-entry while possible inconsistencies with previous entries lead to a warning label and the opportunity for later correction. Entries specifying situations outside the ranges of model applicability usually require the re-entry of the entire data segment.

As discussed in Section III, data input occurs in three or four program segments that load automatically. Once a complete input data sequence has been specified, it is gets summarized on-screen for the user to check. If something needs to be changed, the entire input data sequence has to be re-entered in correct form.

Due to the similarity of data entry for the three subsystems, a common description is given for all input data sequences, except discharge data to which a separate subsection for each CORMIXn subsystem is devoted below. Further guidance on data specification can be obtained from examining the case studies in the Appendices and from the documentation manuals (7,8,9). Following the discussion of input data sequences, units of measure conversion factors and checklists for input preparation are presented.

B. Site/Case Identifier Data

The first input data sequence determines basic information needed for the program to operate. These include: a four-part identifier for labeling output, a computer file name and whether to echo screen output to a printer.

It is necessary to specify four site/case labels that facilitate the rapid identification of printed output and aid in good record-keeping. The system provides for one label called SITE NAME (e.g. Blue River), another called DISCHARGER (e.g. B-Company), another called POLLUTANT (e.g. Mercury), and another called DESIGN CASE (e.g. 7Q10-low-flow, or High-velocity-port).

The user needs to supply a DOS-compatible FILE NAME, up to eight characters long, and without extension (e.g. sdif7q10). CORMIX will use that user-specified file name "fn", and create, transfer, or store intermediate or final data files with that same file name, but with different extensions. The most important of these are the two output data files, "fn".CXS and "fn".CXO which are discussed further in Section V.

Finally, the user is also asked to specify whether a hard-copy record of the CORMIX session should be recorded on an attached printer. Since several iterations of an analysis are typically required to gain a thorough understanding of the mixing processes at a site, it is recommended that this feature be used at all times and that the printouts be retained for later reference.

C. Ambient Data

Ambient conditions are defined by the geometric and hydrographic conditions in the vicinity of the discharge. Due to the significant effect of boundary interactions on mixing processes, the ambient data requirements for the laterally bounded and unbounded analysis situations are presented separately in the discussions below. Following these is a discussion on ambient density specification.

CORMIX requires that the actual cross-section of the ambient waterbody be described by a rectangular channel that may be bounded laterally or unbounded. Furthermore, that channel is assumed to be uniform in the downstream direction, following the mean flow of the actual water body that may be non-uniform or meandering. The process of describing a receiving waterbody's geometry with a rectangular cross-section is herein called schematization.

The first step towards specifying the ambient conditions is to determine whether a receiving waterbody should be considered "bounded" or "unbounded." To do this, as well as answer other questions on the ambient geometry, it is usually necessary to have access to cross-sectional diagrams of the waterbody. These should show the area normal to the ambient flow direction at the discharge site and at locations further downstream. If the waterbody is constrained on both sides by banks such as in rivers, streams, narrow estuaries, and other narrow watercourses, then it should be considered "bounded." However, in some cases the discharge is located close to one bank or shore while the other bank is for practical purposes very far away. When interaction of the effluent plume with that other bank or shore is impossible or unlikely, then the situation should be considered "unbounded." This would include discharges into wide lakes, wide estuaries, and coastal areas.

(1) Bounded Cross-Section- Both geometric (bathymetric) and hydrographic (ambient discharge) data should be used for defining the appropriate rectangular cross-section. This schematization may be quite evident for well-channeled and regular rivers or artificial channels. For highly irregular cross-sections, it may require more judgment and perhaps several iterations of the analysis to get a better feel on the sensitivity of the results to the assumed cross-sectional shape.

In any case, the user is advised to consider the following comments:

a) Be aware that a particular flow condition such as a river discharge is usually associated with a certain water surface elevation or "stage." Data for a stage-discharge relationship is normally available from a USGS office; otherwise it can be obtained from a separate hydraulic analysis or from field measurements.

In the simplest case of a river flow, if river depth is known for a certain flow condition (subscript 1 in the following) corresponding perhaps to the situation at the time of a field study, then the depth for a given design (e.g. low) flow (subscript 2) can be predicted from the Manning's equation

$$HA_2 = HA_1 \left[\frac{QA_2}{QA_1}\right]^{\frac{3}{5}}$$

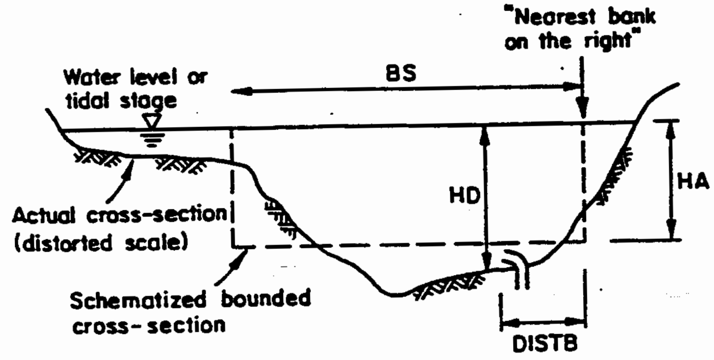
in which QA is the ambient river flow and HA the mean ambient depth. This approach assumes that the both the ambient width and frictional characteristics of the channel (i.e. Manning's n) remain approximately the same during such a stage change.

b) For the given stage/river discharge combination to be analyzed, assemble plots showing the cross-sections at the discharge and several downstream locations. Examine these to determine an "equivalent rectangular cross-sectional area." Very shallow bank areas or shallow floodways may be neglected as unimportant for effluent transport. Also, more weight should be given to the cross-sections at, and close to, the discharge location since these will likely have the greatest effect on near-field processes. Figure 9a provides an example of the schematization process for a river or estuary cross-section.

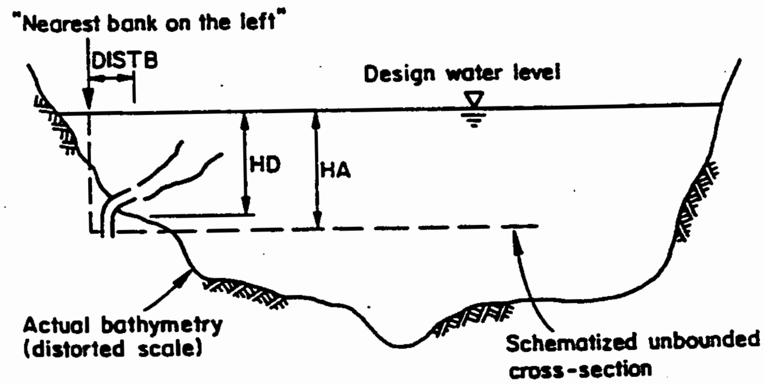
c) The input data values for surface width (BS) and (average) depth (HA) should be determined from the equivalent rectangular cross-sectional area. When ambient discharge and ambient velocity data are available, the reasonableness of the schematization should be checked with the continuity relation. It specifies that ambient discharge equals velocity times cross-sectional area, where the area is given by the product of average width and depth.

d) CORMIX1 and 2 also require specification of the actual water depth (HD) at the submerged discharge location which is uniquely different from the depth to the discharge port centerline. A check is built into CORMIX1 and 2 allowing the local depth HD not to differ from the schematized average depth HA by more than +/- 30%. This restriction is included to prevent CORMIX misuse in several discharge/ambient combinations involving strongly non-uniform channels. Alternative schematizations can be explored by the user to work around the restriction. The choice for these alternatives may be influenced somewhat by the expected plume pattern. As an example, Figure 9b illustrates a small buoyant discharge that is located on the side slope of a deep reservoir and that is rising upward. In this situation, the correct representation of the deeper mean reservoir depth is irrelevant for plume predictions. Although the illustration is for an unbounded example, the comments on choice of HA apply here, too.

When schematizing HA and HD in highly non-uniform conditions, HD is the variable that usually influences near-field



a) Example: Bounded Cross-Section Looking Downstream (River or Estuary)



b) Example: Unbounded Cross-Section Looking Downstream (Small Buoyant Jet Discharge Into Large Lake or Reservoir)

FIGURE 9 EXAMPLES OF THE SCHEMATIZATION PROCESS mixing, while HA is important for far-field transport and never influences the near-field.

CORMIX3 uses the variable HD for the actual water depth which is observed at the channel entry (HD) location and it requires an additional specification for the receiving water bottom slope (THETAB). THETAB is the slope of the receiving water bottom surface in the direction perpendicular to the shoreline. These details are important for identifying cases where plume attachment to the bottom can occur.

e) The ambient discharge (QA) or mean ambient velocity (UA) may be used to specify the ambient flow condition. Depending which is specified, the program will calculate and display the other. The displayed value should be checked to see whether it is consistent with schematizations and continuity principles discussed above.

The simulation of stagnant conditions should usually be avoided. If zero or a very small value for ambient velocity or discharge is entered, CORMIX will label the ambient environment as stagnant. In this case, CORMIX will predict only the near-field of the discharge, since steady-state far-field processes require a mean transport velocity. Although stagnant conditions often, but not necessarily always, represent the extreme limiting case for a dilution prediction, a real waterbody never is truly stagnant. Therefore, a more realistic assumption for natural waterbodies would be to consider a small, but finite ambient crossflow.

f) As a measure of the roughness characteristics in the channel the value of Manning's n, or alternatively of the Darcy-Weisbach friction factor f, must be specified. Friction values are useful for applications in laboratory studies. If Manning's n is given, as is preferable for field cases, CORMIX internally converts it to an f friction value using the following equation

$$f = 8g \frac{n^2}{HA^{1/3}}$$

in which $g = 9.81\ m/s^2$.

The friction parameters influence the mixing process only in the final far-field diffusion stage, and do not have a large impact on the predictions. Generally, if these values can be estimated within +/-30%, the far-field predictions will vary by +/-10% at the most. The following list is a brief guide for specification of Manning's n values; additional details are available in Chow (20).

Channel type	Manning's n
Smooth earth channel, no weeds	0.020
Earth channel, some stones and weeds	0.025
Clean and straight natural rivers	0.025 - 0.030
Winding channel, with pools and shoals	0.033 - 0.040
Very weedy streams, winding, overgrown	0.050 - 0.150
Clean straight alluvial channels	$0.031\ d^{1/6}$
(d = 75% sediment grain size in feet)	

(2) Unbounded Cross-section- Both hydrographic and geometric information are closely linked in this case. The following comments apply:

a) From lake or reservoir elevation or tidal stage data, determine the water depth(s) for the receiving water condition to be analyzed.

b) For the given receiving water condition to be analyzed, assemble plots showing water depth as a function of distance from the shore for the discharge location and for several positions downstream along the ambient current direction.

c) If detailed hydrographic data from field surveys or from hydraulic numerical model calculations are available, determine the "cumulative ambient discharge" from the shore to the discharge location for the discharge cross-section. For each of the subsequent downstream cross-sections, determine the distance from the shore at which the same cumulative ambient discharge has been attained. Mark this position on all cross-sectional profiles. Examine the vertically averaged velocity and the depth at these positions to determine typical values for the ambient depth (HA) and ambient velocity (UA) input specifications. The conditions at, and close to, the discharge location should be given the most weight. The distance from the shore (DISTB) for the outfall location is typically specified as the cumulative ambient discharge divided by the product UA times HA.

When detailed hydrographic data are unavailable, data or estimates of the vertically averaged velocity at the discharge location can be used to specify HA, UA, and DISTB. First, determine the cumulative cross-sectional area from the shore to the discharge location for the discharge cross-section. For each of the subsequent downstream cross-sections, mark the position where the cumulative cross-sectional area has the same value as at the discharge cross-section. Then proceed as discussed in the preceding paragraph.

d) The specification of the actual water depth at the submerged discharge location (HD) in CORMIX1 and 2 is governed by considerations that are similar to those discussed earlier for bounded flow situations (see Subsection IVC(1)d). Figure 9b shows an illustration of the schematization for a small buoyant discharge located on the side slope of a deep reservoir. The plume is expected to rise upward and stay close to one shore, with bottom contact and vertical mixing not expected. In this situation, no emphasis on replicating the mean reservoir depth and the actual width is necessary. However, care must still be taken to specify an ambient mean velocity that is: (a) characteristic of the actual reservoir and (b) not determined using the reduced depth assumption.

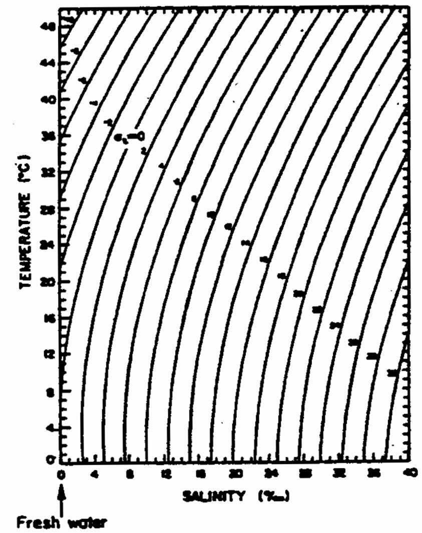
The specification of HD for CORMIX3 is identical to the bounded case.

e) Either Manning's n or the Darcy-Weisbach friction factor f can be specified for the ambient roughness characteristics as described previously for the bounded case (see <u>Subsection IVC(1)e</u>). If the unbounded case represents a large lake or coastal area, it is often preferable to use the friction factor f. Typical f values for such open waterbodies range from 0.020 to 0.030, with larger values for rougher conditions.

(3) <u>Ambient Density Specification:</u> Information about the density distribution in the ambient water body is very important for the correct prediction of effluent discharge plume behavior. CORMIX first inquires whether the ambient water is fresh water or non-fresh (i.e. brackish or saline). If the ambient water is fresh and above 10 °C, the system provides the option of entering ambient temperature data so that the ambient density values can be internally computed from an equation of state. This is the recommended option for specifying the density of fresh water, even though ambient temperature <u>per se</u> is not needed for the analysis of mixing conditions. In the case of salt water conditions, <u>Figure 10</u> is included as a practical guide for specifying the density if "<u>salinity values</u>" in parts-per-thousand (ppt or ‰) are available for the waterbody. Typical open ocean salinities are in the range 33 - 35 ppt.

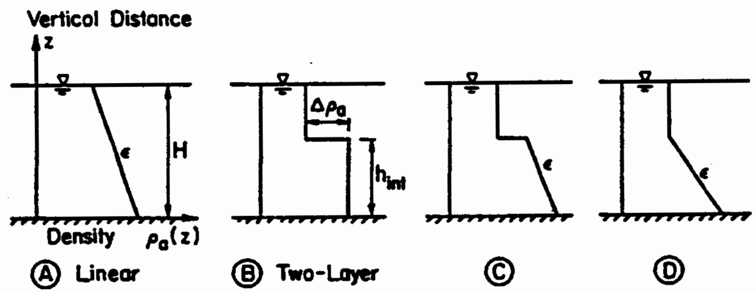
In CORMIX1 and 2, the user then specifies whether the ambient density (or temperature) can be considered as uniform or as non-uniform within the water body, and in particular within the expected plume regions. As a practical guide, vertical variations in density of less than 0.1 kg/m³ or in temperature of less than 1 °C can be neglected. For uniform conditions, the average ambient density or average temperature must be specified.

When conditions are non-uniform, CORMIX1 and 2 require that the actual measured vertical density distribution be approximated by one of four schematic stratification profile types illustrated in <u>Figure 11</u>. These are: Type A, linear density profile; Type B, two-layer system with constant densities and density jump; Type C, constant density surface layer with linear density profile in bottom layer separated by a density jump; and Type D, constant density surface layer with linear density profile in bottom layer without a density jump. Corresponding profile types exist for approximating a temperature distribution when it is used for specifying the density distribution.



Density $[kg/m^3] = 1000.0 + \sigma_t$

<u>FIGURE 10</u> RELATIONSHIP BETWEEN SALINITY, TEMPERATURE AND SEAWATER DENSITY



<u>FIGURE 11</u> POSSIBLE APPROXIMATIONS FOR REPRESENTING AMBIENT DENSITY STRATIFICATION

After selecting the stratification approximation to be used, the user then enters all appropriate density (or temperature) values and pycnocline heights (HINT) to fully specify the profiles. The pycnocline is defined as zone or level of strong density change that separates the upper and lower layers of the water column (see Figure 11b). The program checks the density specification to insure that stable ambient stratification exists (i.e. the density at higher elevations must not exceed that at lower elevations).

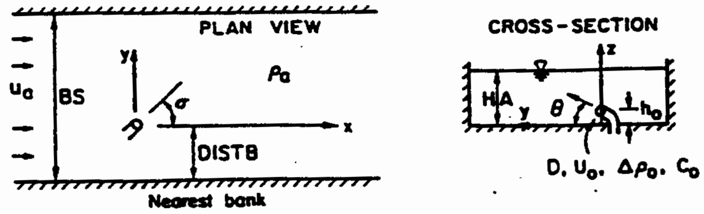
Note that a dynamically correct approximation of the actual density distribution should keep a balance between over- and under-estimation of the actual data similar to a best-fit in regression analysis. If simulation results indicate internal plume trapping, then it is desirable to test --through repeated use of CORMIX-- different approximations (i.e. with different stratification types and/or parameter values) in order to evaluate the sensitivity of the resulting model predictions.

In CORMIX3, only a uniform average ambient density possibility exists because this subsystem applies only to buoyant discharges occurring at or near the water surface. When ambient stratification exists, an average value representing the upper layer of the water body should be specified. For most cases, the depth of the discharge channel is a useful averaging distance. However, for strongly mixing cases including those with near-field instability and bottom attachment, a larger averaging depth may be warranted after inspection of the initial simulation results.

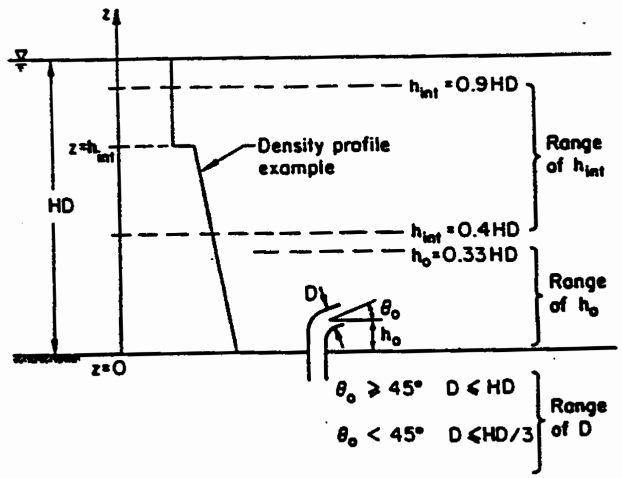
D. Discharge Data: CORMIX1

Figure 12a is a definition sketch giving the geometry and flow characteristics for a submerged single port discharge within the schematized cross-section.

(1) Discharge Geometry- To allow the establishment of a reference coordinate system and orient the discharge to that reference, CORMIX1 requires the specification of 6 data entries. These specifications are illustrated in Figure 12a and include: (a) location of the nearest bank (i.e. left or right) as seen by an observer looking downstream in the direction of the flow, (b) distance to the nearest bank (DISTB), (c) port radius or cross-sectional area for non-circular shaped ports, (d) height of the port (HO) center above the bottom, (e) vertical angle of discharge (THETA) between the port centerline and a horizontal plane, and (f) horizontal angle of discharge (SIGMA) measured counterclockwise from the ambient current direction (x-axis) to the plan projection of the port centerline. Angle THETA may range between -45 and 90 °. As examples, the vertical angle is 90 degrees for a discharge pointing vertically upward, and it is 0 ° for a horizontal discharge. Angle SIGMA may range between 0 and 360 °. As examples, the horizontal angle is 0 ° (or 360 °) when the port points downstream in the ambient flow direction,



a) Definition Diagram CORMIX1 (Special case: HA=HD)



b) Limits of Applicability CORMIX1

FIGURE 12 CORMIX1 DISCHARGE GEOMETRY AND RESTRICTIONS and it is 90 °, when the port points to the left of the ambient flow direction.

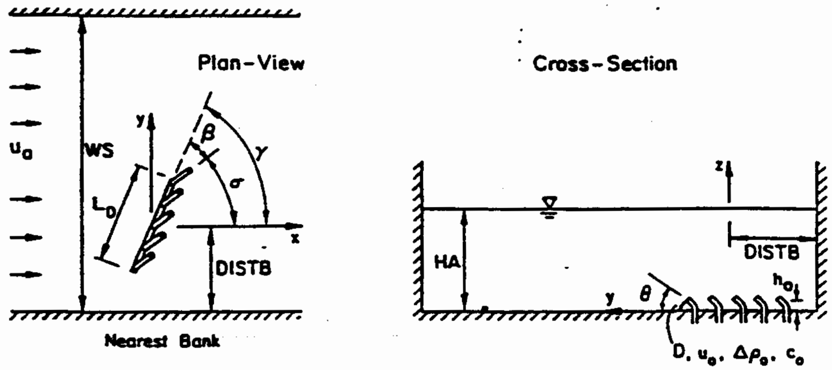
In order to prevent an inappropriate system application, CORMIX1 checks the specified geometry for compliance with the three criteria illustrated in Figure 12b. These are: (a) the port height (HO) value must not exceed one-third of the local water depth (HD) value, (b) the port diameter value must not exceed HD's value for near-vertical designs, and one-third of HD's value for near-horizontal designs, and (c) the pycnocline value must be within the 40 to 90 percent range of HD's value. The port height restriction results from the fact that CORMIX1 only applies to submerged discharge applications. In ordinary design practice, submerged implies a discharge close to the bottom, and not anywhere within the main water column or near the water surface. The port diameter restriction excludes very large discharge diameters relative to the actual water depth since these are unrealistic and/or undesirable. The distance separating the upper and lower layers of the ambient density profile type B, C, or D is restricted in order to prevent: (a) discharges into the upper layer or (b) an unrealistically thick plume relative to a thin upper layer. For those few extreme situations that would normally be limited by the above restrictions, Section 7.4 of Doneker and Jirka (7) contains a number of hints on how to conduct these difficult analyses; only advanced users should attempt these techniques.

(2) Port Discharge Flow- For discharge characteristics, CORMIX1 requires the specification of 3 data entries. These specifications include: (a) the discharge flow rate (Q0) or discharge velocity (U0), (b) the discharge density or discharge temperature for an essentially freshwater discharge, and (c) the discharge concentration of the material of interest. The Q0 and U0 variables are related through the port cross-sectional area and the program computes and displays the alternate value allowing for user inspection and verification. For a freshwater discharge, discharge density can be directly related to temperature via an equation of state since the addition of any pollutant or tracer has negligible effect on density.

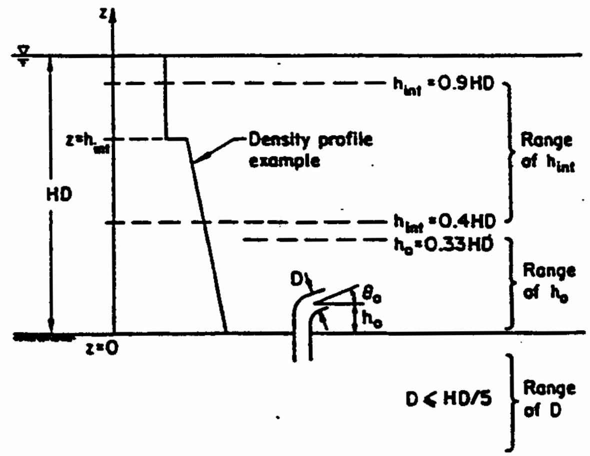
The discharge concentration of the material of interest (pollutant, tracer, or temperature) is defined as the excess concentration above any ambient concentration of that same material. The user can specify this quantity in any units. CORMIX1 predictions should be interpreted as computed excess concentrations in these same units. If no specific pollutant is under consideration, simply specify a discharge concentration of 100%.

E. Discharge Data: CORMIX2

A generalized definition sketch showing the geometry and flow characteristics for a typical multiport diffuser installation is provided in Figure 13a. Due to the great number



a) Definition Diagram CORMIX2 (Special case: HA=HD)



b) Limits of Applicability CORMIX2

FIGURE 13 CORMIX2 DISCHARGE GEOMETRY AND RESTRICTIONS of complexities which may rise in describing an existing or proposed diffuser design, a few definitions are introduced prior to discussing actual data requirements of CORMIX2.

A multiport diffuser is a linear structure consisting of many more or less closely spaced ports or nozzles which inject a series of turbulent jets at high velocity into the ambient receiving waterbody. These ports or nozzles may be connected to vertical risers attached to an underground pipe or tunnel or they may simply be openings in a pipe lying on the bottom.

The diffuser line (or axis) is a line connecting the first port or nozzle and the last port or nozzle. Generally, the diffuser line will coincide with the connecting pipe or tunnel. CORMIX2 will assume a straight diffuser line. If the actual diffuser pipe has bends or directional changes it must be approximated by a straight diffuser line.

The diffuser length is the distance from the first to the last port or nozzle. The origin of the coordinate system used by CORMIX2 is located at the center (mid-point) of the diffuser line. The only exception is when the diffuser line starts at the shore; then the origin is located directly at the shore.

CORMIX2 can analyze discharges from the three major diffuser types used in common engineering practice. These are illustrated in Figure 14 and include: (a) the unidirectional diffuser where all ports (or nozzles) point to one side of the diffuser line and are oriented more or less normally to the diffuser line and more or less horizontally; (b) the staged diffuser where all ports point in one direction generally following the diffuser line with small deviations to either side of the diffuser line and are oriented more or less horizontally; and (c) the alternating diffuser where the ports do not point in a nearly single horizontal direction. In the latter case, the ports may point more or less horizontally in an alternating fashion to both sides of the diffuser line or they may point upward, more or less vertically.

(1) Diffuser Geometry- CORMIX2 assumes uniform discharge conditions along the diffuser line. This includes the local ambient receiving water depth (HD) and discharge parameters such as port size, port spacing and discharge per port, etc. If the actual receiving water depth is variable (e.g. due to an offshore slope), it should be approximated by the mean depth along the diffuser line with a possible bias to the more shallow near-shore conditions. Similarly, mean values should be used to specify variable diffuser geometry when it occurs.

To allow the establishment of a reference coordinate system and orient the discharge to that reference, CORMIX2 requires the specification of 12 data entries. These specifications are illustrated in Figure 13a and include: (a) location of the nearest bank (i.e. left or right) as seen by an observer looking

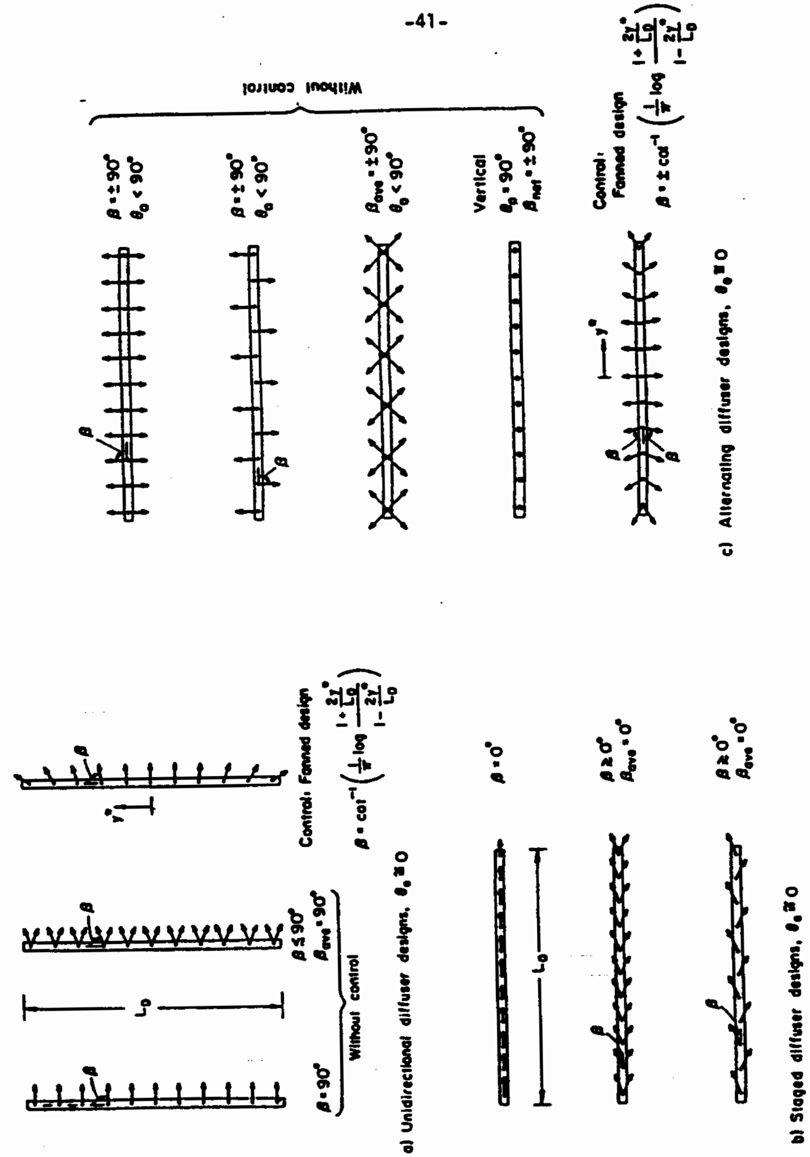


FIGURE 14 CONFIGURATIONS OF COMMON MULTIPORT DIFFUSER TYPES

downstream in the direction of the flow, (b) average distance to the nearest bank (DISTB), (c) average diameter (D0) of the discharge ports or nozzles, (d) average height of the port centers (H0) above the bottom, (e) average vertical angle of discharge (THETA) between the port centerlines and a horizontal plane (-45 and 90 °), (f) for the unidirectional and staged diffusers only, the average horizontal angle of discharge (SIGMA) measured counterclockwise from the ambient current direction (x-axis) to the plan projection of the port centerlines (0 to 360 °), (g) approximate straight-line diffuser length (LD) between the first and last ports or risers, (h) distance from the shore to the first and last ports or risers (YB1, YB2) of the diffuser line, (i) number of ports or risers and the number of ports per riser if risers are present, (j) jet contraction coefficient value, (k) average alignment angle (GAMMA) measured counterclockwise from the ambient current direction (x-axis) to the diffuser axis (0 to 180 °), and (l) for the unidirectional and staged diffusers only, relative orientation angle (BETA) measured either clockwise or counterclockwise from the average plan projection of the port centerlines to the nearest diffuser axis (0 to 90 °). Note that CORMIX2 always assumes a uniform spacing between risers or between ports, and a round port cross-sectional shape.

As examples of angle specifications, THETA is 0 degrees for a horizontal discharge and it is +90 degrees for a vertically upward discharge, SIGMA is 0 degrees (or 360 °) when the ports point downstream in the ambient flow direction and it is 90 degrees when the ports point to the left of the ambient flow direction, GAMMA is 0 degrees (or 180 °) for a parallel diffuser and it is 90 degrees for a perpendicular diffuser, and BETA is 0 degrees for a staged diffuser and it is 90 degrees for a unidirectional diffuser.

CORMIX2 performs a number of consistency checks to ensure the user does not make arithmetical errors when preparing and entering the above data and it also checks the specified geometry for compliance with three criteria to prevent an inappropriate system application. Figure 13b shows the imposed limits of system application for CORMIX2 which are: (a) the port height (H0) value must not exceed one-third of the local water depth (HD) value, (b) the port diameter value must not exceed one-fifth of HD's value, and (c) the pycnocline value must be within the 40 to 90 percent range of HD's value. The restrictions are similar to those shown in Figure 12b for CORMIX1 with the exception of the diameter limit for each port.

(2) Diffuser Discharge Flow- For discharge characteristics, CORMIX2 requires the specification of 3 data entries. These specifications include: (a) the total discharge flow rate (Q0) or discharge velocity (U0), (b) the discharge density or discharge temperature for an essentially freshwater discharge, and (c) the discharge concentration of the material of interest. The Q0 and U0 variables are related through the total cross-sectional area

of all diffuser ports and the program computes and displays the alternate value allowing for user inspection and verification.

The discharge concentration of the material of interest (pollutant, tracer, or temperature) is defined as the excess concentration above any ambient concentration of that same material. The user can specify this quantity in any units. CORMIX2 predictions should be interpreted as computed excess concentrations in these same units.

F. Discharge Data: CORMIX3

A definition sketch for the discharge geometry and flow characteristics for a buoyant surface discharge is provide in Figure 15. In general, CORMIX3 allows for different types of inflow structures, ranging from simple rectangular channels to horizontal round pipes that may be located at or near the water surface. In addition, three different configurations relative to the bank are allowed as illustrated in Figure 16. Discharge structures can be: (a) flush with the bank/shore, (b) protruding from the bank or (c) co-flowing along the bank.

(1) Discharge Geometry- To allow the establishment of a reference coordinate system and orient the discharge to that reference, CORMIX3 requires the specification of 5 data entries. These specifications are illustrated in Figure 15 and include: (a) location of the nearest bank (i.e. left or right) as seen by an observer looking downstream in the direction of the flow, (b) discharge channel width (B0), (c) discharge channel depth (H0), (d) bottom slope (THETAB) in the receiving water body in the vicinity of the discharge channel, and (e) horizontal angle of discharge (SIGMA) measured counterclockwise from the ambient current direction (x-axis) to the plan projection of the port centerline. In all cases, CORMIX3 assumes the discharge is being issued horizontally.

In the case of a circular pipe discharge CORMIX3 assumes the outlet is flowing full and that it is not submerged under the water surface by more than 1/2 of the outlet diameter. If the discharge outlet has an odd cross-sectional shape (e.g. a pipe flowing partially full) then it should be represented schematically as a rectangular outlet of the same cross-sectional area and similar channel depth.

For open channel discharges, considerable care should be exercised when specifying discharge channel depth since this parameter is directly linked to the ambient receiving water depth (stage). This is especially important for tidal situations.

To prevent an inappropriate system application, CORMIX3 only allows for a discharge channel depth-to-width aspect ratio of 0.05 to 2. This prohibits the use of extremely oblong discharge geometries.

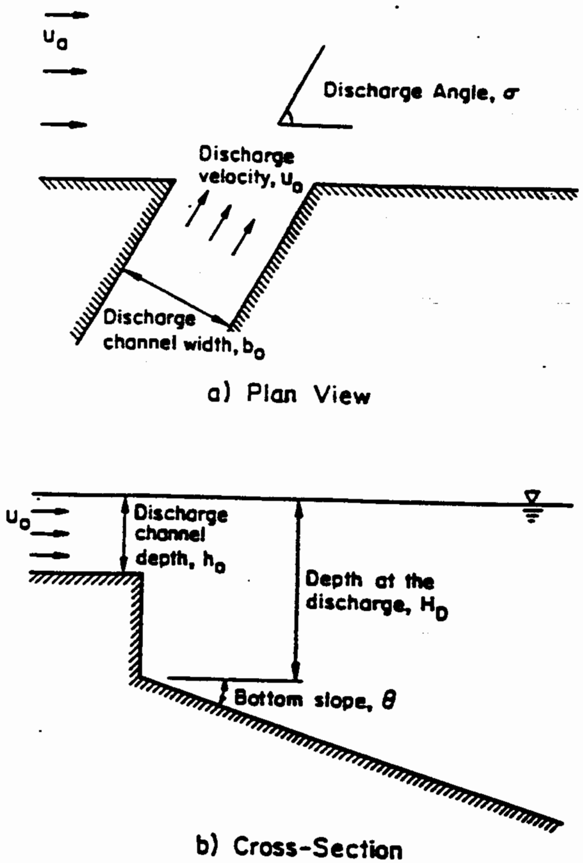


FIGURE 15 CORMIX3 DISCHARGE GEOMETRY

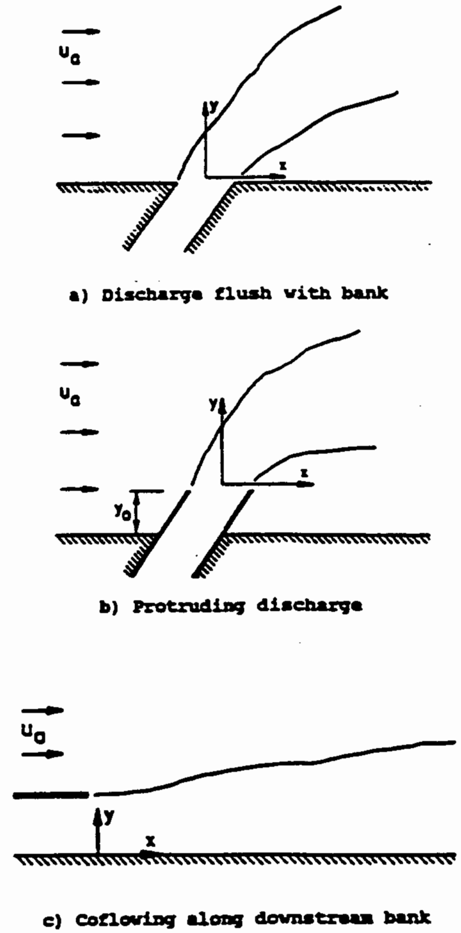


FIGURE 16 POSSIBLE DISCHARGE CONFIGURATION IN CORMIX3

(2) Discharge Flow- For discharge characteristics, CORMIX3 requires the specification of 3 data entries. These specifications include: (a) the total discharge flow rate (Q0) or discharge velocity (U0), (b) the discharge density or discharge temperature for an essentially freshwater discharge, and (c) the discharge concentration of the material of interest. The Q0 and U0 variables are related through the channel cross-sectional area; the program computes and displays the alternate value allowing for user inspection and verification.

The discharge concentration of the material of interest (pollutant, tracer, or temperature) is defined as the excess concentration above any ambient concentration of that same material. The user can specify this quantity in any units. CORMIX3 predictions should be interpreted as computed excess concentrations in these same units.

G. Mixing Zone Data

For the program element SUMn to tailor the hydrodynamic simulation results summary to the situation undergoing analysis, the user must indicate: (a) whether EPA's toxic dilution zone (TDZ) definitions apply, (b) whether an ambient water quality standard exists, (c) whether a regulatory mixing zone (RMZ) definition exists, (d) the spatial region of interest (ROI) over which information is desired, and (e) number of locations (i.e. "grid intervals") in the ROI to display output details. Depending on the responses to the above, several additional data entries may be necessary as described in the following paragraphs.

When TDZ definitions apply, the user must also indicate the criterion maximum concentration (CMC) and criterion continuous concentration (CCC) which are intended to protect aquatic life from acute and chronic effects, respectively. CORMIX will check for compliance with: (a) the CMC standard at the edge of the TDZ and (b) the CMC standard at the edge of the RMZ, proving a RMZ was defined. See Subsection IIIG(2) for additional discussion.

When RMZ definition exists, it can be specified by: (a) a distance from the discharge location, (b) the cross-sectional area occupied by the plume, or (c) the width of the effluent plume.

The ROI, which is a user defined region where mixing conditions are to be analyzed, is specified as the maximum analysis distance in the direction of mixed effluent flow. The level of detail for the output data within the ROI and thus, for the entire hydraulic simulation, is established by specifying the number of grid intervals that will be displayed in the output files. This parameter's allowable range is 3 to 50 and the chosen value does not affect the accuracy of the CORMIX prediction, only the amount of output detail. A low value should be specified for initial calculations to minimize printout lengths

while a large value might be desirable for final predictions to give enough resolution for plotting of plume dimensions.

H. Units of Measure

CORMIX uses the metric system of measurement. When data values are provided to the user in English units, these must be converted to equivalent metric measures. The following list gives the five metric dimensions used by CORMIX in the left column, and on the right, their equivalents in some common English units.

Length:	1 m	= 3.281 ft = 39.37 in = 0.0006214 mile
Velocity:	1 m/s	= 3.281 ft/s (fps) = 2.237 miles/hr (mph) = 1.943 knots
Discharge:	1 m^3/s	= 35.31 ft^3/s (cfs) = 22.82 million-gal/day (mgd)
Density:	1000 kg/m^3	= 62.43 lb/ft^3
Temperature:	°C	= (°F - 32.0) * 0.5556

Pollutant concentrations can be entered in any conventional measure such as mg/L, ppb, bacteria-count, etc.

Considering the potential accuracy of CORMIX predictions, 3 to 4 significant digits are sufficiently accurate for most input data values as suggested in the above conversion list. The only exceptions are the ambient and effluent density values. These may require 5 significant digits, especially when simulating the discharge to an ambient density-stratified receiving waterbody.

I. Checklists for Input Data Preparation

Tables 2, 3, and 4 summarize the data input requirements of CORMIX1, 2, and 3, respectively and have been enclosed to aid in the assembly and preparation of this data. Prior to beginning an analysis, the appropriate table should be used as a check list to verify that all necessary data are available.

V SYSTEM OUTPUT FEATURES

CORMIX is a highly interactive system and conveys information to the user through qualitative descriptions and detailed quantitative numerical predictions. This output can be viewed on screen, can be directed to a printer and is stored in

TABLE 2 INPUT DATA CHECKLIST FOR CORMIX1

CORMIX1 -- Submerged single port discharges -- CORMIX1

SITE/CASE IDENTIFIER:
Site Name ______
Discharger ______
Pollutant ______
Design Case ______
DOS FILE NAME ______

Prepared by:

Date prepared:

AMBIENT DATA:
Bounded or unbounded? ______
Channel width ______ m
Channel depth ______ m
Depth at discharge ______ m
Ambient flowrate ______ m^3/s or: Ambient velocity ______ m/s
Manning's n ______ or: Darcy-Weisbach f ______

Density data:
Fresh or salt water? ______
Density or temp. values? ______
If uniform:
Average density/temp. ______

Density units: kg/m^3
Temperature units: °C

If stratified:
Density/temp. at surface ______
Density/temp. at bottom ______
Stratification type ______
(Pycnocline height ______ m)
(Density/temp. jump ______)

DISCHARGE DATA:
Nearest bank (left/right)? ______
Distance to nearest bank ______ m
Vertical angle (THETA) ______ °
Horizontal angle (SIGMA) ______ °
Port height ______ m
Port diameter ______ m or: Port area ______ m^2
Discharge flow rate ______ m^3/s or: Discharge velocity ______ m/s
Discharge density ______ kg/m^3 or: Discharge temp. ______ °C
Concentration units ______
Discharge concentration ______

MIXING ZONE DATA:
Is effluent toxic? ______ If yes: CMC value ______
CCC value ______

Is there a WQ standard for conventional pollutant? ______ If yes: value of standard ______
Any mixing zone specified? ______ If yes: distance ______ m
or width (% or m) ______
or area (% or m^2) ______

Region of interest ______ m
Grid intervals for display ______

Date of data input into CORMIX1:

TABLE 3 INPUT DATA CHECKLIST FOR CORMIX2

CORMIX2 -- Submerged multiport diffuser discharges -- CORMIX2

SITE/CASE IDENTIFIER:
Site Name ______
Discharger ______
Pollutant ______
Design Case ______
DOS FILE NAME ______

Prepared by:

Date prepared:

AMBIENT DATA:
Bounded or unbounded? ______
Channel width ______ m
Channel depth ______ m
Depth at discharge ______ m
Ambient flowrate ______ m^3/s or: Ambient velocity ______ m/s
Manning's n ______ or: Darcy-Weisbach f ______

Density data:
Fresh or salt water? ______
Density or temp. values? ______
If uniform:
Average density/temp. ______

Density units: kg/m^3
Temperature units: °C

If stratified:
Density/temp. at surface ______
Density/temp. at bottom ______
Stratification type ______
(Pycnocline height ______ m)
(Density/temp. jump ______)

DISCHARGE DATA:
Nearest bank (left/right)? ______ Distance to one endpoint ______ m
Diffuser length ______ m to other endpt. ______ m
Total number of openings ______
Port diameter ______ m with contraction ratio ______
Diffuser arrangement/type ______
Alignment angle (GAMMA) ______ ° Horizontal angle (SIGMA) ______ °
Vertical angle (THETA) ______ ° Rel. orientation (BETA) ______ °
Port height ______ m
Discharge flow rate ______ m^3/s or: Discharge velocity ______ m/s
Discharge density ______ kg/m^3 or: Discharge temp. ______ °C
Concentration units ______
Discharge concentration ______

MIXING ZONE DATA:
Is effluent toxic? ______ If yes: CMC value ______
CCC value ______

Is there a WQ standard for conventional pollutant? ______ If yes: value of standard ______
Any mixing zone specified? ______ If yes: distance ______ m
or width (% or m) ______
or area (% or m^2) ______

Region of interest ______ m
Grid intervals for display ______

Date of data input into CORMIX2:

TABLE 4 INPUT DATA CHECKLIST FOR CORMIX3

CORMIX3 -- Buoyant surface discharges -- CORMIX3			
SITE/CASE IDENTIFIER:		**Prepared by:**	
Site Name	______		
Discharger	______		
Pollutant	______	**Date prepared:**	
Design Case	______		
DOS FILE NAME	______		
AMBIENT DATA:			
Bounded or unbounded?	______		
Channel width	______ m		
Channel depth	______ m		
Ambient flowrate	______ m^3/s	or: Ambient velocity	______ m/s
Manning's n	______	or: Darcy-Weisbach f	______
Density data:			
Fresh or salt water?	______		
Density or temp. values?	______		
Average density	______ kg/m^3	or: Average Temperature	______ °C
DISCHARGE DATA:			
Discharge located on	left/right bank?		
Discharge configuration	______ (flush, protruding, or coflowing)		
Horizontal angle (SIGMA)	______ °	If protruding:	
Depth at discharge	______ m	Distance from bank	______ m
Bottom slope (THETAB)	______ °		
If rectangular cross-section:		If circular cross-section:	
Discharge channel width	______ m	Outlet pipe diameter	______ m
Discharge channel depth	______ m		
Discharge flowrate	______ m^3/s	or: Discharge velocity	______ m/s
Discharge density	______ kg/m^3	or: Discharge temp.	______ °C
Concentration units	______		
Discharge concentration	______		
MIXING ZONE DATA:			
Is effluent toxic?	______	If yes: CMC value	______
		CCC value	______
Is there a WQ standard for conventional pollutant?	______	If yes: value of standard	______
Any mixing zone specified?	______	If yes: distance	______ m
		or width (% or m)	______
		or area (% or m^2)	______
Region of interest	______ m		
Grid intervals for display	______		
Date of data input into CORMIX3:			

subdirectory \CMXn\SIMn\ files. As stated previously, use of CORMIX's hardcopy echo feature is recommended.

A. Qualitative Output: Flow Descriptions

After completion of the input data entry sequences, the system proceeds through the program elements following the flow chart displayed in Figure 8. In addition to the routine operational messages provided during program execution, important qualitative information is displayed on-screen about the ongoing analysis of the given ambient/discharge case. The three general types of descriptive information provided are: (a) descriptive messages, (b) length scale computation results and (c) flow class descriptions. The paragraphs within this Subsection aid in the interpretation of that information.

The program elements PARAMn and CLASSn, in particular, provide essential information on the expected dynamic behavior of the discharge. By actively participating in the interactive process, the novice and intermediate user can derive a substantial educational benefit and a technical appreciation of the physical aspects of initial mixing processes. Although advanced users may find some of the presented material somewhat repetitive, they should still consult the length scale computation results.

(1) Descriptive Messages- These messages provide both physical information and insight into the logic reasoning employed by CORMIX. Three example descriptive messages are:

"The effluent density (1004.5 kg/m^3) is greater than the surrounding water density at the discharge level (997.2 kg/m^3).
Therefore, the effluent is negatively buoyant and will tend to sink towards the bottom."

"STRONG BANK INTERACTION will occur for this perpendicular diffuser type due to its proximity to the bank (shoreline). The shoreline will act as a symmetry line for the diffuser flow field.
The diffuser length and total flow variables are doubled (or approximately doubled, depending on the vicinity to the shoreline).
All of the following length scales are computed on that basis."

"The specified two layer ambient density stratification is dynamically important. The discharge near field flow will be confined to the lower layer by the ambient density stratification.
Furthermore, it may be trapped below the ambient density jump at the pycnocline."

The preceding example output highlights several features of CORMIX's descriptive messages. These include: (a) conveying basic information about the involved mixing processes, (b) using a careful terminology (e.g. "..tend to sink.."), (c) describing key calculation assumptions, and (d) alerting the user to sensitive analysis conditions. In some instances, the provided information may be obvious to the user, while in others it may not, particularly for situations involving linear ambient stratification. The use of a careful terminology is necessary because messages are presented as the analysis proceeds and subsequent tests may alter, or amplify, initial results. For example, near-field instabilities, which are tested for late in the analysis, can prevent an otherwise sinking plume.

(2) Length Scale Computations- The program element PARAMn computes so-called "length scales" which represent important dynamic measures about the relative influence of certain hydrodynamic processes on effluent mixing. These calculated values are subsequently used in program element CLASSn to identify the generic flow class upon which the hydraulic simulations will be based. This flow classification is accomplished through formal dynamic length scale analysis, which is a key aspect of the theoretical underpinnings for the CORMIX approach. The CORMIX documentation manuals (7,8,9) provide the theoretical background on length scale definitions and significance, and three journal references (10,11,12) give summaries of the length scales, their derivation from principles of dimensional analysis, and their use in the CORMIX flow classification approach.

Although flow classification is a formal process using criteria derived from theoretical studies and/or experimental data, a great deal can be deduced about the flow dynamics by comparing the calculated length scales to the actual physical measures of the ambient/discharge situation. Of greatest importance are comparison to such geometric measures as: the available water depth (HD), a pycnocline height (HINT) and the distance to the nearest bank (DISTB). The following discussion provides a brief explanation of the more important length scales and examples on how to make appropriate comparisons in a given application. Users are encouraged to make these comparisons.

a) Some important length scales relating to submerged round buoyant jets (CORMIX1) are described in Table 5. All of these scales are defined from an interplay of the momentum and buoyancy flux quantities of the discharge with each other or with the current velocity and stratification gradient variables.

As an example, consider a vertically discharging buoyant jet into an unstratified ambient receiving water. When both calculated L_m and L_b values are substantially less than the local water depth (HD), this is an immediate indication to the user that the crossflow is very strong, leading to complete bending of the buoyant jet. If the reverse holds true, the crossflow may be so weak that its deflecting effect is negligible, and the buoyant

TABLE 5 LENGTH SCALES USED IN CORMIX1 AND 2

Jet/plume transition length scale (L_M)
definition: $L_M = M_o^{3/4} / J_o^{1/2}$
interpretation: For combined buoyant jet flow, the distance at which the transition from jet to plume behavior takes place in a stagnant uniform ambient.

Jet/crossflow length scale (L_m)
definition: $L_m = M_o^{1/2} / u_a$
interpretation: In the presence of a crossflow, the distance of the transverse (i.e. across ambient flow) jet penetration beyond which the jet is strongly deflected (advected) by the cross flow. For a strictly co-flowing discharge ($\theta=0, \sigma=0$), the length of the region beyond which the flow is simply advected.

Plume/crossflow length scale (L_b)
definition: $L_b = J_o / u_a^3$
interpretation: The vertically upward or downward flotation distance beyond which a plume becomes strongly advected by crossflow.

Jet/stratification length scale (L_m')
definition: $L_m' = M_o^{1/4} / \varepsilon^{1/4}$
interpretation: In a stagnant linearly stratified ambient, the distance at which a jet becomes strongly affected by the stratification, leading to terminal layer formation with horizontally spreading flows.

Plume/stratification length scale (L_b')
definition: $L_b' = J_o^{1/4} / \varepsilon^{3/8}$
interpretation: In a stagnant linearly stratified ambient, the distance at which a plume becomes strongly affected by the stratification, leading to terminal layer formation with horizontally spreading flows.

Notes:
$M_o = U_o Q_o$, kinematic momentum flux
$J_o = g'_o Q_o$, kinematic buoyancy flux
$Q_o = U_o a_o$, source discharge volume flux
a_o = port area
u_a = ambient velocity
U_o = port discharge velocity
ε = ambient buoyancy gradient
g'_o = discharge buoyancy = $g(\rho_a - \rho_o)/\rho_a$

jet will strongly interact (impinge) with the water surface. In the first instance, a situation as depicted in Figures 1b combined with Figure 1a will result, while in the second instance, a flow resembling Figures 2c or 2d may arise, depending on the relation of the two scales with each other.

As another example, consider a buoyant jet discharging into a linearly stratified ambient. If both L_m' and L_b' are both larger than the pycnocline height (HINT) and even the water depth (HA), this would be an indication that the existing stratification is so weak that it will not lead to any trapping of the effluent plume within the available vertical space.

By making such comparisons, users will gradually get a good feel for the behavior of the buoyant jet, and other mixing processes within the space constraints of the ambient environment. Those interested in design can quickly gain an appreciation of the length scale measures and their sensitivity to design choices. However, there are limitations to these simplistic comparisons because the "length scales" are by no means precise measurements for the influence of the different processes. As their name implies they should be taken only as "scale" estimates. The actual CORMIX classification scheme, uses formal criteria when comparing the length scale measures with the geometric constraints or each other.

b) Some important length scales for multiport diffusers (CORMIX2) are described in Table 6. To a large extent, these scales have a similar meaning for the behavior of the plane buoyant jet as the earlier ones discussed for the round buoyant jet (Table 5). However, they are calculated differently because the CORMIX2 system uses the "equivalent slot diffuser" concept to model the overall dynamics of the submerged multiport diffuser (Subsection IIIB). Except for the immediate close-up zone before the individual jets merge (Figure 1d) this concept is a dynamically valid and accurate representation of multiport diffuser flows (8).

There are some exceptions and additional complexities to interpreting the two-dimensional slot length scales measures described in Table 6. In addition to the predominately two-dimensional flow behavior, some of the large scale dynamics of multiport diffusers may also be influenced by other scales depending on the overall diffuser flow pattern. A notable example is circulating motions induced in shallow receiving waters due to intermediate-field effects (Subsection IIA(1)). The immediate close-up zone before the individual jets merge is also not addressed by the two-dimensional length scales. Additional discussion of these and other peculiarities can be found elsewhere (7,14).

c) Some important length scales that describe the near-field dynamics of buoyant surface jets discharging into unstratified receiving waters (CORMIX3) are listed in Table 7. These scales are defined in a similar manner to the submerged discharged cases

TABLE 6 DYNAMIC LENGTH SCALES FOR MULTIPORT DIFFUSER IN THE TWO-DIMENSIONAL "SLOT" DISCHARGE REPRESENTATION

Slot jet/plume transition length scale (ℓ_M)
definition: $\ell_M = m_o / j_o^{2/3}$
interpretation: For combined buoyant jet flow, the distance at which the transition from jet to plume behavior takes place in a stagnant uniform ambient.

Slot jet/crossflow length scale (ℓ_m)
definition: $\ell_m = m_o / u_a^2$
interpretation: In the presence of a crossflow, the distance of the transverse (i.e. across ambient flow) jet penetration beyond which the jet is strongly deflected (advected) by the cross flow. For a strictly co-flowing discharge (θ=0,σ=0), the length of the region beyond which the flow is simply advected.

Slot jet/stratification length scale (ℓ_m')
definition: $\ell_m' = m_o^{1/3} / \varepsilon^{1/3}$
interpretation: In a stagnant linearly stratified ambient, the distance at which a jet becomes strongly affected by the stratification, leading to terminal layer formation with horizontally spreading flows.

Slot plume/stratification length scale (ℓ_b')
definition: $\ell_b' = j_o^{1/3} / \varepsilon^{1/2}$
interpretation: In a stagnant linearly stratified ambient, the distance at which a plume becomes strongly affected by the stratification, leading to terminal layer formation with horizontally spreading flows.

Crossflow/stratification length scale (ℓ_a)
definition: $\ell_a = u_a / \varepsilon^{1/2}$
interpretation: The vertically upward or downward floatation distance beyond which a plume becomes strongly advected by crossflow.

Notes: $m_o = U_o q_o$, kinematic momentum flux per lt.
$j_o = g'_o q_o$, kinematic buoyancy flux per lt.
$q_o = U_o n a_o / L_D$, source discharge volume flux
a_o = port area
u_a = ambient velocity
U_o = port discharge velocity
ε = ambient buoyancy gradient
g'_o = discharge buoyancy = $g(\rho_a - \rho_o)/\rho_a$
n = total number of nozzles
L_D = overall diffuser length

TABLE 7 DYNAMIC LENGTH SCALES FOR BUOYANT SURFACE JETS DISCHARGING INTO UNSTRATIFIED RECEIVING WATER

Jet/plume transition length scale (L_M)
definition: $L_M = M_o^{3/4} / J_o^{1/2}$
interpretation: For stagnant ambient conditions, the extent of the initial jet region before mixing changes over into an unsteady surface spreading motion.

Jet/crossflow length scale (L_m)
definition: $L_m = M_o^{1/2} / u_a$
interpretation: The distance over which a discharging jet intrudes into the ambient cross-flow before it gets strongly deflected.

Plume/crossflow length scale (L_b)
definition: $L_b = J_o / u_a^3$
interpretation: A measure of the tendency for upstream intrusion for a strongly buoyant discharge.

Notes:
$M_o = U_o Q_o$, kinematic momentum flux
$J_o = g'_o U_o$, kinematic buoyancy flux
$Q_o = U_o a_o$, source discharge volume flux
a_o = port area
u_a = ambient velocity
U_o = port discharge velocity
g'_o = discharge buoyancy = $g(\rho_a - \rho_o)/\rho_a$

but due to the discharge location at the surface, they have different interpretations. For example, L_m is compared to the channel width (BS) instead of the local water depth as it was in submerged case examples; if it exceeds BS, the discharge will quickly interact with the opposing bank.

(3) Description of Flow Classes- Program element CLASSn, performs a rigorous classification of the given discharge/ambient situation into one of many generic flow classes with distinct hydrodynamic features. In a way, this amounts to identifying a general pictorial description of the expected flow configuration.

Table 8 lists and describes the broad categories of flow classes available in CORMIX. CORMIX1, 2 and 3, consider 35, 31 and 11 distinct flow classifications, respectively. Each flow class identification consists of an alphanumeric label corresponding to the flow category and a number (e.g. MU2). Text descriptions of the flow classes are available on-screen during the analysis and can printed from the files stored within subdirectory \CMXn\DESCn (Table 1). Pictorial illustrations of the flow classes can be found in Appendix A. As an example,

Figure 17 shows the pictorial illustration and text description for flow class S1, a case of an effluent that becomes trapped in ambient stratification. It is strongly recommended that novice or intermediate users scrutinize these materials to gain a qualitative understanding of the effluent flow's behavior.

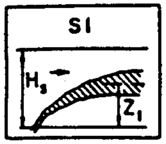
TABLE 8 FLOW CLASS CATEGORIES AND DESCRIPTIONS

CORMIX1 (35 flow classes)	
Classes S:	Flows trapped in a layer within linear stratification.
Classes V,H:	Positively buoyant flows in a uniform density layer.
Classes NV,NH:	Negatively buoyant flows in uniform density layer.
Classes A:	Flows affected by dynamic bottom attachment.
CORMIX2 (31 flow classes)	
Classes MS:	Flows trapped in a layer within linear ambient stratification.
Classes MU:	Positively buoyant flows in a uniform density layer.
Classes MNU:	Negatively buoyant flows in uniform density layer.
CORMIX3 (11 flow classes)	
Classes FJ:	Free jet flows without near-field shoreline interaction.
Classes SA:	Shoreline-attached discharges in crossflow.
Classes WJ:	Wall jets/plumes from discharges parallel to shoreline.
Classes PL:	Upstream intruding plumes.

B. Quantitative Output: Numerical Flow Predictions

After execution of the detailed flow prediction in program element HYDROn, the system provides two types of detailed numerical output on effluent plume trajectory and mixing and on compliance with regulations. A concise summary is available on-screen in the final system element SUMn and a detailed numerical output file is also generated for inspecting and plotting the plume's behavior after the analysis.

(1) Summary Output in SUMn- The self-explanatory summary output which can be displayed on-screen includes: (a) the date and time of the analysis section, (b) a complete echo of the input data, (c) the calculated flux, length scale and non-dimensional



FLOW CLASS S1

This flow configuration is profoundly affected by the linear ambient density stratification. The predominantly jet-like flow gets trapped at some terminal (equilibrium) level. The trapping is also affected by the reasonably strong ambient crossflow. Following the trapping zone, the discharge flow forms an internal layer that is further influenced by buoyant spreading and passive diffusion.

The following flow zones exist:

1) Weakly deflected jet in crossflow: The flow is initially dominated by the effluent momentum (jet-like) and is weakly deflected by the ambient current.

2) Strongly deflected jet in crossflow: The jet has become strongly deflected by the ambient current and is slowly rising toward the trapping level.

3) Terminal layer approach: The bent-over submerged jet/plume approaches the terminal level. Within a short distance the concentration distribution becomes relatively uniform across the plume width and thickness.

*** The zones listed above constitute the HYDRODYNAMIC MIXING ZONE in which strong initial mixing takes place. ***

4) Buoyant spreading in internal layer: The discharge flow within the internal layer spreads laterally while it is being advected by the ambient current. The plume thickness may decrease during this phase. The mixing rate is relatively small. The plume may interact with a nearby bank or shoreline.

5) Passive ambient mixing: After some distance the background turbulence in the ambient shear flow becomes the dominating mixing mechanism. The passive plume is growing in depth and in width. The plume may interact with the upper layer boundary, channel bottom and/or banks.

*** Predictions will be terminated in zone 4 or 5 depending on the definitions of the LEGAL MIXING ZONE or the REGION OF INTEREST. ***

FIGURE 17 EXAMPLE OF A FLOW CLASS DESCRIPTION

parameter values, (d) the flow classification used for predicting plume trajectory and mixing, (e) the coordinate system used in the analysis, (f) a summary of the near-field hydraulic mixing zone (HMZ) conditions, (g) the far-field locations where the plume becomes essentially fully mixed (i.e. uniform concentration) in the horizontal and vertical directions, (h) a summary of the toxic dilution zone (TDZ) conditions, and (i) a summary of the regulatory mixing zone (RMZ) conditions. Although the raw data used to construct this summary output is permanently stored in file 'fn'.CXS within the output sub-directory \CMXn\SIMn, a hard-copy printout should be requested during the analysis session because the raw data file is unformatted and does not contain the explanatory text that is available during program execution; 'fn' is the filename specified by the user during input data entry.

The coordinate system conventions pertain to the origin location and axis direction. In CORMIX1 analyses, the origin is located at the bottom of the receiving water just below the discharge port center and thus, at a depth HD below the water surface. In CORMIX2 analyses, the origin is located at the bottom of the receiving water, at the midpoint of the diffuser line and thus, at a depth HD below the water surface. In CORMIX3 analyses, the origin is located at the water surface where the discharge channel centerline and receiving water shoreline intersect. The x-axis lies in the horizontal plane and points downstream in the direction following the ambient flow; the y-axis lies in the horizontal plane and points to the left as seen by an observer looking downstream along the x-axis; and the z-axis points vertically upward. Note that when the ambient current direction varies (e.g. due to reversing tidal flows), the interpretation of simulation results becomes more involved since the x-axis and the y-axis will change depending on flow direction.

In addition to the numerical predictions of the plume size, location and chemical concentration, the summary of the near-field hydraulic mixing zone (HMZ) conditions describes other relevant plume features such as bottom attachment, bank interaction and the degree of upstream intrusion. This information is useful for both engineering design and for determining whether important resource areas may be exposed to undesirable chemical concentrations.

In case of a discharge containing potentially toxic chemicals, the summary toxic dilution zone (TDZ) conditions will indicate the location along the plume where the local concentration begins to fall below the specified CMC. This location's distance from the discharge structure is compared to the three applicable U.S. EPA criteria (see Subsection IIB(3)) and the results of these comparisons are displayed.

When regulatory mixing zone (RMZ) criteria have been specified during input data entry, the geometric, dilution and concentration conditions at the edge of the specified or proposed RMZ are compared to these criteria and/or to the applicable CCC concentration following the practices discussed in Subsection IIB(4). The results of these comparisons are displayed.

(2) Detailed Prediction Output File 'fn'.CXO- The file 'fn'.CXO stored within sub-directory \CMXn\SIMn contains the same kinds of information available in the summary output plus the detailed numerical predictions on plume geometry and mixing produced during the hydraulic simulation. Data in that file form the basis for further analysis, inspection, evaluation, and plotting of the plume shape and trajectory. Presently, plotting must be done separately by the user using either another applications program or manually.

During program execution, the user has several opportunities to display on-screen or print out this file. It can also be printed at a later date by using the DOS PRINT command. CORMIX will not erase any of the files with .CXO (or .CXS) extension that get stored in the \CMXn\SIMn sub-directory. Consequently, periodic directory maintenance is recommended to remove old and superfluous files.

The 'fn'.CXO file is a FORTRAN output file generated by the HYDROn prediction program. As is typical of many FORTRAN outputs, its display features are terse with tight format control and data items labeled in symbolic form only (e.g. "Q0" for discharge flow rate). Complete output file examples can be inspected in Appendices B, C and D.

All three CORMIXn subsystems produce a 'fn'.CXO output file with common appearance and features as described in the following paragraphs.

a) Lead-in information: The output starts (and ends) with a "111...111", "222...222", or "333...333" banner line to accentuate which subsystem has been used. The date and time of the analysis session and all important input data are the next items in the file. These are subsequently followed by the calculated length scale values, non-dimensional numbers of interest to the specialist, the flow class identification, and the coordinate system is displayed.

b) Prediction results for each flow "module": As was mentioned previously in Subsection IIIG, the CORMIX prediction methodology utilizes of a number of simulation modules that are executed sequentially and that correspond to the different flow processes and associated spatial regions which occur within a given flow class. The 'fn".CXO output reflects that sequence and is a arranged in output blocks for each module.

Each simulation module has a "MODnxx" label where "n" is 1, 2, or 3 corresponding to CORMIXn, and "xx" is a two-digit identification number. The two general types of modules are continuous flow and control volume.

The continuous flow module type describes the continuous evolution of a flow region along a trajectory. Depending on the number of grid intervals specified by the user, information on plume geometry, flow, and mixing information along the plume trajectory may be available for a few or many waterbody locations.

Figure 18 provides examples of typical output from continuous flow modules. The annotations along the right margin illustrate important features of the output format. Figure 18a was taken from a CORMIX1 simulation output file and shows an example of a submerged jet region module. The output contains labeling information on the module, and explanatory notes on profile definitions. It also gives a numerical list on the predictions, first repeating the final values from the preceding flow module and then one line for each user-specified grid interval. This information gives the x-y-z position of the jet/plume centerline, the dilution (S) and concentration (C) at the centerline, and the jet width (B). Dilution is defined as the inverse of the fractional reduction in concentration between the discharge structure and a given location.

Another example of a continuous flow module output is shown in Figure 18b. It was abstracted from a CORMIX2 simulation output file and shows predictions for the far-field process of buoyant ambient spreading (Figure 6). Although it is terse, the output file values and commentary generally provide a complete picture of flow conditions. In this example output (Figure 18b), evidence of this completeness includes: (a) the prediction output is separated in two stages corresponding to before and after bank interaction, respectively; due to the typical oblong cross-section of the plume in this stage, width dimensions for the vertical and lateral extent are given and defined; the coordinates for the upper and lower boundaries of the plume are listed as a convenience for plotting; and the system searches for criteria that apply to mixing zone regulations and when a criterion is satisfied, a remark gets inserted in the output list at the appropriate spatial position.

Some mixing flow processes are so complicated that no mechanistically-based mathematical description of them is presently available in state-of-the-art science. Those processes are most reliably analyzed with control volume modules. In this modeling approach, the outflow values for a region are computed as a function of the inflow values and are based on conservation principles.

Two output examples for control volumes modules are illustrated in Figure 19. The first example (Figure 19a) was

```
BEGIN MOD111: WEAKLY DEFLECTED JET IN CROSSFLOW                     < module label and description
CO-FLOWING DISCHARGE                                                < some geometric information

PROFILE DEFINITIONS:
  B = Gaussian 1/e (37%) half-width, normal to trajectory
  C = centerline concentration
  S = corresponding dilution

PREDICTION
     X        Y      Z       S       C        B                     < XYZ - trajectory coordinates
    .00      .00    1.00    1.0   .100E+03   .11                    < initial values from
   2.08      .00    2.18    3.4   .291E+02   .37                          preceding  module
   4.16      .00    2.23    5.9   .170E+02   .64                    < user has specified 5 lines
   6.23      .00    4.51    8.3   .120E+02   .90                          of display on plume
   8.33      .00    6.08   10.6   .828E+01  1.17                          properties
  10.41      .00    8.13   13.2   .737E+01  1.43

END OF MOD111: WEAKLY DEFLECTED JET IN CROSSFLOW                    < end of module
```

(a) Prediction Module within Near-Field of a Round Buoyant Jet Discharge (CORMIX1)

```
BEGIN MOD241: BUOYANT AMBIENT SPREADING
 PROFILE DEFINITIONS:
  BV = TOP-HAT THICKNESS, MEASURED VERTICALLY                       < two width definitions since
  BH = TOP-HAT HALF-WIDTH, MEASURED HORIZONTALLY IN Y-DIRECTION            plume cross-section is
  C = AVERAGE CONCENTRATION                                                rectangular
  S = CORRESPONDING DILUTION

PREDICTION: STAGE 1: NOT BANK ATTACHED                              < plume is not yet bank
     X        Y      Z       S       C       BV      BH     ZU     ZL       attached
   70.53     .00    5.20   10.0   .200E+03  5.20   17.04   5.20    .00
  100.16     .00    5.20   11.1   .180E+03  3.80   28.78   5.20   1.40
  128.78     .00    5.20   11.8   .168E+03  3.14   34.36   5.20   2.06

** WATER QUALITY STANDARD OR CCC HAS BEEN FOUND  **
 THE POLLUTANT CONCENTRATION IN THE PLUME FALLS BELOW THE WATER QUALITY    < CORMIX informs when mixing
STANDARD OR CCC VALUE OF .17E+03 DUE TO MIXING IN THIS INTERVAL.                zone criteria get
                                                                                satisfied
  141.40     .00    5.20   12.4   .162E+03  2.74   41.34   5.20   2.46
  162.02     .00    5.20   12.8   .156E+03  2.48   47.87   5.20   2.74
  182.64     .00    5.20   13.2   .151E+03  2.25   54.00   5.20   2.85

PLUME IS ATTACHED TO RIGHT BANK/SHORE.
  PLUME WIDTH IS NOW DETERMINED FROM RIGHT BANK/SHORE.

PLUME IS LATERALLY FULLY MIXED AT THE END OF THE BUOYANT SPREADING REGIME.
PROFILE DEFINITIONS:
  BV = top-hat thickness, measured vertically
  BH = top-hat half-width, measured horizontally in y-direction
  C = average concentration
  S = corresponding dilution

PREDICTION: STAGE 2: BANK ATTACHED                                  < bank interaction has occurred
     X        Y      Z       S       C       BV      BH     ZU     ZL
  182.64  -54.00    5.20   13.2   .151E+03  2.25  108.01   5.20   2.85
  233.40  -54.00    5.20   13.8   .147E+03  2.05  121.08   5.20   3.15    < Y-coordinate of trajectory
  284.18  -54.00    5.20   14.0   .143E+03  1.88  135.18   5.20   3.30          has changed after bank
  334.81  -54.00    5.20   14.3   .140E+03  1.78  147.78   5.20   3.42          interaction
  385.87  -54.00    5.20   14.8   .137E+03  1.67  158.67   5.20   3.53
  436.43  -54.00    5.20   14.8   .135E+03  1.38  171.50   5.20   3.81

END OF MOD241: BUOYANT AMBIENT SPREADING
```

(b) Prediction Module within Far-Field of a Multiport Diffuser Discharge (CORMIX2)

FIGURE 18 EXAMPLES OF CONTINUOUS FLOW MODULE


```
BEGIN MOD131: LAYER BOUNDARY/TERMINAL LAYER APPROACH                          < module label and description

  CONTROL VOLUME                                                              < reminder that this is a
PROFILE DEFINITIONS:                                                                control volume
  BV = TOP-HAT THICKNESS, MEASURED VERTICALLY                                 < explanations refer to outflow
  BH = TOP-HAT WIDTH, MEASURED HORIZONTALLY IN Y-DIRECTION                          cross-section
  C = AVERAGE CONCENTRATION
  S = CORRESPONDING DILUTION

PREDICTION
     X         Y      Z        S       C        B
 CONTROL VOLUME INFLOW
   14.04     13.38  13.58    35.1   .285E+02   3.97                           < initial values repeated from
     X         Y      Z        S       C        BV       BH     ZU     ZL           preceding module
 CONTROL VOLUME OUTFLOW
   21.99     13.38  13.58    59.7   .168E+02   6.69     6.69  15.93   9.24   < outflow plume values

END OF MOD131: LAYER BOUNDARY/TERMINAL LAYER APPROACH
```

(a) Prediction Module for a Round Buoyant Jet Approach to the Water Surface (CORMIX1)

```
BEGIN MOD 331 : UPSTREAM INTRUDING PLUME

      UPSTREAM INTRUSION LENGTH =                    153.27 m          < several important geometric
      WIDTH OF PLUME AT DISCHARGE =                  398.51 m                feature of the upstream
      WIDTH OF PLUME AT END OF NEAR-FIELD =          100.00 m                intrusion are summarized
      DEPTH OF PLUME AT DISCHARGE =                     .30 m
      DILUTION JUST DOWNSTREAM OF DISCHARGE =          1.50 m

  CONTROL VOLUME
PROFILE DEFINITIONS:
  BV = TOP-HAT THICKNESS, MEASURED VERTICALLY
  BH = TOP-HAT WIDTH, MEASURED HORIZONTALLY IN Y-DIRECTION
  C = AVERAGE CONCENTRATION
  S = CORRESPONDING DILUTION

PREDICTION
     X          Y          BH         BV        C          S

    .00        .00        5.00       1.00    .10E+03    .10E+01       < inflow and outflow plume
  153.27       .00      100.00        .30    .67E+02    .15E+01             values are listed

END MOD 331 : UPSTREAM INTRUDING PLUME
```

(b) Prediction Module for an Upstream Intruding Plume From a Buoyant Surface Discharge (CORMIX3)

FIGURE 19 EXAMPLES OF CONTROL VOLUME MODULE

taken from a CORMIX1 simulation output file and gives predictions for a flow case corresponding to an unstable near-field (Figure 2c). Note that a separate listing of inflow variables and outflow variables is given with appropriate explanations. The second example (Figure 19b) from a CORMIX3 simulation output file, shows the predictions for the initial stage of an upstream intruding plume, important plume dimensions, and inflow and outflow values.

c) Numerous other supplementary messages on plume behavior (e.g. bottom attachment, bank contact, etc.) and on possible model restrictions (e.g. ambient dilution limitations in a flow-restricted river) are contained in the output as warranted; Figures 18 and 19 provide but a few examples of these user aids.

(3) Comments on Plotting of Plume Features- The case study materials in the Appendices should be consulted for some possibilities on how to graphically display the plume features described in the 'fn'.CXO output file. As shown above, the plume is characterized by its centerline trajectory, dilution, and width values. If added detail in the plume cross-section is desired, then the plotting aids shown in Figure 20 may be useful. These give the cross-sectional distribution of concentration for many of the commonly occurring plume cross-sections in the various regions predicted by the CORMIXn subsystems.

By and large, all CORMIXn predictions are continuous from module to module satisfying the conservation of mass, momentum and energy principles. Occasionally, some mismatches in plume width can occur as a consequence of enforcing these principles. Most of these will be barely noticeable with the usual plotting resolution and they can usually be safely ignored. In addition, when bottom attachment or bank interaction occurs, the plume trajectory is assumed to (and simulation predictions do) shift suddenly to the boundary. In actuality, that shift would be much more gradual and this should be considered when plotting plume features.

VI CLOSURE

A. Synopsis

The Cornell Mixing Zone Expert System (CORMIX) is a series of software subsystems for the analysis, prediction, and design of aqueous pollutant discharges into diverse water bodies. The major emphasis is on the geometry and dilution characteristics of the initial mixing zone including compliance with regulatory constraints. The system also predicts the behavior of the discharge plume at larger distances. The highly user-interactive CORMIX system is implemented on IBM-PC compatible microcomputers and consists of three subsystems. These are: CORMIX1 for submerged single port discharges, CORMIX2 for submerged

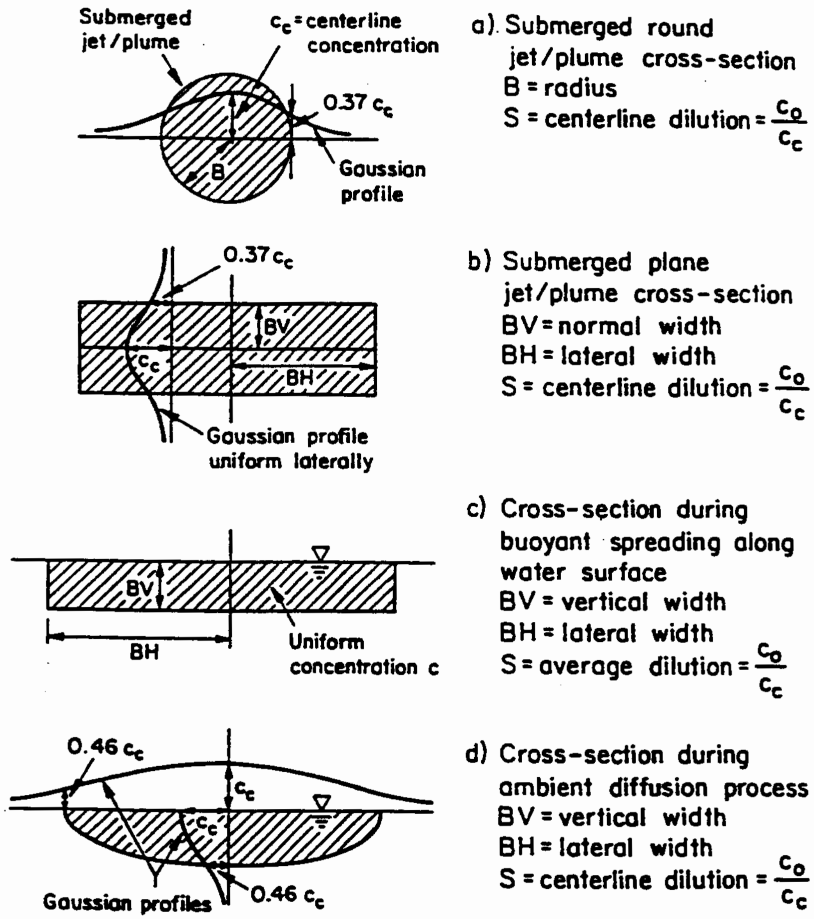


FIGURE 20 USEFUL RELATIONSHIPS FOR PLOTTING DATA FROM CORMIX SIMULATIONS

multiport diffuser discharges and CORMIX3 for buoyant surface discharges.

This user's guide gives a comprehensive and uniform description of all three CORMIX subsystems; it provides advice for assembly and preparation of required input data; it delineates ranges of applicability of the three subsystems; it provides instruction for the interpretation and graphical display of system output; and it illustrates practical system application through several case studies.

B. System and Documentation Availability

The CORMIX system programs can be obtained from the U.S. EPA Center for Environmental Assessment Modeling (CEAM), USEPA-ERL, Athens, Georgia 30613-7799, Telephone (404)546-3590. Presently (August 1991), the following versions of CORMIX are available: CORMIX1, Version 1.30 (August 1991); CORMIX2, Version 1.20 (August 1991); and CORMIX3, Version 1.10 (August 1991). The models can be obtained by mail or over the electronic bulletin board operated by CEAM. Information on program installation and computer configuration are also provided by CEAM.

The distribution versions of CORMIXn contain only the executable code of the FORTRAN program HYDROn; they do not include the source code. The source code can be requested separately by writing to CEAM at U.S. EPA-ERL and giving the reason for code inspection and possible manipulation. The full code, while made up of simple individual modules, is complex with multiple interdependencies; only experienced research personnel should attempt this work when engaged in comparison of model predictions to new field or laboratory data.

The documentation manual are available as U.S. EPA and NTIS publications, and have also been issued as technical reports of the DeFrees Hydraulics Laboratory.

C. Future Developments and Enhancements

Data entry features: Current CORMIX experience and user response has shown that while the system is well adapted to the novice and intermediate user its use becomes cumbersome for the more advanced user. This problem lies primarily in the present data entry interface. A simple spreadsheet-like data entry might be preferable. However, this would be dangerous since the change of one variable can effect the consistency of the entire data set and require changes in several other variables. A FAST-CORMIX system version is under development that uses fast spreadsheet-like data entry while preserving the safety and screening features of the knowledge base logic of the expert system.

Graphical output display: A graphical output interface for CORMIX is a complicated task due to the diverse possibilities of

plume shapes and mixing zone patterns that can exist. Present development plans will evaluate trade-offs for plotting of at least the more common discharge/ambient situations.

Hydrodynamic model features: (1) CORMIX, as all initial mixing models, is a steady-state model. In many situations of tidal reversing flows it is uncertain to apply such models for the low-speed slack tide conditions that are the most conservative on one hand, but highly transient on the other (for a discussion, see Jirka, 1991). Research is underway to define the appropriate design base for such highly transient conditions. (2) The predictive elements of the CORMIXn subsystems represent a great number of often complex hydrodynamic mixing processes. Many of the sub-models have been extensively tested and are well verified. For others, the data base is either poor, missing, or conflicting. Additional investigations are underway to improve and/or support these elements of CORMIX in the following problem areas: upstream and lateral spreading phases after the plume impingement process; merging of individual plumes of multiport diffuser to form a line plume; transition between stratified flow (buoyant spreading) and passive diffusion in the far-field; and pollutant decay processes (CORMIX currently assumes a conservative pollutant).

Future updated versions of CORMIXn, expected over the 1992/93 time frame will include these additional features. Any high-quality field or laboratory data on effluent mixing processes is a valuable asset in this development. Transmittal of such data to the DeFrees Hydraulics Laboratory, c/o Prof. G.H. Jirka, Cornell University, Ithaca, New York 14853, is greatly appreciated.

VII GLOSSARY

Actual Water Depth (HD) - the actual water depth at the submerged discharge location. It is also called local water depth. For surface discharges it is the water depth at the channel entry location.

Alignment Angle (GAMMA) - the angle measured counterclockwise from the ambient current direction to the diffuser axis.

Allocated Impact Zone - see mixing zone.

Alternating Diffuser - a multi-port diffuser where the ports do not point in a nearly single horizontal direction.

Ambient Conditions - the geometric and dynamic characteristics of a receiving water body that impact mixing zone processes. These include plan shape, vertical cross sections, bathymetry, ambient velocity, and density distribution.

Ambient Currents - A velocity field within the receiving water which tends to deflect a buoyant jet into the current direction.

Ambient Discharge (QA) - the volumetric flow rate of the receiving waterbody.

Average Diameter (D0) - the average diameter of the discharge ports or nozzles for a multi-port diffuser.

Average Depth (HA) - the average depth of the receiving waterbody determined from the equivalent cross sectional area during schematization.

Bottom Slope (THETAB) - the slope of the bottom that extends from a surface discharge into the receiving waterbody.

Buoyant Jet - a discharge where turbulent mixing is caused by a combination of initial momentum flux and buoyancy flux. It is also called a forced plume.

Buoyant Spreading Processes - far-field mixing processes which arise due to the buoyant forces caused by the density difference between the mixed flow and the ambient receiving water.

Buoyant Surface Discharge - the release of a positively or neutrally buoyant effluent into a receiving water through a canal, channel, or near-surface pipe.

Coanda Attachment - a dynamic interaction between the effluent plume and the water bottom that results from the entrainment demand of the effluent jet itself and is due to low pressure effects.

Cumulative Ambient Discharge - refers to the volumetric flow rate which occurs between the shore line and a submerged discharge into a coastal waterbody.

Darcy-Weisbach Friction Factor - a measure of the roughness characteristics in a channel.

Deep Conditions - see near-field stability.

Density Stratification - the presence of a vertical density profile within the receiving water.

Diffuser Length (LD) - The distance between the first and last port of a multi-port diffuser line. See diffuser line.

Diffuser Line - a hypothetical line between the first and last ports of a multi-port diffuser.

Discharge Velocity (U0) - the average velocity of the effluent being discharged from the outfall structure.

Discharge from Shore (DISTB) - the average distance between the outfall location and the shoreline. It is also specified as a cumulative ambient discharge divided by the product UA times HA.

Discharge Flow Rate (Q0) - the volumetric flow rate from the discharge structure.

Discharge Channel Width (B0) - the average width of a surface discharging channel.

Discharge Channel Depth (H0) - the average depth of a surface discharging channel.

Discharge Conditions - the geometric and flux characteristics of an outfall installation that effect mixing processes. These include port area, elevation above the bottom and orientation, effluent discharge flow rate, momentum flux, and buoyancy flux.

Distance from Shore (YB1, YB2) - the distance from the shore line to the first and last ports of a multi-port diffuser.

Far-field - the region of the receiving water where buoyant spreading motions and passive diffusion control the trajectory and dilution of the effluent discharge plume.

Far-field Processes - physical mixing mechanisms that are dominated by the ambient receiving water conditions particularly ambient current velocity and density differences between the mixed flow and the ambient receiving water.

Flow Classification - the process of identifying the most appropriate generic qualitative description of the discharge flow undergoing analysis. This is accomplished by examining known relationships between flow patterns and certain calculated physical parameters.

Flux Characteristics - the properties of effluent discharge flow rate, momentum flux and buoyancy flux for the effluent discharge.

Forced Plume - see buoyant jet.

Generic Flow Class - a qualitative description of a discharge flow situation that is based on known relationships between flow patterns and certain physical parameters.

Height of Port (H0) - the average distance between the bottom and the average nozzle centerline.

Horizontal Angle (SIGMA) - the angle measured counterclockwise from the ambient current direction to the plane projection of the port center line.

Hydrodynamic Mixing Processes - the physical processes that determine the fate and distribution of effluent once it is discharged.

Hydrodynamic Mixing Zone (HMZ) - the zone of strong initial mixing where the so called near-field processes occur. It is the region of the receiving water where outfall design conditions are most likely to have an impact on in-stream concentrations.

Input Data Sequence - a group of questions from one of four topical areas.

Intermediate-field Affects - induced flows in shallow waters which extend beyond the strictly near-field region of a multi-port defuser.

Jet - see pure jet.

Laterally Bounded - refers to a waterbody which is constrained on both sides by banks such as rivers, streams, estuaries and other narrow water courses.

Laterally Unbounded - a waterbody which for practical purposes is constrained on at most one side. This would include discharges into wide lakes, wide estuaries and coastal areas.

Legal Mixing Zone (LMZ) - see regulatory mixing zone.

Length Scale - a dynamic measure of the relative influence of certain hydrodynamic processes on effluent mixing.

Length Scale Analysis - an approach which uses calculated measures of the relative influence of certain hydrodynamic processes to identify key aspects of a discharge flow so that a generic flow class can be identified.

Local Water Depth (HD) - see actual water depth.

Manning's n - a measure of the roughness characteristics in a channel.

Mean Ambient Velocity (UA) - the average velocity of the receiving waterbody's cross flow.

Merging - the physical interaction of the discharge plumes from adjacent ports of a multi-port diffuser.

Mixing Zone - an administrative construct which defines a limited area or volume of the receiving water where the initial dilution of a discharge is allowed to occur.

Mixing Zone Regulations - The administrative construct that intends to prevent any harmful impact of a discharged effluent on the aquatic environment and its designated uses.

Momentum Jet - see pure jet.

Multi-port Diffuser - a structure with many closely spaced ports or nozzles that inject more than one buoyant jet into the ambient receiving waterbody.

Near-field - the region of a receiving water where the initial jet characteristic of momentum flux, buoyancy flux and outfall geometry influence the jet trajectory and mixing of an effluent discharge.

Near-field Stability - the amount of local recirculation and reintrainment of already mixed water back into the buoyant jet region. Stable discharge conditions are associated with weak momentum and deep water and are also sometimes called deep water conditions. Unstable discharge conditions have localized recirculation patterns and are also called shallow water conditions.

Negative Buoyancy - the measure of the tendency of an effluent discharge to sink in a receiving water.

Non-buoyant Jet - see pure jet.

Open Format - data input which does not require precise placement of numerical values in fixed fields and which allows character strings to be entered in either upper or lower case letters.

Passive Ambient Diffusion Processes - far-field mixing processes which arise due to existing turbulence in the ambient receiving water flow.

Plume - see buoyant jet.

Positive Buoyancy - the measure of the tendency of an effluent discharge to rise in the receiving water.

Pure Jet - a discharge where only the initial momentum flux in the form of a high velocity injection causes turbulent mixing. It is also called momentum jet or non-buoyant jet.

Pure Plume - a discharge where only the initial buoyancy flux leads to local vertical accelerations which then lead to turbulent mixing.

Pycnocline - a horizontal layer in the receiving water where a rapid density change occurs.

Pycnocline Height (HINT) - the average distance between the bottom and a horizontal layer in the receiving waterbody where a rapid density change occurs.

Region Of Interest (ROI) - a user defined region of the receiving waterbody where mixing conditions are to be analyzed.

Regulatory Mixing Zone (RMZ) - the region of the receiving water where mixing zone regulations are applied. It is sometimes referred to as the legal mixing zone.

Relative Orientation Angle (BETA) - the angle measured either clockwise or counterclockwise from the average plan projection of the port centerline to the nearest diffuser axis.

Schematization - the process of describing a receiving waterbody's actual geometry with a rectangular cross section.

Shallow Water Conditions - see near-field stability.

Stable Discharge - see near-field stability.

Staged Diffuser - a multi-port diffuser where all ports point in one direction, generally following the diffuser line.

Stagnant Conditions - the absence of ambient receiving water flow. A condition which rarely occurs in actual receiving waterbodies.

Submerged Multi-port Diffuser - an effluent discharge structure with more than one efflux opening that is located substantially below the receiving water surface.

Submerged Single Port Discharge - an effluent discharge structure with a single efflux opening that is located substantially below the receiving water surface.

Surface Buoyant Jets - positively or neutrally buoyant effluent discharges occurring horizontally at the water surface from a latterly entering channel or pipe.

Surface Width (BS) - the equivalent average surface width of the receiving waterbody determined from the equivalent rectangular cross sectional area during schemation.

Toxic Dilution Zone (TDZ) - the region of the receiving water where the concentration of a toxic chemical may exceed the acute effects concentration.

Unidirectional Diffuser - a multi-port diffuser with all ports pointing to one side of the diffuser line and all ports oriented more or less normally to the diffuser line.

Unstable Discharge - see near-field stability.

Vertical Angle (THETA) - the angle between the port centerline and the horizontal plane.

Wake Attachment - a dynamic interaction of the effluent plume with the water bottom that is forced by the receiving water crossflow.

Zone of Initial Dilution (ZID) - a term used in old EPA documents to describe the mixing zone for the discharge of municipal waste water into the coastal ocean and some large lakes; its use can cause confusion and should be avoided in the future.

VIII LITERATURE REFERENCES

(1) "U.S. EPA Cooperative Agreement No. CR813093," U.S. EPA, Environmental Research Laboratory, Athens, GA, 1986.

(2) "Technical Support Document for Water Quality-based Toxics Control," U.S. EPA, Office of Water, Washington, DC, September, 1991.

(3) "Draft: Management Plan for Puget Sound," Puget Sound Water Quality Authority, Olympia, WA, 1989.

(4) "Draft: Assessment and Control of Bioconcentratable Contaminants in Surface Waters," U.S. EPA, Office of Water, Washington, DC, March, 1991.

(5) Muellenhoff, W. P., et al.,"Initial Mixing Characteristics of Municipal Ocean Discharges (Vol I & II)," USEPA, Environmental Research Laboratory, Narragansett, RI.

(6) "Technical Support Document for Water Quality-based Toxics Control," U.S. EPA, Office of Water, Washington, DC, 1985.

(7) Doneker, R. L., and G. H. Jirka, "CORMIX1: An Expert System for Mixing Zone Analysis of Conventional and Toxic Single Port Aquatic Discharges (EPA 600/600/3-90/012)," U.S. EPA, Environmental Research Laboratory, Athens, GA, 1990.

(8) Akar, P. J. and G. H. Jirka, "CORMIX2: An Expert System for Hydrodynamic Mixing Zone Analysis of Conventional and Toxic Submerged Multiport Discharges," Technical Report of the DeFrees Hydraulics Laboratory, School of Civil and Environmental Engineering, Cornell University, Ithaca, N.Y. (Also in print as an NTIS report, USEPA, Environmental Research Laboratory Athens, Ga.), 1991.

(9) Jones, G. R. and G. H. Jirka, "CORMIX3: An Expert Sytem for the Analysis and Prediction of Buoyant Surface Discharges," Technical Report, DeFrees Hydraulics Laboratory, School of Civil and Environmental Engineering, Cornell University, Ithaca, N.Y., 1991.

(10) Jirka G. H. and R. L. Doneker (1991), "Hydrodynamic Classification of Submerged Single Port Discharges", J. Hydraulic Engineering, ASCE, Vol.117, HY9.

(11) Jirka G. H. and P. J. Akar, "Hydrodynamic Classification of Submerged Multiport Diffuser Discharges," J. Hydraulic Engineering, ASCE, (117), HY9, 1991.

(12) Jones, G.R. and G.H. Jirka, "Classification and Prediction of Buoyant Surface Discharges," J. Hydraulic Engineering, ASCE, (in review, 1991).

(13) Jirka, G. H., "Use of Mixing Zone Models in Estuarine Waste Load Allocation," Part III of Technical Guidance Manual for Performing Waste Load Allocations, Book III: Estuaries, Ed. by R. A. Ambrose and J. L. Martin, U.S. EPA, Washington, D.C. (in print), 1991.

(14) Jirka G. H., "Multiport Diffusers for Heat Disposal: A Summary," J. Hydraulics Division, ASCE, (108), HY12, pp. 1423-68, 1982.

(15) Holley, E. R. and G. H. Jirka, "Mixing in Rivers," Technical Report E-86-11, U.S. Army Corps of Engineers, Washington, DC, 1986.

(16) Fischer, H. B. et al., Mixing in Inland and Coastal Waters, Academic Press, New York, 1979.

(17) "Water Quality Standards Handbook," U.S. EPA, Office of Water Regulations and Standards, Washington, DC, 1984.

(18) "Technical Guidance Manual for the Regulations Promulgated Pursuant to Section 301 (g) of the Clean Water Act of 1977 (Draft)," U.S. EPA, Washington, DC, August, 1984.

(19) "Revised Section 301 (h) Technical Support Document," EPA 430/9-82-011, U.S. EPA, Washington, DC, 1982.

(20) Chow, V. T., Open Channel Hydraulics, McGraw-Hill, New York, 1959.

APPENDIX A

FLOW CLASSIFICATION LOGIC DIAGRAMS AND PICTORIAL ILLUSTRATIONS OF FLOW CLASSES FOR CORMIXn SUBSYSTEMS

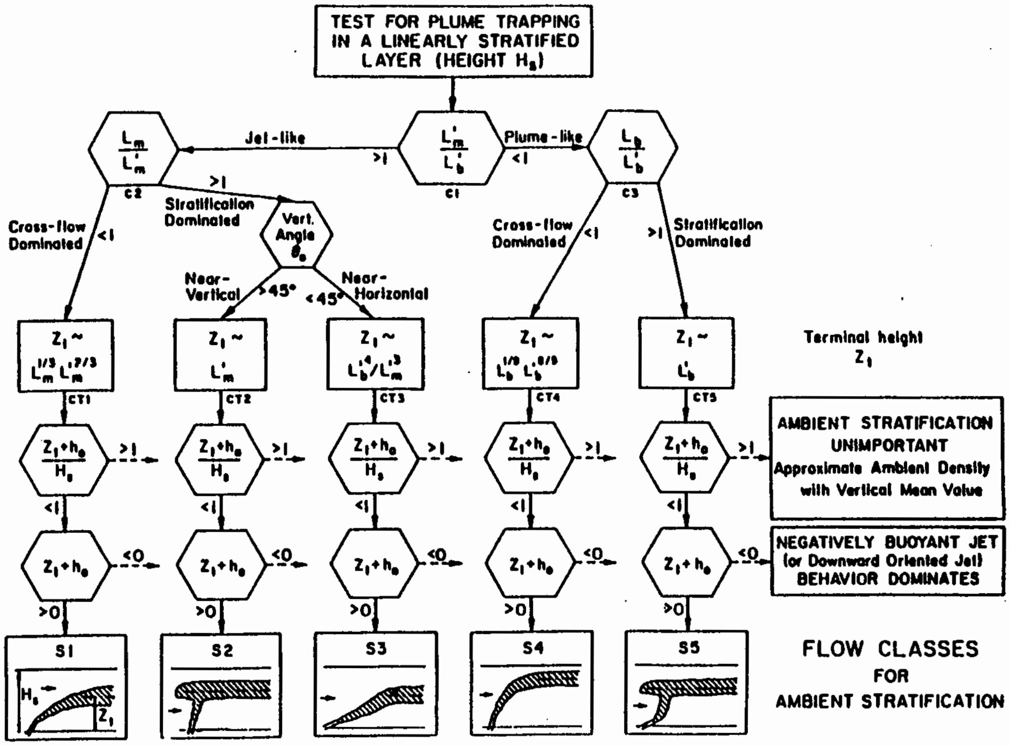


FIGURE A-1 CORMIX1'S SUB CLASSIFICATION: ASSESSMENT OF AMBIENT DENSITY STRATIFICATION AND DIFFERENT FLOW CLASSES FOR INTERNALLY TRAPPED DISCHARGES (FLOW CLASSES S)

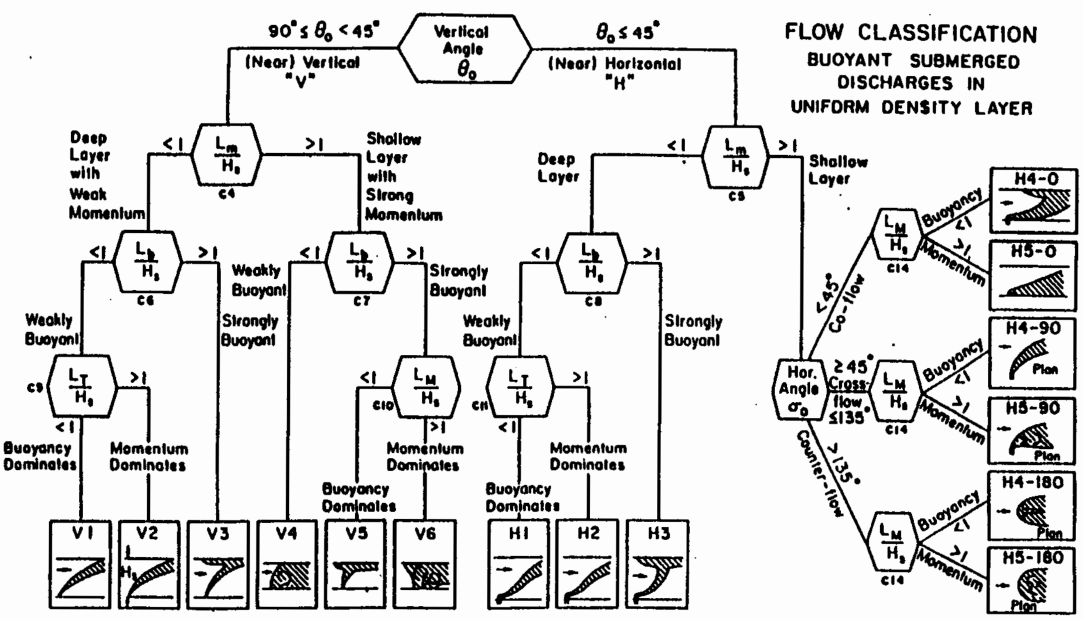


FIGURE A-2 CORMIX1'S SUB CLASSIFICATION: BEHAVIOR OF POSITIVELY BUOYANT DISCHARGES IN UNIFORM AMBIENT LAYER (FLOW CLASSES V AND H)

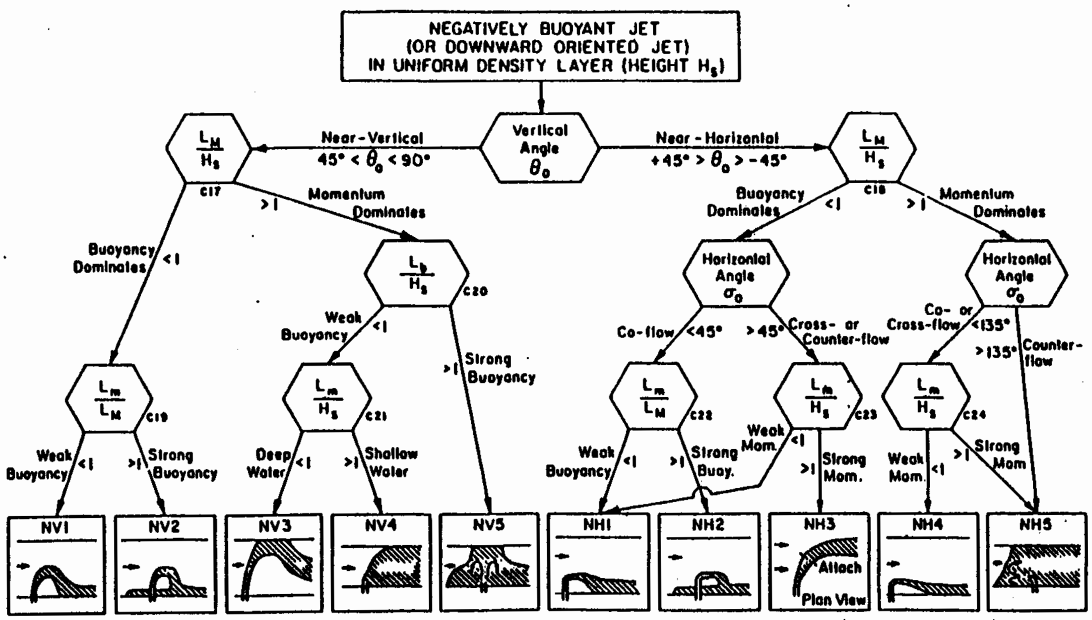


FIGURE A-3 CORMIX1'S SUB CLASSIFICATION: BEHAVIOR OF NEGATIVELY BUOYANT DISCHARGES IN UNIFORM AMBIENT LAYER (FLOW CLASSES NV AND NH)

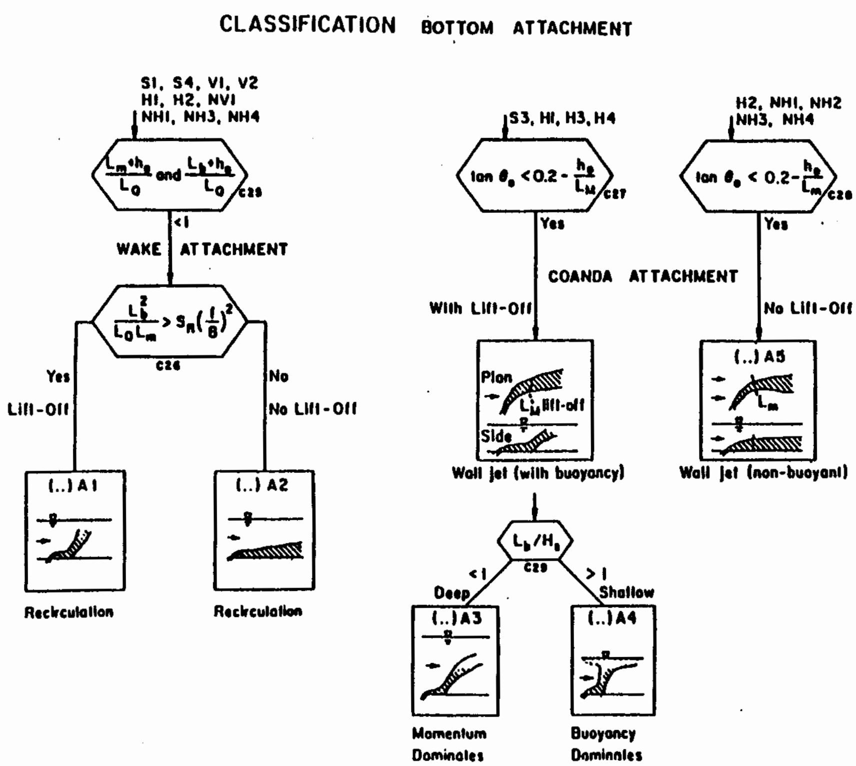


FIGURE A-4 CORMIX1'S SUB CLASSIFICATION: DYNAMIC BOTTOM ATTACHMENT OF DISCHARGE DUE TO WAKE OR COANDA EFFECTS (FLOW CLASSES A)

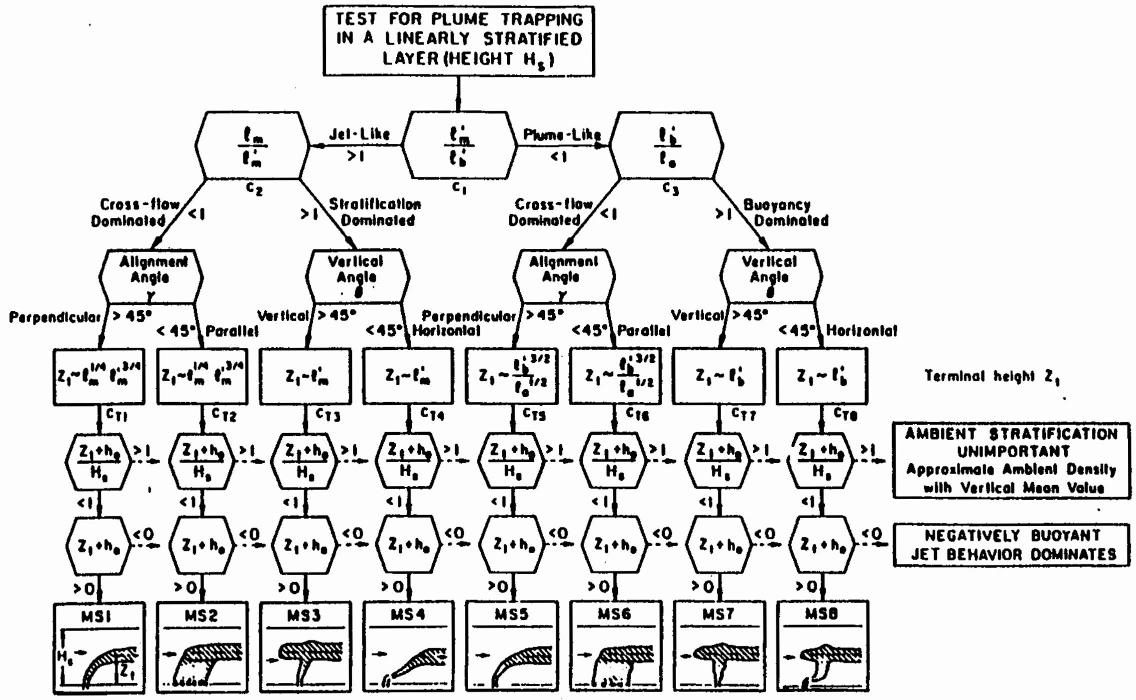


FIGURE A-5 CORMIX2'S SUB CLASSIFICATION: ASSESSMENT OF AMBIENT DENSITY STRATIFICATION AND DIFFERENT FLOW CLASSES FOR INTERNALLY TRAPPED DISCHARGES (FLOW CLASSES MS)

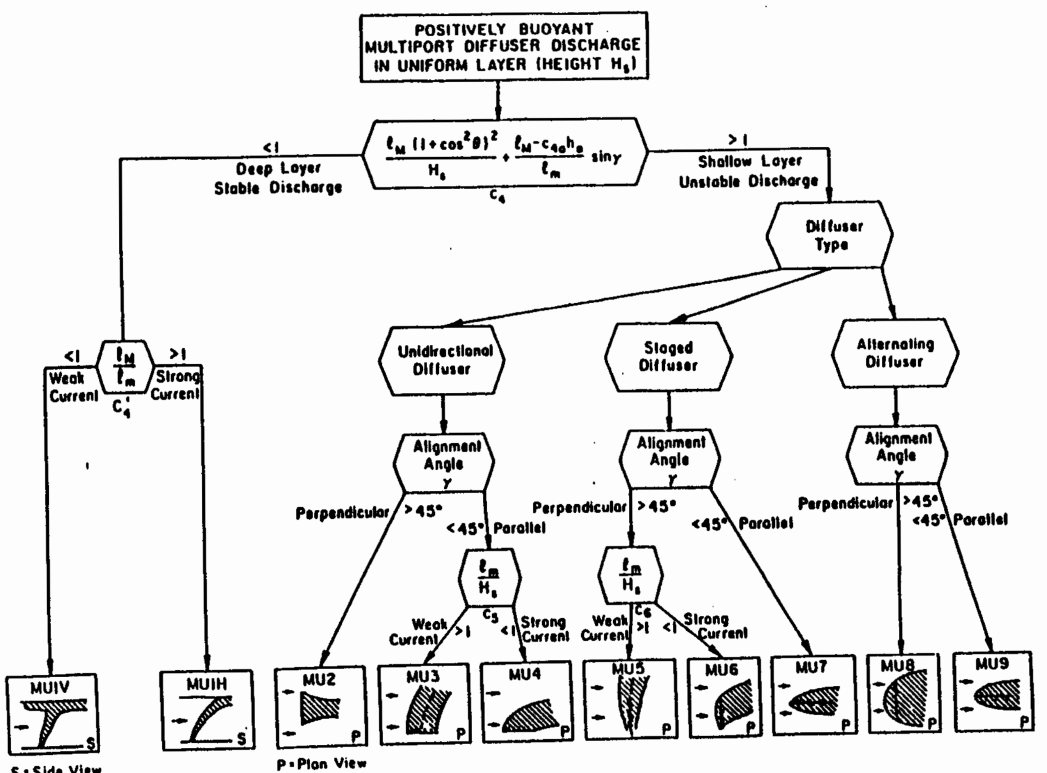


FIGURE A-6 CORMIX2'S SUB CLASSIFICATION: BEHAVIOR OF POSITIVELY BOUYANT MULTIPORT DIFFUSER DISCHARGES IN UNIFORM AMBIENT LAYER (FLOW CLASSES MU)

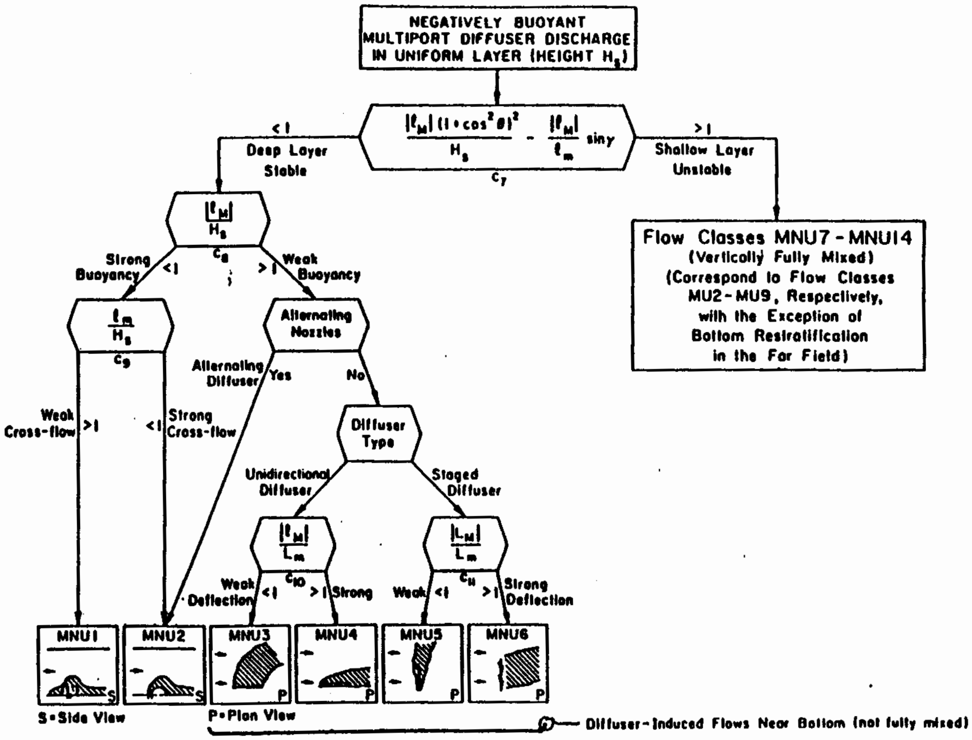


FIGURE A-7 CORMIX2'S SUB CLASSIFICATION: BEHAVIOR OF NEGATIVELY BOUYANT MULTIPORT DIFFUSER DISCHARGES IN UNIFORM AMBIENT LAYER (FLOW CLASSES MNU)

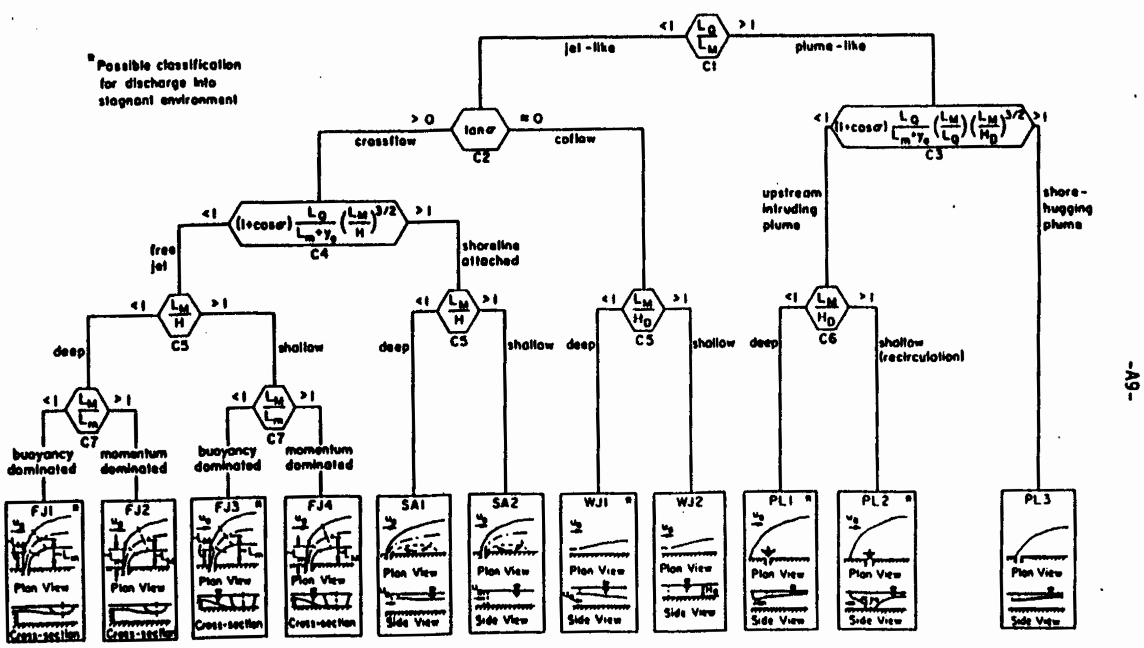


FIGURE A-8 CORMIX3'S CLASSIFICATION SCHEME FOR POSITIVELY AND NEUTRALLY BUOYANT SURFACE DISCHARGES

APPENDIX B

CORMIX1 CASE STUDY: SUBMERGED SINGLE PORT DISCHARGE IN A DEEP RESERVOIR

This case study illustrates the application of CORMIX1 to the prediction of the effluent from a small manufacturing plant into a large and deep, stratified reservoir.

B.1 Problem Statement

A manufacturing plant (A-Plant) is discharging its effluent into an adjacent deep reservoir. The plant design flowrate is 3.5 mgd (≈ 0.153 m^3/s). The effluent contains chlorides at a concentration of 3500 ppm, and is released at a temperature of 68° F (= 20 °C).

The existing reservoir has been formed by flooding a river valley. The reservoir length is approximately 60 miles. The water level in the reservoir is fluctuating depending on the release operation at the downstream dam with its hydropower installation. During summer conditions, the reservoir level is typically at an elevation of 710 ft above sea level. This results in a reservoir width of about 4000 ft (≈ 1200 m) and a maximum depth of 310 ft (≈ 95 m) at the discharge location. The mean river flow into the reservoir during the summer low-flow conditions is about 9000 cfs (≈ 255 m^3/s). The typical temperature of the inflowing river water is 55 °F (≈ 13 °C).

Figure B-1 shows the local bathymetry (as obtained from a USGS map) in the vicinity of the proposed discharge. Since the discharge is very small relative to the reservoir size and the ambient flowrate, it is expected that mostly local conditions will be important, and not overall reservoir dimensions. (Note: Any such conjecture has to be verified against the final simulation results, and adjustments have to be made if needed.)

Temperature data as a function of depth obtained from field measurements in the center of the reservoir show a significant temperature stratification (see Figure B-2), as is typical for such deep reservoirs during summer conditions. The stratification can be expected to be horizontally uniform and therefore similar conditions will hold at the discharge site. Also, the river inflow is colder than the surface layer of the stratified reservoir. Therefore, it can be expected that the river water will flow predominantly in a vertically limited layer. In this example, the layer is assumed to extend from a depth of about 30 m to about 16 m below the surface. The velocity of that flow is estimated at about 1.5 cm/s (= 0.015 m/s), given the 14 m thick layer and an about 1200 m width at that elevation. (Note:

More detailed hydrodynamic investigations, using available models for stratified reservoir dynamics, can be used to obtain more precise estimates of the velocity field. Generally, however, it cannot be assumed that the velocity in stratified reservoirs is given by the simple average of the flowrate divided by the cross-sectional area.)

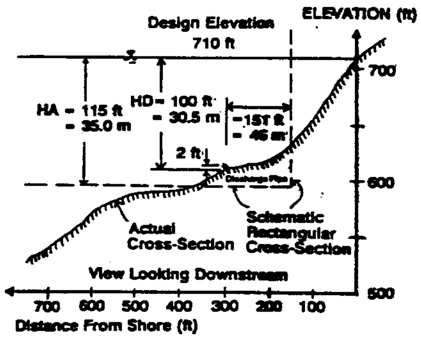


FIGURE B-1 LOCAL DETAILS OF RESERVOIR CROSS-SECTION AND CORMIX1 SCHEMATIZATION

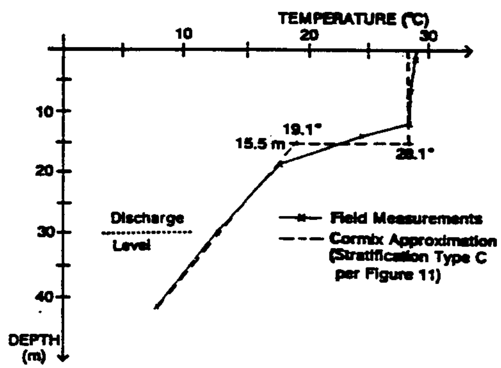


FIGURE B-2 TEMPERATURE FIELD DATA AS A FUNCTION OF DEPTH AND CORMIX1 SCHEMATIZATION OF TEMPERATURE PROFILE

The proposed discharge location on the side slope of the cross-section is also shown in Figure B-1: a submerged single port discharge at an elevation of 610 ft above sea level, i.e. at a local depth of 100 ft (≈ 30.5 m) below the surface, is proposed in the initial design phase. The port diameter is 10 in (= 0.254 m) and is located 2 ft (≈ 0.6 m) above the local bottom. The discharge is pointing perpendicular to the shoreline and is angled upward at 10 °.

The discharge is subject to State mixing zone regulations whereby the mixing zone width is less than 10 % of the width of the water body. Furthermore, the chlorides in the effluent are considered as toxic with CMC and CCC limits of 1200 and 600, respectively.

B.2 Problem Schematization and Data Preparation

Table B-1 is the data sheet that summarizes the CORMIX1 input for the present problem. The ambient water body has been characterized as unbounded in line with the expectation that the discharge plume will be small in size relative to the reservoir width. Furthermore, since (a) the discharge elevation is well above the lowest point of the reservoir and (b) the plume is expected to rise toward the surface, the ambient water depth is taken as 115 ft (= 35.0 m) only. The depth at the discharge corresponds to the elevation at the discharge location. The ambient velocity corresponds to the estimate made above for the stratified water body. A Manning's n value of 0.02 describes the smooth bottom.

Density data is simply entered via the temperature values of the fresh water body. A Stratification Type C is chosen to describe the actual temperature profile.

The discharge data values summarize the discharge situation as described above. Finally, the mixing zone specifications include a width value of 120 m, corresponding to 10 % of the water body width of approximately 1200 m. Information is desired over about one mile (≈ 1600 m) which represents the region of interest (ROI) limitation.

B.3 CORMIX1 Session and Results

If desired by the user, CORMIX1 provides a summary of the data as they are entered, and then a full record of the simulation sequence and final results. This session summary is shown in Table B-2. Of particular interest to the user are the evaluations in program element PARAM and CLASS. Note, that the computed length scales L_m' and L_b' are quite small, indicating that the jet or plume will be trapped quickly by the ambient stratification; thus, this is the first numerical indication that the near-field jet/plume will indeed be small relative to the reservoir. The crossflow related scales L_m and L_b are quite

TABLE B-1 DATA SHEET FOR SUBMERGED SINGLE PORT DISCHARGE IN DEEP RESERVOIR

CORMIX1 -- Submerged single port discharges -- CORMIX1

SITE/CASE IDENTIFIER:

Site Name	DEEP RESERVOIR	Prepared by: GHJ
Discharger	A-PLANT	
Pollutant	CHLORIDES	Date prepared:
Design Case	SUMMER - STRATIFICATION	
DOS FILE NAME	SUMMER1	10/18/91

AMBIENT DATA:

Bounded or unbounded?	UNBOUNDED				
Channel width	---	m			
Channel depth	35.0	m			
Depth at discharge	30.5	m			
Ambient flowrate	---	m^3/s	or: Ambient velocity	0.015	m/s
Manning's n	0.02		or: Darcy-Weisbach f	---	

Density data:		Density units:	kg/m^3
Fresh or salt water?	FRESH	Temperature units:	°C
Density or temp. values?	TEMP		
If uniform:		If stratified:	
Average density/temp.	---	Density/temp. at surface	28.1
		Density/temp. at bottom	11.0
		Stratification type	C
		(Pycnocline height	15.5 m)
		(Density/temp. jump	9.0)

DISCHARGE DATA:

Nearest bank (left/right)?	RIGHT				
Distance to nearest bank	46.0	m			
Vertical angle (THETA)	10	°			
Horizontal angle (SIGMA)	90	°			
Port height	0.6	m			
Port diameter	0.254	m	or: Port area	---	m^2
Discharge flow rate	0.153	m^3/s	or: Discharge velocity	---	m/s
Discharge density	---	kg/m^3	or: Discharge temp.	20.0	°C
Concentration units	PPM				
Discharge concentration	3500				

MIXING ZONE DATA:

Is effluent toxic?	YES	If yes: CMC value	1200
		CCC value	600
Is there a WQ standard for conventional pollutant?	NO	If yes: value of standard	---
Any mixing zone specified?	YES	If yes: distance	--- m
		or width (% or m)	120 m
		or area (% or m^2)	
Region of interest	1600 m		
Grid intervals for display	20		

Date of data input into CORMIX1: 10/18/91

TABLE B-2 CORMIX1 SESSION RECORD FOR DEEP RESERVOIR

```
XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX
                    CORNELL MIXING ZONE EXPERT SYSTEM
CORMIX1: Submerged Single Port Discharges        CU Version 1.30 (August 1991
Start of session:          2- 5-1992 15:11:23
*************************************************************************
SITE/CASE DESCRIPTION SUMMARY:
  Site name                 = DEEP RESERVOIR
  Effluent discharger       = A-PLANT
  Pollutant                 = CHLORIDES
  DESIGN CASE               = SUMMER-STRATIFICATION
  FILE NAME                 = SUMMER1
*************************************************************************
SUMMARY OF AMBIENT CONDITIONS:
The ambient water body is unbounded.
  Average water depth                HA     = 35.0 m
  Depth at discharge                 HD     = 30.5 m
  Ambient velocity                   UA     = 0.015 m/s
  Manning's n                        N      = 0.02
Ambient Density Stratification Type C: Constant density surface layer with
  linear density profile in bottom layer separated by a density jump.
  Surface density                    RHOAS  = 996.340881 kg/m^3
  Bottom density                     RHOAB  = 999.691284 kg/m^3
  Intermediate level                 HS     = 15.5 m above bottom (pycnocline)
  Pycnocline density jump            DRHOJ  = 2.222656 kg/m^3
  Top lower layer density            RHOAP  = 998.563538 kg/m^3
  Buoyancy gradient                  E      = 0.000714 1/s^2
  Buoyancy gradient                  ES     = 0.002120 1/s^2 (surrogate)
*************************************************************************
SUMMARY OF DISCHARGE CONDITIONS:
Distance to nearest bank             DISTB  = 46.0 m
Nearest bank/shore                          = right
Port height                          H0     = 0.6 m
Vertical angle                       THETA  = 10 deg
Horizontal angle                     SIGMA  = 90 deg
Port diameter                        D0     = 0.254 m
Port area                            A0     = 0.050669 m^2
Discharge flowrate                   Q0     = 0.153 m^3/s
Discharge velocity                   U0     = 3.019598 m/s
Discharge density                    RHO0   = 998.387207 kg/m^3
Effluent concentration               C0     = 3500
Concentration units                         = PPM
*************************************************************************
SUMMARY OF MIXING ZONE PARAMETERS:
The effluent is toxic as defined by USEPA standards.
  The discharge is subject to Toxic Dilution Zone (TDZ) restrictions.
  CMC  = 1200 PPM.
  CCC  = 600 PPM.
Regulatory Mixing Zone (RMZ) is specified by width.
  Mixing zone extent = 120 (m if length or width or m^2 if area).
Region Of Interest (ROI) is specified by downstream distance.
  XINT = 1600 m  from discharge point
Number of intervals for prediction display = 20
*************************************************************************

*************************************************************************
Program Element PARAM:                  Design Case: SUMMER-STRATIFICATION

Relative density differences between discharge and ambient:

Ambient density at the discharge level        RHOAH0  = 999.647644 kg/m^3
Surrogate ambient density at discharge level  RHOAH0S = 999.561584 kg/m^3
Vertical mean ambient density                 RHOAM   = 997.757019 kg/m^3
The effluent density (998.387207 kg/m^3) is less than the surrounding
```

TABLE B-2 (Continued)

```
  ambient water density at the discharge level (999.647644 kg/m^3).
Therefore, the effluent is positively buoyant and will tend to rise towards
  the surface.

Flow bulk parameters:
Discharge volume flux                        Q0 = 0.153 m^3/s
Discharge momentum flux                      M0 = 0.461999 m^4/s^2.
Discharge buoyancy flux                      J0 = 0.001892 m^4/s^3.
Surrogate discharge buoyancy flux            J0S = 0.001763 m^4/s^3.

Flow length scales:
Discharge length scale                       LQ = 0.225098 m
Jet crossflow length scale                   Lm = 45.313648 m
Plume to crossflow length scale              Lb = 560.546753 m
Jet to plume transition length scale         LM = 12.883611 m
Jet stratification length scale               Lm' = 5.043541 m
Surrogate jet stratification length scale     Lm's = 3.842166 m
Plume stratification length scale             Lb' = 3.155618 m
Surrogate plume stratification length scale   Lb's = 2.098193 m
The surrogate length scales assume an equivalent linear stratification
  including the density jump in the lower layer.

Non-dimensional parameters:
Densimetric Froude number                    FR0 = 53.880943
Jet/Crossflow Velocity Ratio                 R   = 201.306534
*****************************************************************************

*****************************************************************************
Program Element CLASS:               Design Case: SUMMER-STRATIFICATION

The near field flow configuration will have the following features:

If flow trapping occurs, then the flow is jet-like and is strongly affected
  by the ambient density stratification with a weak crossflow effect (if any)

The specified two layer ambient density stratification is dynamically
  important. The discharge near field flow will be confined to the lower
  layer by the ambient density stratification.
Furthermore, it may be trapped below the ambient density jump at the
  pycnocline.

The discharge near-field behavior is dominated by either the positive
  buoyancy of the discharge or the upward vertical orientation of the
  discharge port.
There is the possibility of dynamic bottom attachment.

The following conclusion on the flow configuration applies to the lower
  layer only of the specified ambient stratification condition C.
Note that the lower layer will be overlaid by the surface layer of the
  ambient density stratification. The surface layer will remain undisturbed
  by the near field discharge flow (with the exception of some possible
  intrusion along the pycnocline):
    +-------------------------------------------------------+
              Flow class              S3
              Applicable layer depth  HS = 15.5 m
    +-------------------------------------------------------+
*****************************************************************************
```


TABLE B-2 (Continued)

```
*****************************************************************************
Program Element SUM:                  Design Case: SUMMER-STRATIFICATION
-----------------------------------------------------------------------------
X-Y-Z Coordinate system:
  Origin is located at the bottom below the port center:
    46.0 m from the right bank/shore.
-----------------------------------------------------------------------------
HYDRODYNAMIC MIXING ZONE (HMZ) CONDITIONS :
Note: The HMZ is the zone of strong initial mixing.  It has no legal
  implication.  However, this information may be useful for the discharge
  designer because the mixing in the HMZ is usually sensitive to the
  discharge design conditions.
  Pollutant concentration at edge of HMZ =          264.7477 PPM
  Dilution at edge of HMZ                =           13.2
  HMZ Location:                        x =      62.45 m
  (centerline coordinates)             y =      10.39 m
                                       z =       2.41 m
  HMZ plume dimensions:       half-width =     124.91 m
                               thickness =        .53 m
-----------------------------------------------------------------------------
UPSTREAM INTRUSION SUMMARY:
Plume exhibits upstream intrusion due to low ambient velocity or strong
  discharge buoyancy.

  Intrusion length                         =      62.45 m
  Intrusion stagnation point               =     -62.45 m
  Intrusion thickness                      =       1.34 m
  Intrusion half width at impingement      =     124.91 m
  Intrusion half thickness at impingement  =        .53 m

********************** TOXIC DILUTION ZONE SUMMARY ***********************
Recall: The TDZ corresponds to the three (3) criteria issued in the USEPA
  Technical Support Document (TSD) for Water Quality-based Toxics Control,
  1991 (EPA/505/2-90-001).
  Criterion maximum concentration (CMC)  =      1200.000000 PPM
  Corresponding dilution                 =           2.9
The CMC was encountered at the following plume position:
  Plume location:                      x =        .00 m
    (centerline coordinates)           y =       2.37 m
                                       z =        .75 m
  Plume dimension:            half-width =        .40 m

 Criterion 1: This location is within 50 times the discharge length scale of
              Lq =        .22 m.
 +++++ The discharge length scale test for the TDZ has been SATISFIED. ++++++

 Criterion 2: This location is within 5 times the ambient water depth
              HD =       30.50 (m).
 ++++++++++ The ambient depth test for the TDZ has been SATISFIED.+++++++++++

 Criterion 3: This location is within one tenth the distance of the extent
              of the Regulatory Mixing Zone of     41.33 m downstream.
 +++++ The Regulatory Mixing Zone test for the TDZ has been SATISFIED. ++++++

 The diffuser discharge velocity is equal to        3.01 m/s.
   This exceeds the value of 3.0 m/s recommended in the TSD.
 ++ The discharge velocity recommendation for the TDZ has been SATISFIED. +++

 *** All three CMC criteria for the TDZ are SATISFIED for this discharge. ***

********************* REGULATORY MIXING ZONE SUMMARY **********************
The plume conditions at the boundary of the specified RMZ are as follows:
```

TABLE B-2 (Continued)

```
   Pollutant concentration                     =       300.556700 PPM
   Corresponding dilution                      =         11.9
   Plume location:                           x =      41.33 m
     (centerline coordinates)                y =     -46.00 m
                                             z =       2.41 m
   Plume dimensions:              half-width =      120.00 m
                                   thickness =         .49 m

 At this position, the plume is contacting to the right bank.
 Furthermore, the CCC for the toxic pollutant has indeed been met
   within the RMZ. In particular:
 The CCC was encountered at the following plume position:
 The CCC for the toxic pollutant was encountered at the following
   plume position:
   CCC                                         =       600.000000 PPM
   Corresponding dilution                      =          5.8
   Plume location:                           x =        .00 m
     (centerline coordinates)                y =       5.96 m
                                             z =       1.27 m
   Plume dimension:               half-width =         .80 m

 ********************************************************************
   DESIGN CASE: SUMMER-STRATIFICATION                   FILE NAME: SUMM
   End of session:   2- 5-1992 15:31: 5
 CORMIX1: Submerged Single Port Discharges       CU Version 1.30 (August 1991
 XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX
```

Note: Programs distributed by US EPA and Cornell University with the same version numbers may not always provide identical simulation results due to a lag time between the discovery of subtle program code errors and the transmission of corrections from Cornell to the EPA software distribution network. This case study represents such a situation; the EPA ver 1.30 provides simulation results which differ slightly from those show above.

large, indicating that the ambient velocity is very weak. The resulting flow class S3 is dominated by the ambient stratification; the plume will be limited to the lower layer of the stratification. The user should also consult the description of flow class S3 that is available during the CORMIX1 session (not reproduced here). The detailed plume properties are computed in program element HYDRO, and are displayed in the Fortran output file (see Table B-3, discussed in more detail further below).

Many important features of the plume prediction are summarized in program element SUM of the session record (see Table B-2). Notably, all aspects pertaining to mixing zone regulations are contained in that summary. For example, it can be seen quickly from that summary that the present discharge configuration meets all three toxic dilution zone (TDZ) criteria

TABLE B-3 OUTPUT FILE FOR DETAILED CORMIX1 PREDICTION FOR DEEP RESERVOIR

```
1111111111111111111111111111111111111111111111111111111111111111111111111111
                        CORNELL MIXING ZONE EXPERT SYSTEM
CORMIX1: Submerged Single Port Discharges          CU Version 1.30 (August 199
----------------------------------------------------------------------------

CASE DESCRIPTION
 SITE NAME:               DEEP RESERVOIR
 DISCHARGER NAME:         A-PLANT
 POLLUTANT NAME:          CHLORIDES

 DESIGN CASE:             SUMMER-STRATIFICATION
 DOS FILE NAME:           SUMMER1.CXO
 DATE AND TIME OF FORTRAN SIMULATION:   1-16-1992_18:19:31

ENVIRONMENT PARAMETERS (METRIC UNITS)
 UNBOUNDED SECTION
 HA      =     35.00  HD      =     30.50
 UA      =      .015  F       =      .010

 DENSITY STRATIFIED ENVIRONMENT
 STRCND =  C
 RHOAS   =    996.34  RHOAB   =    999.69  DRHOJ   =     2.22    RHOA    = 99999.
 RHOAHO =     999.65  HINT    =     15.50
 E       = .7140E-03  ES      = .2120E-02

DISCHARGE PARAMETERS (METRIC UNITS)
 BANK    =  RIGHT     DISTB   =     46.00
 D0      =      .254  A0      =      .051 H0      =       .60
 THETA   =     10.00  SIGMA   =     90.00
 U0      =     3.020  Q0      =      .153 (Q0     = .1530E+00)
 RHO0    =    998.39  DRHO0   =      1.26  GP0    = .1237E-01
 C0      = .3500E+04  UNITS   =  PPM

FLUX VARIABLES
 Q0      = .1530E+00  M0      = .4620E+00  J0     = .1892E-02  SIGNJ0 = 1.0
 ASSOCIATED LENGTH SCALES (METERS)
 LQ      =       .23  LM      =     12.88  Lm     =     45.31  Lb      =    560.
                                           Lmp    =      5.04  Lbp     =      3.

NON-DIMENSIONAL PARAMETERS
 FR0     =     53.88  R       =    201.31

FLOW CLASSIFICATION
*************************************************************************
*          FLOW CLASS                      =    S3                      *
*          APPLICABLE LAYER DEPTH HS       =    15.50                   *
*************************************************************************

MIXING ZONE / TOXIC DILUTION / REGION OF INTEREST PARAMETERS
 C0      = .3500E+04  CUNITS = PPM
 NTOX    =  1         CMC     = .1200E+04  CCC     =     CSTD
 NSTD    =  1         CSTD    = .6000E+03
 LEGMZ   =  1         LEGSPC =  2          LEGVAL =      120.00
 XLEG    =       .00  WLEG    =    120.00  ALEG    =       .00
 XINT    =   1600.00
 XMAX    =   1600.00
 NSTEP   = 20

 X-Y-Z COORDINATE SYSTEM:
    ORIGIN is located at the bottom and below the center of the port:
       -46.00 m  from the RIGHT bank/shore.
    X-axis points downstream, Y-axis points to left, Z-axis points upward.
```

TABLE B-3 (Continued)

BEGIN MOD101: DISCHARGE MODULE

PROFILE DEFINITIONS:
B = Gaussian 1/e (37%) half-width, normal to trajectory
C = centerline concentration
S = corresponding centerline dilution

PREDICTION

X	Y	Z	S	C	B
.00	.00	.60	1.0	.350E+04	.13

END OF MOD101: DISCHARGE MODULE

BEGIN MOD114: NEAR-HORIZONTAL JET IN LINEAR STRATIFICATION

CROSS-FLOWING DISCHARGE
PROFILE DEFINITIONS:
B = Gaussian 1/e (37%) half-width, normal to trajectory
C = centerline concentration
S = corresponding centerline dilution

PREDICTION

X	Y	Z	S	C	B
.00	.00	.60	1.0	.350E+04	.14
.00	.52	.62	1.4	.246E+04	.20
.00	1.04	.65	1.8	.190E+04	.25
.00	1.56	.68	2.3	.154E+04	.31
.00	2.08	.72	2.7	.130E+04	.37

** CMC HAS BEEN FOUND **
THE POLLUTANT CONCENTRATION IN THE PLUME FALLS BELOW THE CMC VALUE OF .12E
IN THE CURRENT PREDICTION INTERVAL.
THIS IS THE EXTENT OF THE TOXIC DILUTION ZONE.

.00	2.60	.77	3.1	.113E+04	.43
.00	3.12	.83	3.5	.991E+03	.49
.00	3.64	.90	4.0	.885E+03	.54
.00	4.16	.97	4.4	.800E+03	.60
.00	4.68	1.05	4.8	.729E+03	.66
.00	5.20	1.13	5.2	.670E+03	.72
.00	5.72	1.23	5.6	.620E+03	.78

** WATER QUALITY STANDARD OR CCC HAS BEEN FOUND **
THE POLLUTANT CONCENTRATION IN THE PLUME FALLS BELOW THE WATER QUALITY STAND
OR CCC VALUE OF .60E+03 IN THE CURRENT PREDICTION INTERVAL.
THIS IS THE SPATIAL EXTENT OF CONCENTRATIONS EXCEEDING THE WATER QUALITY
STANDARD OR CCC VALUE.

.00	6.24	1.33	6.1	.577E+03	.83
.00	6.76	1.44	6.5	.539E+03	.89
.00	7.28	1.56	6.9	.507E+03	.95
.00	7.80	1.68	7.3	.477E+03	1.01
.00	8.32	1.81	7.8	.451E+03	1.07
.00	8.84	1.95	8.2	.428E+03	1.12
.00	9.36	2.10	8.6	.407E+03	1.18
.00	9.88	2.26	9.0	.388E+03	1.24
.00	10.40	2.42	9.4	.371E+03	1.30

END OF MOD114: NEAR-HORIZONTAL JET IN LINEAR STRATIFICATION

TABLE B-3 (Continued)

BEGIN MOD137: TERMINAL LAYER INJECTION/UPSTREAM SPREADING

CONTROL VOLUME

UPSTREAM INTRUSION PROPERTIES:

MAX/MIN ELEVATION OF JET/PLUME RISE/FALL	=	3.72 m
LAYER THICKNESS IN INJECTION REGION	=	1.30 m
UPSTREAM INTRUSION LENGTH	=	62.46 m
X-POSITION OF UPSTREAM STAGNATION POINT	=	-62.45 m
THICKNESS IN UPSTREAM INTRUSION REGION	=	1.35 m
HALF-WIDTH AT DOWNSTREAM END	=	124.91 m
THICKNESS AT DOWNSTREAM END	=	.54 m

PROFILE DEFINITIONS:
BV = top-hat thickness,measured vertically
BH = top-hat half-width, measured horizontally in Y-direction
C = average (bulk) concentration
S = corresponding average (bulk) dilution
ZU = upper plume boundary (Z-coordinate)
ZL = lower plume boundary (Z-coordinate)

PREDICTION

X	Y	Z	S	C	B			
CONTROL VOLUME INFLOW								
.00	10.40	2.42	9.4	.371E+03	1.30			
X	Y	Z	S	C	BV	BH	ZU	ZL
CONTROL VOLUME OUTFLOW								
62.46	10.40	2.42	13.2	.265E+03	.54	124.91	2.69	2.15

END OF MOD137: TERMINAL LAYER INJECTION/UPSTREAM SPREADING

*** END OF HYDRODYNAMIC MIXING ZONE (HMZ) ***

IN THIS DESIGN CASE, THE DISCHARGE IS LOCATED TOO CLOSE TO BANK/SHORE.
SOME BOUNDARY INTERACTION OCCURS AT END OF NEAR-FIELD.
THIS MAY BE RELATED TO A DESIGN CASE WITH A VERY LOW AMBIENT VELOCITY.
THE DILUTION VALUES IN ONE OR MORE OF THE PRECEDING ZONES MAY BE TOO HIGH.

CAREFULLY EVALUATE RESULTS IN NEAR-FIELD AND DETERMINE DEGREE OF INTERACTION

CONSIDER LOCATING OUTFALL FURTHER AWAY FROM BANK OR SHORE.
IN THE NEXT PREDICTION MODULE, THE PLUME CENTERLINE WILL BE SET
TO FOLLOW THE BANK/SHORE.

BEGIN MOD142: BUOYANT TERMINAL LAYER SPREADING

PLUME IS ATTACHED TO RIGHT BANK/SHORE.
PLUME WIDTH IS NOW DETERMINED FROM RIGHT BANK/SHORE.

SIMULATION LIMIT BASED ON MAXIMUM SPECIFIED DISTANCE = 1600.00m.
THIS IS THE REGION OF INTEREST LIMITATION.

PROFILE DEFINITIONS:
BV = top-hat thickness,measured vertically
BH = top-hat half-width, measured horizontally in Y-direction
C = average (bulk) concentration
S = corresponding average (bulk) dilution
ZU = upper plume boundary (Z-coordinate)
ZL = lower plume boundary (Z-coordinate)

PREDICTION: STAGE 2: BANK ATTACHED

X	Y	Z	S	C	BV	BH	ZU	ZL

TABLE B-3 (Continued)

```
** REGULATORY MIXING ZONE BOUNDARY **
IN THIS PREDICTION INTERVAL THE PLUME WIDTH MEETS OR EXCEEDS
     THE REGULATORY VALUE =    120.00 m
     THIS IS THE EXTENT OF THE REGULATORY MIXING ZONE.
```

62.46	-46.00	2.42	13.2	.265E+03	.74	181.31	2.79	2.05
139.33	-46.00	2.42	13.5	.260E+03	.70	195.85	2.77	2.07
216.21	-46.00	2.42	13.7	.255E+03	.67	209.61	2.75	2.09
293.09	-46.00	2.42	13.9	.251E+03	.64	222.74	2.74	2.10
369.96	-46.00	2.42	14.1	.248E+03	.61	235.30	2.72	2.11
446.84	-46.00	2.42	14.3	.245E+03	.59	247.38	2.71	2.12
523.72	-46.00	2.42	14.5	.242E+03	.57	259.04	2.70	2.13
600.60	-46.00	2.42	14.6	.240E+03	.55	270.31	2.69	2.14
677.47	-46.00	2.42	14.8	.237E+03	.54	281.24	2.69	2.15
754.35	-46.00	2.42	14.9	.235E+03	.52	291.86	2.68	2.16
831.23	-46.00	2.42	15.0	.233E+03	.51	302.20	2.67	2.17
908.11	-46.00	2.42	15.1	.231E+03	.49	312.28	2.67	2.17
984.98	-46.00	2.42	15.3	.229E+03	.48	322.12	2.66	2.18
1061.86	-46.00	2.42	15.4	.228E+03	.47	331.75	2.65	2.18
1138.74	-46.00	2.42	15.5	.226E+03	.46	341.16	2.65	2.19
1215.61	-46.00	2.42	15.6	.225E+03	.45	350.39	2.65	2.19
1292.49	-46.00	2.42	15.7	.223E+03	.45	359.43	2.64	2.20
1369.37	-46.00	2.42	15.8	.222E+03	.44	368.31	2.64	2.20
1446.25	-46.00	2.42	15.9	.220E+03	.43	377.03	2.63	2.20
1523.12	-46.00	2.42	16.0	.219E+03	.42	385.61	2.63	2.21

```
SIMULATION LIMIT AS SPECIFIED HAS BEEN REACHED; PREDICTION TERMINATES
   AT THIS STAGE.
```

1600.00	-46.00	2.42	16.1	.218E+03	.42	394.04	2.63	2.21

```
END OF MOD142: BUOYANT TERMINAL LAYER SPREADING
-----------------------------------------------------------------------------
-----------------------------------------------------------------------------
End of Output

CORMIX1: Submerged Single Port Discharges         CU Version 1.30 (August 199
1111111111111111111111111111111111111111111111111111111111111111111111111111111
```

Note: Programs distributed by US EPA and Cornell University with the same version numbers may not always provide identical simulation results due to a lag time between the discovery of subtle program code errors and the transmission of corrections from Cornell to the EPA software distribution network. This case study represents such a situation; the EPA ver 1.30 provides simulation results which differ slightly from those show above.

and also the regulatory mixing zone (RMZ) limitation. Obviously, other ambient conditions and discharge variations should be considered in additional simulations before a design such as this should be deemed fully satisfactory.

B.4 Graphical Displays of Detailed Plume Predictions

As for most engineering studies it is desirable to produce graphical displays for visualization of the predicted results. The data contained in the Fortran output file (Table B-3) form the basis for such plots. Unfortunately, it is often difficult to display all plume features in one single plot because the plume may contain a lot of near-field details while extending over large distances into the far-field. A short examination of Table B-3 proves that point: The plume gets quickly trapped within a very limited near-field but with considerable mixing (see MOD114 of the CORMIX1 prediction). Yet after that the plume extends over large distances into the far-field forming a wide thin layer within the stratified reservoir (see MOD142).

Two plots have been prepared to display the detailed near-field mixing (Figure B-3) and the larger scale far-field spreading (Figure B-4) of the effluent plume. Figure B-3 shows

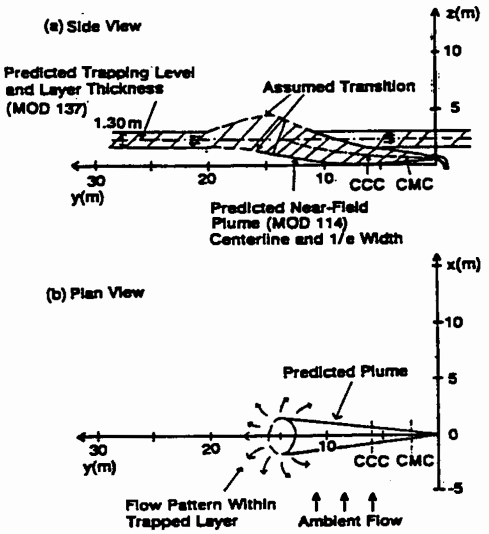


FIGURE B-3 NEAR-FIELD DETAILS OF CORMIX1 PLUME PREDICTION

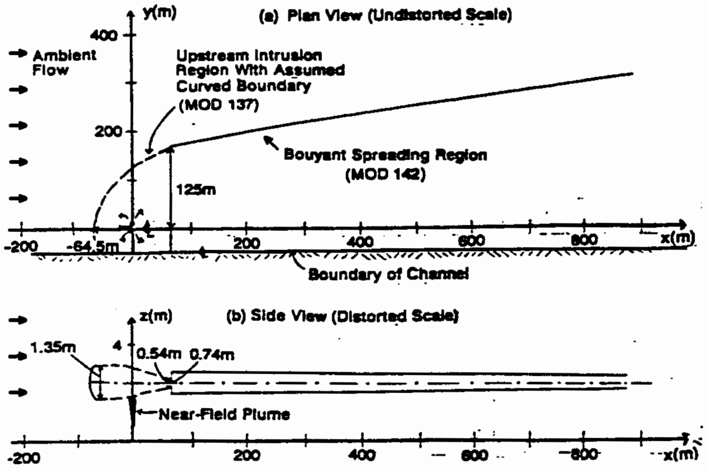


FIGURE B-4 OVERALL SHAPE OF PLUME PREDICTED BY CORMIX1

the initial trajectory of the slightly upward curved jet (MOD114) that gets trapped at an elevation of 2.34 m above the local bottom. In the trapping stage the jet undergoes a complicated transition (MOD 137) to the horizontally spreading layer. No detailed models are available to predict that transition; CORMIX1 predicts only a few parameters such as the upstream intrusion length, downstream width, etc. As indicated in Figure B-3, reasonable transition boundaries can be assumed to provide smooth transitions to the far-field processes.

The far-field plot (Figure B-4) shows the wide and thin layer that forms as the plume collapses laterally within the ambient stratification while it is advected by the weak ambient flow. Again, a discontinuity occurs in the transition from the control volume (MOD137) describing upstream spreading to the continuous prediction for ambient buoyant spreading (MOD142). The cause for this discontinuity is the simultaneous interaction of the plume with the channel boundary that occurs within MOD137. CORMIX1 detects such complicated simultaneous processes and warns the user who then can compensate by providing reasonable, mass-conserving transitions.

It is also possible to include concentration values, e.g. along the centerline, in plots of this type. This has not been done in these figures in order not to overload them. Only the locations where the CMC and CCC values are met have been indicated.

CORMIX1 AND 2 CASE STUDY: SUBMERGED SINGLE PORT DISCHARGE AND MULTIPORT DIFFUSER IN A SHALLOW RIVER

The design modification of an existing (hypothetical situation) discharge from a plant into a shallow river is considered in this case study. This affords an opportunity to demonstrate the joint use of CORMIX and of a dye field study in order to analyze an existing effluent plume from a single port discharge and to suggest a design conversion to a multiport diffuser with improved mixing characteristics.

C.1 Problem Statement

An industrial plant (B-Plant) is currently discharging its effluent into an adjacent shallow river. The design flowrate is quite small at 2.1 mgd ($\approx$ 0.092 m^3/s). The river is about 200 to 300 ft wide at the discharge location and the following downstream reach. Water depth is, of course, dependent on the river discharge that is seasonally variable. An examination of available streamflow records (USGS data) suggests a 7Q10 low flow discharge of 285 cfs ($\approx$ 8.1 m^3/s).

Recent water quality studies in the discharge reach performed during low flow summer conditions have shown occasional coloration problems in the discharge plume that seem to be related to inadequate mixing characteristics of the present submerged single port discharge. For that reason the plant operator is considering an improvement of the discharge structure.

C.2 Existing Single Port Discharge: Dye Field Study and CORMIX1 Comparison:

An initial field study was conducted in order (a) to measure the geometric and hydraulic characteristics of the discharge reach with special emphasis on the first 1000 ft downstream, and (b) to determine plume concentrations by means of a dye injection into the plant effluent.

Figure C-1 shows the plan geometry of the discharge reach. River cross-sections were determined by depth measurements at several stations as indicated. For example, Figure C-2 gives the cross-section at the discharge location. All cross-sections exhibit quite some non-uniformity as is typical for a gently meandering alluvial (gravel) river. The indicated water level corresponds to the river discharge of 840 cfs ($\approx$ 23.7 m^3/s) that was measured during the field survey using the usual USGS streamgaging methods. The 8 in ($\approx$ 0.2 m) diameter discharge pipe is

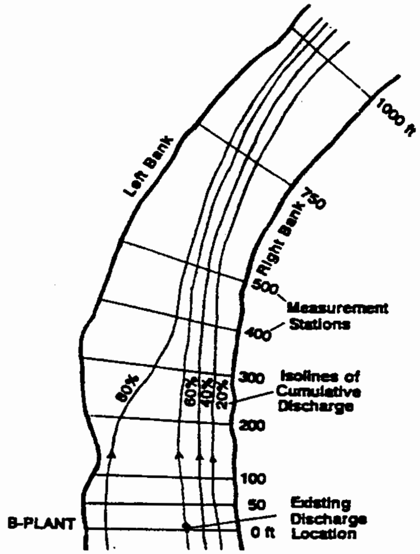


FIGURE C-1 PLAN VIEW OF DOWNSTREAM REACH OF SHALLOW RIVER WITH MEASUREMENT CROSS-SECTIONS AND CUMULATIVE DISCHARGE ISOLINES

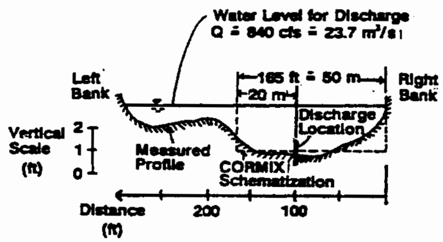


FIGURE C-2 RIVER CROSS-SECTION MEASURED AT DISCHARGE LOCATION

located about 95 ft from the right bank, and is pointing in the downstream direction.

To obtain a detailed description of the flow field in the river reach, discharge measurements were conducted at several more downstream stations (200, 400, 750, and 1000 ft, respectively). Figure C-1 includes the cumulative discharge isolines, expressed in % of the total discharge as measured from the right bank, for the reach. These lines provide a useful indication of the mean flow pattern in such a winding river for subsequent interpretation of observed plume features (see also comments on cumulative discharge method in Subsection IVC(2)c).

A dye test was carried out by continuously discharging a fluorescein dye solution into the plant effluent. The dye concentration exiting the discharge pipe was 560 ppb. Dye concentrations were measured at the transects indicated in Figure C-1, and have been plotted in Figure C-3 as a function of distance from the right bank. The observed concentration profiles show decreasing peak (maximum) values with increasing downstream distances. Observations indicated a vertically mixed plume at

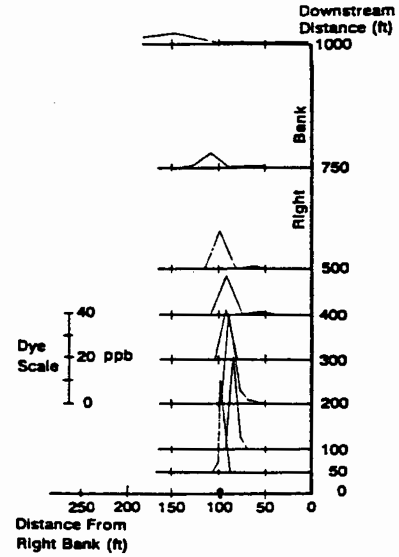


FIGURE C-3 DYE CONCENTRATION FROM FIELD TEST PLOTTED AS A FUNCTION OF DISTANCE FROM RIGHT BANK all locations. In the display of Figure C-3 the plume centerline position is clearly shifting relative to the right bank, and the plume width occasionally appears to contract in width.

An initial CORMIX1 evaluation was carried out to ascertain its applicability in this somewhat irregular flow environment. For this purpose, the cross-section was schematized as a rectangular cross-section putting emphasis on the depth conditions around the discharge location. Information from the cumulative discharge data was used. Note that the cumulative discharge data shows the discharge located at the 60 % isoline. Although it appears geographically closer to the right bank, the discharge is hydraulically closer to the left bank. This is reflected in the schematization: Within the assumed 165 ft (≈ 50 m) wide by 2 ft (≈ 0.6 m) deep rectangular channel, the discharge is located 20 m from the left bank. The roughness of the slightly winding, but otherwise clean natural channel has been specified by a Manning's n value of 0.03.

Table C-1 summarizes the data prepared for the CORMIX1 session, while Table C-2 represents the session record. The detailed plume prediction is given in Table C-3. CORMIX1 predicts that the plume gets rapidly mixed over the shallow depth, and is primarily influenced by far-field mixing processes, a feature that is quite consistent with observations. The dye concentration distribution predicted by CORMIX1 in the schematic rectangular channel are plotted in Figure C-4 and show a much more regular mixing pattern than the earlier Figure C-3. However, matters can be readily reconciled when both field data and CORMIX1 predictions are interpreted as a function of cumulative discharge. This has been done in Figure C-5 where both distributions are directly superimposed on the cumulative discharge pattern. The agreement is excellent. This entire procedure points out the need for high-quality field data if detailed interpretations and predictions of discharge plumes are desired.

C.3 Proposed Multiport Diffuser Discharge Under 7Q10 Flow Conditions: CORMIX2 Predictions

The following strategy is pursued in order to improve the near-field mixing characteristics of the existing discharge: (a) Utilization of a multiport diffuser to increase the initial entrainment of ambient water into the multiple effluent jets, and (b) shifting of the discharge location toward the right bank to delay the contact with the left shoreline that with the present installation, seems to occur at a downstream distance of 1000 ft.

The design study is carried out for the low flow ambient condition given by the 7Q10 discharge (285 cfs ≈ 8.1 m^3/s) as is typical for water quality studies on riverine sites. Assuming channel roughness is unchanged, the Manning equation presented in Subsection IVC(1)a suggests a water depth of approximately 0.3 m occurs during the 7Q10 design flow condition.

TABLE C-1 DATA SHEET FOR SUBMERGED SINGLE PORT DISCHARGE IN SHALLOW RIVER

CORMIX1 -- Submerged single port discharges -- CORMIX1

SITE/CASE IDENTIFIER:		Prepared by:
Site Name	SHALLOW-RIVER	GHJ
Discharger	B-PLANT	Date prepared:
Pollutant	TRACER	10/18/91
Design Case	DYE-TEST	
DOS FILE NAME	DYE1	

AMBIENT DATA:					
Bounded or unbounded?	BOUNDED				
Channel width	50.0	m			
Channel depth	0.60	m			
Depth at discharge	0.60	m			
Ambient flowrate	23.7	m^3/s	or: Ambient velocity	---	m/s
Manning's n	0.03		or: Darcy-Weisbach f	---	
Density data:			Density units:	kg/m^3	
Fresh or salt water?	FRESH		Temperature units:	°C	
Density or temp. values?	TEMP				
If uniform:			If stratified:		
Average density/temp.	20.0		Density/temp. at surface	---	
			Density/temp. at bottom	---	
			Stratification type	---	
			(Pycnocline height	---	m)
			(Density/temp. jump	---	)

DISCHARGE DATA:					
Nearest bank (left/right)?	LEFT				
Distance to nearest bank	20.0	m			
Vertical angle (THETA)	0	°			
Horizontal angle (SIGMA)	0	°			
Port height	0.15	m			
Port diameter	0.2	m	or: Port area	---	m^2
Discharge flow rate	0.092	m^3/s	or: Discharge velocity	---	m/s
Discharge density	---	kg/m^3	or: Discharge temp.	22.0	°C
Concentration units	PPB				
Discharge concentration	560				

MIXING ZONE DATA:					
Is effluent toxic?	NO		If yes: CMC value	---	
			CCC value	---	
Is there a WQ standard for conventional pollutant?	NO		If yes: value of standard	---	
Any mixing zone specified?	NO		If yes: distance	---	m
			or width (% or m)	---	
			or area (% or m^2)	---	
Region of interest	1000	m			
Grid intervals for display	20				

Date of data input into CORMIX1: 10/18/91

TABLE C-2 CORMIX1 SESSION RECORD FOR SHALLOW RIVER

```
XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX
                     CORNELL MIXING ZONE EXPERT SYSTEM
CORMIX1: Submerged Single Port Discharges          CU Version 1.30 (August 1991)
Start of session:          10-18-1991 17:23:40
*******************************************************************************
SITE/CASE DESCRIPTION SUMMARY:
  Site name           = SHALLOW-RIVER
  Effluent discharger = B-PLANT
  Pollutant           = TRACER
  DESIGN CASE         = DYE-TEST
  FILE NAME           = DYE1
*******************************************************************************
SUMMARY OF AMBIENT CONDITIONS:
The ambient water body is bounded.
  Channel width               W      = 50.0 m
  Average water depth         HA     = 0.60 m
  Depth at discharge          HD     = 0.60 m
  Ambient velocity            UA     = 0.790000 m/s
  Ambient flowrate            QA     = 23.7 m^3/s
  Manning's n                 N      = 0.03
The ambient is unstratified.
  Ambient density             RHOA   = 998.387207 kg/m^3
*******************************************************************************
SUMMARY OF DISCHARGE CONDITIONS:
Distance to nearest bank      DISTB  = 20.0 m
Nearest bank/shore                   = left
Port height                   H0     = 0.15 m
Vertical angle                THETA  = 0 deg
Horizontal angle              SIGMA  = 0 deg
Port diameter                 D0     = 0.2 m
Port area                     A0     = 0.031415 m^2
Discharge flowrate            Q0     = 0.092 m^3/s
Discharge velocity            U0     = 2.928537 m/s
Discharge density             RHO0   = 997.958801 kg/m^3
Effluent concentration        C0     = 560
Concentration units                  = PPB
*******************************************************************************
SUMMARY OF MIXING ZONE PARAMETERS:
The effluent is not toxic as defined by USEPA standards.
No ambient water quality standard has been specified.
No Regulatory Mixing Zone (RMZ) has been specified.
Region Of Interest (ROI) is specified by downstream distance.
  XINT = 1000 m  from discharge point
Number of intervals for prediction display = 20
*******************************************************************************

*******************************************************************************
Program Element PARAM:                Design Case: DYE-TEST

Relative density differences between discharge and ambient:
The effluent density (997.958801 kg/m^3) is less than the surrounding
  ambient water density at the discharge level (998.387207 kg/m^3).
Therefore, the effluent is positively buoyant and will tend to rise towards
  the surface.

Flow bulk parameters:
Discharge volume flux                           Q0 = 0.092 m^3/s
Discharge momentum flux                         M0 = 0.269425 m^4/s^2.
Discharge buoyancy flux                         J0 = 0.000387 m^4/s^3.
```


TABLE C-2 (Continued)

```
Flow length scales:
Discharge length scale                        LQ = 0.177243 m
Jet crossflow length scale                    Lm = 0.657040 m
Plume to crossflow length scale               Lb = 0.000785 m
Jet to plume transition length scale          LM = 19.006311 m

Non-dimensional parameters:
Densimetric Froude number                     FR0 = 100.948029
Jet/Crossflow Velocity Ratio                  R   = 3.707009
*******************************************************************************

*******************************************************************************
Program Element CLASS:                Design Case: DYE-TEST

The near field flow configuration will have the following features:

The discharge near-field behavior is dominated by either the positive
  buoyancy of the discharge or the upward vertical orientation of the
  discharge port.
There is the possibility of dynamic bottom attachment.

The discharge flow will experience instabilities with full vertical mixing
  in the near-field.
  There may be benthic impact of high pollutant concentrations.

The following conclusion on the flow configuration applies to a layer
  corresponding to the full water depth at the discharge site:
     +------------------------------------------------------+
              Flow class                H5-0
              Applicable layer depth    HS = 0.60 m
     +------------------------------------------------------+
*******************************************************************************

*******************************************************************************
Program Element SUM:                 Design Case: DYE-TEST

-------------------------------------------------------------------------------
X-Y-Z Coordinate system:
  Origin is located at the bottom below the port center:
    20.0 m from the left bank/shore.
-------------------------------------------------------------------------------
HYDRODYNAMIC MIXING ZONE (HMZ) CONDITIONS :
Note: The HMZ is the zone of strong initial mixing.  It has no legal
  implication.  However, this information may be useful for the discharge
  designer because the mixing in the HMZ is usually sensitive to the
  discharge design conditions.
  Pollutant concentration at edge of HMZ =         329.4117 PPB
  Dilution at edge of HMZ                =         1.7
  HMZ Location:                        x =         .60 m
  (centerline coordinates)             y =         .00 m
                                       z =         .60 m
  HMZ plume dimensions:       half-width =         .16 m
                               thickness =         .60 m

-------------------------------------------------------------------------------
FAR-FIELD MIXING SUMMARY:
  Plume becomes fully vertically mixed at          .6 m downstream.

*********************** TOXIC DILUTION ZONE SUMMARY ***********************
```

TABLE C-2 (Continued)

```
No TDZ was specified for this simulation.

********************* REGULATORY MIXING ZONE SUMMARY ***********************
No RMZ and no ambient water quality standard have been specified.

***************************************************************************
  DESIGN CASE: DYE-TEST                          FILE NAME: DYE1
  End of session:  10-18-1991 17:31:49
CORMIX1: Submerged Single Port Discharges        CU Version 1.30 (August 1991)
XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX
```

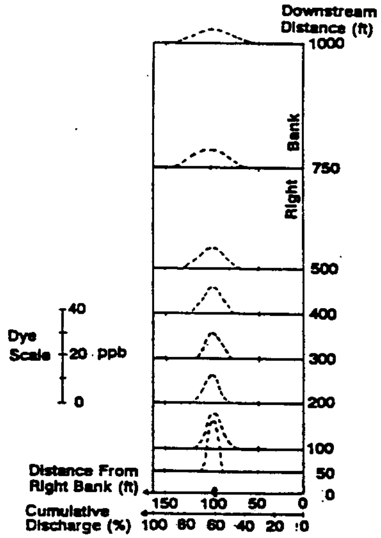


FIGURE C-4 DYE CONCENTRATIONS PREDICTED BY CORMIX1 PLOTTED AS A FUNCTION OF DISTANCE FROM RIGHT BANK IN SCHEMATIZED CHANNEL

TABLE C-3 OUTPUT FILE FOR DETAILED CORMIX1 PREDICTION FOR SHALLOW RIVER

```
1111111111111111111111111111111111111111111111111111111111111111111111111111111
                        CORNELL MIXING ZONE EXPERT SYSTEM
CORMIX1: Submerged Single Port Discharges        CU Version 1.30 (August 1991)
-------------------------------------------------------------------------------
CASE DESCRIPTION
 SITE NAME:                SHALLOW-RIVER
 DISCHARGER NAME:          B-PLANT
 POLLUTANT NAME:           TRACER

 DESIGN CASE:              DYE-TEST
 DOS FILE NAME:            DYE1    .CXO
 DATE AND TIME OF FORTRAN SIMULATION:  10-18-1991_17:30:57

ENVIRONMENT PARAMETERS (METRIC UNITS)
 BOUNDED SECTION
 BS     =      50.00  AS     =     30.00  QA     =      23.70
 HA     =        .60  HD     =       .60
 UA     =       .790  F      =      .084

 UNIFORM DENSITY ENVIRONMENT
 RHOA   =     998.39

DISCHARGE PARAMETERS (METRIC UNITS)
 BANK   =  LEFT        DISTB  =     20.00
 D0     =        .200  A0     =      .031  H0     =        .15
 THETA  =        .00   SIGMA  =      .00
 U0     =       2.929  Q0     =      .092  (Q0    = .9200E-01)
 RHO0   =      997.96  DRHO0  =       .43  GP0    = .4208E-02
 C0     = .5600E+03    UNITS  =  PPB

FLUX VARIABLES
 Q0     = .9200E-01  M0     = .2694E+00  J0     = .3871E-03  SIGNJ0 = 1.0
 ASSOCIATED LENGTH SCALES (METERS)
 LQ     =        .18  LM     =     19.01  Lm     =        .66  Lb     =        .00
                                          Lmp    =   99999.90  Lbp    =   99999.90

NON-DIMENSIONAL PARAMETERS
 FR0    =     100.95  R      =      3.71

FLOW CLASSIFICATION
 ***********************************************************************
 *        FLOW CLASS                     =    H5-0                      *
 *        APPLICABLE LAYER DEPTH HS      =     .60                      *
 ***********************************************************************

MIXING ZONE / TOXIC DILUTION / REGION OF INTEREST PARAMETERS
 C0     = .5600E+03  CUNITS = PPB
 NSTD   = 0          CSTD   = .1000E+07
 LEGMZ  = 0          LEGSPC = 0            LEGVAL =   99999.90
 XLEG   =        .00 WLEG   =        .00   ALEG   =        .00
 XINT   =    1000.00
 XMAX   =    1000.00
 NSTEP  = 20

 X-Y-Z COORDINATE SYSTEM:
    ORIGIN is located at the bottom and below the center of the port:
```

TABLE C-3 (Continued)

20.00 m from the LEFT bank/shore.
X-axis points downstream, Y-axis points to left, Z-axis points upward.

BEGIN MOD101: DISCHARGE MODULE

COANDA ATTACHMENT IMMEDIATELY FOLLOWING THE DISCHARGE.

PROFILE DEFINITIONS:
B = Gaussian 1/e (37%) half-width, normal to trajectory
C = centerline concentration
S = corresponding centerline dilution

PREDICTION

X	Y	Z	S	C	B
.00	.00	.00	1.0	.560E+03	.14

END OF MOD101: DISCHARGE MODULE

BEGIN MOD112: WEAKLY DEFLECTED WALL JET IN CROSSLOW

CO-FLOWING OR COUNTER-FLOWING WALL JET

THIS FLOW REGION IS INSIGNIFICANT IN SPATIAL EXTENT AND WILL BE BY-PASSED.

END OF MOD112: WEAKLY DEFLECTED WALL JET IN CROSSLOW

BEGIN MOD117: STRONGLY DEFLECTED WALL JET IN CROSSFLOW

SPECIAL CO-FLOWING CASE: THIS FLOW REGION DOES NOT OCCUR.

END OF MOD117: STRONGLY DEFLECTED WALL JET IN CROSSFLOW

BEGIN MOD133: LAYER BOUNDARY IMPINGEMENT/FULL VERTICAL MIXING

CONTROL VOLUME
PROFILE DEFINITIONS:
BV = layer depth (vertically mixed)
BH = top-hat half-width, in horizontal plane normal to trajectory
C = average (bulk) concentration
S = corresponding average (bulk) dilution

PREDICTION

CONTROL VOLUME INFLOW

X	Y	Z	S	C	B
.00	.00	.00	1.0	.560E+03	.14

CONTROL VOLUME OUTFLOW

X	Y	Z	S	C	BV	BH	ZU	ZL
.60	.00	.60	1.7	.329E+03	.60	.16	.60	.00

END OF MOD133: LAYER BOUNDARY IMPINGEMENT/FULL VERTICAL MIXING

*** END OF HYDRODYNAMIC MIXING ZONE (HMZ) ***

TABLE C-3 (Continued)

BEGIN MOD141: BUOYANT AMBIENT SPREADING

DISCHARGE IS NON-BUOYANT OR WEALY BUOYANT.
THEREFORE BUOYANT SPREADING REGIME IS ABSENT.

END OF MOD141: BUOYANT AMBIENT SPREADING

BEGIN MOD161: PASSIVE AMBIENT MIXING IN UNIFORM AMBIENT

VERTICAL DIFFUSIVITY OF AMBIENT FLOW: EDIFFV = .009697 m**2/s
HORIZONTAL DIFFUSIVITY OF AMBIENT FLOW: EDIFFH = .024242 m**2/s

THE PASSIVE DIFFUSION PLUME IS VERTICALLY FULLY MIXED AT BEGINNING OF REGION.

SIMULATION LIMIT BASED ON MAXIMUM SPECIFIED DISTANCE = 1000.00m.
THIS IS THE REGION OF INTEREST LIMITATION.

PROFILE DEFINITIONS:
BV = Gaussian s.d. (46%) thickness, measured vertically
= or equal to layer depth, if fully mixed
BH = Gaussian s.d. (46%) half-width, measured horizontally in Y-direction
C = centerline concentration
S = corresponding centerline dilution
ZU = upper plume boundary (Z-coordinate)
ZL = lower plume boundary (Z-coordinate)

PREDICTION: STAGE 1: NOT BANK ATTACHED

X	Y	Z	S	C	BV	BH	ZU	ZL
.60	.00	.60	1.7	.329E+03	.60	.16	.60	.00
50.57	.00	.60	22.7	.247E+02	.60	2.20	.60	.00
100.54	.00	.60	32.0	.175E+02	.60	3.11	.60	.00
150.51	.00	.60	39.2	.143E+02	.60	3.81	.60	.00
200.48	.00	.60	45.3	.124E+02	.60	4.39	.60	.00
250.45	.00	.60	50.6	.111E+02	.60	4.91	.60	.00
300.42	.00	.60	55.4	.101E+02	.60	5.38	.60	.00
350.39	.00	.60	59.9	.935E+01	.60	5.81	.60	.00
400.36	.00	.60	64.0	.875E+01	.60	6.21	.60	.00
450.33	.00	.60	67.9	.825E+01	.60	6.59	.60	.00
500.30	.00	.60	71.5	.783E+01	.60	6.94	.60	.00
550.27	.00	.60	75.0	.746E+01	.60	7.28	.60	.00
600.24	.00	.60	78.4	.715E+01	.60	7.60	.60	.00
650.21	.00	.60	81.6	.687E+01	.60	7.92	.60	.00
700.18	.00	.60	84.6	.662E+01	.60	8.21	.60	.00
750.15	.00	.60	87.6	.639E+01	.60	8.50	.60	.00
800.12	.00	.60	90.5	.619E+01	.60	8.78	.60	.00
850.09	.00	.60	93.3	.600E+01	.60	9.05	.60	.00
900.06	.00	.60	96.0	.584E+01	.60	9.31	.60	.00
950.03	.00	.60	98.6	.568E+01	.60	9.57	.60	.00

SIMULATION LIMIT AS SPECIFIED HAS BEEN REACHED; PREDICTION TERMINATES
AT THIS STAGE.

X	Y	Z	S	C	BV	BH	ZU	ZL
1000.00	.00	.60	101.2	.554E+01	.60	9.82	.60	.00

END OF MOD161: PASSIVE AMBIENT MIXING IN UNIFORM AMBIENT

End of Output

CORMIX1: Submerged Single Port Discharges CU Version 1.30 (August 1991)

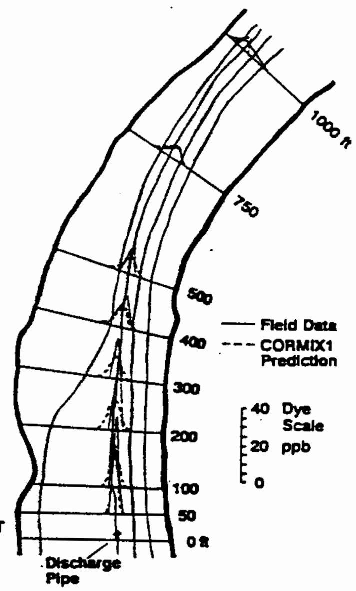


FIGURE C-5 COMPARISON OF DYE FIELD DATA AND CORMIX1 PREDICTIONS WITHIN CUMULATIVE DISCHARGE PATTERN

State water quality regulations call for a demonstration of plume concentrations at the edge of a mixing zone that is limited to one fourth (1/4) of the river width. Assuming an average river width of 250 ft, this corresponds to a width limitation of 250/4 ≈ 62 ft (≈ 19 m). (Note: For the schematic channel width of 50 m this represents a 19/50 = 38 % width limitation as used in CORMIX2.)

Obviously, a number of design solutions, with different diffuser configurations and locations, need to be investigated. One of several feasible solutions is presented in the following: A 15 m (≈ 49 ft) long diffuser consisting of 7 nozzles is installed in a perpendicular, coflowing arrangement and is centered at the 40 % cumulative discharge position. (Note: In the actual coordinate position, this corresponds to a distance of about 70 ft from the right bank; see Figure C-1.) The nozzle diameter is 2.5 in (= 0.0635 m).

The CORMIX2 simulation is summarized in Table C-4 (data sheet), Table C-5 (session record), and Table C-6 (detailed output file). Inspection of the session record and detailed results shows that the plume becomes rapidly mixed over the very shallow water depth. Furthermore, a high initial dilution of 29.8 is attained in a short region (labelled the "acceleration zone", MOD271) following the high velocity multiport discharge. Figure C-6 displays the predicted plume concentrations (expressed in % of the initial concentration) within the river geometry, using the cumulative discharge coordinates. After the rapid initial mixing in the near-field the plume is growing only very slowly in the far-field (MOD261). At the 1000 ft transect, the plume stays clear of the left bank.

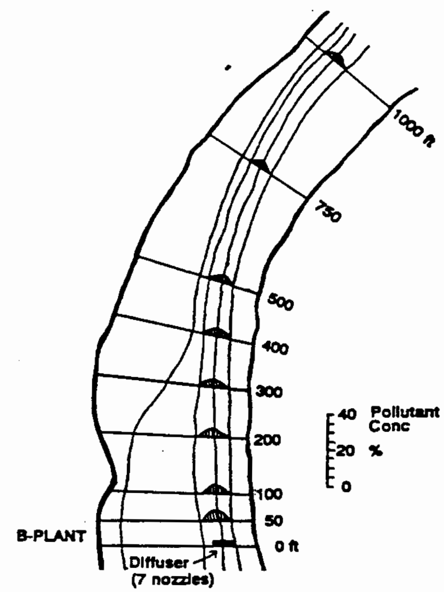


FIGURE C-6 CORMIX2 PLUME PREDICTION FOR PROPOSED MULTIPORT DIFFUSER UNDER 7Q10 LOW FLOW IN SHALLOW RIVER

TABLE C-4 DATA SHEET FOR SUBMERGED MULTIPORT DIFFUSER DISCHARGE IN SHALLOW RIVER

CORMIX2 – Submerged multiport diffuser discharges – CORMIX2

SITE/CASE IDENTIFIER:

Site Name	SHALLOW - RIVER	Prepared by: GHJ
Discharger	B-PLANT	
Pollutant	GENERIC	Date prepared: 10/18/91
Design Case	LOW -FLOW - 7Q10	
DOS FILE NAME	DIFF1	

AMBIENT DATA:

Bounded or unbounded?	BOUNDED				
Channel width	50.0	m			
Channel depth	0.30	m			
Depth at discharge	0.30	m			
Ambient flowrate	8.1	m³/s	or: Ambient velocity	---	m/s
Manning's n	0.03		or: Darcy-Weisbach f	---	

Density data:		Density units:	kg/m³	
Fresh or salt water?	FRESH	Temperature units:	°C	
Density or temp. values?	TEMP			
If uniform:		If stratified:		
Average density/temp.	20.0	Density/temp. at surface	---	
		Density/temp. at bottom	---	
		Stratification type	---	
		(Pycnocline height	---	m)
		(Density/temp. jump	---	)

DISCHARGE DATA:

Nearest bank (left/right)?	RIGHT		Distance to one endpoint	12.5	m
Diffuser length	15.0	m	to other endpt.	27.5	m
Total number of openings	7				
Port diameter	0.0635	m	with contraction ratio	1	
Diffuser arrangement/type	UNIDIRECTIONAL				
Alignment angle (GAMMA)	90	°	Horizontal angle (SIGMA)	0	°
Vertical angle (THETA)	0	°	Rel. orientation (BETA)	90	°
Port height	0.09	m			
Discharge flow rate	0.092	m³/s	or: Discharge velocity	---	m/s
Discharge density	---	kg/m³	or: Discharge temp.	22.0	°C
Concentration units	PERCENT				
Discharge concentration	100				

MIXING ZONE DATA:

Is effluent toxic?	NO		If yes: CMC value	---	
			CCC value	---	
Is there a WQ standard for conventional pollutant?	NO		If yes: value of standard	---	
Any mixing zone specified?	YES		If yes: distance		m
			or width (% or m)	38%	
			or area (% or m²)		
Region of interest	1000	m			
Grid intervals for display	20				

Date of data input into CORMIX2: 10/18/91

TABLE C-5 CORMIX2 SESSION RECORD FOR SHALLOW RIVER

```
XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX
                    CORNELL MIXING ZONE EXPERT SYSTEM
CORMIX2: Submerged Multiport Diffusers       EPA Version 1.20 (October 1991)
  Start of session:       1- 9-1992 16:52:29
**************************************************************************
SITE/CASE DESCRIPTION SUMMARY:
  Site name                = SHALLOW-RIVER
  Effluent discharger      = B-PLANT
  Pollutant                = GENERIC
  DESIGN CASE              = LOW-FLOW-7Q10
  FILE NAME                = DIFF11
**************************************************************************
SUMMARY OF AMBIENT CONDITIONS:
The ambient water body is bounded.
  Channel width                 W       = 50 m
  Average water depth           HA      = .3 m
  Depth at discharge            HD      = .3 m
  Ambient velocity              UA      = 0.540000 m/s
  Ambient flowrate              QA      = 8.1 m^3/s
  Manning's n                   N       = .03
The ambient is unstratified.
  Ambient density               RHOA    = 998.387207 kg/m^3
**************************************************************************
SUMMARY OF DISCHARGE CONDITIONS:
The diffuser type is unidirectional perpendicular.
  Diffuser length               LD      = 15 m
  Nearest bank/shore located on           right.
  Diffuser endpoints            YB1     = 12.5 m;  YB2    = 27.5 m
  Total number of openings      NUMBER  = 7 ports/nozzles
  Average spacing               SPAC    = 2.500000 m
  Actual diameter                       = .0635 m
  Effective diameter            DO      = 0.063500 m (with contraction)
  Port cross-sectional area     AO      = 0.003167 m^2
  Port height                   HO      = .09 m
  Total discharge               QO      = .092 m^3/s
  Discharge velocity            UO      =  4.1499 m/s (with contraction)
The port/nozzle arrangement is unidirectional without fanning.
  Diffuser alignment angle      GAMMA   = 90 deg
  Vertical discharge angle      THETA   = 0 deg
  Horizontal angle of discharge SIGMA   = 0 deg
  Relative orientation angle    BETA    = 90 deg
  Discharge density             RHOO    =  997.95 kg/m^3
  Effluent concentration        CO      = 100
  Concentration units                   = PERCENT
**************************************************************************
SUMMARY OF MIXING ZONE PARAMETERS
The effluent is not toxic as defined by USEPA standards.
No ambient water quality standard has been specified.
Regulatory Mixing Zone (RMZ) is specified by width.
  Mixing zone extent = 19 (m if length or width or m^2 if area).
Region Of Interest (ROI) is specified by downstream distance.
  XINT = 1000 m  from discharge point
Number of intervals for prediction display = 20
**************************************************************************

**************************************************************************
Program Element PARAM:                  Design Case: LOW-FLOW-7Q10
Relative density differences between discharge and ambient:
```

TABLE C-5 (Continued)

```
The effluent density ( 997.95 kg/m^3) is less than the surrounding
  ambient water density at the discharge level (998.387207 kg/m^3).
Therefore, the effluent is positively buoyant and will tend to rise towards
  the surface.

Flow bulk parameters:
  (Bulk variables are defined on a 2-D basis, i.e. per unit diffuser length)
Ambient momentum flux                          ma = 0.087480 m^3/s^2
Discharge volume flux                          q0 = 0.006133 m^2/s
Discharge momentum flux                        m0 = 0.025451 m^3/s^2
  This flux has a net component in the discharge direction (orientation of
  ports/nozzles).
Discharge buoyancy flux                        j0 = 0.000026 m^3/s^3

Flow length scales:
  (Length scales are defined on a 2-D basis, i.e. per unit diffuser length)
Jet crossflow length scale                     lm = 0.087281 m
Discharge length scale                         lq = 0.001478 m
Jet to plume transition length scale           lM = 28.854139 m

Non-dimensional parameters:
Slot densimetric Froude number                 FR0  = 1647.287476
CORMIX2 uses the equivalent two-dimensional slot diffuser concept
  to represent the actual three-dimensional diffuser. Thus, CORMIX2
  simplifies the details of the merging processes of the actual diffuser.
Equivalent slot width                          B0   = 0.001478 (m)
Port/nozzle densimetric Froude number          FRD0 = 251.315765
Jet/Crossflow velocity ratio                   R    = 7.684999
*****************************************************************************

*****************************************************************************
Program Element CLASS:                  Design Case: LOW-FLOW-7Q10

The near field flow configuration will have the following features:

The discharge near-field behavior is dominated by either the positive
  buoyancy of the discharge or the upward vertical orientation of the
  discharge port.

The following conclusion on the flow configuration applies to a layer
  corresponding to the full water depth at the discharge site:
     +------------------------------------------------------+
              Flow class                MU2
              Applicable layer depth    HS = .3 m
     +------------------------------------------------------+

NOTE ON THE APPLICABILITY OF CORMIX2:
The spacing between adjacent nozzles/ports/risers is somewhat greater than
  the discharge water depth.  In fact, the spacing falls into the range
  between three times to ten times the discharge depth.
It is unlikely that sufficient lateral interaction of adjacent jets will
  occur in the near-field. However, the individual jets/plumes may merge
  soon after in the intermediate-field or in the far-field.

CORMIX2 may have limited applicability for this discharge situation.
  The results may be somewhat unrealistic in the near-field (minimum
  dilution may be overpredicted), but appear to be applicable for the
  intermediate- and far-field processes.
```


TABLE C-5 (Continued)

```
The user is advised to proceed with the present CORMIX2 analysis, but
  should use a subsequent CORMIX1 (single port discharge) analysis, using
  discharge data for an individual diffuser jet/plume, in order to compare
  to the near field prediction.

*****************************************************************************

*****************************************************************************
Program Element SUM:                    Design Case: LOW-FLOW-7Q10
-----------------------------------------------------------------------------
X-Y-Z Coordinate system:
  Origin is located at the bottom and diffuser mid-point:
    20 m from the right bank/shore.
-----------------------------------------------------------------------------
HYDRODYNAMIC MIXING ZONE (HMZ) CONDITIONS :
Note: The HMZ is the zone of strong initial mixing.  It has no legal
  implication.  However, this information may be useful for the discharge
  designer because the mixing in the HMZ is usually sensitive to the
  discharge design conditions.
  Pollutant concentration at edge of HMZ =          3.353781 PERCENT
  Dilution at edge of HMZ                =         29.8
  HMZ Location:                        x =       7.50 m
  (centerline coordinates)             y =       0.00 m
                                       z =       0.30 m
  HMZ plume dimensions:       half-width =       8.46 m
                               thickness =       0.30 m

-----------------------------------------------------------------------------
FAR-FIELD MIXING SUMMARY:
  Plume becomes fully vertically mixed at         7.5 m downstream.

************************ TOXIC DILUTION ZONE SUMMARY ************************
No TDZ was specified for this simulation.

********************** REGULATORY MIXING ZONE SUMMARY ***********************
The plume conditions at the boundary of the specified RMZ are as follows:
  Pollutant concentration                =          2.988999 PERCENT
  Corresponding dilution                 =         33.4
  Plume location:                      x =     350.74 m
    (centerline coordinates)           y =       0.00 m
                                       z =       0.30 m
  Plume dimensions:           half-width =       9.50 m
                               thickness =       0.30 m

***************** FINAL MESSAGE ON APPLICABILITY OF CORMIX2 *****************
The diffuser discharge situation computed above is characterized by FAIRLY
  LARGE SPACING between adjacent risers/ports/nozzles, in the range between
  three to ten water depths.
It is unlikely that sufficient lateral interaction of adjacent jets will
  occur in the near-field.  This is somewhat inconsistent with the assumption
  of CORMIX2 of a laterally fully merged diffuser plume.
Thus, CORMIX2 may have limited applicability for this discharge situation.
  The results may be somewhat unrealistic in the near-field (minimum
  dilution may be overpredicted), but should be well applicable for the
  intermediate- and far-field processes.

A subsequent CORMIX1 (single port discharge) analysis is recommended, using
```

TABLE C-5 (Continued)

discharge data for an individual jet/plume, in order to compare to the above near-field results.

```
*****************************************************************************
 DESIGN CASE: LOW-FLOW-7Q10                            FILE NAME: DIFF11
 End of session:    1- 9-1992 17: 3: 9
CORMIX2: Submerged Multiport Diffusers          EPA Version 1.20 (October 1991)
XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX
```


TABLE C-6 OUTPUT FILE FOR DETAILED CORMIX2 PREDICTION FOR SHALLOW RIVER

```
2222222222222222222222222222222222222222222222222222222222222222222222222222

                    CORNELL MIXING ZONE EXPERT SYSTEM
CORMIX2: Submerged Multiport Diffusers        EPA Version 1.20 (October 1991)

-----------------------------------------------------------------------------

CASE DESCRIPTION
 SITE NAME:                  SHALLOW-RIVER
 DISCHARGER NAME:            B-PLANT
 POLLUTANT NAME:             GENERIC

 DESIGN CASE:                LOW-FLOW-7Q10
 DOS FILE NAME:              DIFF11 .CXO
 DATE AND TIME OF FORTRAN SIMULATION:      1- 9-1992_17: 0:33

ENVIRONMENT PARAMETERS (METRIC UNITS)
 BOUNDED SECTION
 BS     =       50.00 AS      =      15.00 QA     =       8.10
 HA     =        0.30 HD      =       0.30
 UA     =       0.540 F       =      0.105

 UNIFORM DENSITY ENVIRONMENT
 RHOA   =     998.39

DIFFUSER DISCHARGE PARAMETERS (METRIC UNITS)
 DITYPE = unidirectional_perpendicular
 BETYPE = unidirectional_without_fanning
 BANK   =  RIGHT      DISTB   =      20.00 YB1    =     -12.50  YB2    =     -27.50
 LD     =       15.00 NOPEN   =    7       SPAC   =       2.50
 DO     =       0.064 AO      =      0.003 HO     =       0.09
 GAMMA  =       90.00 THETAO  =       0.00
 SIGMA  =        0.00 BETA    =      90.00
 (ALPHA =        0.00 DELTA   =       0.00)
 UO     =       4.150 QO      =      0.092 (QO    =0.9200E-01)
 RHOO   =      997.95 DRHOO   =       0.44 GPO    =0.4294E-02
 CO     =0.1000E+03   UNITS   =  PERCENT

FLUX VARIABLES - PER UNIT DIFFUSER LENGTH
 q0     =0.6133E-02   m0      =0.2545E-01  j0     =0.2600E-04  SIGNJO =  1.
 ASSOCIATED 2-D LENGTH SCALES (METERS)
 lq=B   =       0.001 lM      =      28.85 lm     =       0.09
 lmp    =    99999.90 lbp     =   99999.90 la     =   99999.90

FLUX VARIABLES - ENTIRE DIFFUSER
 Q0     =0.9200E-01   M0      =0.3818E+00  J0     =0.3900E-03
 ASSOCIATED 3-D LENGTH SCALES (METERS)
 Lq     =        0.15 LM      =      24.44 Lm     =      1.14 Lb     =       0.00
                                           Lmp    =  99999.90 Lbp    =   99999.90

NON-DIMENSIONAL PARAMETERS
 FRO    =     1647.29 FRDO    =     251.32 R      =       7.68
 (SLOT)               (PORT/NOZZLE)

 FLOW CLASSIFICATION
 ********************************************************************
 *        FLOW CLASS                    =       MU2                 *
```

TABLE C-6 (Continued)

```
*       APPLICABLE LAYER DEPTH HS    =      0.30                   *
*******************************************************************
MIXING ZONE / TOXIC DILUTION / REGION OF INTEREST PARAMETERS
 CO     =0.1000E+03  CUNITS = PERCENT
 NSTD   =  0         CSTD   =0.1000E+07
 LEGMZ  =  1         LEGSPC =  2          LEGVAL =     19.00
 XLEG   =      0.00  WLEG   =      19.00  ALEG   =  -   0.00
 XINT   =   1000.00
 XMAX   =   1000.00
 NSTEP  = 20

X-Y-Z COORDINATE SYSTEM:
 ORIGIN is located at the bottom and the diffuser mid-point:
      -20.00 m  from the RIGHT bank/shore.
   X-axis points downstream, Y-axis points to left, Z-axis points upward.
-------------------------------------------------------------------------
BEGIN MOD201: DISCHARGE MODULE

PROFILE DEFINITIONS:
  BV = Gaussian 1/e (37%) half-width, in vertical plane normal to trajectory
  BH = top-hat half-width, in horizontal plane normal to trajectory
  C  = centerline concentration
  S  = corresponding centerline dilution

PREDICTION
     X         Y      Z        S      C        BV       BH
    0.00      0.00   0.09     1.0 0.100E+03   0.00     7.50

END OF MOD201: DISCHARGE MODULE
-------------------------------------------------------------------------
-------------------------------------------------------------------------
BEGIN MOD271: ACCELERATION ZONE OF UNIDIRECTIONAL CO-FLOWING DIFFUSER

IN THIS ZONE THE DIFFUSER PLUME BECOMES VERTICALLY FULLY MIXED
  OVER THE ENTIRE LAYER DEPTH (HS =    0.30m).
  FULL MIXING IS ACHIEVED AFTER A PLUME DISTANCE OF ABOUT FIVE
  LAYER DEPTHS FROM THE DIFFUSER.

PROFILE DEFINITIONS:
  BV = layer depth (vertically mixed)
  BH = top-hat half-width, in horizontal plane normal to trajectory
  C  = average (bulk) concentration
  S  = corresponding average (bulk) dilution

PREDICTION
     X         Y      Z        S      C        BV       BH
    0.00      0.00   0.30     1.0 0.100E+03   0.30     7.50
    0.37      0.00   0.30    29.8 0.335E+01   0.30     7.39
    0.75      0.00   0.30    29.8 0.335E+01   0.30     7.30
    1.12      0.00   0.30    29.8 0.335E+01   0.30     7.22
    1.50      0.00   0.30    29.8 0.335E+01   0.30     7.15
    1.87      0.00   0.30    29.8 0.335E+01   0.30     7.09
    2.25      0.00   0.30    29.8 0.335E+01   0.30     7.04
    2.62      0.00   0.30    29.8 0.335E+01   0.30     6.99
    3.00      0.00   0.30    29.8 0.335E+01   0.30     6.95
    3.37      0.00   0.30    29.8 0.335E+01   0.30     6.91
    3.75      0.00   0.30    29.8 0.335E+01   0.30     6.87
```


TABLE C-6 (Continued)

```
    4.12      0.00   0.30    29.8 0.335E+01   0.30     6.84
    4.50      0.00   0.30    29.8 0.335E+01   0.30     6.82
    4.87      0.00   0.30    29.8 0.335E+01   0.30     6.80
    5.25      0.00   0.30    29.8 0.335E+01   0.30     6.78
    5.62      0.00   0.30    29.8 0.335E+01   0.30     6.76
    6.00      0.00   0.30    29.8 0.335E+01   0.30     6.75
    6.37      0.00   0.30    29.8 0.335E+01   0.30     6.74
    6.75      0.00   0.30    29.8 0.335E+01   0.30     6.74
    7.12      0.00   0.30    29.8 0.335E+01   0.30     6.74
    7.50      0.00   0.30    29.8 0.335E+01   0.30     6.73

END OF MOD271: ACCELERATION ZONE OF UNIDIRECTIONAL CO-FLOWING DIFFUSER
-------------------------------------------------------------------------
-------------------------------------------------------------------------
BEGIN MOD251: DIFFUSER PLUME IN CO-FLOW

PHASE 1: VERTICALLY MIXED, PHASE 2: RE-STRATIFIED

PHASE 1: THE DIFFUSER PLUME IS VERTICALLY FULLY MIXED OVER THE
         ENTIRE LAYER DEPTH.
THIS FLOW REGION IS INSIGNIFICANT IN SPATIAL EXTENT AND WILL BE BY-PASSED.
END OF PHASE 1
-------------------------------------------------------------------------
PHASE 2: THE FLOW HAS RESTRATIFIED AT THE BEGINNING OF THIS ZONE

THIS FLOW REGION IS INSIGNIFICANT IN SPATIAL EXTENT AND WILL BE BY-PASSED.

END OF MOD251: DIFFUSER PLUME IN CO-FLOW
-------------------------------------------------------------------------
-------------------------------------------------------------------------
*** END OF HYDRODYNAMIC MIXING ZONE (HMZ) ***
-------------------------------------------------------------------------
THE WIDTH PREDICTIONS IN THE FOLLOWING MODULE WILL BE ADJUSTED BY A FACTOR
  OF   1.26 IN ORDER TO SATISFY MASS FLUX CONSERVATION IN THE FAR-FIELD.

-------------------------------------------------------------------------
BEGIN MOD241: BUOYANT AMBIENT SPREADING

DISCHARGE IS NON-BUOYANT OR WEAKLY BUOYANT.
  THEREFORE BUOYANT SPREADING REGIME IS ABSENT.

END OF MOD241: BUOYANT AMBIENT SPREADING
-------------------------------------------------------------------------
-------------------------------------------------------------------------
BEGIN MOD261: PASSIVE AMBIENT MIXING IN UNIFORM AMBIENT

  VERTICAL DIFFUSIVITY OF AMBIENT FLOW:   EDIFFV =    0.003720 m**2/s
  HORIZONTAL DIFFUSIVITY OF AMBIENT FLOW: EDIFFH =    0.009299 m**2/s

SIMULATION LIMIT BASED ON MAXIMUM SPECIFIED DISTANCE =      1000.00 m
  THIS IS THE REGION OF INTEREST LIMITATION.

PROFILE DEFINITIONS:
  BV = Gaussian s.d. (46%) thickness, measured vertically
     = or equal to layer depth, if fully mixed
  BH = Gaussian s.d. (46%) half-width, measured horizontally in y-direction
  C  = centerline concentration
```

TABLE C-6 (Continued)

S = corresponding centerline dilution
ZU = upper plume boundary (Z-coordinate)
ZL = lower plume boundary (Z-coordinate)

PREDICTION: STAGE 1: NOT BANK ATTACHED

X	Y	Z	S	C	BV	BH	ZU	ZL

PLUME INTERACTS WITH SURFACE
THE PASSIVE DIFFUSION PLUME BECOMES VERTICALLY FULLY MIXED WITHIN THIS PREDICTION INTERVAL.

X	Y	Z	S	C	BV	BH	ZU	ZL
7.50	0.00	0.30	29.8	0.335E+01	0.30	8.47	0.30	0.00
57.12	0.00	0.30	30.4	0.329E+01	0.30	8.62	0.30	0.00
106.75	0.00	0.30	30.9	0.323E+01	0.30	8.78	0.30	0.00
156.37	0.00	0.30	31.4	0.318E+01	0.30	8.93	0.30	0.00
206.00	0.00	0.30	32.0	0.313E+01	0.30	9.08	0.30	0.00
255.62	0.00	0.30	32.5	0.308E+01	0.30	9.23	0.30	0.00
305.25	0.00	0.30	33.0	0.303E+01	0.30	9.37	0.30	0.00

** REGULATORY MIXING ZONE BOUNDARY **
IN THIS PREDICTION INTERVAL THE PLUME WIDTH MEETS OR EXCEEDS
THE REGULATORY VALUE = 19.00 m
THIS IS THE EXTENT OF THE REGULATORY MIXING ZONE.

X	Y	Z	S	C	BV	BH	ZU	ZL
354.87	0.00	0.30	33.5	0.299E+01	0.30	9.51	0.30	0.00
404.50	0.00	0.30	34.0	0.294E+01	0.30	9.65	0.30	0.00
454.12	0.00	0.30	34.5	0.290E+01	0.30	9.79	0.30	0.00
503.75	0.00	0.30	35.0	0.286E+01	0.30	9.93	0.30	0.00
553.37	0.00	0.30	35.4	0.282E+01	0.30	10.06	0.30	0.00
603.00	0.00	0.30	35.9	0.279E+01	0.30	10.19	0.30	0.00
652.62	0.00	0.30	36.4	0.275E+01	0.30	10.32	0.30	0.00
702.25	0.00	0.30	36.8	0.272E+01	0.30	10.45	0.30	0.00
751.87	0.00	0.30	37.3	0.268E+01	0.30	10.58	0.30	0.00
801.50	0.00	0.30	37.7	0.265E+01	0.30	10.71	0.30	0.00
851.12	0.00	0.30	38.1	0.262E+01	0.30	10.83	0.30	0.00
900.75	0.00	0.30	38.6	0.259E+01	0.30	10.95	0.30	0.00
950.37	0.00	0.30	39.0	0.256E+01	0.30	11.08	0.30	0.00

SIMULATION LIMIT AS SPECIFIED HAS BEEN REACHED; PREDICTION TERMINATES AT THIS STAGE.

X	Y	Z	S	C	BV	BH	ZU	ZL
1000.00	0.00	0.30	39.4	0.254E+01	0.30	11.20	0.30	0.00

END OF MOD261: PASSIVE AMBIENT MIXING IN UNIFORM AMBIENT

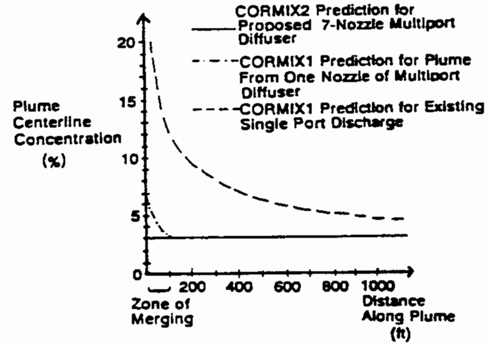
BECAUSE OF THE FAIRLY LARGE SPACING BETWEEN ADJACENT RISERS/NOZZLES/PORTS, THE ABOVE RESULTS MAY BE UNRELABLE IN THE IMMEDIATE NEAR-FIELD OF THE DIFFUSER.
A SUBSEQUENT APPLICATION OF CORMIX1 IS RECOMMENDED TO PROVIDE MORE DETAIL FOR ONE OF THE INDIVIDUAL JETS/PLUMES IN THE INITIAL REGION BEFORE MERGING.

End of Output
CORMIX2: Submerged Multiport Diffusers EPA Version 1.20 (October 1991)

2222222222222222222222222222222222222222222222222222222222222222222222222222

With regards to the regulatory mixing zone (RMZ), the prediction results indicate that it will be encountered at a considerable distance downstream, at about 305 m (≈ 1000+ ft), i.e. outside the region plotted in Figure C-6. The dilution at that location is 33.0, corresponding to a local centerline concentration value of 3.0 %.

Finally, it is illuminating to compare the performance of the proposed multiport diffuser design with the existing single port situation, both under 7Q10 low flow. This is done in Figure C-7 by plotting the plume centerline concentration. (Note: The data sheet, session record, and output file for this CORMIX1 application are omitted for space reasons.) Clearly, the multiport design achieves much more rapid initial mixing by capturing more of the ambient entrainment flow as the diffuser is spread over portion of the river width. Figure C-7 also includes an additional CORMIX1 prediction for a single plume out of the 7-nozzle arrangement to provide more detail in the near-field; the user has been prompted by several messages within CORMIX2 to perform this additional prediction.



Note: The concentrations shown assume that the chemical's ambient receiving water concentration is zero. If the ambient concentration is greater than zero, the in-stream concentrations are calculated by adding the ambient concentration to the diluted discharge concentrations depicted in this figure.

FIGURE C-7 PREDICTED PLUME CENTERLINE CONCENTRATIONS UNDER 7Q10 LOW FLOW IN SHALLOW RIVER: COMPARISON OF SINGLE PORT (CORMIX1) AND MULTIPORT DIFFUSER (CORMIX2) DESIGNS

APPENDIX D

CORMIX3 CASE STUDY: BUOYANT SURFACE DISCHARGE IN AN ESTUARY

Estuarine conditions are characterized by highly variable ambient conditions during the tidal cycle. This case study provides a short application example for a buoyant surface discharge from a large manufacturing plant discharging its process water into an estuary.

D.1 Problem Statement

A manufacturing plant (C-Plant) is discharging process water at a volumetric rate of 2.2 m^3/s (≈ 50 mgd). The process water is essentially fresh water with a discharge temperature of 20.0 °C and it contains copper at a concentration of 80 µg/L.

The plant is located at the shore of an estuary. Figure D-1 shows the bottom bathymetry near the plant location; two transects have been measured and both show a relatively rapid drop-off from the MSL line to a depth of about 5 m below MSL. It is proposed to build a discharge channel with a bottom elevation of about 1.0 m below MSL and a width of 2.0 m. Thus, given the tidal variation indicated in Figure D-1, the actual channel depth will vary from a maximum of about 2.0 m at MHW to a minimum of about 0.25 m at MLW, with corresponding adjustments in the discharge velocity.

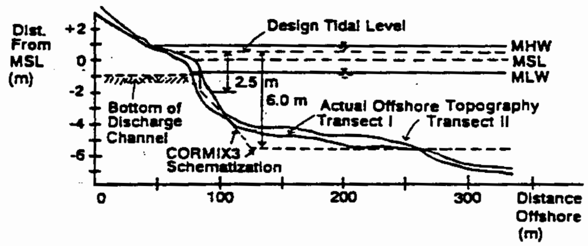


FIGURE D-1 BATHYMETRIC CONDITIONS IN ESTUARY IN VICINITY OF SURFACE DISCHARGE

Figure D-2 shows data from oceanographic field surveys near the discharge site with velocity variations from about +0.7 m/s

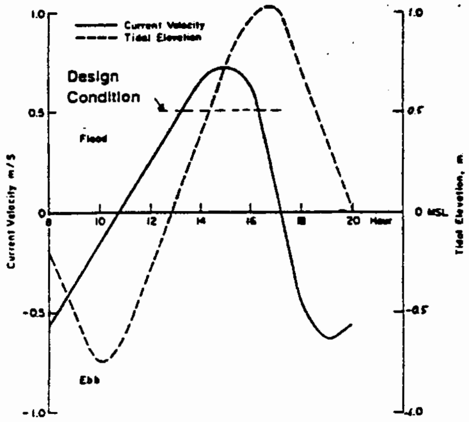


FIGURE D-2 OCEANOGRAPHIC DATA FOR ESTUARINE SITE

for flood tide and to about -0.7 m/s for ebb tide. The estuary contains 20 °C brackish water with mean salinity of about 26 ppt, yielding a density of about 1018 kg/m³ (see Figure 10 as an aid).

State regulations specify a mixing zone of about 300 m extending in any direction from the discharge point. The CMC and CCC values for copper are 25 and 15 µg/L, respectively.

D.2 CORMIX3 Application

It is to be expected that the surface discharge plume for this situation will be quite variable in appearance and mixing characteristics since both ambient velocity conditions change and, perhaps more importantly, the discharge exit velocity will vary greatly depending on tidal stage. Clearly, several applications of CORMIX3 should be performed to assess the behavior at different stages in the tidal cycle. (Note: Some suggestions on how to combine these individual steady-state plume predictions toward a complete description of the actual unsteady plume behavior can be found in the USEPA Technical Guidance Manual (13). A complete theoretical framework for unsteady plume behavior in tidal situations is presently lacking, however.)

The plume condition with low discharge velocity that will prevail during the high flood stage may be considered as most critical in its environmental impact since it may be associated with high concentration levels due to poor mixing and with shoreline contact. For this reason, a CORMIX3 simulation for an average flood tide condition is performed first and presented here.

Figure D-2 shows the design condition with a water level of 0.5 m above MSL and flood velocity of about 0.5 m/s.

As shown in Figure D-1, the ambient water body is schematized as unbounded with an average depth of 6 m for this tidal stage. The conditions in front of the discharge channel are described by a local depth of 2.5 m and a bottom slope of 11 °. All input data have been prepared in the data sheet (Table D-1).

The results of the CORMIX3 session are summarized in the session record (Table D-2) and in the detailed output file (Table D-3); plume conditions are plotted in Figure D-3. The results show that the weak plume, with its low exit velocity and high discharge buoyancy, becomes rapidly deflected by the large tidal current. A deflected jet-like stage (MOD321) is followed by the deflected plume stage (MOD323). During the latter stage, the plume already experiences shoreline interaction as the user is informed in a special message. At about 45 m downstream, the now shoreline-attached plume undergoes a lateral spreading motion where its width increases while the plume becomes thinner with little additional mixing in this stage (MOD341).

The concentration drop-off along the plume centerline is also shown on Figure D-3. The CMC condition is quickly met and satisfies all three applicable TDZ criteria. However, the limited mixing in the far-field does not satisfy the CCC condition within the RMZ defined by a distance of 300 m. In fact, that condition is encountered only at a somewhat larger distance of 457 m as indicated in the Table D-2 summary.

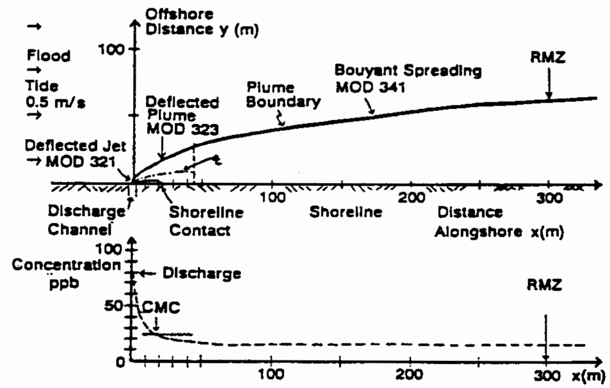


FIGURE D-3 BUOYANT SURFACE DISCHARGE PLUME BEHAVIOR PREDICTED BY CORMIX3 IN ESTUARY

TABLE D-1 DATA SHEET FOR BOUYANT SURFACE DISCHARGE IN ESTUARY

CORMIX3 – Buoyant surface discharges – CORMIX3

SITE/CASE IDENTIFIER:

Site Name	ESTUARY	Prepared by: GHJ
Discharger	C-PLANT	
Pollutant	COPPER	Date prepared: 10/28/91
Design Case	FLOOD-TIDE	
DOS FILE NAME	FLOOD1	

AMBIENT DATA:

Bounded or unbounded?	UNBOUNDED				
Channel width	---	m			
Channel depth	6.0	m			
Ambient flowrate	---	m³/s	or: Ambient velocity	0.5	m/s
Manning's n	---		or: Darcy-Weisbach f	0.025	

Density data:

Fresh or salt water?	SALT				
Density or temp. values?	DENSITY				
Average density	1018.0	kg/m³	or: Average Temperature	---	°C

DISCHARGE DATA:

Discharge located on	~~left~~/right ✓ bank?				
Discharge configuration	FLUSH	(flush, protruding, or coflowing)			
Horizontal angle (SIGMA)	90	°	If protruding:		
Depth at discharge	2.5	m	Distance from bank	---	m
Bottom slope (THETAB)	11.0	°			
If rectangular cross-section:			If circular cross-section:		
Discharge channel width	2.0	m	Outlet pipe diameter	---	m
Discharge channel depth	1.5	m			
Discharge flowrate	2.2	m³/s	or: Discharge velocity	---	m/s
Discharge density	---	kg/m³	or: Discharge temp.	20.0	°C
Concentration units	MUG-P-L				
Discharge concentration	80				

MIXING ZONE DATA:

Is effluent toxic?	YES		If yes: CMC value	25.0	
			CCC value	15.0	
Is there a WQ standard for conventional pollutant?	NO		If yes: value of standard	---	
Any mixing zone specified?	YES		If yes: distance	300	m
			or width (% or m)		
			or area (% or m²)		
Region of interest	1000	m			
Grid intervals for display	10				

Date of data input into CORMIX3: 10/28/91

TABLE D-2 CORMIX3 SESSION RECORD FOR ESTUARY

```
XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX
                      CORNELL MIXING ZONE EXPERT SYSTEM
CORMIX3: Buoyant Surface Discharges               CU Version 1.10  (August 1991)
Start of session:          10-28-1991 15:50:17
******************************************************************************
SITE/CASE DESCRIPTION SUMMARY:
  Site name                   = ESTUARY
  Effluent discharger         = C-PLANT
  Pollutant                   = COPPER
  DESIGN CASE                 = FLOOD-TIDE
  FILE NAME                   = FLOOD1
******************************************************************************
SUMMARY OF AMBIENT CONDITIONS:

The ambient water body is bounded.
   Average water depth                 HA     = 6.0 m
   Ambient velocity                    UA     = 0.5 m/s
   Darcy-Weisbach f                    F      = 0.025
   Ambient density                     RHOA   = 1018.0 kg/m^3
******************************************************************************
SUMMARY OF DISCHARGE CONDITIONS:

The discharge configuration is:  flush discharge.
   Discharge located on          right   bank/shoreline.
   Discharge angle               SIGMA = 90 deg
   Depth at discharge            HD     = 2.5 m
   Bottom slope at discharge     THETAB= 11.0 deg
   Discharge cross-section area  A0     = 3 m^2
   Discharge aspect ratio        AR     = 0.750000
   Discharge flowrate            Q0     = 2.2 m^3/s
   Discharge velocity            U0     = 0.733333 m/s
   Discharge density             RHO0   = 998.387207 kg/m^3
   Effluent concentration        C0     = 80.0 MUG-P-L
******************************************************************************
SUMMARY OF MIXING ZONE PARAMETERS:

The effluent is toxic as defined by USEPA standards.
  The discharge is subject to Toxic Dilution Zone (TDZ) restrictions.
  CMC   = 25.0 MUG-P-L
  CCC   = 15.0 MUG-P-L

Regulatory Mixing Zone (RMZ) is specified by distance.
  Mixing zone extent = 300 (m if length or width or m^2 if area).

Region Of Interest (ROI) is specified by downstream distance.
  ROI = 1000 m  from discharge point

Number of intervals for prediction display = 10

******************************************************************************
Program Element PARAM:                    Design Case: FLOOD-TIDE

The effluent density (998.387207 kg/m^3) is less than the surrounding
  ambient water density (1018.0 kg/m^3).
Therefore, the effluent is positively buoyant.

Flow bulk parameters:
    Discharge volume flux                    Q0    = 2.199999 m^3/s
    Discharge momentum flux                  M0    = 1.613332 m^4/s^2
```

TABLE D-2 (Continued)

```
   Discharge buoyancy flux              J0   = 0.415657 m^4/s^3
   Reduced gravitational acceleration   GP0  = 0.188935 m/s^2

Non-dimensional parameters:
   Velocity ratio                       R    = 1.466666
   Densimetric Froude number            FR0  = 1.281931 (based on LQ)
   Channel densimetric Froude number    FRCH = 1.377525 (based on outlet
                                                         channel depth)

Flow length scales:
   Discharge length scale               LQ   = 1.732051 m
   Jet to crossflow length scale        Lm   = 2.540340 m
   Plume to crossflow length scale      Lb   = 3.325254 m
   Jet to plume transition length scale LM   = 2.220369 m

*****************************************************************************
Program Element CLASS:                  Design Case: FLOOD-TIDE

The near field flow configuration will have the following features:

     The discharge is jet-like.
     The discharge orientation is crossflowing.
     The flow is a free jet, unattached to the shoreline.
     The ambient is deep.

     +------------------------------------------------------+
              Flow class               FJ2
     +------------------------------------------------------+
*****************************************************************************
Program Element SUM:                   Design Case:   FLOOD-TIDE

-----------------------------------------------------------------------------
X-Y Coordinate system:
Origin is located at the water surface in center of discharge channel/outlet:
         0.0 m from the right bank/shore.
     X-axis points in downstream direction
     Y-axis points to the left, as seen by an observer looking downstream

**************** HYDRODYNAMIC CLASSIFICATION AND SIMULATION ****************
Flow Class =       FJ2

----------------------------------------------------------------------------
HYDRODYNAMIC MIXING ZONE (HMZ) CONDITIONS :
Note: The HMZ is the zone of strong initial mixing.  It has no legal
  implication.  However, this information may be useful for the discharge
  designer because the mixing in the HMZ is usually sensitive to the
  discharge design conditions.

Pollutant concentration at edge of HMZ =     18.36 MUG-P-L
Dilution at edge of HMZ                =      4.35

Plume Location:                        x =     45.07 m
  (centerline coordinates)             y =     10.52 m

Plume Dimensions:             half-width =     23.10 m
                              thickness  =       .41 m

----------------------------------------------------------------------------
PLUME BANK CONTACT SUMMARY:
Plume is in contact with the nearest bank throughout simulation.
```


TABLE D-2 (Continued)

```
Plume does not contact the far bank in the simulation.

                              **WARNING**
Shore line interaction occurs in the near-field.  Prediction values may
not be accurate.

*********************** TOXIC DILUTION ZONE SUMMARY ***********************
Recall: The TDZ corresponds to the three (3) criteria issued in the USEPA
  Technical Support Document (TSD) for Water Quality-based Toxics Control,
  1991 (EPA/505/2-90-001).

  Criterion maximum concentration (CMC)  =      25.0 MUG-P-L
  Corresponding dilution                 =       3.20

Plume Location:                          x =     18.13 m
  (centerline coordinates)               y =      6.56 m

Plume Dimensions:           plume half-width =     6.77 m
                            plume thickness  =     1.04 m

  Criterion 1: This location is within 50 times the discharge length scale of
               LQ =       1.7 m.
 ++++++ The discharge length scale test for the TDZ has been SATISFIED. +++++

  Criterion 2: This location is within 5 times the ambient water depth
               HA =       6.00 m.
 ++++++++++ The ambient depth test for the TDZ has been SATISFIED.+++++++++++

  Criterion 3: This location is within one tenth the distance of the extent
               of the regulatory mixing zone of    300.00 m downstream.
 +++++ The regulatory mixing zone test for the TDZ has been SATISFIED. ++++++

 *** All three CMC criteria for the TDZ are SATISFIED for this discharge ***

 The discharge velocity is equal to       .7 m/s.
     This is less than the minimum of 3.0 m/s recommended in the TSD.
 ++++ The discharge velocity recommendation for the TDZ has NOT been met.++++

********************* REGULATORY MIXING ZONE SUMMARY **********************
The plume conditions at the boundary of the specified RMZ are as follows:
  Pollutant concentration                =     15.81 MUG-P-L
  Corresponding dilution                 =      5.06

  Plume location:                        x =    300.00 m
    (centerline coordinates)             y =       .00 m

  Plume dimensions:           half-width =     61.40 m
                               thickness =       .36 m

However, the CCC for the toxic pollutant has not been met within the RMZ.
  In particular:
The CCC was encountered at the following plume position:
  CCC                                    =     15.000000 MUG-P-L
  Corresponding dilution                 =       5.3

  Plume location:                        x =   457.641600 m
    (centerline coordinates)             y =      .000000 m

  Plume dimensions:           half-width =     75.722640 m
```

TABLE D-2 (Continued)

```
                    thickness =       .310520 m

***************************************************************************
  DESIGN CASE:       FLOOD-TIDE
  FILE NAME:         FLOOD1
  End of session:   10-28-1991 15:56:29
CORMIX3: Buoyant Surface Discharges           CU Version 1.10  (August 1991)
XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX
```


TABLE D-3 OUTPUT FILE FOR DETAILED CORMIX3 PREDICTION FOR ESTUARY

```
333333333333333333333333333333333333333333333333333333333333333333333333333333
                    CORNELL MIXING ZONE EXPERT SYSTEM
CORMIX3: Buoyant Surface Discharges                  CU Version 1.10 (August 1991)
------------------------------------------------------------------------------

SIMULATION / CASE DESCRIPTION
 SITE NAME:            ESTUARY
 DISCHARGER NAME:      C-PLANT
 POLLUTANT NAME:       COPPER

 DESIGN CASE:          FLOOD-TIDE
 DOS FILE NAME:        FLOOD1 .CXO
 DATE AND TIME OF FORTRAN SIMULATION:  10-28-1991_15:33:26

ENVIRONMENT PARAMETERS (METRIC UNITS)
 UNBOUNDED SECTION
 HA     =       6.00  HD     =       2.50
 UA     =        .50  F      =        .025
 RHOA   =    1018.00

DISCHARGE PARAMETERS (METRIC UNITS)
 BANK   =         DISTB  =        .00
 RECTANGULAR DISCHARGE
 B0     =      2.000  H0     =      1.500  A0     =      3.000
 SIGMA  = 90.00       THETAB = 11.00
 U0     =       .733  Q0     =      2.200  (Q0    = .2200E+01)
 RHO0   =     998.39  DRHO0  =      19.61  GP0    = .1889E+00
 C0     = .8000E+02   UNITS  = MUG-P-L

FLUX PARAMETERS (METRIC UNITS)
 Q0     = .2200E+01   M0     = .1613E+01   J0     = .4157E+00

ASSOCIATED LENGTH SCALES (METERS)
 LQ     =       1.73  LM     =       2.22  Lm     =       2.54  Lb     =      3.33
 LQ2D   =        .00  LM2D   =        .00  Lm2D   =        .00

NON-DIMENSIONAL PARAMETERS
 FR0    =       1.28  R      =       1.47

FLOW CLASSIFICATION
_********************************************************************
 *        FLOW CLASS                        =          FJ2           *
 ********************************************************************

MIXING ZONE / TOXIC DILUTION / REGION OF INTEREST PARAMETERS
 C0     = .8000E+02   UNITS  = MUG-P-L
 NTOX   =   1         CMC    = .2500E+02   CCC    = CSTD
 XMAX   =    1000.00
 LEGMZ  = 1           LEGSPC = 1           LEGVAL =     300.00
 XLEG   =     300.00  WLEG   =        .00  ALEG   =        .00
 ROI    =    1000.00
 NSTD   =   1         CSTD   = .1500E+02
 NSTEP  = 10

 X-Y COORDINATE SYSTEM:
    ORIGIN is located at the water surface and at the center of the discharge
    channel/outlet:       .00 m  from the  rightbank/shore.
```

TABLE D-3 (Continued)

X-axis points downstream
Y-axis points to left

BEGIN MOD301 : DISCHARGE MODULE

PROFILE DEFINITIONS:
BH = Gaussian 1/e (37%) horizontal half-width, normal to trajectory
BV = Gaussian 1/e (37%) vertical thickness
C = centerline concentration
S = corresponding centerline dilution

PREDICTION

X	Y	BH	BV	C	S
.00	.00	1.38	1.38	.800E+02	1.0

END OF MOD301 : DISCHARGE MODULE

BEGIN MOD311 : WEAKLY DEFLECTED JET (3-D)

SIGMA > 45 DEGREES
SURFACE JET INTO A CROSSFLOW

THIS FLOW REGION IS INSIGNIFICANT IN ITS SPATIAL EXTENT AND WILL BE BY-PASSED.

END OF MOD311 : WEAKLY DEFLECTED JET (3-D)

BEGIN MOD321 : STRONGLY DEFLECTED JET (3-D)

PROFILE DEFINITIONS:
BH = Gaussian 1/e (37%) horizontal half-width, normal to trajectory
BV = Gaussian 1/e (37%) vertical thickness
C = centerline concentration
S = corresponding centerline dilution

PREDICTION

X	Y	BH	BV	C	S
.00	.00	1.62	1.62	.800E+02	1.0
.52	.47	1.77	1.77	.677E+02	1.2
1.05	.88	1.89	1.89	.593E+02	1.3
1.57	1.24	2.00	2.00	.530E+02	1.5
2.09	1.56	2.09	2.09	.482E+02	1.7
2.61	1.86	2.18	2.18	.444E+02	1.8
3.14	2.13	2.26	2.26	.412E+02	1.9
3.66	2.39	2.34	2.34	.386E+02	2.1
4.18	2.63	2.41	2.41	.363E+02	2.2
4.70	2.85	2.48	2.48	.343E+02	2.3
5.23	3.07	2.54	2.54	.326E+02	2.5

END OF MOD321 : STRONGLY DEFLECTED JET (3-D)

BEGIN MOD323 : STRONGLY DEFLECTED PLUME (3-D)

PROFILE DEFINITIONS:
BH = Gaussian 1/e (37%) horizontal half-width, normal to trajectory
BV = Gaussian 1/e (37%) vertical thickness

TABLE D-3 (Continued)

C = centerline concentration
S = corresponding centerline dilution

PREDICTION

X	Y	BH	BV	C	S
5.23	3.07	2.32	2.32	.326E+02	2.5
9.21	4.43	3.55	1.69	.293E+02	2.7
13.19	5.49	4.83	1.34	.272E+02	2.9
17.18	6.38	6.36	1.09	.254E+02	3.2
** CMC HAS BEEN FOUND **					
THE POLLUTANT CONCENTRATION IN THE PLUME FALLS BELOW THE CMC VALUE OF .25E+02 DUE TO MIXING IN THIS INTERVAL.					
18.14	6.57	6.78	1.05	.250E+02	3.2
THIS IS THE EXTENT OF THE TOXIC DILUTION ZONE.					
21.16	7.16	8.12	.91	.239E+02	3.4
25.15	7.84	10.10	.77	.226E+02	3.5
29.13	8.47	12.31	.67	.215E+02	3.7
33.12	9.04	14.72	.58	.206E+02	3.9
37.10	9.57	17.32	.51	.197E+02	4.1
41.09	10.06	20.12	.46	.190E+02	4.2
45.07	10.53	23.11	.41	.184E+02	4.4

END OF MOD323 : STRONGLY DEFLECTED PLUME (3-D)

*** END HYDRODYNAMIC MIXING ZONE (HMZ) ***

BEGIN MOD341 : BUOYANT SPREADING

THE PLUME INTERACTS WITH THE NEAR BANK SOMEWHERE IN THE PREVIOUS MODULE. FOR THIS REASON THE FAR-FIELD PREDICTIONS MAY BE INACCURATE. THE WIDTH AND DEPTH WILL BE RECALCULATED.

BANK ATTACHMENT

PROFILE DEFINITIONS:
BH = top-hat half-width, measured horizontally from bank/shoreline
BV = top-hat thickness,measured vertically
C = average (bulk) concentration
S = corresponding average (bulk) dilution

PREDICTION

X	Y	BH	BV	C	S
45.07	.00	33.63	.57	.184E+02	4.4
140.56	.00	45.07	.46	.171E+02	4.7
236.06	.00	55.20	.39	.162E+02	4.9
** REGULATORY MIXING ZONE BOUNDARY **					
IN THIS PREDICTION INTERVAL THE PLUME DISTANCE MEETS OR EXCEEDS THE REGULATORY VALUE = 300.00 m					
300.00	.00	61.40	.36	.158E+02	5.1
THIS IS THE EXTENT OF THE REGULATORY MIXING ZONE.					
331.55	.00	64.47	.35	.156E+02	5.1
427.04	.00	73.11	.32	.151E+02	5.3
** WATER QUALITY STANDARD OR CCC HAS BEEN FOUND **					
THE POLLUTANT CONCENTRATION IN THE PLUME FALLS BELOW THE WATER QUALITY STANDARD OR CCC VALUE OF .15E+02 IN THE CURRENT PREDICTION INTERVAL.					
457.64	.00	75.72	.31	.150E+02	5.3

THIS IS THE SPATIAL EXTENT OF CONCENTRATIONS EXCEEDING THE WATER QUALITY

TABLE D-3 (Continued)

STANDARD OR CCC VALUE.

522.54	.00	81.27	.29	.147E+02	5.4
618.03	.00	89.03	.27	.144E+02	5.6
713.52	.00	96.48	.26	.141E+02	5.7
809.01	.00	103.64	.24	.139E+02	5.8
904.51	.00	110.57	.23	.136E+02	5.9
1000.00	.00	117.28	.22	.134E+02	6.0

THE REGION OF INTEREST HAS BEEN REACHED.
END OF SIMULATION.

END OF MOD341 : BUOYANT SPREADING

End of Output

CORMIX3: Buoyant Surface Discharges CU Version 1.10 (August 1991)

333333333333333333333333333333333333333333333333333333333333333333333333333333333

Draft
August 1991

CORMIX3: AN EXPERT SYSTEM FOR THE ANALYSIS AND PREDICTION OF BUOYANT SURFACE DISCHARGES

by

Gilbert R. Jones and Gerhard H. Jirka

Technical Report

DeFrees Hydraulics Laboratory
School of Civil and Environmental Engineering
Cornell University
Ithaca, New York 14853-3501

ABSTRACT

In an era of growing environmental concern, the need to accurately predict the impact of aqueous discharges into our natural waters has become essential. To aid the design, analysis, and prediction of discharges into watercourses, a series of software systems has been developed, coined CORMIX for Cornell Mixing Zone Expert System. The first and second subsystems of CORMIX, CORMIX1 and CORMIX2, deal with submerged single port and multiport discharges, respectively. The objective of this work is to develop a third subsystem, CORMIX3, which is to be used as an engineering tool for the prediction, analysis and design of buoyant discharges at the surface of the receiving water body.

CORMIX3 is designed to give both qualitative and quantitative descriptions of the effluent flow resulting from buoyant surface discharges. Given the myriad of possible discharge and ambient conditions, it is necessary to develop a classification scheme which properly categorizes the flow according to its most significant characteristics. This classification gives the user a qualitative "picture" of the flow. Coupled with the flow classification is a mathematical simulation which provides the quantitative predictions of the dilution, trajectory, width, and depth of the effluent stream.

CORMIX3 is intended to be a user friendly system, which facilitates the input and analysis of a discharge situation by providing ample instructions, explanations of the results, and suggestions for improving the dilution characteristics. However, its greatest advantage over other currently available models developed for surface discharges is its ability to predict a wide variety of complex flow phenomenons, including interaction with boundaries, buoyant upstream intrusion, and recirculation zones.

CONTENTS

List of Tables

List of Figures

List of Symbols

α = diffusivity constant

β = density current constant

$\Delta\rho_o$ = discharge density deficit = $\rho_a - \rho_o$

ΔT = surface water temperature difference

ΔT_o = discharge temperature difference

θ = bottom slope of receiving water body at the discharge

ρ_a = ambient water density

ρ_o = discharge density

σ = discharge angle

σ_s = standard deviation

a_o = discharge area

AR = aspect ratio = h_o/b_o

b, BH_{3xx}, BV_{3xx} = width constant

b_h = horizontal half-width of jet/plume

b_o = width of discharge channel

BS = width of ambient water body

b_v = depth of jet/plume

$b_{v,max}$ = maximum flow depth

C = centerline pollutant concentration

C_D = coefficient of drag

C_n = concentration of non-conservative pollutant

C_o = discharge pollutant concentration

d_o = diameter of the discharge

E_y = transverse diffusivity

E_z = vertical diffusivity

f = Darcy-Weisbach friction factor

Fr_{ch} = channel densimetric Froude number = $u_o/(g_o' h_o)^{1/2}$

Fr_o = discharge densimetric Froude number = $u_o/(g_o' d_o)^{1/2}$

Fr_o' = overall densimetric Froude number = $u_o/(g_o' a_o^{1/2})^{1/2}$

g = gravitational acceleration

g_o' = initial reduced gravitational acceleration = $g(\rho_a - \rho_o)/\rho_a$

H = local ambient water depth

H_a = average ambient water depth

H_D = ambient water depth at the discharge

HMZ = hydrodynamic mixing zone

h_o = depth of flow in discharge channel

h_s = depth of flow at head of upstream intrusion

j_o = discharge buoyancy flux per unit depth = J_o/H

J_o = discharge buoyancy flux = $g_o' Q_o$

J_o^* = alternate discharge buoyancy flux = $J_o/2^{1/2}$

K_r = reaction constant

L_b = plume-to-crossflow length scale = J_o/u_a^3

L_b^* = alternate plume-to-crossflow length scale = $L_b/2^{1/2}$

L_I = intrusion length scale

l_m = 2-dimensional jet-to-crossflow length scale = m_o/u_a^2

l_M = 2-dimensional jet-to-plume length scale = $m_o/j_o^{2/3}$

L_m = jet-to-crossflow length scale = $M_o^{1/2}/u_a$

L_m^* = alternate jet-to-crossflow length scale = $L_m/\pi^{1/2}$

L_M = jet-to-plume length scale = $M_o^{3/4}/J_o^{1/2}$

l_q = 2-dimensional discharge length scale = q_o^2/m_o

L_Q = discharge length scale = $Q_o/M_o^{1/2}$

m' = line impulse momentum per unit depth = M'/H

M' = line impulse momentum

M_e = excess momentum flux

m_o = discharge momentum flux per unit depth = M_o/H

M_o = discharge momentum flux = $u_o Q_o$

M_o^* = alternate discharge momentum flux = $M_o/\pi^{1/2}$

q_e = entrained fluid at head of density current

q_o = discharge volume flux per unit depth = Q_o/H

Q_o = discharge volume flux = $u_o a_o$

R = velocity ratio = u_o/u_a

s = distance along flow trajectory

s, S_{3xx} = dilution constant

S = dilution

t, T_{3xx} = trajectory constant

TC_{3xx} = transition rule constant

TR = transition rule

u = flow velocity

u_* = friction velocity

u_a = ambient velocity

u_c = centerline flow velocity

u_o = discharge velocity

v_B = velocity of density current front

w_e = net velocity of fluid across density current front

x = horizontal downstream coordinate in global coordinate system

x', y', z' = local (primed) coordinate system

x_R = length of recirculation region

x_s = length of upstream intrusion

x_v, y_v = coordinates of virtual origin

y = horizontal coordinate in global coordinate system perpendicular to ambient crossflow

y_{max} = distance to maximum depth of flow

y_o = distance discharge channel protrudes into ambient water

z = vertical coordinate in global coordinate system

Acknowledgments

This study was conducted at the DeFrees Hydraulics Laboratory, Cornell University, in cooperation with the United States Environmental Protection Agency. The authors would like to extend their appreciation to ____ for their support for the project. The authors acknowledge the assistance of Dr. Robert Doneker, Assistant Professor of Civil Engineering at the University of Portland, and Paul Akar, Graduate Research Assistant, in the development of the expert system. The work was carried out using the facilities of the DeFrees Hydraulics Laboratory. Cameron Wilkens, Electronics Technician, generously assisted with solutions for computer hardware and software problems. Figures in this report were prepared by Ali Avcisoy, ___(title)___.

This report was submitted with essentially similar contents by Gilbert R. Jones, Associate Engineer with ENVIRON International Corporation, to the Graduate School of Cornell University in partial fulfillment of requirements for the degree of Master of Science. Dr. Gerhard Jirka, Professor of Civil and Environmental Engineering, was project supervisor.

Chapter I

Introduction and Historical Background

1.1. Introduction

In an era of growing environmental concern, accurately predicting the impact of aqueous discharges into our natural riverine, estuarine and coastal waters has become essential. Discharges issued at or near the water surface are subject to particular attention because of their conspicuous nature. One common surface discharge situation which prompted extensive research is the disposal of cooling water from power plants into natural watercourses via open channels. More recently, combined sewer overflows have become a major surface discharge concern. Naturally occurring surface discharges also exist, such as the flow of fresh water from rivers into saline coastal environments. These scenarios in and of themselves implicate a need for accurate methods for the prediction of major flow characteristics, such as the trajectory, dilution and geometry of the effluent stream.

In a majority of surface discharge situations, the effluent is positively buoyant with respect to the receiving water. For this reason, this study is limited to positively or neutrally buoyant discharges issued at or near the surface of the ambient water body. These discharges are subsequently referred to as buoyant surface discharges or buoyant surface jets as is common in the field of hydraulics.

All aqueous discharges in the United States are subject to federal, state, and local regulation. An important aspect of this regulation is based on the concept of a mixing zone. A mixing zone is a specified area allocated for the initial mixing of a pollutant discharge. At the edge of the legal mixing zone, standards must be met. In addition to legal mixing zone specifications, strict pollutant level are set for areas that are sensitive to any pollutant interference. For example, shellfish beds require that the ambient water body remains pristine to assure healthy catches. Regulations of this type require reliable predictive techniques.

The intent of this work is to create a robust modelling software system for the prediction and analysis of surface buoyant discharges. It involves the development of an expert system capable of handling the most common discharge situations as well as some uncommon scenarios. This expert system, CORMIX3, is intended for use by practicing engineers, regulators, and the scientific community.

The motivation behind designing an expert system to predict and analyze buoyant surface discharges is discussed in the following section, Section 1.1.1. Section 1.1.2 describes the scope, objectives and results of the CORMIX3 Expert System (Section 1.1.2).

The remainder of the chapter is devoted to the history of the modelling technique employed in this study. Included in this discussion is a general description of the available modelling techniques, the historical development of the length scale relationships used for submerged round buoyant jets, and the extension of these relationships to surface buoyant jet problems.

1.1.1. Motivation for the Development of an Expert System

Given the wide variety of possible discharge and ambient conditions, a spectrum of diverse flow patterns can result each with distinctly different mixing characteristics. Yet with continually stricter regulations and increasing public awareness of environmental pollution problems, there is an growing interest in improving discharge designs to maximize initial dilution of discharged effluent.

Up until now, there have been few guidelines under which a practicing engineer could predict the general characteristics of a buoyant surface discharges thereby making difficult the task of selecting an appropriate modelling technique. Often the limitations and restrictions of some models are overlooked, and these models are subsequently applied to situations for which they are not applicable.

Another difficulty the practicing engineer faces in using most currently available models is the considerable task of assembling the data base. Most models require a high degree of familiarity with the input requirements and format, making the use of such models by the unexperienced analyst formidable.

In contrast, an expert system is intended to facilitate the input of the required information and should assure the use of the correct modelling technique for the situation at hand. An expert system is a computer package that is intended to mimic the decision making process of a trained "expert" by using information stored in a "knowledge base", which is a set of predetermined rules and facts, to properly characterize the situation. Once the system has determined the important factors and parameters of the problem according to the knowledge base, it invokes the appropriate modelling techniques to accurately simulate the scenario.

Expert systems promise significant advantages over the pick-and-choose method of model selection. The following advantages of the expert system approach have been noted by Doneker and Jirka (1990):

- "[An expert system] assures the proper choice of model for a given situation.
- It assures the chosen model is applied methodically without skipping essential elements.

- It guides the acquisition or estimation of data for model prediction.

- It allows a flexible application of design strategies for a given point source, screening of alternatives, and if necessary, switching to predictive models thus avoiding rigid adherence to a single model.

- It flags borderline cases for which no predictive model exists suggesting either avoidance of such designs or caution by assigning a degree of uncertainty.

- It allows a continuous update of the knowledge base as improved predictive models, experimental data, and field experience with particular designs become available.

- It provides a documented analysis listing the knowledge and decision logic that have lead to the problem solution. Thus, unlike conventional programs or computer algorithms, an expert system is not a "black box."

- It provides a common framework whereby both regulators (federal or state), applicants, and the scientific community can arrive at a consensus on the state-of-the-art hydrodynamic mixing and pollution control.

- It gives pollutant concentration at the specified regulatory mixing zones.

- Finally, and perhaps most importantly, it provides a teaching environment whereby the initially inexperienced analyst, through repeated interactive use, gains physical insight and understanding about initial mixing processes."

1.1.2. CORMIX3 - An Expert System for the Analysis and Prediction of Buoyant Surface Discharges

The following section provides the scope and objectives of the present study and a summary of this work.

1.1.2.1. Scope and Objectives

It is the purpose of this study to create an expert system which utilizes a flow classification scheme that assures the use of applicable modelling techniques for the variety of possible flow patterns that may result. The system, dubbed CORMIX3 for Cornell Mixing Zone Expert System - Subsystem 3, is meant to be an engineering tool

which may be used to analyze a majority of positively and neutrally buoyant surface discharges.

CORMIX3 is the third of a series of subsystems which make up the CORMIX Expert System. The first subsystem, CORMIX1, was developed for the prediction and analysis of submerged single port discharges (see Doneker and Jirka, 1990). The second subsystem, CORMIX2, was subsequently developed for multiport diffusers (see Akar, 1990).

As an expert system, CORMIX3 is designed to be a user-friendly program intended to guide the user through the analysis of a particular discharge configuration. To facilitate its use ample instructions should be provided, suggestions for improving dilution characteristics are to be included, and warning messages need to be displayed when undesirable or uncommon flow conditions occur. In addition, a qualitative description of the flow is to be included to give the user a "visual picture" of the flow. Finally, the system should provide accurate predictions of the trajectory, dilution and geometry of the flow.

1.1.2.2. Summary of Present Study

The structure of CORMIX3 is similar to that of CORMIX1 and CORMIX2. It utilizes the expert system shell VP-Expert (Paperback Software, Inc.) to collect the data, parameterize the problem, and classify the discharge situation. Fortran is used to mathematically simulate the flow.

Chapter II discusses the theoretical development of the relationships used in the hydrodynamic simulation of the flow. It also includes a general description of the four flow categories which CORMIX3 distinguishes.

Chapter III provides a description of the structure of CORMIX3 and discusses the two programming languages, VP-Expert and Fortran, used to construct the different segments of the system. Included in this Chapter is a discussion on the five program elements that make up CORMIX3.

Chapter IV details the procedure used to classify the flow. Also included in this Chapter is a discussion of the required input data and the simplified schematizations of the ambient and discharge conditions.

Chapter V describes the structure of the hydrodynamic simulation and lists the expressions used to predict the dilution, trajectory, width and depth of the effluent flow.

Chapter VI is devoted to evaluation and verification of the model. Included are several comparisons between CORMIX3 predictions and corresponding laboratory and field studies. These comparisons are intended to provide verification of the model and to illustrate the flexibility of CORMIX3 to predict a wide variety of flow situations.

Chapter VII illustrates the application of CORMIX3 to a hypothetical scenario. Also included in Chapter VII is a discussion of the limitations of CORMIX3 and some additional comments for the advanced user.

1.2. History of Length-scale Models

This section provides a history of the mathematical technique used in this study to predict the important quantitative characteristics of buoyant surface jets. It is divided into three subsections. Section 1.2.1 explains the three basic groups of mathematical models for buoyant surface jet analysis. Section 1.2.2 reviews the history of the simple analytical expressions used to describe submerged round buoyant jets. Section 1.2.3 discusses various attempts to apply these simple analytical expressions to buoyant surface jets and suggests how models based on similar analyses may be developed for buoyant surface jet problems.

1.2.1. Description of Three General Model Types

The several types of mathematical models which have been developed for submerged round buoyant jets may be grouped into the following categories: jet-integral models, three-dimensional numerical models, and "length-scale" models. Jet-integral models are described by Jirka et al. (1981) as follows:

> "Jet-integral models consist of a set of ordinary differential equations derived from the cross-sectional (normal to the jet trajectory) integration of jet-properties such as mass, momentum, and buoyancy fluxes. Empirical formulations for internal jet behavior such as buoyant damping of turbulence and cross-sectional distortion (lateral spreading) are included. The equation systems are parabolic and are solved by simple forward-marching numerical schemes along the jet trajectory."

Jet-integral models perform satisfactorily for simple flows with no shoreline interaction or attachment. However, strong crosscurrents or limited depths causing attachment with the downstream bank or strong initial buoyancy causing intrusion of the effluent along the upstream bank render these models invalid. In addition, jet-integral models predict only the jet-like behavior of the flow near the source. They are incapable of simulating any far-field processes that occur after a certain transition distance (Jirka et al., 1981).

Three dimensional numerical models attempt to approximate the system of Reynolds equations through finite element or finite difference schemes. To a large

extent, these methods have been unsuccessful. Jirka et al. (1975) summarizes the difficulties with numerical models as follows:

> "The major problem seems to be the specification of boundary conditions, in addition, the formulation of turbulent transport terms is unknown... . From a practical viewpoint the models are highly complicated, difficult to check, have no instructions for the user and are expensive to use (even in moderate size test cases)."

The third class of modeling techniques, called herein "length-scale models," provides the basic methodology utilized in this study. Surface discharge flows can be divided up into different regimes each dominated by particular flow properties such as the initial momentum, the buoyancy flux, or the ambient crossflow. Within each regime, the flow may be approximated with simple asymptotic relationships derived from basic equations describing the simplified problem for which only the most significant properties are accounted for, and then adding perturbation terms to account for lesser effects. The models that use these asymptotic solutions are referred to as "length-scale" models because of the use of specific length scales to delineate the extent of the regimes for which these analytical expressions are valid.

The following subsections discuss the historical development of the asymptotic solutions used to describe the flow regimes of buoyant flows. Initially, analytical expressions and length scales were developed for submerged round buoyant jets. However, they have subsequently been extended to buoyant surface jet analysis.

1.2.2. Review of Submerged Round Buoyant Jet Expressions

Most of the earlier work leading up to our present length-scale models originated from studies of smokestack plumes which grew out of the increasing need for air pollution control. Early studies, such as those by Schmidt (1941), Yih (1951), Rouse, Yih, and Humphreys (1952), Railston (1954), Priestly and Ball (1955), Morton, Taylor, and Turner (1956), and Slawson and Csanady (1967), focused on pure plumes for which the initial momentum of the discharge can be considered negligible. Much of this work pointed to a height-width relationship of $b \propto z$ and a trajectory relationship of plumes in a crossflow of $z \propto x^{2/3}$, where b is the plume half-width, z is the vertical height of the plume centerline, and x is the horizontal coordinate downwind.

Scorer (1958, 1959) recognized the potential importance of an initial momentum regime. Scorer (1959) concluded that a buoyant jet has three regimes dominated by momentum, buoyancy, and passive advection successively. Using simple physical arguments and dimensional analysis he determined that the trajectory relationships for a buoyant jet were $z \propto x^{1/3}$ for the momentum dominated regime and $z \propto x^{2/3}$ for the buoyancy dominated regime. Similarly, he concluded that $b \propto z$ for both of these near-field regimes. Scorer's idea of the two distinguishable regimes in the near-field received little attention for nearly a decade.

Csanady (1961) was the first to introduce length scales into buoyant jet analysis. He used a plume-to-crossflow length scale, $L_b^* = J_o^*/u_a^3$, where $J_o^* = u_o(d_o/2)^2 g_o'$ and u_o is the effluent exit velocity, d_o is the diameter of the discharge outlet, and g_o' is the reduced gravitational acceleration. Note, this definition of a plume-to-crossflow length scale differs from the definition used in the remainder this study by a factor of $2^{1/2}$, i.e.: $L_b = 2^{1/2}L_b^*$. Previously, Morton (1959) had non-dimensionalized his results with the flux ratio $M_o^{3/4}/J_o^{1/2}$, where M_o is the initial momentum flux and J_o is the initial buoyancy flux. This term would later be dubbed a momentum-buoyancy length scale, however Morton never recognized it as a significant independent term. Csanady reasoned through dimensional analysis that the trajectory of a buoyant jet had the functional form of:

$$\frac{z}{L_b} = f\left(\frac{x}{L_b}, Fr_o, \frac{d_o}{L_b}\right) \qquad (1.1)$$

where Fr_o is the discharge Froude number defined as:

$$Fr_o = \frac{u_o}{\sqrt{g' d_o}} \qquad (1.2)$$

At sufficiently large values of z/L_b, the influence of Fr_o and d_o/L_b becomes negligible, resulting in the following simplified relationship:

$$\frac{z}{L_b} = f\left(\frac{x}{L_b}\right) \qquad (1.3)$$

Csanady also noted that this was an asymptotic formula and that "the effective origin of z may have to be moved to allow for a 'momentum rise'." He compared Eqn. 1.3 with the asymptotic forms of the equations derived by Priestley (1956) and Sutton (1953) which suggested that $(z/L_b) \propto (x/L_b)^{3/4}$ and $(z/L_b) \propto (x/L_b)^{2/3}$ respectively and found both fit observed results reasonably well considering the scatter of data.

Csanady's study was followed up by both Briggs (1965) and Moore (1966). Briggs used a plume-to-crossflow length scale in equations obtained through dimensional analysis to describe pure plumes in a crossflow. He concluded the rise due to buoyancy without the effect of ambient stratification was governed by the relationship:

$$\frac{z}{L_b} = 2.0\left(\frac{x}{L_b}\right)^{\frac{2}{3}} \qquad (1.4)$$

He determined the terminal height of the plume was also a function of the plume-to-crossflow length scale. In addition, Briggs gave relationships that were determined in the same manner for stratified ambient conditions which were not of the form of Eqn. 1.3.

Moore, on the other hand, used a plume-to-crossflow length scale in relationships developed from the conservation equations. He also obtained trajectory results in the form of Eqn. 1.3 for continuous plumes.

Hoult, Fay, and Forney (1969) were the first to use a jet-to-crossflow length scale and to develop simple formulae for the momentum dominated regime. They used the jet-to-crossflow length scale, $L_m^* = (d_o/2)(u_o/u_a)$ where u_a is the ambient flow velocity, and the plume-to-crossflow length scale, L_b, in asymptotic relationships developed from conservation and entrainment equations. Note that this definition of the jet-to-crossflow length scale differs from the jet-to-crossflow length scale used in the remainder of this study by a factor of $\pi^{1/2}$, i.e.: $L_m = \pi^{1/2}L_m^*$. They recognized three near-field flow regimes of a buoyant jet, two which were governed by the relationships:

$$\frac{z}{L_m} = K_1\left(\frac{x}{L_m}\right)^{\frac{1}{2}} \quad ; \quad z \ll L_m \qquad (1.5)$$

$$\frac{z}{L_b} = K_2\left(\frac{x}{L_b}\right)^{\frac{2}{3}} \quad ; \quad z \gg L_b \,\&\, L_m \qquad (1.6)$$

They contested that "no simple formula exists" for the intermediate regime where $L_m < z < L_b$. The constants K_1 and K_2 are dependent on the entrainment coefficients which were determined experimentally. Subsequent work by Fay (1973) attempted to extrapolate Hoult, Fay, and Forney's analysis to stratified ambient environments and to motor vehicles and aircraft wakes.

Chu and Goldberg (1974) attempted to link the momentum dominated and buoyancy dominated regimes into one relationship by using an alternate entrainment hypothesis, claiming that "the transition between the two regimes cannot be derived from dimensional analysis." Their proposed relationships for the transition were:
where S is the dilution along the plume centerline, defined as C/C_o; C being the centerline pollutant concentration anywhere along the trajectory and C_o the initial pollutant concentration at the discharge. Note that the trajectory relationship reduces to

$$\frac{z}{L_b} = C_1 \left\{ \frac{1}{2} \left(\frac{x}{L_b} \right)^2 + \left(\frac{L_m}{L_b} \right)^2 \left(\frac{x}{L_b} \right) \right\}^{\frac{1}{3}} \quad (1.7)$$

$$\frac{b}{L_b} = C_2 \left\{ \frac{1}{2} \left(\frac{x}{L_b} \right)^2 + \left(\frac{L_m}{L_b} \right)^2 \left(\frac{x}{L_b} \right) \right\}^{\frac{1}{3}} \quad (1.8)$$

$$S = C_3 \left(\frac{u_o}{u_a} \right) \left\{ \frac{1}{2} \left(\frac{L_b}{L_m} \right) \left(\frac{x}{L_m} \right)^2 + \left(\frac{x}{L_m} \right) \right\}^{\frac{1}{3}} \quad (1.9)$$

Eqns. 1.5 and 1.6 for their respective regimes of dominance. Chu and Goldberg were able to show a decent fit to experimental data with these relationships.

An extension of Hoult, Fay, and Forney's work was carried out by Hoult and Weil (1972). Using the same method as Hoult, Fay, and Forney but a different entrainment hypothesis, they were able to determine the following relationships for the three different regimes:

weakly deflected jet
$$\frac{z}{L_m} = K_1 \left(\frac{x}{L_m} \right)^{\frac{1}{2}} \quad ; \quad z << L_m \quad (1.10)$$

strongly deflected plume
$$\frac{z}{L_b} = K_2 \left(\frac{x}{L_b} \right)^{\frac{2}{3}} \quad ; \quad x >> L_m \ \& \ L_b \quad (1.11)$$

strongly deflected jet
$$\frac{z}{L_b} = K_3 \left(\frac{x}{L_m} \right)^{\frac{1}{3}} \quad ; \quad z >> L_m \quad (1.12)$$

K_1, K_2, and K_3 are constants which were determined experimentally. Hoult and Weil obtained the relationship for the strongly deflected jet region by taking the asymptotic result for a non-buoyant discharge for large values of x/L_b. Note Hoult and Weil assume that $L_m < L_b$, thereby neglecting the possibility of a weakly deflected plume regime that might exist in the intermediate region.

Hoult and Weil also determined transition criteria for the different regimes. For a pure jet, they concluded that the transition criteria from the weakly deflected jet region to the strongly deflected jet region occurs at $x \propto L_m$, the transition from the weakly deflected jet region to the strongly deflected plume region occurs at $x \propto L_m^3/L_b^2$, and if the intermediate strongly deflected jet regime exists, then the transition from the weakly deflected jet region to the strongly deflected jet region occurs at $x \propto L_m^2/L_b$.

Wright (1977) generalized Hoult and Weil's length-scale model using dimensional analysis in a comprehensive study on vertical buoyant jets in a crossflow. Wright used four length scales: the jet-to-crossflow length scale, L_m, the plume-to-crossflow length scale, L_b, a discharge length scale, $L_Q = Q_o/M_o^{1/2}$, and a jet-to-plume length scale, $L_M = M_o^{3/4}/J_o^{1/2}$. Wright proposed that any jet property, ϕ, can be described as a function of three length scale ratios:

$$\phi = f\left(\frac{z}{L_m}, \frac{L_b}{L_m}, \frac{L_Q}{L_m} \right) \quad (1.13)$$

L_M is not present in Eqn. 1.13 since it is a combination of L_m and L_b such that $L_M = (L_m^3/L_b)^{1/2}$. Using simple physical arguments and dimensional considerations he obtained solutions for the different flow regimes for a vertical buoyant jet in a crossflow as shown in Table 1.1. The dilution equations were obtained using the assumption of similarity between velocity and concentration profiles, a condition that has proven accurate in many previous studies for all but the strongly deflected plume regime. He supported the use of these asymptotic equations and found values for the constants C_1 - C_8 by comparing them with both his own data and experiments run by Pan (1967). Table 1.2 shows the sequence of regimes determined by the length-scale ratio L_m/L_b.

Since the publication of Wright's paper, investigators have attempted to extend these simple formulae and length scales to other buoyant jet scenarios. One such study conducted by Bühler and Hauenstein (1979) extended Wrights analysis to buoyant jets discharged horizontally perpendicular to the crossflow. Making certain analogies to Wright's trajectory relationships and using simple physical arguments they obtained the relationships for 3-dimensional trajectories and dilutions. The results of their study revealed that the horizontal trajectory (i.e.: the trajectory seen in a plan view) is a function of one parameter, L_m/L_b, and the independent variable, x/L_m:

$$\frac{y}{L_M} = f\left(\frac{x}{L_M}, \frac{L_m}{L_b} \right) \quad (1.14)$$

Bühler and Hauenstein also used a similar analysis to describe buoyant jets discharged into stagnant ambient environments.

More recent work by Knudsen (1988) extended Wright's (1977) work to include horizontal jets in a still ambient, co-flow, crossflow, and counterflow. However,

Table 1.1

Wright's (1977) Trajectory and Dilution Relationships for a Submerged Buoyant Jet Discharged Vertically in a Crossflow

Flow Regime	Trajectory Relationship	Dilution Relationship
weakly deflected jet	$\frac{z}{L_m} = C_1\left(\frac{x}{L_m}\right)^{\frac{1}{2}}$	$S\frac{Q_e x_e}{M_e} = C_5\left(\frac{x}{L_m}\right)$
strongly deflected jet	$\frac{z}{L_m} = C_2\left(\frac{x}{L_m}\right)^{\frac{1}{3}}$	$S\frac{Q_e x_e}{M_e} = C_6\left(\frac{x}{L_m}\right)^{2}$
weakly deflected plume	$\frac{z}{L_b} = C_3\left(\frac{x}{L_b}\right)^{\frac{3}{4}}$	$S\frac{J_e}{x_e^{5}} = C_7\left(\frac{x}{L_b}\right)^{[illegible]}$
strongly deflected plume	$\frac{z}{L_b} = C_4\left(\frac{x}{L_b}\right)^{\frac{2}{3}}$	$S\frac{J_e}{x_e^{5}} = C_8\left(\frac{x}{L_b}\right)^{-4}$

Table 1.2

Sequence of Flow Regimes as Defined by Wright (1977) for Submerged Buoyant Jets

$\frac{L_m}{L_b} >> 1$	momentum dominated:	weakly deflected jet → strongly deflected jet → strongly deflected plume
$\frac{L_m}{L_b} << 1$	buoyancy dominated:	weakly deflected jet → weakly deflected plume → strongly deflected plume
$\frac{L_m}{L_b} = 1$	small middle region:	weakly deflected jet → strongly deflected plume

Table 1.3

Buhler and Hauenstein's (1979) Trajectory Relationships

Flow Regime	y-Trajectory Relationships	z-Trajectory Relationships
weakly deflected jet	$\frac{y}{L_m} = C_9\left(\frac{x}{L_m}\right)^{\frac{1}{2}}$	$\frac{z}{L_m} = C_{13}\left(\frac{L_m^{1/2}}{L_M^{2/3}}x^{3/2}\right)^{1/2}$
strongly deflected jet	$\frac{y}{L_m} = C_{10}\left(\frac{x}{L_m}\right)^{\frac{1}{3}}$	$\frac{z}{L_m} = C_{14}\left(\frac{L_b^{5/9}}{L_M^{2/3}}x^{3/2}\right)^{1/2}$
weakly deflected plume	$\frac{y}{L_M} = C_{11}$	$\frac{z}{L_b} = C_{15}\left(\frac{x}{L_b}\right)^{3/4}$
strongly deflected plume	$\frac{y}{L_M} = C_{12}$	$\frac{z}{L_b} = C_{16}\left(\frac{x}{L_b}\right)^{2/3}$

Table 1.4

Buhler and Hauenstein's (1979)
Dilution Relationships

Flow Regime	Dilution Relationship
weakly deflected jet	$S = C_{17}\left(\frac{L_M^2}{L_Q^3} z\right)^{1/3}$
strongly deflected jet	$S = C_{18}\left(\frac{L_M^{4/3}}{L_Q^2 L_b^{1/3}} z\right)^{1/2}$
weakly deflected plume	$S = C_{19}\left(\frac{1}{L_Q^3 L_M^2} z^5\right)^{1/3}$
strongly deflected plume	$S = C_{20}\left(\frac{1}{L_Q L_m} z^2\right)$

Knudsen used an excess momentum flux, $M_e = (\pi/4)d_o^2u_o(\underline{u}_o-\underline{u}_a)$, where $\underline{u}_o$ and $\underline{u}_a$ are vector quantities. The use of the excess momentum flux instead of the absolute momentum flux was proposed earlier by List and Imberger (1973).

The use of these length-scale models have proven successful in predicting the bulk characteristics of smokestack plumes, sewer outfalls, and similar round buoyant jet scenarios. Fischer et al. (1979) gives a comprehensive review of turbulent jets and plumes and the application of length-scale analysis to these problems. Length-scale analysis have since been extended to submerged discharges in stratified environments (Doneker and Jirka, 1990), multiport diffuser problems (Akar and Jirka, 1990), and buoyant surface jets (Chu and Jirka, 1986). The following subsection discusses the use of length-scale models for buoyant surface jets.

1.2.3. Applying Length-scale Models to Surface Buoyant Jets

Attempts to extend length-scale models to buoyant surface jets have also been made, although their use in this capacity has not been studied as extensively as for submerged round jets. One of the first applications of length-scale models to buoyant surface jets was proposed by Jirka et al. (1981). They compared trajectory data of free surface jets to the weakly deflected jet and strongly deflected jet regimes of a submerged jet and found similar power laws but with deviations corresponding to the product of the initial densimetric Froude number and the inverse of the velocity ratio, R, which is defined as u_o/u_a. They suggest that for the weakly deflected region of the surface jet, the trajectory has the relation:

$$\frac{y}{L_m} = C_1\left(\frac{x}{L_m}\right)^{\frac{1}{2}} \qquad (1.15)$$

while in the strongly deflected regime, the relationship is:

$$\frac{y}{L_M} = C_2\left(\frac{x}{L_m}\right)^{\frac{1}{3}} \qquad (1.16)$$

The constants of proportionality, C_1 and C_2, are dependent on the quantity Fr_o/R.

Different scaling laws have been proposed by Abdelwahed and Chu (1981). The relationships use a concept of a line impulse which makes this approach more applicable to strongly bent over flows where the current is the primary advecting mechanism and the lateral penetration into the flow is included as a perturbation.

Length scales have also been used to determine empirical expressions describing upstream intruding plumes (Jones et al., 1985) and the extent of recirculation of shoreline attached jets (Chu and Abdelwahed, 1990). However, a comprehensive study of length-scale model applications to buoyant surface jets has not been made. Due to the differences between surface buoyant jets and submerged jets, care must be taken when applying such length-scale analysis as originally proposed by Wright to surface buoyant jets. However, since fundamental similarities exist and some applicability has been recognized, it is reasonable to expect that by developing simple analytical expressions to describe their relative regimes of dominance, practical predictions can be made of the flow behavior in most buoyant surface jet scenarios.

Chapter II

Theoretical Background

The analysis of buoyant surface jets can be simplified by recognizing two separate regions: the "near-field" and the "far-field" (see Figure 2.1). The near-field designates the extent of the flow near the discharge in which the mixing is highly dependent on the discharge conditions, whereas the mixing in the far-field is dependent solely on the ambient conditions.

The dilution in the near-field is highly dependent on the initial volume, momentum, and buoyancy flux of the discharge. Different discharge configurations can lead to fundamentally different flow characteristics in the near-field. This forms the basis for classifying buoyant surface jets. Four categories of near-field flow patterns can be distinguished: free jets, wall jets, shoreline attached flows, and upstream intruding plumes. These four major flow categories are qualitatively described in Section 2.1.

In the far-field, ambient turbulence, stratification, wind shear, and many other factors dependent on the ambient conditions play a role in determining the rate of mixing. Lateral spreading due to buoyancy and passive diffusion caused by ambient turbulence are the predominant flow processes in a majority of practical situations. Since most other processes are difficult to model and/or are generally insignificant, only buoyant spreading and passive diffusion will be considered in the far-field.

It is apparent that no clear transition from the near-field to the far-field exists for buoyant surface jets. Although the transition is gradual, an approximate point of transition can be estimated by using particular length scales. Length scales can be used to delineate regimes within the flow in which particular mixing processes dominate. These length scales are described in Section 2.2.

Section 2.3 and 2.4 describe the theoretical considerations and development of the near-field and far-field regimes respectively. The near-field region includes various flow regimes that make up free jets, wall jets, shoreline attached jets, and upstream intruding plumes. The far-field includes the two processes of buoyant spreading and passive diffusion.

2.1. General Description of Flow Patterns

The distinction of four major flow categories is based on observations in the field and laboratory. Three of the four near-field flow patterns (free jets, shoreline attached jets, and upstream intruding plumes) were first quantitatively defined by Chu and Jirka (1986). The fourth, wall jets, are special cases of free jets and will be discussed along with free jets in Section 2.1.1. The following discussion describes each of these patterns and the processes that are involved.

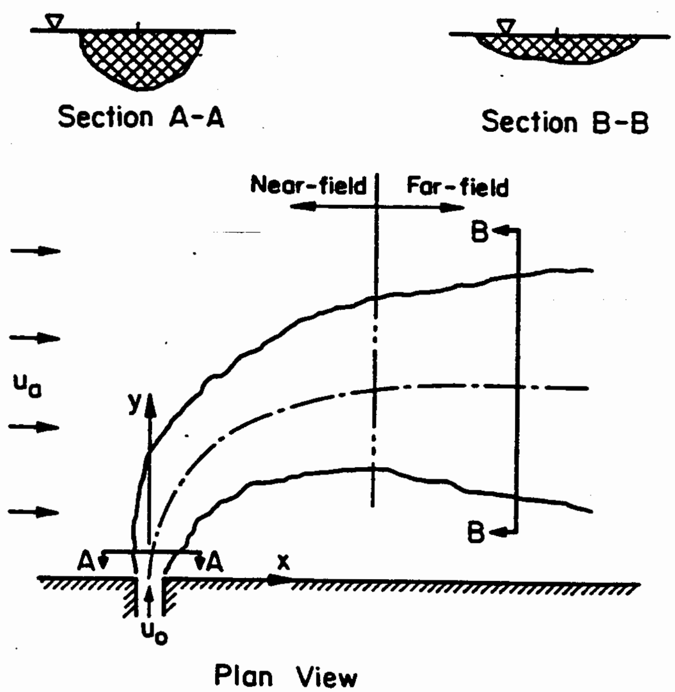


Figure 2.1 - Near and Far-field Regimes for a Typical Buoyant Surface Jet

2.1.1. Free Jets and Wall Jets

Free jets are characterized by a gradual bending so that the flow does not interact with the near shoreline. Figure 2.2 shows the surface isotherms of a typical free jet produced by a heated water discharge. Chu and Jirka (1986) have described the crossflow as having two effects on a free jet. The first is to entrain ambient momentum into the jet causing a gradual "bending" of the jet, and the second is to discourage unsteady buoyant spreading by advecting the plume-like spreading downstream.

Free-jets can be divided into two regimes analogous to submerged jets: a weakly deflected regime and a strongly deflected regime. The weakly deflected regime is characterized by strong initial jet-like mixing which is slightly advected downstream. Within this regime, there may be a transition from this strong jet-like mixing to a buoyancy-induced spreading mechanism, however the trajectory is still similar to that of a weakly deflected submerged jet dictated by the initial momentum. The theoretical development for the weakly deflected region is discussed in Section 2.3.3.1.

In the strongly deflected regime, the flow is advected downstream with the ambient current, but still retains some horizontal momentum which carries it further outward into the ambient flow. In this regime, jet mixing or buoyant spreading mechanisms may dominate. The trajectory remains similar to that of a strongly deflected submerged jet. The strongly deflected regime ends where its lateral progression becomes negligible and far-field processes take over. The theoretical development for the strongly deflected region is discussed in Section 2.3.3.2.

In stagnant ambient environments, only the weakly deflected regime exists. Figure 2.3 demonstrates a typical buoyant surface jet into a stagnant environment. At the end of the weakly deflected regime, a pool of buoyant effluent is established which spreads laterally unsteadily in all directions. The flow near the discharge is characterized by a constantly increasing depth representing the jet-like mixing, while further from the source a decrease in the plume depth occurs where buoyancy-induced spreading takes over. The transition distance in Figure 2.3 represents the end of the weakly deflected regime and the beginning of the unsteady buoyant spreading. The development of the mathematical relationships for buoyant surface jets discharged into stagnant environments is described in Section 2.3.2.

Bottom interaction may occur in free jets. However, if bottom interaction occurs too close to the discharge, it may block off the ambient flow, forcing the jet against the near bank causing shoreline attachment. If bottom interaction does occur without causing shoreline attachment, both the weakly deflected regime and the strongly deflected regime can exist, but will not be dominated by buoyant spreading since buoyant spreading results in detachment from the bottom and restratification of the plume.

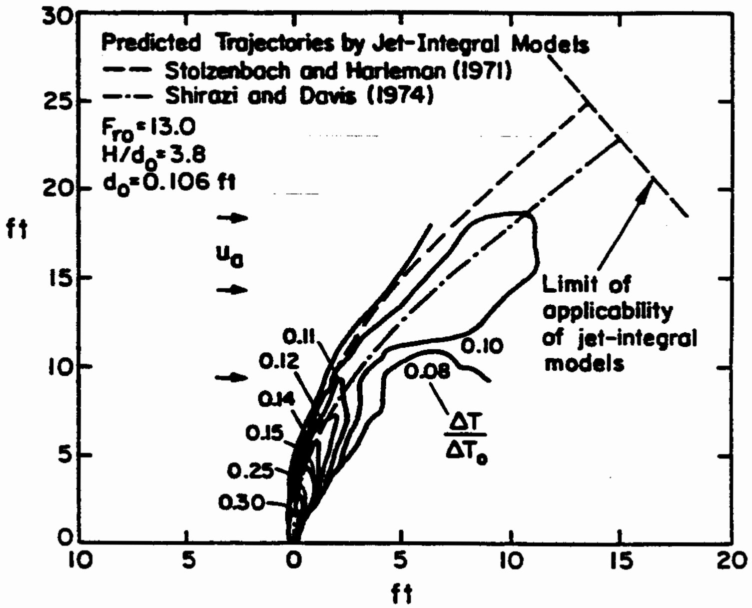


Figure 2.2 - Surface Isotherms of a Buoyant Surface Jet (from Koester, 1974)

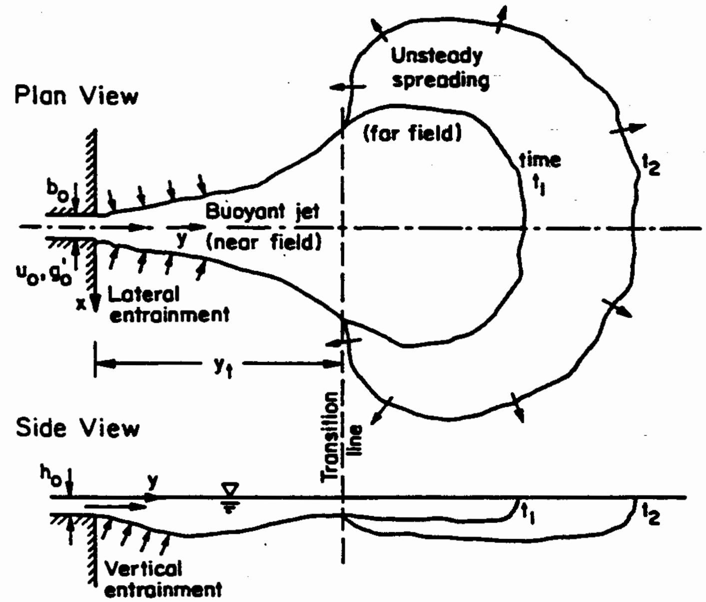


Figure 2.3 - Typical Buoyant Surface Jet in a Stagnant Environment

Wall jets can be considered weakly deflected jets which are discharged in a coflow along the bank (see Figure 2.4). The bank then acts as a reflective boundary along which a mirror image can be created. As with free jets, the initial mixing within the weakly deflected regime is jet-like. However, at the transition to the far-field, this jet-mixing becomes secondary, and buoyant spreading and/or passive diffusion becomes dominant. The theoretical development for wall jets is discussed in Section 2.3.4.

2.1.2. Shoreline Attached Jets

There are two phenomenon that cause dynamic attachment of the flow to the downstream shoreline. First, a strong crosscurrent can bend the jet over far enough to cause it to dynamically attach to the bank. Second, discharging over the whole depth of the receiving water can effectively "block off" the ambient current causing the flow to be pushed against the downstream shoreline. Characteristic of shoreline attached jets is the recirculation of effluent along the downstream bank caused by the wake effects in the lee of the jet. This is illustrated in Figure 2.5.

Shoreline attachment reduces the lateral progression of the jet. However, similar flow regimes to those found in free jets can be recognized: weakly deflected shoreline attached jet regime and strongly deflected shoreline attached jet/plume regime. It is unlikely that in the weakly deflected shoreline attached regime buoyancy will dominate since it is usually very short in extent due to the extreme initial bending. However, in the strongly deflected regime, buoyancy may take over producing buoyancy-induced lateral spreading. This will occur only in situations with no bottom attachment. The mathematical relationships for these regimes are developed in Section 2.3.5.

2.1.3. Upstream Intruding Plumes

In cases where strongly buoyant effluent is discharged into a slowly moving environment, upstream intrusion may develop. In upstream intruding plumes, a front is formed where the buoyant upstream intrusion is balanced by the shearing force at the head of the plume. The distance the plume intrudes along the upstream bank is denoted by the symbol x_u. (see Figure 2.6a). The near-field is limited to the area of the plume upstream of the discharge and a short distance downstream. At a distance x_s downstream from the discharge, the plume exhibits the far-field processes of buoyant spreading and then passive diffusion.

If the depth at the discharge is shallow and the effluent is discharged with sufficiently high momentum and buoyancy, the flow may be unstable and full vertical mixing with recirculation may occur in the immediate vicinity of the discharge. This is illustrated in Figure 2.6b. Restratification generally occurs just downstream of the point of discharge where far-field processes take over.

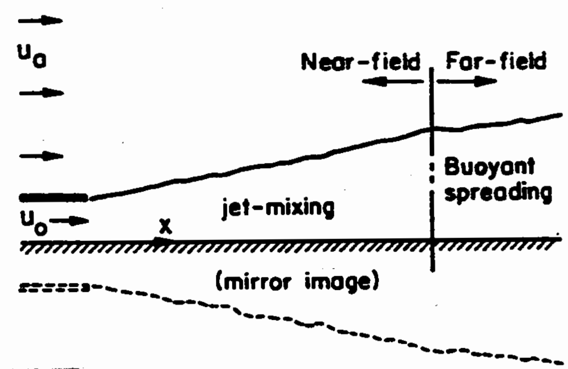


Figure 2.4 - Typical Wall Jet. The mirror image likens the flow to a surface jet issued in a coflow.

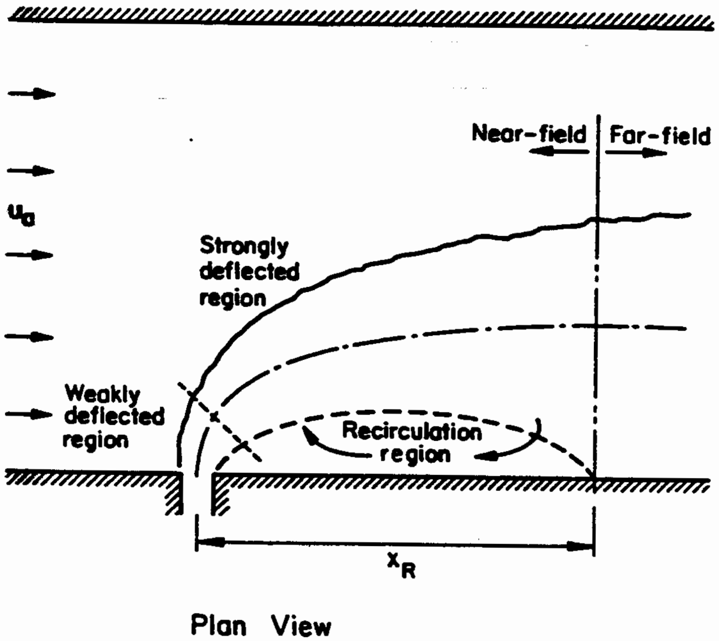


Figure 2.5 - Schematic of a Shoreline Attached Flow

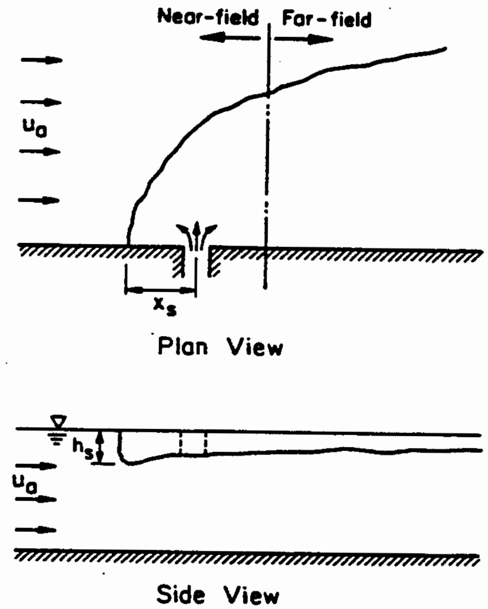


Figure 2.6a - Upstream Intruding Plume In a Deep Environment

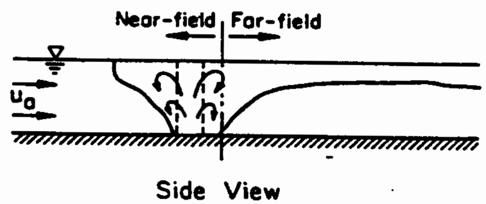


Figure 2.6b - Unstable Upstream Intruding Plume In a Shallow Environment

2.2. Length Scales

Length scales measure the relative importance of the initial volume flux, momentum flux, buoyancy flux, and crossflow velocity. Four length scales have practical meaning for use in buoyant surface jets analysis: the discharge length scale, jet-to-plume length scale, jet-to-crossflow length scale, and plume-to-crossflow length scale. Two dimensional definitions of the first three length scales are also used for situations where there is bottom interaction and the flow can be considered two dimensional. Each of these length scales are described in detail in the following subsections.

2.2.1. Discharge Length Scale

The discharge length scale measures the relative significance of the volume flux as compared to the momentum flux, and is defined as:

$$L_Q = \frac{Q_o}{M_o^{1/2}} \quad (2.1)$$

The discharge length scale defines the region for which discharge channel geometry strongly influences the flow characteristics. This comprises the zone of flow establishment, and is generally insignificant in extent. Note that this length scale plays a critical role when measured against the jet-to plume length scale (discussed below) in determining whether upstream intrusion occurs or not.

2.2.2. Jet-to-Plume Length Scale

The jet-to-plume length scale measures the relative importance of initial momentum and initial buoyancy. It is defined as:

$$L_M = \frac{M_o^{3/4}}{J_o^{1/2}} \quad (2.2)$$

In the region where $y << L_M$ momentum dominates the flow and therefore jet mixing prevails. Where $y >> L_M$, buoyancy dominates and strong lateral spreading prevails. For this reason, the jet-to-plume length scale is an important measure of where regimes characterized by jet mixing end and regimes characterized by buoyancy-induced lateral spreading begin (see Figure 2.7).

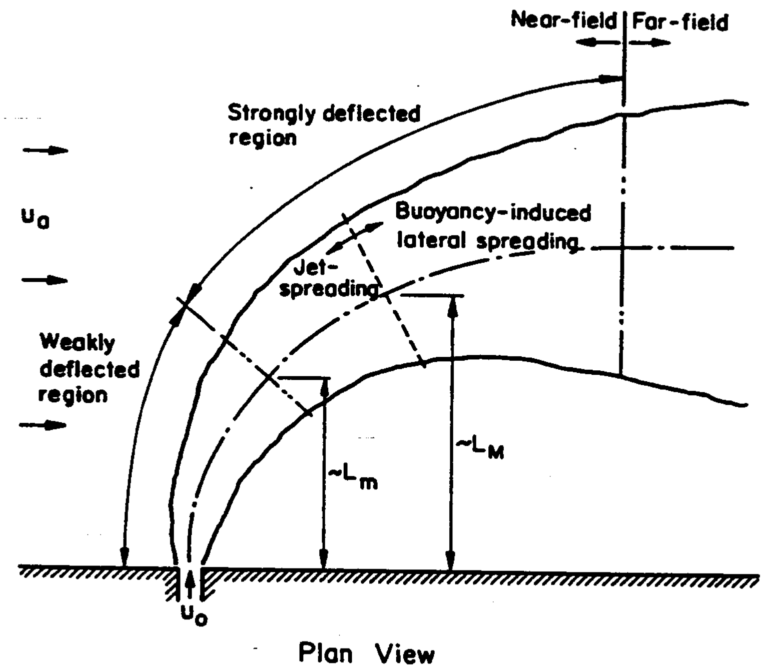


Figure 2.7 - Example of Length Scale Delineation of Flow Regimes for a Free Jet

2.2.3. Jet-to-Crossflow Length Scale

The jet-to-crossflow length scale measures the relative significance of the initial momentum and the ambient crossflow velocity. It is defined as:

$$L_m = \frac{M_o^{1/2}}{u_a} \tag{2.3}$$

The jet-to-crossflow length scale is a measure of where the flow changes from the weakly deflected regime to the strongly deflected regime (see Figure 2.7).

2.2.4. Plume-to-Crossflow Length Scale

The plume-to-crossflow length scale measures the relative importance of the initial buoyancy flux to the ambient crossflow velocity. It is defined as:

$$L_b = \frac{J_o}{u_a^3} \tag{2.4}$$

The plume-to-crossflow length has a significantly different meaning for surface plumes than for submerged buoyant jets. Since this length scale represents an interaction of the initial buoyancy of the effluent and the velocity of the crossflow, its most apparent measure is the extent of upstream spreading that a surface plume may exhibit. It also plays a role in the increased lateral progression of free jets caused by the thinning of a buoyant surface jet due to buoyancy.

2.2.5. Two-dimensional Length Scales

When a flow is mixed over the entire water depth, it can be considered two-dimensional. In this case, all the flow parameters can be defined per unit depth. The appropriate flux definitions for two-dimensional situations are as follows:

$$q_o = Q_o/H \tag{2.5}$$

$$m_o = M_o/H \tag{2.6}$$

$$j_o = J_o/H \tag{2.7}$$

where H is the characteristic local ambient water depth. Using these two-dimensional fluxes, two-dimensional length scales can be defined as follows:

$$l_q = \frac{q_o^2}{m_o} \tag{2.8}$$

$$l_M = \frac{m_o}{j_o^{2/3}} \tag{2.9}$$

$$l_m = \frac{m_o}{u_a^2} \tag{2.10}$$

Note that a two-dimensional plume-to-crossflow length scale is not defined since it cannot not exist on dimensional grounds (Akar, 1990).

2.2.6. Common Non-dimensional Numbers

Certain combinations of these length scales give some commonly used non-dimensional numbers, specifically the discharge densimetric Froude number, defined as:

$$Fr_o' = \frac{u_0}{\sqrt{g_o' h_o^{1/2}}} = \frac{L_M}{L_Q} \tag{2.11}$$

and the velocity ratio:

$$R = \frac{u_o}{u_a} = \frac{L_m}{L_Q} \qquad (2.12)$$

Also, the quantity Fr_o'/R equals:

$$\frac{Fr_o'}{R} = \left(\frac{L_m}{L_b}\right)^{1/2} \qquad (2.13)$$

which is an important factor in determining the trajectory of free jets as discussed in Section 2.3.3.3.

2.3. Near-field Flow Regime Analysis

The following sections provide the theoretical framework on which the analytical expressions used in the near-field flow regimes are based. These expressions are derived through the use of simple dimensional analysis which is discussed in Section 2.3.1. The subsequent section, Section 2.3.2, describes the mixing processes of surface buoyant jets discharging into a deep stagnant ambient environment. Sections 2.3.3 through 2.3.6 incorporate ambient crossflow and shallow water effects into the analysis of buoyant free jets, wall jets, shoreline-attached flows, and upstream intruding plumes respectively.

2.3.1. Dimensional Analysis of Surface Buoyant Surface Jets

The application of dimensional analysis to surface buoyant jets is based on two important assumptions. First, only fully turbulent flows are considered, and therefore the effects of viscosity can be neglected. Second, the Boussinesq approximation is assumed, that is, the density difference between the effluent and the ambient environment is small and is only important in terms of buoyancy forces.

The nine variables that effect the near-field flow of a surface buoyant jet are: the initial volume, momentum and buoyancy fluxes, Q_o, M_o, and J_o; the ambient velocity, u_a; the distance along the trajectory, s; the local ambient water depth, H; the width and depth of the discharge channel, b_o and h_o; and the discharge angle, σ. Therefore any flow variable, ϕ, can be described as a function of these independent variables:

$$\phi = f(Q_o, M_o, J_o, u_a, s, H, h_o, b_o, \sigma) \qquad (2.14)$$

The independent variables may be manipulated into different dimensionless groups which may differ from regime to regime depending on which parameters are significant to the particular flow. Then the form of the solution for a particular regime is obtained by

describing only the particular processes that dominate the flow in that regime and solving the simplified problem.

This is an asymptotic approach which provides solutions that are only valid within certain specified regimes and require experimentally determined coefficients. However, these solutions may be linked together so that appropriate expressions are used in succession providing an overall prediction for the entire problem.

The cartesian coordinate system used in this study is oriented with the origin at the mouth of the discharge, the x-axis pointing downstream, and the y-axis pointing across the current perpendicular to the ambient crossflow. This is illustrated in Figure 2.1.

2.3.2. Buoyant Surface Jet in a Stagnant Ambient Environment

A brief description of a free jet into a stagnant ambient environment is given in Section 2.1.1. As described in that section, the flow is comprised of two regimes: an initial regime of strong jet mixing with growth of the jet in both the vertical and horizontal directions, followed by a regime of increased buoyancy induced spreading for which the plume thickness decreases yet retains enough of the initial momentum to prevent the unstable buoyant pool which develops at the transition distance. This exemplified in Figure 2.3.

The transition between these two regimes is characterized by the jet-to-plume length scale L_M. As discussed in Section 2.2.2, the jet-to-plume length scale is a relative measure of the initial momentum and the initial buoyancy. For $y/L_M << O(1)$, the flow is dominated by the initial momentum and therefore is characterized by strong jet mixing. For $y/L_M >> O(1)$, the flow is dominated by the buoyancy and the lateral plume-like spreading becomes prevalent. In the case that $L_M << L_Q$ there will be no momentum dominated flow and the flow will be entirely plume-like.

In applying dimensional analysis to this problem, the ambient velocity u_a, depth parameter H, and the discharge angle σ are neglected, therefore reducing Eqn. 2.14 to:

$$\phi = f(Q_o, M_o, J_o, s, h_o, b_o) \qquad (2.15)$$

The nondimensionalized form of the flow parameter, ϕ^*, can then be described as a function of the following non-dimensional ratios:

$$\phi^* = f\left(\frac{s}{L_M}, \frac{L_Q}{L_M}, AR\right) \tag{2.16}$$

where AR is the discharge channel aspect ratio defined as h_o/b_o. Previous experience has indicated that the aspect ratio plays an insignificant role in flows with high local dilutions (Jirka et al., 1981).

2.3.2.1. Initial Jet-like Flow Characteristics

The initial regime, dominated by strong jet mixing, is analogous to one-half of a round submerged non-buoyant jet. After an initial zone of flow establishment (which will be neglected in the following analyses), the jet displays a full Gaussian velocity profile in the horizontal direction and a half Gaussian velocity profile in the vertical direction. Figure 2.8 demonstrates these profiles. The pollutant concentration exhibits similar Gaussian profiles. The centerline velocity, u_c, decreases with increasing distance along the centerline, s. However, total momentum flux, M, is conserved throughout this region. For jet-like flows with a Gaussian profile, the half-width b_h and vertical depth b_v of the flow are defined to be where the concentration is 1/e (37%) of the centerline concentration.

From dimensional considerations, u_c is found to be a function of the initial momentum, M_o, and the distance along the trajectory centerline, s, as follows:

$$u_c = c_1 \frac{M_o^{1/2}}{s} \tag{2.17}$$

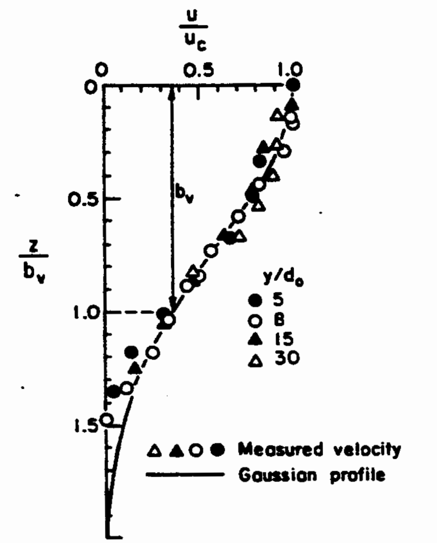
where c_1 is a constant. The only possible expression for the half-width that may be obtained from dimensional analysis is:

$$b = b_1 s \tag{2.18}$$

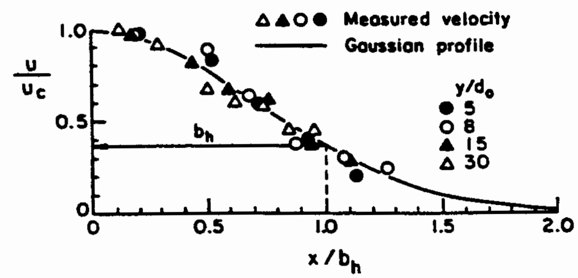
where b_1 is a constant. If the centerline dilution, S, is defined as C_o/C, where C_o is the initial discharge concentration and C is the centerline concentration, then the only dimensionally consistent relationship for S is:

$$S = s_1 \frac{M_o^{1/2}}{Q_o} s = s_1 \frac{s}{L_Q} \tag{2.19}$$

where s_1 is a constant. The constants c_1, b_1, and s_1 must be determined experimentally.



a) Vertical Velocity Profile



b) Horizontal Velocity Profile

Figure 2.8 - Gaussian Velocity Profile of a Non-buoyant Surface Jet (adapted from Rajaratnam and Humphries, 1984)

2.3.2.2. Jet-like Flow with Superimposed Buoyant Spreading

The following regime retains the initial momentum, therefore preserving the centerline velocity relationship given by Eqn. 2.17. However, the vertical bulk buoyant force acts on the flow that results in continuous deformation of the jet cross-section, increasing the horizontal spreading and vertical thinning. This buoyant spreading process can be considered a perturbation which may be superimposed on the jet-like centerline velocity (see Figure 2.3).

The buoyant spreading perturbation assumes the plume acts as a density current. Density currents generally entrain fluid in the frontal zones located at the edge of the plume, which spread laterally with a velocity v_B. Benjamin (1968) derived an equation for this spreading velocity:

$$v_B = \left(\frac{g' b_v}{C_D}\right)^{1/2} \qquad (2.20)$$

C_D is the coefficient of drag for the flow and ranges from 0.5 to 2.0 (Doneker and Jirka, 1990). The density current is modelled as having a top-hat velocity profile. Therefore, the half-width b_h and depth b_v are defined at the edge of the flow as shown in Figure 2.9.

Along a streamline the spreading velocity can be written as $v_B = u_c(db_h/ds)$. Substituting this into Eqn. 2.20, we obtain:

$$\left(\frac{db_h}{ds}\right) = \frac{1}{u_c}\left(\frac{g' b_v}{C_D}\right)^{1/2} \qquad (2.21)$$

Since buoyancy flux is conserved according to the identity $J_o = 2u_c g' b_v b_h$, the term g' in Eqn. 2.21 may be replaced by $J_o/(2u_c b_v b_h)$. If the centerline velocity relationship for a pure jet as given by Eqn. 2.17 is substituted in to Eqn. 2.21, the resulting expression is:

$$b_h \frac{db_h}{ds} = c_4 s^{1/2} \left(\frac{1}{2C_D}\right)^{1/2} \frac{J_o^{1/2}}{M_o^{3/4}} \qquad (2.22)$$

Integrating, we obtain the final horizontal spreading relationship:

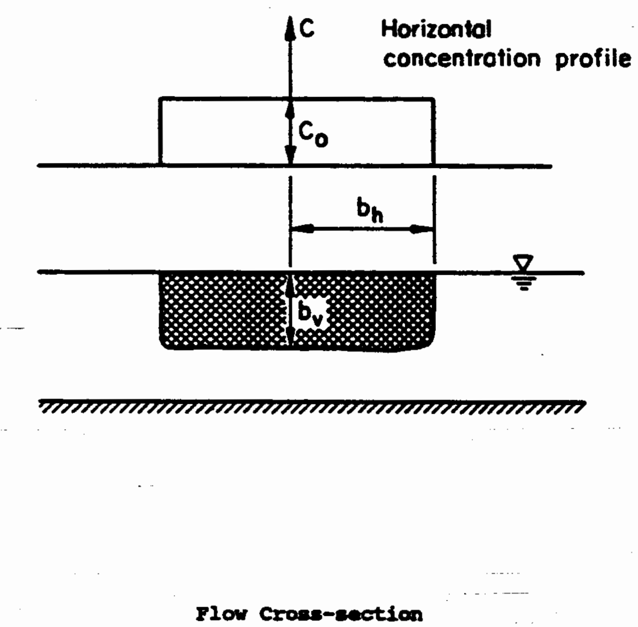


Figure 2.9 - Top-hat concentration profile as defined for flows dominated by buoyant spreading.

$$b_h = b_{hi} + b_2\left(\frac{1}{2C_D}\right)^{1/3}\left(\frac{1}{L_M}\right)^{2/3}\left(s^{3/2} - s_i^{3/2}\right)^{2/3} \tag{2.23}$$

where b_{hi} and s_i are the initial half-width and distance along the trajectory at the beginning of this region and b_2 is a constant. Comparison to various laboratory results have proven this to be an accurate description of the buoyancy-induced spreading process.

Adapting the entrainment relationship $q_e(s) = \beta v_B b_v$ where β is a constant within the range 0.15 and 0.25 (Simpson and Britter, 1979; Jirka and Arita, 1987) and applying them as they are for far-field processes (Section 2.4.1), the vertical depth of the plume is obtained:

$$b_v = b_{vi}\left(\frac{b_h}{b_{hi}}\right)^{\beta - 1} \tag{2.24}$$

If both buoyancy flux and pollutant flux are conserved, the ratio g'/g_o' can be used as an indicator of the dilution. As discussed in Section 2.4.1 for far-field processes, the dilution relationship is as follows:

$$S = S_i\left(\frac{b_h}{b_{hi}}\right)^{\beta} \tag{2.25}$$

where S_i is the initial dilution.

2.3.3. Free Jets in a Crossflow

By analogy to submerged buoyant jets, the trajectory of a buoyant free jet can be expected to pass through two phases (Jirka et al., 1981). The first is the weakly deflected region where the trajectory of the jet is similar to that of a pure momentum jet which is laterally deflected by the crossflow. In the second, the crossflow has bent the flow over and the jet/plume behaves like a line impulse which is gradually propagating perpendicular to the crossflow. Each of these regions are detailed in the following subsections 2.3.3.1 and 2.3.3.2.

The proper length scale to measure the transition between these two regions is the jet-to-crossflow length scale, L_m, discussed in Section 2.2.3. For $y/L_m << O(1)$, the crossflow is relatively unimportant and is treated as a small perturbation on the two regimes described in the stagnant case. This region is termed the "weakly deflected region." For $y/L_m >> O(1)$, the crossflow becomes the primary advecting mechanism

for which alternate theories will be developed (Section 2.3.3.2) and is termed the "strongly deflected region."

Similar flow phases have been established for the trajectories of submerged jets. By comparing surface buoyant jet trajectory data with the expressions developed for submerged jets, Jirka et al. (1981) showed that it would be reasonable to use similar flow region delineation for surface buoyant jets. An updated version of this comparison has been duplicated in Figure 2.10. Note that there appears to be some systematic effect due to the ratio $Fr_o/R = (L_m/L_Q)^{1/2}$. Correction for this effect is discussed in Section 2.3.3.3.

2.3.3.1. Weakly Deflected Flows

For a weakly deflected jet in a crossflow, the centerline velocity, half-width and dilution relationships developed for the initial jet-like mixing region in a stagnant environment (Eqns. 2.17, 2.18, and 2.19) still hold for this regime. However, the following perturbation is included to account for the downstream advection caused by the crossflow:

$$\frac{u_c}{u_a} = \frac{dy}{dx} \tag{2.26}$$

Substituting the centerline velocity given by Eqn. 2.17 into this expression and integrating gives the following trajectory relationship:

$$\frac{y}{L_m} = t_1\left(\frac{x}{L_m}\right)^{1/2} \tag{2.27}$$

where t_1 is a constant which must be determined experimentally. This the same dependency found for the weakly deflected region of a submerged jet (Wright, 1977). It is also consistent with the dimensional analysis discussed in Section 2.3.1 and the data for the initial phase of surface buoyant jets shown in Figure 2.10.

For two dimensional flow, the same crossflow perturbation is applied as for the 3-dimensional case Eqn. 2.26, but the centerline velocity exhibits the following relationship (Holley and Jirka, 1986):

$$u_c = c_4\left(\frac{m_o^{1/2}}{s^{1/2}}\right) \tag{2.28}$$

Substituting this centerline velocity definition into Eqn. 2.26 and integrating results in the following relationship:

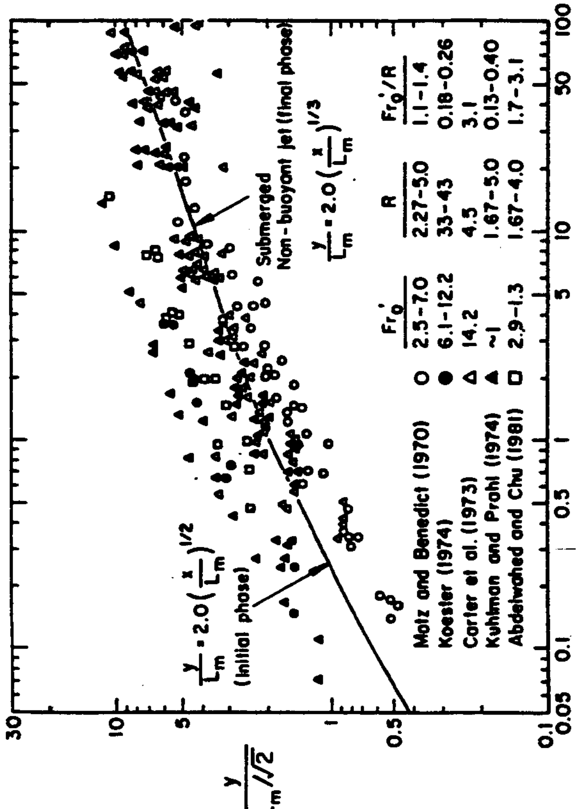


Figure 2.10 - Trajectories of Free Jets in Comparison to Trajectory Laws for Submerged Non-buoyant Jets (adapted from Chu and Jirka, 1986)

$$\frac{y}{l_m} = z_4\left(\frac{x}{l_m}\right)^{2/3} \tag{2.29}$$

The dilution is derived analogously to that of the 3-dimensional case but using the 2-dimensional flux and length scale definitions. This results in:

$$S = s_4\left(\frac{y}{l_q}\right)^{1/2} \tag{2.30}$$

The horizontal half-width relationship is similar to Eqn. 2.18:

$$b = b_4 y \tag{2.31}$$

but uses a different constant, b_4.

For cases where $L_M' < L_m$, buoyancy induced spreading occurs in the weakly deflected region. In this case, the same trajectory relationship applies as for the weakly deflected jet (Eqn 2.27), but the half-width, depth and dilution relationships that apply are the same as for the buoyancy induced spreading in the second flow regime of the stagnant case, and are given by Eqns. 2.23, 2.24, and 2.25. This, in effect, superimposes the governing equations for a density current onto a flow whose trajectory is still momentum controlled.

2.3.3.2. Strongly Deflected Flows

In the strongly deflected region, where $y/L_m >> O(1)$, the flow is advected downstream with the ambient current at a velocity u_a. However, it still exhibits some lateral deflection due to some residual momentum force. This is modelled as an instantaneous release of non-buoyant fluid issued horizontally from a line source. This conceptualization, as described by Scorer (1959), can be described with appropriate dimensional analysis, where the significant variables are the line impulse M', the horizontal progression y, and the time after release t. The resulting expression is:

$$\frac{M't}{y^3} = const. \tag{2.32}$$

In applying this analogy to a pure jet, M' is replaced by M_o/u_a and t is replaced by y/u_a. This results in the following dimensionally consistent expression:
This is the same form as for a submerged non-buoyant jet which is strongly deflected and is consistent with the data shown in Figure 2.10. Again, the half-width relationship for

$$\frac{y}{L_m} = t_5 \left(\frac{x}{L_m} \right)^{1/3} \qquad (2.33)$$

a jet holds:

$$b = b_5 y \qquad (2.34)$$

If we use the identity $C_o Q_o \propto C_e b_h b_v u_a$ to describe the mass conservation, then in terms of length scales the dilution $S = C_o/C_e$ is given by:

$$S = s_5 \left(\frac{y^2}{L_m L_Q} \right) \qquad (2.35)$$

For 2-dimensional strongly deflected jets, a similar line impulse model is used resulting in the following relationship:

$$\frac{m' t^2}{y^3} = const. \qquad (2.36)$$

where m' is the line impulse per unit depth of discharge, $m' = M'/H$. Therefore, substituting m_o/u_a and y/u_a into Eqn. 2.36 as before, the following relationship is obtained:

$$\frac{y}{l_m} = t_6 \left(\frac{x}{l_m} \right)^{1/2} \qquad (2.37)$$

The dilution is analogous to that of the 3-dimensional case but uses the 2-dimensional flux and length-scale definitions. The dilution for a strongly deflected 2-dimensional jet is then:

$$S = s_6 \frac{y}{(l_q l_m)^{1/2}} \qquad (2.38)$$

The horizontal half-width equation is similar to Eqn. 2.18:

$$b = b_6 y \qquad (2.39)$$

The above equations only apply for jet-like flows in the strongly deflected region, i.e.: when $L_m < y < L_M$. However, once buoyancy starts to deform the flow and

buoyancy induced spreading becomes the dominate mixing process in this region, i.e.: when $y > L_m$ & L_M, the half-width and dilution expressions developed in Section 2.3.2.2 for buoyancy driven lateral spreading apply (Eqns. 2.23, 2.24, and 2.25). However, the trajectory relationship remains the same as developed for a strongly deflected jet (Eqn. 2.33).

2.3.3.3. Correction for Trajectory Constant

When buoyancy-induced spreading causes the plume to thin, the flow will tend to penetrate further into the crossflow. This can be seen by the systematic effect of Fr_o'/R on the trajectories of free jets in Figure 2.10. To estimate this trend of variability, run-averaged trajectory constants were plotted against Fr_o'/R as shown in Figure 2.11. Using regression, a best-fit line was obtained that can be approximated by:

$$t = 2.27 \left(\frac{1}{Fr_d'/R} \right)^{1/3} \qquad (2.40)$$

The minimum trajectory constant was taken as 1.6, the theoretical value for a non-buoyant wall jet (Holley and Jirka, 1986).

2.3.4. Wall Jets

As noted in Section 2.1.1, wall jets are considered a special case of weakly deflected free jets. When the mirror image of a wall jet is considered, the flow is identical to that of a free jet issue in a coflow. Since the discharge is issued in a coflow, however, no strongly deflected region exists and any buoyancy induced spreading can be considered a far-field process.

Therefore, the two possible regimes that exist in the near-field of a wall jet are analogous to the 3-dimensional and 2-dimensional weakly deflected jet regimes. Using identical formulations as for the weakly deflected free jets but including a mirror image, the following dilution relationships are obtained for the 3-dimensional and 2-dimensional cases, respectively:

$$S = s_7 \left(\frac{x}{L_Q} \right) \qquad (2.41)$$

$$S = s_8 \left(\frac{x}{l_q} \right)^{1/2} \qquad (2.42)$$

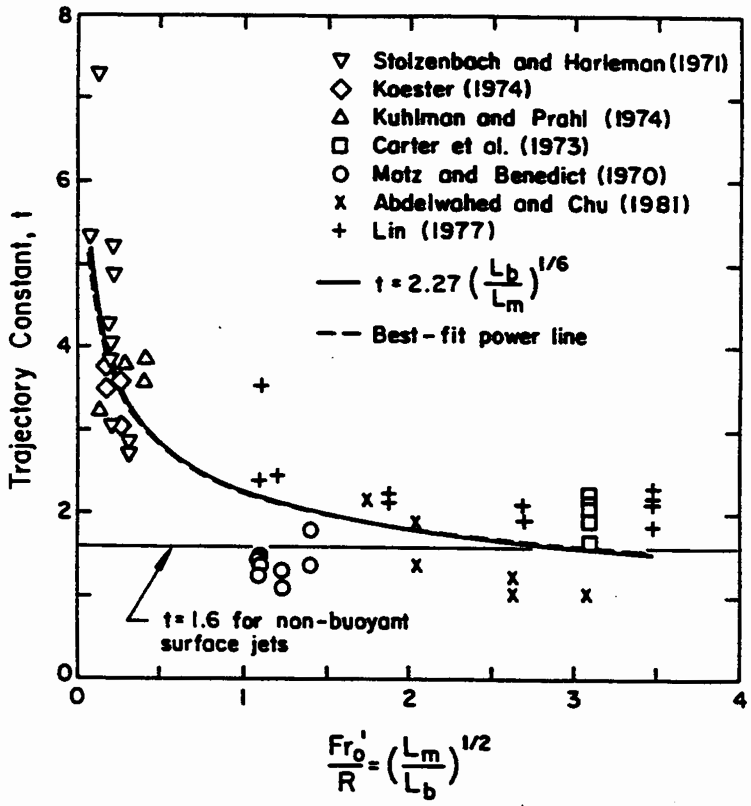


Figure 2.11 - Variation of the Trajectory Constant With the Ratio $Fr_o'/R = (L_m/L_b)^{1/2}$

where s_7 and s_8 are constants. The horizontal half-width retains a similar linear half-width relationship for both the 3-dimensional and 2-dimensional cases:

$$b_h = b_{7,8}\,x \qquad (2.43)$$

where b_7 and b_8 are constants.

2.3.5. Shoreline Attached Flows

A qualitative description of shoreline attached flows is given in Section 2.1.2. Shoreline attached flows are typically strongly bent over with a recirculation region between the jet-like structure and the near bank. This is exhibited in Figure 2.5.

The bulk of the discharged fluid follows a trajectory similar to that of a free jet yet reduced in lateral penetration into the crossflow. Very often, the flow is strongly bent over very near the discharge so no weakly deflected region exists. Plots of shoreline attached flow trajectories show that the same power laws developed for free jets apply to shoreline attached flows as shown in Figure 2.12. Note, that the lateral progression is approximately half of that found for free jets. The lack of data for $y/L_m < 1$ is due to the fact that this weakly deflected region is typically very small and often negligible. As with free jets, there seems to be some systematic effect due to Fr_o'/R.

Since the trajectories of shoreline attached jets are analogous to free jets, similar trajectory relationships may be used for the 3-dimensional flows in the weakly deflected region and strongly deflected region, respectively:

$$\frac{y}{L_m} = t_9\left(\frac{x}{L_m}\right)^{1/2} \qquad (2.44)$$

$$\frac{y}{L_m} = t_{10}\left(\frac{x}{L_m}\right)^{1/3} \qquad (2.45)$$

where t_9 and t_{10} are constants which are approximately half of their free jet counterparts. For 2-dimensional jets, relationships similar to those for free jets also apply for the weakly deflected and strongly deflected regions, respectively:

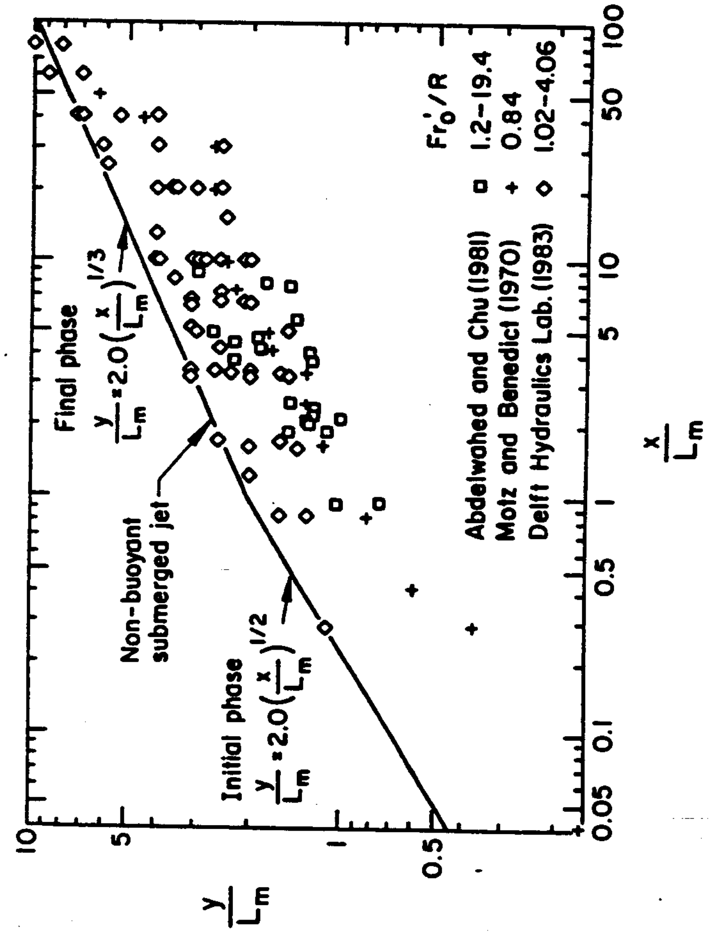


Figure 2.12 - Trajectory Data for Shoreline Attached Flows and Comparison to Submerged Jet Trajectory Laws.

$$\frac{y}{l_m} = t_{11}\left(\frac{x}{l_m}\right)^{2/3} \qquad (2.46)$$

$$\frac{y}{l_m} = t_{12}\left(\frac{x}{l_m}\right)^{1/2} \qquad (2.47)$$

where t_{11} and t_{12} are also approximately one-half of the corresponding free-jet constants.

Similarly, the dilution and half-width relationships will be analogous to the free jet relationships. However, the dilution constants will be reduced due to the recirculation of the effluent along the downstream bank. The half-width definition only applies between the centerline and the outside edge of the flow since there is no definable half-width between the trajectory centerline and the near bank (see Figure 2.5).

2.3.6. Upstream Intruding Plumes

For very buoyant discharges into weak crossflows, the plume may spread upstream against the current. These upstream intruding plumes are qualitatively described in Section 2.1.3. The theoretical development of these plumes comes largely from studies by Jones et al. (1985) performed for radial surface and subsurface discharges.

Jones et al. define an intrusion length scale, L_I, which describes the interaction between the buoyant spreading force and the ambient crossflow.

$$L_I = \frac{J_o}{\pi C_{DI} u_a^3} = \frac{L_b}{\pi C_{DI}} \qquad (2.48)$$

where C_{DI} is a drag coefficient on the order of unity (i.e.: O(1)).

Jones et al. provide a numerical solution for the upstream intrusion length, x_s, which can be approximated as follows:

$$\frac{x_s}{L_I} = 3.77\left(\frac{L_M}{L_I}\right)^{2/3} \qquad for \quad \frac{L_M}{L_I} \geq 0.356 \qquad (2.49)$$

$$\frac{x_s}{L_T} = 1.9 \qquad for \quad \frac{L_M}{L_T} < 0.356 \tag{2.50}$$

Jones et al. also developed a relationship for the bulk dilution at the end of the "intermediate region" which is approximately located at $x_f = x_s$. The relationship is approximated as:

$$S_f = \frac{0.81}{(\pi C_{DI})^{1/3}} \frac{L_M^{2/3}}{L_Q} L_b^{1/3} \tag{2.51}$$

Jones et al. give the typical depth of flow in the upstream intrusion region, h_s, as:

$$h_s = \frac{C_{D2} u_a^2}{g'} \tag{2.52}$$

where $C_{D2} \approx 0.8$. Since buoyancy flux is conserved, $g' = g_o'/S$. Therefore, using the above definition of dilution and writing Eqn. 2.51 in terms of length scales, h_s is equivalent to:

$$h_s = \left(\frac{0.405\, C_{D2}}{(\pi C_{DI})^{1/3}}\right) \frac{L_M L_Q}{L_m} \tag{2.53}$$

The width of the plume b_h at the source is predicted as approximately 2.6 times the length of the upstream intrusion length. The width of the plume at the end of the near-field region is estimated as approximately $4.0x_s$.

$$b_{h,x=0} = 2.6\, x_s \tag{2.54}$$

$$b_M = 4.0\, x_s \tag{2.55}$$

By continuity, the vertical depth of the plume at the end of this region can be computed as:

$$b_v = \frac{S_f L_m L_Q}{b_M} \tag{2.56}$$

If the depth at the discharge is shallow and the effluent is discharged with reasonably high momentum and buoyancy, the flow may be unstable and full vertical mixing may occur as shown in Figure 2.6b. Because of recirculation, dilutions are reduced. From dimensional analysis, the dilution of this flow pattern can be concluded to be in the following form:

$$S = s_{13} \frac{H_D^{5/3}}{L_M^{2/3} L_Q} \tag{2.57}$$

where s_{13} must be experimentally determined.

Restratification will generally occur in this plume-like flow just downstream of the point of discharge. The point of restratification will be used for the end of this regime and the beginning of the far-field. Restratification of the flow occurs at a distance approximately H_D downstream of the discharge (Doneker and Jirka, 1990), therefore $x_f = H_D$. The same half-width, depth, and upstream intrusion length as used for the stable case apply to this unstable regime.

2.4. Far-field Flow Regime Analysis

The two far-field processes that may occur are buoyant spreading followed by passive diffusion. Although buoyant spreading may or may not occur depending on the buoyancy and hydrodynamics of the flow, all discharges, if taken far enough downstream, are affected by ambient turbulence and therefore become passively diffused. The following two subsections (Sections 2.4.1 and 2.4.2) describe the theoretical development of these two processes.

2.4.1. Buoyant Spreading Process

For strongly buoyant discharges, the far-field may exhibit strong lateral spreading and vertical thinning. This is similar to the buoyancy-driven spreading process described in Section 2.3.2.2 for the near-field. However, in the far-field there is no net lateral movement of the plume and the plume is advected downstream with the ambient current at a velocity u_a.

The definition diagram and structure of a surface buoyant spreading process in unstratified crossflow is shown in Figure 2.13. The laterally spreading flow behaves like a density current and entrains ambient fluid in the "head region" of the current. The mixing rate is usually relatively small. Furthermore, the flow may interact with a nearby bank or shoreline. The flow depth may decrease during this phase. The analysis of this

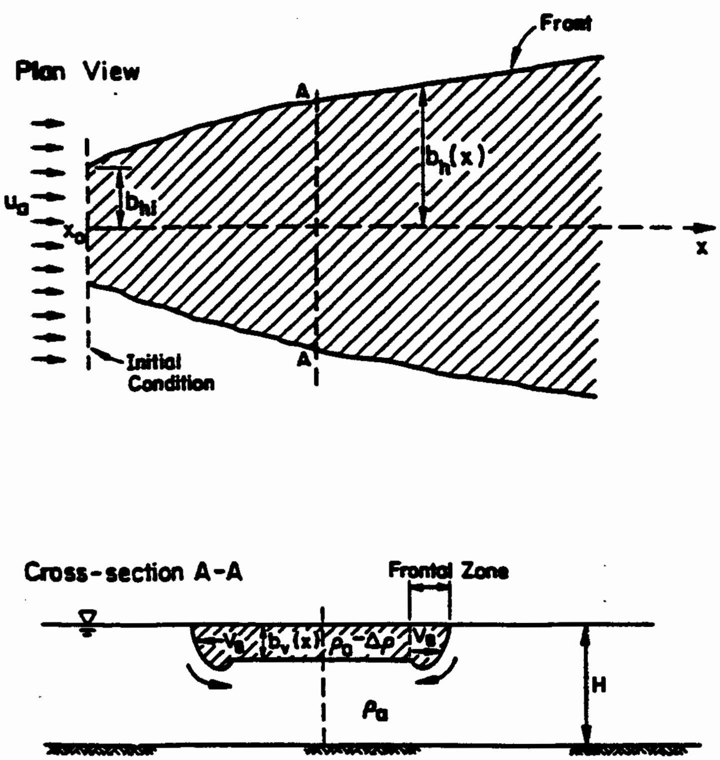


Figure 2.13 - Buoyant Surface Spreading Process (from Doneker and Jirka, 1990)

region is analogous to the arguments presented in Section 2.3.2.2 for buoyancy-induced spreading in the near-field.

The continuity equation for the density current is:

$$u_a \frac{\partial b_v}{\partial x} + \frac{\partial (v b_v)}{\partial y} = w_e \tag{2.58}$$

where w_e is the net velocity across the interface and v(x,y) is the local transverse velocity. Combining the equation for the spreading velocity v_B developed by Benjamin (Eqn.2.20) with Eqn. 2.58 and integrating laterally over the density current half-width gives:

$$u_a \frac{d(b_h b_v)}{dx} = q_e(x) \tag{2.59}$$

where $q_e(x)$ is the localized head entrainment representative of the dominant mixing mechanism.

The localized head entrainment of the density current is parameterized as $q_e(x) = \beta v_B b_v$ where β is a constant with a range of 0.15 to 0.25 (Simpson and Bitter, 1979; Jirka and Arita, 1987).

The flow half-width b_h is obtained for any downstream distance x by using the boundary condition for the streamline ($v_B = u_a db_h/dx$) and integrating Eqn. 2.58.

$$b_h = \left[b_{hi}^{3/2} + \frac{3}{2}\left(\frac{L_b}{2C_D}\right)^{1/2}(x - x_i)\right]^{2/3} \tag{2.60}$$

where x_i is the downstream distance at the beginning of the buoyant spreading region, and b_{hi} is the initial density current half-width. C_D is the coefficient of drag for the head region of the flow and ranges from 0.5 to 2.0 (Doneker and Jirka, 1990). This 2/3 power law of flow spreading is in agreement with the previous work of Larsen and Sorensen (1968).

The vertical flow half-width b_v is given by integrating Eqn. 2.59 to obtain:

$$b_v = b_{vi}\left(\frac{b_h}{b_{hi}}\right)^{\beta - 1} \tag{2.61}$$

Due to mixing in the head region, the local concentration C and local buoyancy g' gradually change with distance x. The bulk dilution S, given by C_0/C, is equivalent to the ratio g_0'/g'. Since buoyancy flux is conserved, the identity $u_a g' b_v b_h$ = constant may be combined with the initial conditions to obtain the following expression for dilution:

$$S = S_i \left(\frac{b_h}{b_{hi}}\right)^{\beta} \qquad (2.62)$$

where S_i is the initial dilution.

2.4.2. Passive Ambient Diffusion

The existing turbulence in the ambient environment becomes the dominating mixing mechanism at sufficiently large distances from the discharge point. In general, the passively diffusing flow is growing in width and in thickness (see Figure 2.14). Furthermore, it may interact with the channel bottom and/or banks.

The analysis of this region follows classical diffusion theory (e.g.: Fischer et al., 1979). The standard deviation σ_z of a diffusing plume in crossflow can be written in terms of the transverse turbulent diffusivity E:

$$\sigma_z^2 = \frac{2Ex}{u_a} \qquad (2.63)$$

in which x is the distance following the ambient flow with the point release located at x = 0. The coefficient of eddy diffusivity depends on the turbulence conditions in the environment and may be a function of distance x (or plume size σ_z).

2.4.2.1. Diffusion in Bounded Channel Flow

In open channel flow the eddy diffusivity can be related to the friction velocity u_* and the channel depth H

$$E_z = 0.2 u_* H \qquad (2.64)$$

for vertical diffusivity, and

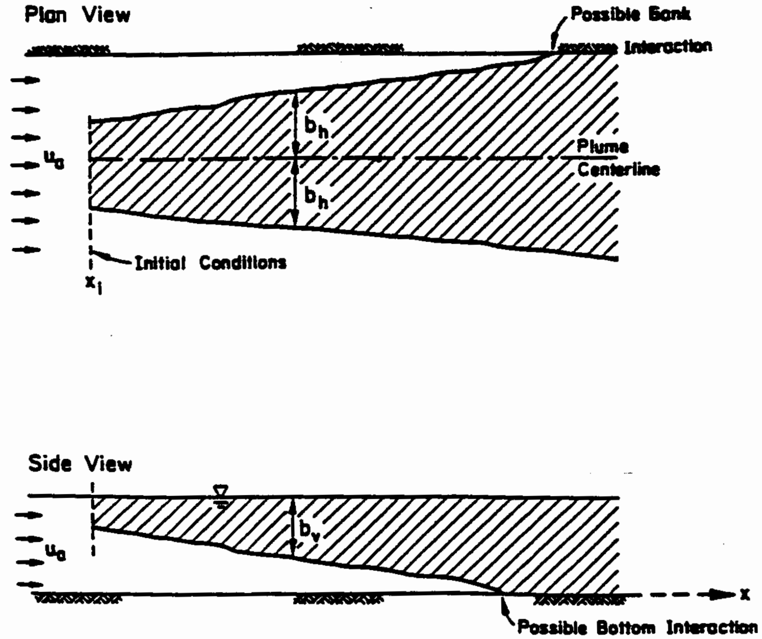


Figure 2.14 - Passive Ambient Diffusion Process (from Doneker and Jirka, 1990)

$$E_y = 0.6 u_* H \qquad (2.65)$$

for horizontal diffusivity. The friction velocity is given by $u_* = (f/8)^{1/2} u_a$ where f is the Darcy-Weisbach friction factor. Due to some anisotropy in a typical channel flow, the diffusivity in the horizontal transverse direction is usually larger than the diffusivity in the vertical direction. The coefficients included in Eqns. 2.64 and 2.65 are average values for seasonably uniform channels. The coefficients may be considerably larger (up to a factor of 2) for highly non-uniform cross-sections and/or strongly curved channels (see also Holley and Jirka, 1986).

Solution of Eqn. 2.63 with these diffusivities and with initial flow half-width conditions specified at x_i (see Figure 2.14) gives the vertical thickness b_v and half-width b_h, respectively:

$$b_v = \left[\frac{\pi E_z (x - x_i)}{u_a} + b_{vi}^2\right]^{1/2} \qquad (2.66)$$

$$b_h = \left[\frac{\pi E_y (x - x_i)}{u_a} + b_{hi}^2\right]^{1/2} \qquad (2.67)$$

where x_i, b_{hi}, and b_{vi} are the distance, half-width, and depth of the plume at the beginning of the passive diffusion region. The above definitions are related to the vertical and horizontal standard deviations by a factor of $(\pi/2)^{1/2}$: $b_v = (\pi/2)^{1/2}\sigma_{zv}$ and $b_h = (\pi/2)^{1/2}\sigma_{yh}$, assuming an equivalent top-hat plume with same centerline concentration and pollutant mass flux.

Applying the continuity equation $2u_a b_v b_h \approx SQ_o$ yields the dilution:

$$S = \frac{2 b_v b_h}{L_M L_Q} \qquad (2.68)$$

Beyond the distance at which the flow becomes fully mixed ($b_v = H$), the dilution expression is:

$$S = \frac{2 H b_h}{L_M L_Q} \qquad (2.69)$$

2.4.2.2. Horizontal Diffusion in Unbounded Channel Flow

Many environmental flows without any significant limitation on the transverse dimension (coastal water, large lakes, etc.) exhibit an accelerating turbulent diffusive growth pattern. The horizontal diffusivity is often specified by the so called "4/3 law" (see Fischer et al., 1979):

$$E_y = \alpha (3\sigma_{yh})^{4/3} \qquad (2.70)$$

in which α is a coefficient equal to 0.01 $cm^{2/3}/s$ (appropriate for small plume sizes) and E_y is in units of [cm^2/s] and σ_{yh} in [cm]. Integration of the applicable diffusion equation with this variable E_y yields a solution for plume growth (Brooks, 1960, and Fischer et al., 1979):

$$b_h = b_{hi}\left[1 + \left(\frac{\pi}{3}\right)\frac{E_{yi}(x - x_i)}{u_a b_{hi}^2}\right]^{3/2} \qquad (2.71)$$

using the present notation and half-width convention. E_{yi} is the initial value of diffusivity, so from Eqn. 2.70 at position x_i:

$$E_{yi} = 0.0015\, b_{hi}^{4/3} \qquad (2.72)$$

with units of [m^2/s] for E_{yi} and [m] for the initial half-width b_{hi}. The dilution expressions are the same as before, given by Eqns. 2.68 and 2.69.

Chapter III

Program Structure

This chapter provides a general overview of the structure of CORMIX3. Section 3.1 provides some general comments on the language and structure of the system and Section 3.2 details the separate elements that make up CORMIX3.

3.1 General Comments and Description

CORMIX3 (*Cornell Mixing* Zone Expert System, Subsystem *3*) is intended to be a tool to aid design and analysis of buoyant surface discharges. CORMIX3 is designed to give the user a qualitative "picture" of the flow through ample descriptions of the flow characteristics. CORMIX3 also gives quantitative results including estimates of the flow trajectory, dilution, width and depth.

The system uses two programming languages: VP-Expert (Paperback Software,Inc.) and Fortran. VP-Expert is an expert system shell and is used to construct the knowledge base. VP-Expert is efficient in data collection, logic programming, and user aid. It is used to collect the input data and classify the flow according to the classification scheme described in Chapter IV. However, VP-Expert is poor in performing mathematical calculations, and therefore Fortran is used to perform the numerical simulation of the flow. For a complete description of expert systems and logic programming as applied to CORMIX Expert Systems, see Doneker and Jirka (1990).

3.2 Knowledge Base System Elements

Figure 3.1 shows the overall structure of CORMIX3. CORMIX3 is comprised of five knowledge base elements: DATIN3, PARAM3, CLASS3, HYDRO3, and SUM3. Each of these elements are described in more detail in the following sub-sections. The Fortran hydrodynamic simulation is accessed through the HYDRO3 element and is discussed in more detail in Chapter V.

Information is passed from one element to the next by writing all the variables and their respective values into a text file called a "cache" file, which is then read by the following element. These cache files, as well as the other components of CORMIX3, are stored in separate DOS subdirectories. The CORMIX3 program file directories are listed and described in Table 3.1.

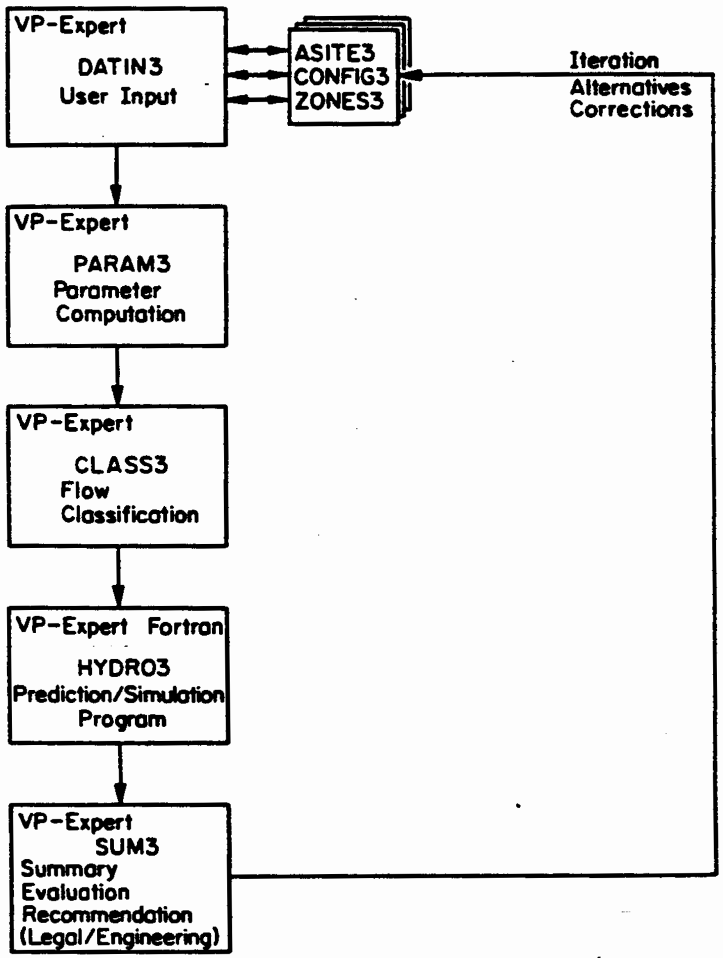


Figure 3.1 - Overall Structure of CORMIX3

Table 3.1

CORMIX3 Program File Directories

Directory	Comments
C:\cmx3	System root directory. Contains VP-Expert system files and the knowledge base system driver CORMIX3.
C:\cmx3\advice3	Contains user-requested advice files
C:\cmx3\bat3	Contains batch files for program execution, data file manipulation, and program control.
C:\cmx3\cache3	Stores cache "fact" files created by knowledge base elements.
C:\cmx3\data3	Contains constant files and other case-specific data files.
C:\cmx3\desc3	Contains descriptions of each flow class.
C:\cmx3\kbs3	Contains the executable knowledge base programs.
C:\cmx3\pgms3	Contains executable Fortran hydrodynamic simulation program and other Fortran file manipulation programs.
C:\cmx3\sim3	Stores hydrodynamic simulation result files.

3.2.1. DATIN3

DATIN3 collects all the input data from the user. It is divided into three segments: ASITE3, CONFIG3, and ZONES3. The user is prompted for one piece of information at a time and instructions for entering the data are provided with each segment if requested by the user.

The first segment, ASITE3, collects general site and case identifier information. It is in this segment that the user gives the present run a DOS name for which all of the data files are saved under. The second segment, CONFIG3, collects the data on the geometry and flow conditions of the receiving water body and the discharge channel. This segment also requires information of the area immediately surrounding the discharge outlet. The details of the ambient and discharge requirements are discussed in Section 4.1. The fourth segment, ZONES3, allows for a legal mixing zone, toxic dilution zone, and region of interest to be specified.

3.2.2. PARAM3

The second knowledge base element, PARAM3, calculates the physical *parameters* necessary to classify the flow and perform the hydrodynamic simulation. These parameters and their definitions are listed in Table 3.2. Note that if any of the parameters are infinite, i.e.: $L_m = \infty$ when $u_a = 0$ m/s, the parameter is assigned a value of 99999.9.

The length scales calculated in this element are then used in the following element, CLASS3, to classify the flow. For a description of the meaning of these length scales, see Section 2.2.

3.2.3. CLASS3

The knowledge base element CLASS3 classifies the flow according to the discharge and ambient characteristics which are measured by the length scales calculated in the previous element. A more detailed description of the classification scheme is given in Section 4.2. By exploiting the logic programming strengths of the expert system shell, this element can efficiently classify the flow into a particular flow class, and in the process display the decisions for making its conclusion. For a more detailed explanation of the logic programming used, the reader is referred to Doneker and Jirka (1990).

Once the flow has been classified the user may obtain a complete description of the flow class if so desired. These flow class descriptions can be found in Appendix A.

Table 3.2

Parameters Calculated in PARAM3

Parameter Description	Parameter Definition
Velocity ratio	$R = u_o/u_a$
Discharge density deficit	$\Delta\rho_o = \rho_a - \rho_o$
Overall Densimetric Froude number	$Fr'_o = u_o/\sqrt{(g'_o a_o)^{1/2}}$
Channel Densimetric Froude number	$Fr'_{ch} = u_o/\sqrt{g'_o h_o}$
Discharge volume flux	$Q_o = u_o a_o$
Discharge momentum flux	$M_o = u_o Q_o$
Discharge buoyancy flux	$J_o = g_o' Q_o$
Discharge length scale	$L_Q = Q_o / M_o^{1/2}$
Jet-to-plume length scale	$L_M = M_o^{3/4} / J_o^{1/2}$
Jet-to-crossflow length scale	$L_m = M_o^{1/2} / u_a$
Plume-to-crossflow length scale	$L_b = J_o / u_a^3$

3.2.4. HYDRO3

The expert system element HYDRO3 simply links to the Fortran hydrodynamic simulation program. The Fortran simulation is discussed in more detail in Chapter V. When the Fortran simulation is run, no results are shown. Instead, once the hydrodynamic simulation is complete, the program returns to the HYDRO3 expert system element where, if the user so desires, the results of the simulation may be viewed.

To obtain a hardcopy of the hydrodynamic simulation, the user must exit CORMIX3 and return to DOS. A hardcopy may then be obtained by typing:

print c:\cmx3\sim3\'fn'.cxo

where: 'fn' is the DOS file name of the run.

These instructions are repeated at the end of CORMIX3 in the SUM3 element.

3.2.5. SUM3

The final expert system element, SUM3, summarizes the results of the analysis. It first provides a summary of the information obtained from DATIN3 and the corresponding parameters calculated in PARAM3. It then provides a summary of the hydrodynamic simulation, including the conditions at the legal mixing zone, toxic dilution zone, and region of interest. The simulation summary also provides information on bank attachment, full vertical mixing, and full horizontal mixing if they occur.

Towards the end of the SUM3 element, recommendations on improving the design may be obtained if so desired. Instructions for obtaining hardcopies of the hydrodynamic simulation and flow class description are also given.

The final step of CORMIX3 allows the user to return to different points in the run to change site information, and/or ambient and discharge information, and/or mixing zone specifications. This allows for the user to iterate on a given scenario.

Chapter IV

Flow Classification

The first section of this chapter, Section 4.1, details the ambient and discharge data requirements and schematization used in CORMIX3. The next section, Section 4.2, describes the process used in the CLASS3 element to classify the flow according to its near-field flow characteristics.

4.1. Ambient and Discharge Data Requirements

Due to the complexity of geometries and flow conditions of most receiving water bodies, simplifying assumptions and schematizations must be made in order to perform the analysis and simulation of most situations. Section 4.1.1 discusses the schematization of the receiving water body and the required ambient parameters.

The discharge and immediate area surrounding the outlet may also require some simplification. Section 4.1.2 describes the discharge schematization and data requirements for the discharge.

The hydrodynamic simulation of the flow is based on the assumption that ambient conditions are steady state. Although this is generally not true over long periods of time for most water bodies, it is usually a suitable assumption for the time scale of near-field mixing processes.

For all cases, the coordinate system is arranged with the x-coordinate pointing downstream and the y-coordinate pointing perpendicularly across the ambient flow away from the discharge. The origin is assumed to be at the mouth of the discharge.

4.1.1. Schematization of the Receiving Water Body

CORMIX3 assumes the receiving water body is rectangular in cross-section with a constant width, BS, and depth, H_a, (see Figure 4.1). Although this simplification may be readily apparent for well channeled rivers or artificial channels, it may require more judgement and experience for highly irregular channels. CORMIX3 does not account for meandering effects, and therefore can be seen as assuming the river is "stretched out" to be straight. In addition, a uniform velocity profile equal to the depth averaged velocity, u_a, is assumed.

The receiving water can be assumed to "bounded" if it has an finite width which can be specified. On the other hand, if the section can be assumed to be infinite in width

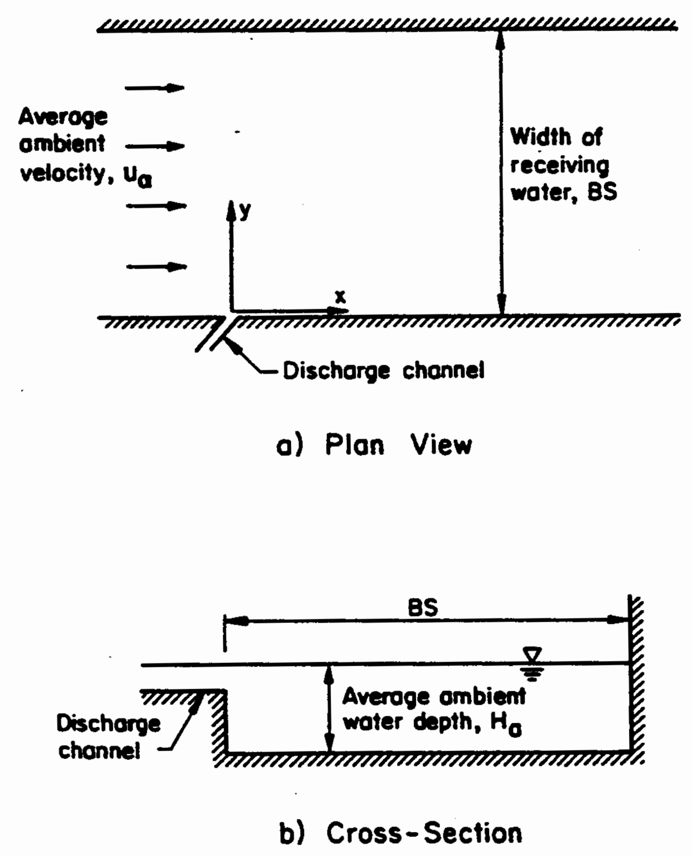


Figure 4.1 - Schematization of the Receiving Water Body

for all practical purposes, then it is considered "unbounded". This may b reasonable for discharges into large estuaries or lakes or the ocean. For unbounded sections, CORMIX3 sets the width of the receiving water body to 88888.8 meters.

The ambient density, ρ_a, of the receiving water is assumed to be constant throughout the entire water depth. If ambient stratification exists near the discharge and is of the same magnitude as the initial discharge density difference, then significant errors may cause. Therefore, special consideration should be taken when such situations occur.

4.1.2. Discharge Configurations

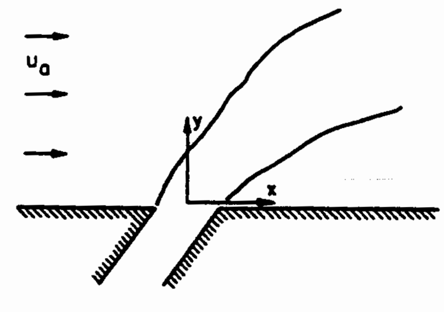
The three discharge configurations possible in CORMIX3 are shown in Figure 4.2. CORMIX3 also allows for both rectangular discharge channels and circular (pipe) discharges assuming that the pipe is flowing full. Pipe discharges may be submerged by no more than one-half pipe diameter for them to be considered a surface discharge. If the cross-section is an odd shape (i.e.: a pipe flowing partially full), then an equivalent rectangular cross section should be used with an equivalent discharge area and width-to-depth ratio.

Discharge channel geometry may vary with time if the elevation of the receiving water changes. This is common when the effluent is discharged into a tidal environment where significant daily changes in water elevation may exist. In this case, separate calculations for each elevation of the receiving water must be made as long as steady state can be assumed.

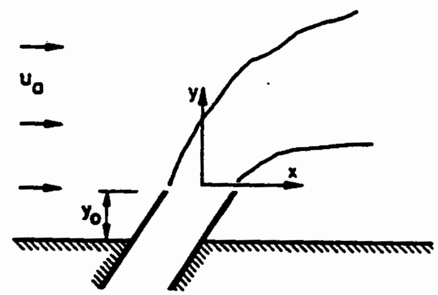
In the immediate area of the discharge, the actual depth of the receiving water, H_D, may be less than average ambient depth. In addition, the bottom of the channel may be sloping away form the discharge bank at an angle θ. This geometric schematization is depicted in Figure 4.3. The bottom slope is considered to be important only in the near-field, and for all far-field computations the average ambient depth is assumed to be the characteristic depth. The user must be careful not to specify an ambient depth that will not be reached in the near-field since this may lead to inconsistencies when passing from the near-field to the far-field. CORMIX3 will note these inconsistencies if they occur, but will make no effort to correct any inaccuracies they may cause.

The discharge angle σ has been limited to the range of $0° \leq \sigma \leq 120°$ since upstream oriented surface discharges are highly undesirable and difficult to model. The most common orientation is $\sigma \approx 90°$.

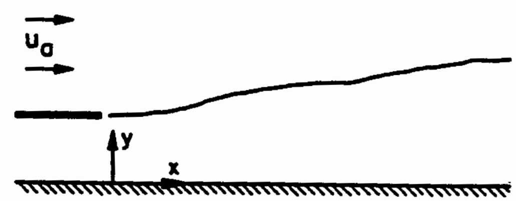
The discharge density, ρ_o, can be entered directly or computed from the discharge temperature. If the discharge channel densimetric Froude number, Fr_{ch}, is less than unity, a wedge of ambient of water may intrude upstream into the channel changing the



a) Discharge flush with bank



b) Protruding discharge



c) Coflowing along downstream bank

Figure 4.2 - Three Possible Discharge Configurations

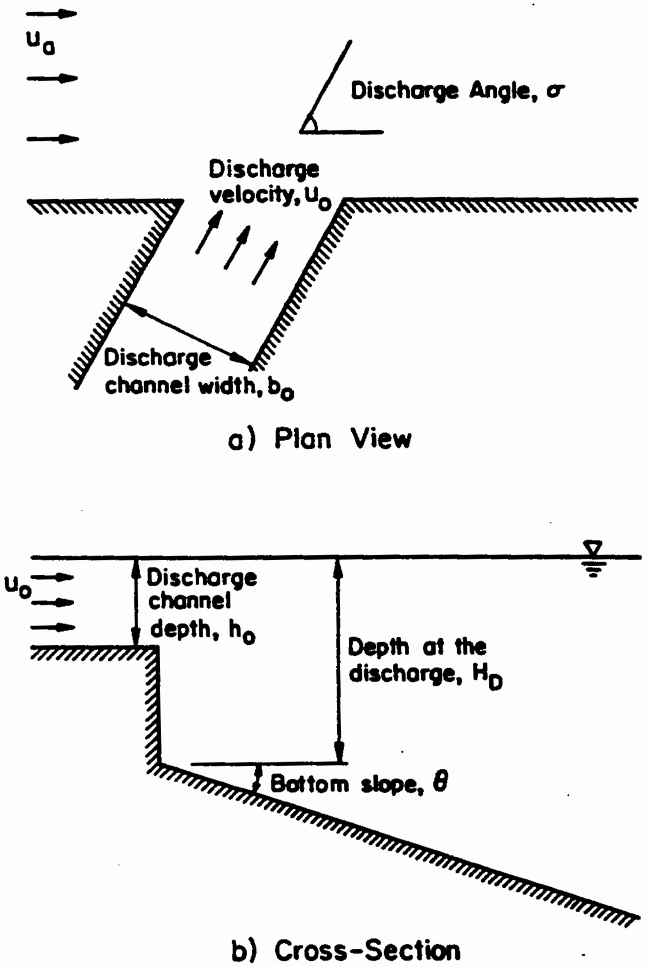


Figure 4.3 - Near-field Discharge Schematization (The discharge channel is shown as rectangular for illustrative purposes. It may also be circular with diameter d_o.)

discharge conditions. CORMIX3 recognizes this possibility and recalculates the discharge conditions if intrusion does occur.

4.2. Flow Classification Scheme

In this section, a general description of the flow classification process is described (Section 4.2.1) and then the rationale behind the classification criteria is discussed (Section 4.2.2).

4.2.1. General Description of Classification Scheme

Just as ambient and discharge conditions can be highly variable, so can the resulting flow patterns. With a wide variety of flows possible, each with very different characteristics, it is necessary to classify the flow in order to determine the method of simulation. For example, a non-buoyant shore-hugging plume will have very different mixing and dilution properties than will a buoyant free jet discharging into a stagnant environment. Therefore, to classify the flow, measures of the discharge and ambient conditions must be used.

CORMIX3 uses length scales, ambient depth measures, and the discharge angle to classify the flow. Figure 4.4 shows the CORMIX3 classification scheme. This classification scheme has been derived from the classification diagram first proposed by Chu and Jirka (1986) shown in Figure 4.5.

The CORMIX3 classification scheme is a tree-like structure consisting of a series of criteria or decisions. The decision criteria shown in Figure 4.4 indicate that the criteria are measured against unity. In actuality, they are measured against their respective criterion constants (C1, C2, etc.) which are O(1). For example, The criteria used to distinguish between jet-like flows and plume-like flows, L_Q/L_M, is actually compared against the constant C1, i.e.: $L_Q/L_M < C1$ for jet-like flows and $L_Q/L_M > C1$ for plume-like flows. The values for these constants are listed in Table 4.1. The rationale for these criteria are discussed in the next section, Section 4.2.2.

The classification of the flow is designed so that it makes one decision at a time, outputs that decision to the user, and then proceeds to the next decision. As seen in Figure 4.4, up to five decisions may be required to before a classification of the flow is reached. The resulting flow class then gives the user an idea of the general characteristics of the flow. For example, if the flow is classified as an SA2, then the flow is attached to the downstream bank and is initially fully vertically mixed. A description of the four general flow classes can be found in Section 2.1.

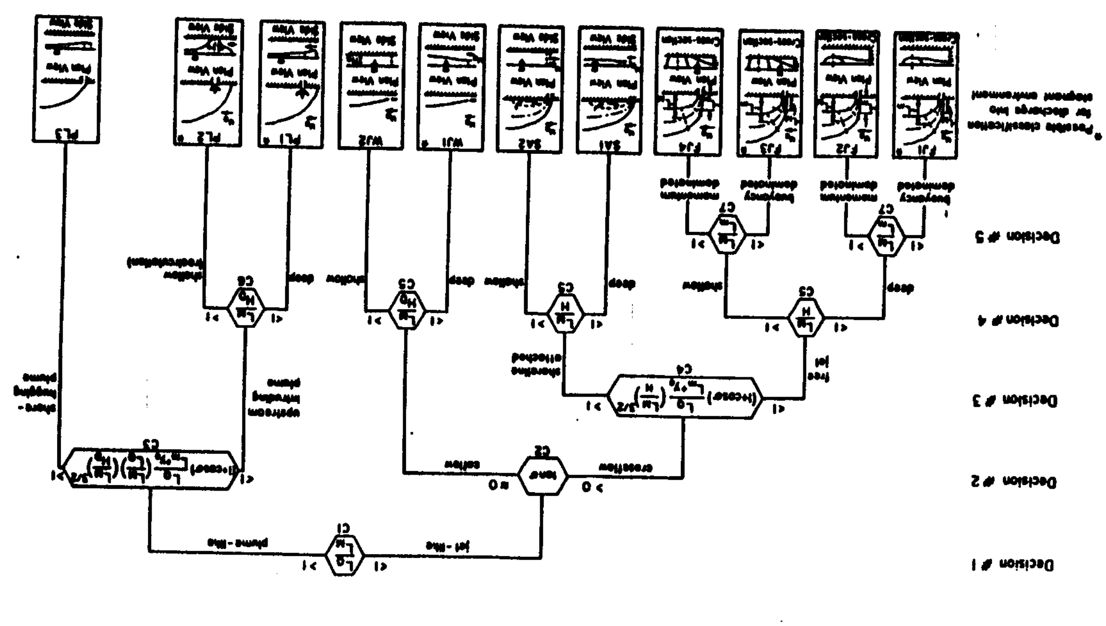


Figure 4.4 - CORMIX3 Classification Scheme

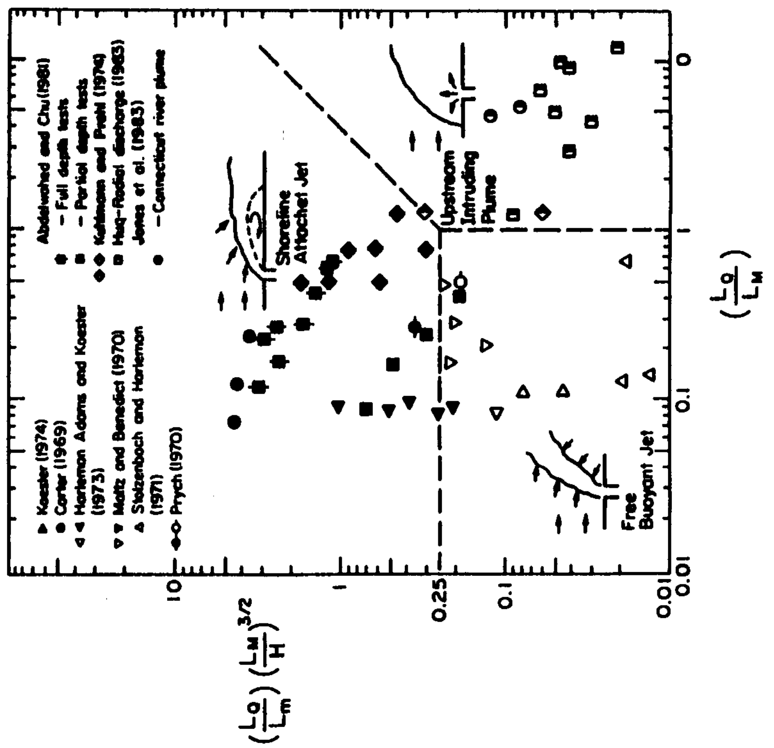


Figure 4.5 - Classification Diagram from Chu and Jirka (1986)

4.2.2. Classification Criteria

Several basic classification criteria make up the CORMIX3 flow classification scheme (see Figure 5.4). Each of these criteria and the rationale behind them are discussed below.

4.2.2.1. Jet-like vs. Plume-like (C1)

The first decision to be made in the classification scheme is to determine whether the flow exhibits stronger jet-like characteristics or more plume-like characteristics. The appropriate parameter in this case is the overall densimetric Froude number Fr_o,' for which the equivalent length scale ratio is L_M/L_Q. For $L_M/L_Q > O(1)$, the flow is highly buoyant with the initial discharge momentum being secondary, therefore being more plume-like. The converse is true for $L_M/L_Q < O(1)$.

Note that in actuality, the classification scheme uses the inverse of the Froude number, L_Q/L_M, to remain consistent with Chu and Jirka's classification diagram, Figure 4.5.

4.2.2.2. Crossflow vs. Coflow (C2)

This criterion identifies the flows which are issued near parallel to the bank or at a discharge angle σ small enough to cause Coanda attachment to the downstream bank. For these cases, the flow is classified as a wall jet.

4.2.2.3. Upstream Intruding Plume vs. Shore-hugging Plume (C3)

This criterion combines several effects to determine whether a plume-like flow will intrude along the upstream bank against the current. This criterion is given as:

$$(1 + \cos\sigma)\frac{L_Q}{L_M + y_o}\left(\frac{L_M}{L_Q}\right)\left(\frac{L_M}{H_D}\right)^{\frac{3}{2}} = C \qquad (4.1)$$

The terms $(1+\cos\sigma)$ and y_o are correction terms to account for the discharge angle and any protrusion of the discharge channel into the receiving water. Eliminating these factors reduces the criterion to that shown in Figure 4.5.

$$\left(\frac{L_m}{L_b}\right)^{\frac{1}{2}}\left(\frac{L_M}{H_D}\right)^{\frac{3}{2}} = C \qquad (4.2)$$

Table 4.1

Flow Classification Criteria

Criterion	Criterion Constant	Source(s) or Comments
$\frac{L_Q}{L_M} \lessgtr C1$	C1 = 1	A) From Chu and Jirka (1981) classification diagram (Fig. 4.5)
$\tan\sigma \gtrless C2$	C2 = 0.2	Davidson (1989), and Sharp and Vyas (1977)
$(1+\cos\sigma)\frac{L_Q}{L_m+y_o}\left(\frac{L_M}{L_Q}\right)\left(\frac{L_M}{H_D}\right)^{3/2} \gtrless C3$	C3 = 0.25	See A
$(1+\cos\sigma)\frac{L_Q}{L_m+y_o}\left(\frac{L_M}{H}\right)^{3/2} \gtrless C4$	C4 = 0.25	See A
$\frac{L_M}{H} \gtrless C5$	C5 = 3.33	B) Criteria for maximum surface jet depth as per Chu and Jirka (1981)
$\frac{L_M}{H_D} \gtrless C6$	C6 = 2.0	See B
$\frac{L_M}{L_m} \gtrless C7$	C7 = 0.42	Simple dimensional analysis

Assuming that the criterion constant, C3, reflects the depth dependency of L_M/H_D, Eqn. 4.5 may be rewritten as:

$$\left(\frac{L_m}{L_b}\right) = C \qquad where:\ C = C(L_M/H_D) \qquad (4.3)$$

The physical significance of L_b is different for buoyant surface jets than for submerged buoyant jets. For a buoyant surface jet, L_b is a measure of the upstream intrusion. If L_b is small relative to L_m, i.e.: L_m/L_b > C, upstream intrusion will be negligible and a shore-hugging plume results. On the other hand, if buoyancy is strong and L_m/L_b < C, upstream intrusion will exist.

4.2.2.4. Free Jets vs. Shoreline Attached Flows (C4)

This criterion combines several effects to distinguish free jets from shoreline attached jets, and is given as follows:

$$(1 + \cos\sigma)\frac{L_Q}{L_M + y_o}\left(\frac{L_M}{H}\right)^{\frac{3}{2}} = C \qquad (4.4)$$

As with the previous criterion, the terms (1+cosσ) and y_o are corrections added to the criterion proposed by Chu and Jirka (1986). Without these factors the criterion reduces to:

$$\left(\frac{L_Q}{L_m}\right)^{\frac{1}{2}}\left(\frac{L_M}{H}\right)^{\frac{3}{2}} = C \qquad (4.5)$$

The same assumption regarding the ratio of (L_M/H) will be made as described for the upstream intruding plume vs. shore-hugging plume criterion, that is, the criterion constant C4 reflects the depth dependency of L_M/H. As noted in Section 4.2.2.3, the ambient depth H is calculated according to the location of maximum depth, y_{max} = 3.89 L_M. Therefore, the resulting criterion reduces to:

$$\left(\frac{L_Q}{L_m}\right) = C \qquad where:\ C = C(L_M/H) \qquad (4.6)$$

L_m is a measure of where the initial discharge momentum becomes secondary and the ambient crossflow predominates. If L_m is small, i.e.: less than the discharge length scale L_Q, then the jet is very rapidly bent over and attachment to the downstream bank will

result. On the other hand, if L_m is very large, it will take a much greater distance for the jet to become bent over and it will remain free from shoreline attachment.

The constants in these two criteria have been determined empirically by Chu and Jirka and contain some uncertainty. However, as shown in Figure 4.5, they appear to provide a good delineation for the different flow patterns.

4.2.2.5. Deep vs. Shallow (C5, C6)

Jirka et al. (1981) found that the maximum depth of a buoyant surface jet is proportional to the length scale L_M:

$$b_{v,max} = 0.30\ L_M \qquad (4.7)$$

If H is the ambient water depth at the point of maximum jet depth, the ratio L_M/H can be considered a relative measure of the maximum jet depth as compared to the ambient depth.

For the plume-like flows and wall jets, the ambient depth H is taken as the ambient water depth at the point of discharge, H_D, since the near-field is located in the vicinity of the discharge or along the bank. For free jets and shoreline attached jets, the ambient depth H is calculated at the location of maximum jet depth. Jirka et al. (1981) determined the location of maximum jet depth to be proportional to L_M:

$$y_{max} = 3.89\ L_M \qquad (4.8)$$

for a stagnant environment. Therefore, for free jets and shoreline attached jets, the ambient depth is calculated at this point.

$$H = Max\ (\ H_o,\ H_D + y_{max}\sin\theta) \qquad (4.9)$$

4.2.2.6. Momentum Dominated vs. Buoyancy Dominated (C7)

Although the flow may originally be determined to be jet-like, buoyancy may still play a significant role in the mixing process near the discharge. For this reason, it is important to distinguish the flows for which buoyant spreading becomes predominant in the weakly deflected region from the flows where it does not become important until the strongly deflected region.

To distinguish these two types of flows, the ratio L_M/L_m is employed. L_M measures the transition distance where buoyant spreading becomes the predominant flow characteristic and the jet-induced mixing becomes secondary. L_m measures the transition from the weakly deflected region to the strongly region. Therefore, for $L_M/L_m < O(1)$, buoyant spreading occurs in the weakly deflected region. For $L_M/L_m > O(1)$, jet-mixing extends into the strongly deflected region.

Chapter V

Hydrodynamic Simulation

This chapter discusses the hydrodynamic simulation of the flow. Section 5.1 describes the protocols used for the different flow classes. Section 5.2 provides the relationships used to estimate the dilution, trajectory, width, and depth in the different segments of the simulation. The final section, Section 5.3, specifies how the location of the transition from one module to the next is determined.

5.1. Flow Protocols

The hydrodynamic simulation is a Fortran program linked to the expert system through the HYDRO3 expert system element. It consists of a series of subroutines, each of which describes a different region of the flow. Each of these subroutines, termed "modules", is used for a region of the flow for which a particular process or processes dominate. The theoretical development of the different types of flows is detailed in Chapter II. Table 5.1 lists the different modules used in CORMIX3. There are 17 modules which fall into four categories: The weakly deflected region, the strongly deflected region, the upstream spreading processes, and the far-field processes. The following section, Section 5.2, describes the relationships used for each module.

A "flow protocol" refers to the specific series of modules a particular flow class uses. For example, a flow classified as SA2 will use Modules 301, 318, 328, 341, and 351 sequentially. Tables 5.2a-d list the protocols for each flow class. Note that Module 301, the discharge module, is actually the first module for every flow simulation. It was excluded from Tables 5.2a-d for the sake of brevity. The transition between modules are determined according to specific "transition rules" discussed in Section 5.3.

5.2. Simulation Modules

The relationships used to estimate the trajectory, dilution, width, and depth of the flow in each module are discussed in the following subsections. Note that most of the modules use continuous relationships, that is, they describe the evolution of a flow process along its trajectory. However, the two upstream spreading modules use a control volume approach where only the upstream intrusion length, the width of the flow at the discharge, and the final conditions at the transition are calculated.

The initial conditions at the beginning of each module are subscripted with "i" while the final conditions are subscripted with "f". Each module predicts the trajectory

Table 5.1

CORMIX3 Simulation Modules

Module No.	Description
301	Discharge module
311	Weakly deflected jet, 3-D
312	Weakly deflected jet, 2-D
313	Weakly deflected plume
314	Wall jet, 3-D
315	Wall jet, 2-D
317	Weakly deflected shoreline attached jet, 3-D
318	Weakly deflected shoreline attached jet, 2-D
321	Strongly deflected jet, 3-D
322	Strongly deflected jet, 2-D
323	Strongly deflected plume
327	Strongly deflected shoreline attached jet, 3-D
328	Strongly deflected shoreline attached jet, 2-D
329	Strongly deflected shoreline attached plume
331	Upstream spreading
332	Upstream spreading with unstable recirculation
341	Buoyant spreading
351	Passive diffusion

Table 5.2a

Flow Protocols for Free Jets

Flow Class	Flow Zone	Module No.	Transition Rule
FJ1	Weakly deflected jet, 3-D	311	12
	Weakly deflected plume	313	15
	Strongly deflected plume	323	24
	Buoyant spreading	341	41
	Passive diffusion	351	
FJ2	Weakly deflected jet, 3-D	311	13
	Strongly deflected jet, 3-D	321	22
	Strongly deflected plume	323	24
	Buoyant spreading	341	41
	Passive diffusion	351	
FJ3	Weakly deflected jet, 3-D	311	11
	Weakly deflected jet, 2-D	312	14
	Weakly deflected plume	313	15
	Strongly deflected plume	323	24
	Buoyant spreading	341	41
	Passive diffusion	351	
FJ4	Weakly deflected jet, 3-D	311	11
	Weakly deflected jet, 2-D or	312	10
	Strongly deflected jet, 3-D	321	21
	Strongly deflected jet, 2-D	322	23
	Strongly deflected plume, 3-D	323	24
	Buoyant spreading	341	41
	Passive diffusion	351	

Table 5.2b

Flow Protocols for Wall Jets

Flow Class	Flow Zone	Module No.	Transition Rule
WJ1	Wall jet, 3-D	314	16
	Buoyant spreading	341	41
	Passive diffusion	351	
WJ2	Wall jet, 3-D	314	20
	Wall jet, 2-D	315	17
	Buoyant spreading	341	41
	Passive diffusion	351	

Table 5.2c

Flow Protocols for Shoreline-attached Flows

Flow Class	Flow Zone	Module No.	Transition Rule
SA1	Weakly deflected shoreline attached jet, 3-D	317	18
	Strongly deflected shoreline attached jet, 3-D	327	25
	Strongly deflected shoreline attached plume	329	27
	Buoyant spreading	341	41
	Passive diffusion	351	
SA2	Weakly deflected shoreline attached jet, 2-D	318	19
	Strongly deflected shoreline attached jet, 2-D	328	-
	Strongly deflected shoreline attached plume	329	27
	Buoyant spreading	341	41
	Passive diffusion	351	

Table 5.2d

Flow Protocols for Upstream Intruding Plumes

Flow Class	Flow Zone	Module No.	Transition Rule
PL1	Upstream intrusion	331	31
	Buoyant spreading	341	41
	Passive diffusion	351	
PL2	Upstream intrusion with recirculation	332	31
	Buoyant spreading	341	41
	Passive diffusion	351	
PL3	Buoyant spreading	341	41
	Passive diffusion	351	

coordinates (x,y), the centerline concentration and dilution (C and S), and half-width and depth (b_h and b_v).

Each of the near-field modules are applied in a local (primed) coordinate system. This arises from the assumption on which the near field analyses are based, that the discharge is a point source (for 3-dimensional flows) or a line source (for 2-dimensional flows). To account for this, a virtual origin is assumed with coordinates (x_v,y_v). The conversion from the local coordinate system to the global coordinate system is simply:

$$(x,y) = (x',y') + (x_v,y_v) \qquad (5.1)$$

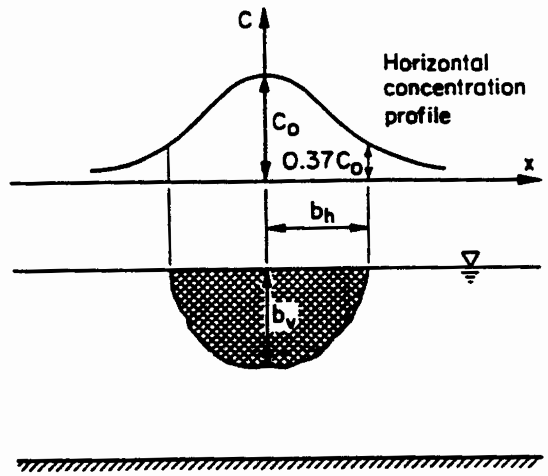
The position of the virtual source is computed by taking the dilution at the end of the previous module and back calculating the location of the virtual origin using the dilution equation for the present module. This assures continuity of dilution and concentration. However, small discontinuities in width and depth may occur.

Discontinuities in width and depth are due to the different definitions used to describe the concentration profile. For jet-like flows, the concentration profile is Gaussian and the width and depth are defined to be where the concentration is 1/e (37%) of the centerline concentration. However, for flows where buoyancy-induced lateral spreading dominates, the concentration profile is assumed to be a top-hat profile, and in this case the width and depth definitions are simply defined at the limits of the profile. Figure 5.1 depicts these two width definitions.

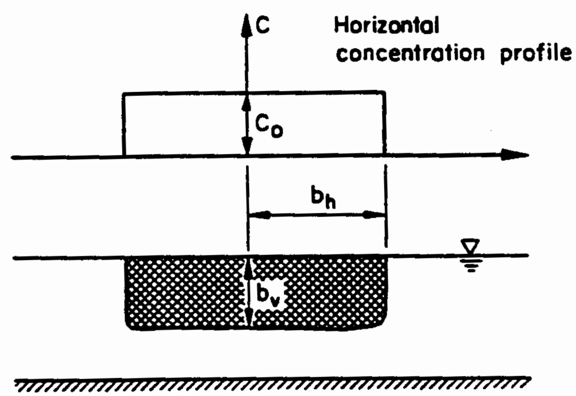
For the passive diffusion region, the concentration profile is assumed to be Gaussian again, only the width and depth of the flow is defined at 46% of the centerline value, which is $(\pi/2)^{1/2}$ times the standard deviation. This width definition was chosen so that it matches the top-hat width definition of the buoyant spreading region while maintaining the same centerline concentration.

5.2.1. Discharge Module (MOD301)

The discharge module is the first module of every flow class protocol. For free jets (flow classes FJ1, FJ2, FJ3, and FJ4) and the 3-dimensional shoreline-attached jet (flow class SA1) it converts the uniform velocity profile of the discharge into a full gaussian velocity profile (see Figure 5.2a). For these flows, the final width and depth of the plume are calculated as:

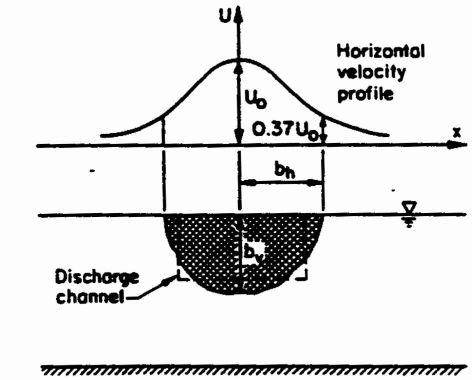


a) Gaussian concentration profile

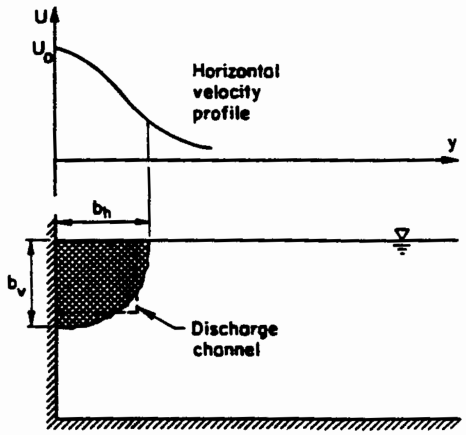


b) Top-hat concentration profile

Figure 5.1 - Gaussian and Top-hat Width Definitions



a) Full gaussian velocity profile



b) Wall jet velocity profiles

Figure 5.2 - Discharge Velocity Profiles for a) Full Gaussian Discharge and b) Wall Jet

$$b_{hf} = b_{vf} = \left(2\frac{a_o}{\pi}\right)^{\frac{1}{2}} \qquad (5.2)$$

For the 3-dimensional wall jet (flow class WJ1), this module converts the discharge velocity profile into a half Gaussian profile (see Figure 5.2b).

$$b_{hf} = b_{vf} = \left(4\frac{a_o}{\pi}\right)^{\frac{1}{2}} \qquad (5.3)$$

For the remaining flows, the depth and width relationships are given simply as:

$$b_{hf} = \frac{b_o}{2} \qquad (5.4)$$

$$b_{vf} = h_o \qquad (5.5)$$

No dilution is assumed to occur in this module, therefore $S_f = 1$ and $C_f = C_o$. The final trajectory coordinates are assumed to be at the origin, $(x_f, y_f) = (0,0)$.

5.2.2. Weakly Deflected Flow Modules

Table 5.3 lists the relationships used in the weakly deflected flow modules. The theoretical development of these relationships are presented in Section 2.3.3.1. They use the local coordinate system for the sake of clarity. All but the wall jet relationships were developed for the two asymptotic cases of $\sigma \approx 90°$ and $\sigma \approx 0°$, which are applied to discharges with $\sigma \geq 45°$ and $\sigma < 45°$ respectively. The constants (T_{311}, S_{311}, etc.) correspond to the constants used in theoretical development found in Chapter II. The values of these constants are listed in Table 5.7.

Modules 311, 312, 314, 315, 317, and 318 all use jet mixing processes, therefore the width and depth (3-dimensional cases only) relationships are based on a Gaussian profile definitions. Module 313 incorporates buoyancy-induced spreading and therefore the width and depth definitions are based on a top-hat profile .

5.2.3. Strongly Deflected Flow Modules

Table 5.4 lists the relationships used in the strongly deflected flow modules. The theoretical development of these relationships are presented in Section 2.3.3.2. They use

Table 5.3

Flow Relationships for Weakly Deflected Region

Modules	Flow Relationships		
311 & 317*	$\sigma \geq 45°$:	$x' = y'\cot\sigma + \frac{y'^2}{T_{311}^2 L_m}$	(5.6)
		$S = \frac{S_{311}}{L_Q}\frac{y'}{\sin\sigma}$	(5.7)
		$b_h = BH_{311}\frac{y'}{\sin\sigma}$	(5.8)
		$b_v = BV_{311}\frac{y'}{\sin\sigma}$	(5.9)
	$\sigma < 45°$	$y' = x'\tan\sigma - \frac{x'^2}{T_{311}^2 L_m}\tan\sigma$	(5.10)
		$S = \frac{S_{311}}{L_Q}\frac{x'}{\cos\sigma}$	(5.11)
		$b_h = BH_{311}\frac{x'}{\cos\sigma}$	(5.12)
		$b_v = BV_{311}\frac{x'}{\cos\sigma}$	(5.13)

* For Module 317 the constants have the subscript 317

Table 5.3(cont.)

Flow Relationships for Weakly Deflected Region

Module	Flow Relationships		
312 & 318*	$\sigma \geq 45°$:	$x' = y'\cot\sigma + \frac{\sin^{\frac{1}{2}}\sigma}{T_{312}\, l_m^{1/2}}\, y'^{\frac{3}{2}}$	(5.14)
		$S = \frac{S_{312}\, y'^{3/2}}{l_q^{1/2}\sin^{1/2}\sigma}$	(5.15)
		$b_h = BH_{312}\frac{y'}{\sin\sigma}\frac{b_v}{b_{vi}}$	(5.16)
		$b_v = Min\ (H_e\ ,\ H_D + y\sin\theta\ )$	(5.17)
	$\sigma < 45°$:	$y' = \frac{\cos^{1/3}\sigma\,\sin\sigma}{T_{312}\, l_m^{1/2}}\, x'^{\frac{2}{3}} - \frac{\sin\sigma}{\cos\sigma}\, x'$	(5.18)
		$S = \frac{S_{312}}{l_q^{1/2}}\frac{x'^{\frac{1}{2}}}{\cos^{1/2}\sigma}\frac{b_v}{b_{vi}}$	(5.19)
		$b_h = BH_{312}\frac{x'}{\cos\sigma}$	(5.20)
		$b_v = Min\ (H_e\ ,\ H_D + y\sin\theta\ )$	(5.21)

* For Module 318 the constants have the subscript 318

Table 5.3(cont.)

Flow Relationships for Weakly Deflected Region

Module	Flow Relationships		
313	$\sigma \geq 45°$:	$x' = y'\cot\sigma + \frac{y'^2}{T_{313}^2 L_m}$	(5.22)
		$S = S_i\left(\frac{b_h}{b_{hi}}\right)^{\beta}$	(5.23)
		$b_h = b_{hi} + \left(\frac{BH_{313}}{(2C_{D43})^{1/2}L_M}\right)^{\frac{2}{3}} \frac{\left(y'^{3/2} - y_i'^{3/2}\right)^{2/3}}{\sin^{2/3}\sigma}$	(5.24)
		$b_v = b_{vi}\left(\frac{b_{hi}}{b_h}\right)\left(\frac{S}{S_i}\right)$	(5.25)
	$\sigma < 45°$:	$y' = x'\tan\sigma - \frac{x'^2}{T_{313}^2 L_m}\tan\sigma$	(5.26)
		$S = S_i\left(\frac{b_h}{b_{hi}}\right)^{\beta}$	(5.27)
		$b_h = b_{hi} + \left(\frac{BH_{313}}{(2C_{D43})^{1/2}L_M}\right)^{\frac{2}{3}} \frac{\left(x'^{3/2} - x_i'^{3/2}\right)^{2/3}}{\cos^{2/3}\sigma}$	(5.28)
		$b_v = b_{vi}\left(\frac{b_{hi}}{b_h}\right)\left(\frac{S}{S_i}\right)$	(5.29)

Table 5.3(cont.)

Flow Relationships for Weakly Deflected Region

Module	Flow Relationships	
314	$S = S_{314} \cos^{1/2}\sigma \frac{x'}{L_Q}$	(5.30)
	$b_h = BH_{314}\, x'$	(5.31)
	$b_v = BV_{314}\, x'$	(5.32)
315	$S = S_{314}\left(\frac{\cos\sigma}{l_q}\right)^{1/2} x'^{\frac{1}{2}}$	(5.33)
	$b_h = BH_{315}\, x'$	(5.34)
	$b_v = H_D$	(5.35)

Table 5.4

Flow Relationships for Strongly Deflected Region

Module	Flow Relationships	
321	$y' = T_{321} L_m^{\frac{2}{3}} \sin^{\frac{1}{3}}\sigma \, x^{\frac{1}{3}}$	(5.36)
	$S = S_{321} \dfrac{y'^2}{L_m L_Q}$	(5.37)
	$b_h = BH_{321} \, y'$	(5.38)
	$b_v = BV_{321} \, y'$	(5.39)
322 & 328*	$y' = T_{322} \, l_m^{\frac{1}{2}} \sin^{\frac{1}{2}}\sigma \, x^{\frac{1}{2}}$	(5.40)
	$S = S_{322} \dfrac{y'}{(l_m l_q)^{1/2}} \left(\dfrac{b_v}{b_{vi}}\right)$	(5.41)
	$b_h = BH_{322} \, y'$	(5.42)
		(5.43)

*For Module 328 the constants have a subscript 328 and the dilution is multiplied by an additional factor of $(b_h+y)/2b_h$.

Table 5.4(cont.)

Flow Relationships for Strongly Deflected Region

Module	Flow Relationships	
323 & 329*	$y' = T_{323} L_m^{\frac{2}{3}} \sin^{\frac{1}{3}}\sigma \, x^{\frac{1}{3}}$	(5.44)
	$S = S_i \left(\dfrac{b_h}{b_{hi}}\right)^{\beta}$	(5.45)
	$b_h = b_{hi} + \left(\dfrac{BH_{323}}{(2C_{D41})^{1/2} L_M}\right)^{\frac{2}{3}} \left(x^{3/2} - x_i^{3/2}\right)^{\frac{2}{3}}$	(5.46)
	$b_v = b_{vi} \left(\dfrac{b_{hi}}{b_h}\right)\left(\dfrac{S}{S_i}\right)$	(5.47)
327	$y' = T_{327} L_m^{\frac{2}{3}} \sin^{\frac{1}{3}}\sigma \, x^{\frac{1}{3}}$	(5.48)
	$S = S_{327} \dfrac{y'^2}{L_m L_Q} \left(\dfrac{b_h+y}{2b_h}\right)$	(5.49)
	$b_h = BH_{327} \, y'$	(5.50)
		(5.51)

*For Module 329 the constants have a subscript 329 and the dilution is multiplied by an additional factor of $(b_h+y)/2b_h$.

the local coordinate system for the sake of clarity. These relationships are not dependent on the initial horizontal discharge angle since these flows are bent over in the strongly deflected regimes. The values of the constants are listed in Table 5.7.

Modules 321, 322, 327, and 328 all use jet mixing processes, therefore the width and depth relationships (3-dimensional cases only) are based on a Gaussian profile. Modules 323 and 329 incorporate buoyant spreading and therefore use the top-hat profile.

5.2.4. Upstream Intruding Modules

Table 5.5 lists the relationships used in the upstream intruding flow modules. The theoretical development of these relationships is presented in Section 2.3.6. These relationships use a control volume approach, therefore, only the initial and final values are computed. The drag coefficients used in these relationships are listed in Table 5.7. The width and depth at the end of this module are defined according to a top-hat concentration profile.

5.2.5. Far-field Modules

Table 5.6 lists the relationships used in the far-field modules. The theoretical development of these relationships are presented in Section 2.4. Since these modules do not use a virtual origin, the relationships are expressed in the global coordinates. The centerline trajectory is a straight line with no lateral progression. However, if the plume interacts with a bank, the centerline shifts over to that bank and the spreading process is limited to one frontal zone. The drag coefficients used in the width expressions are listed in Table 5.7.

5.3. Transition Rules

Transition rules define the extent of each module. The location of the transition from one module to the next depends on both the current module and the succeeding module. Table 5.8 lists the transition rules used in CORMIX3 and the modules for which they apply. The constants for the transition rules are listed in Table 5.9. Except for some possible discontinuities in width and depth, the transition from one module to the next is generally a smooth one.

Some of the transition rules use the primed coordinate system while others use the global coordinate system. It is possible that the location of the transition as determined by the transition rule may already be exceeded, that is, the starting point of the module is farther downstream than the ending point for that module. This means the flow process described by that module is insignificant and therefore the module is skipped.

Table 5.5

Flow Relationships for Upstream Intruding Modules

Module	Flow Relationship
331	$x_s = 4.2\ L_I \left(\frac{L_I}{L_M}\right)^{-2/3} + \frac{b_o}{2} \quad ; \frac{L_I}{L_M} \leq 2.81$ (5.52)
	$x_s = 1.9 + \frac{b_o}{2} \quad ; \frac{L_I}{L_M} > 2.81$ (5.53)
	$S = \frac{0.81}{(\pi C_{D31})^{1/3}} \frac{L_M^{2/3}}{L_Q} L_b^{1/3}$ (5.54)
	$b_f = 4.0\, x_s$ (5.55)
	$b_{vf} = \frac{Q_o S}{b_{hf} u_a}$ (5.56)
332	$S = S_{332} \frac{H_D^{5/3}}{L_M^{2/3} L_Q}$ (5.57) (The remaining equations are the same as for Module 331 shown above)

Table 5.6

Flow Relationships for Far-field Processes

Module	Flow Relationship	
341	$S = S_i \left(\frac{b_h}{b_{hi}} \right)^{\beta}$	(5.58)
	$b_v = b_{vi} \left(\frac{b_h}{b_{hi}} \right)^{\beta - 1}$	(5.59)
	if not bank attached:	
	$b_h = \left[b_i^{3/2} + \frac{3}{2} \frac{1}{(2C_{Dd})^{1/2}} L_b^{1/2} (x - x_i) \right]^{2/3}$	(5.60)
	if bank attached:	
	$b_h = \left[b_i^{3/2} + \frac{3}{2} \frac{1}{(2C_{Dd})^{1/2}} (2L_b)^{1/2} (x - x_i) \right]^{2/3}$	(5.61)

Table 5.6(cont.)

Flow Relationships for Far-field Processes

Module	Flow Relationship	
332	$b_v = \left[\frac{E_v \pi}{u_a} (x - x_1) + b_{v1}^2 \right]^{1/2}$	(5.62)
	if section is unbounded:	
	$b_h = b_{hi} \left[1 + \frac{\pi E_{yo} (x - x_1)}{3 u_a b_{hi}^2} \right]^{3/2}$	(5.63)
	if section is bounded:	
	$b_h = \left[\frac{E_h \pi}{u_a} (x - x_i) + b_{hi}^2 \right]^{1/2}$	(5.64)
	if not attached to bank:	
	$S = \frac{2 b_v b_h}{L_m L_Q}$	(5.65)
	if bank attached:	
	$S = \frac{b_v b_h}{L_m L_Q}$	(5.66)

Table 5.7

Hydrodynamic Simulation Constants

Coefficient(s)	Value	Source(s) or Comments
T_{311}, T_{313}, T_{321}, T_{322}, T_{323}	1.6	A) Adapted form Jirka et al. (1981), and Chu and Jirka (1986)
S_{311}, S_{314}, S_{317}	0.13	B) Adapted from Wright (1977), Fischer et al. (1979), List (1982), Holley and Jirka (1986), and Lee et al. (1987)
BH_{311}, BV_{311}, BH_{314}, BV_{314}, BH_{317}, BV_{317}	0.11	See B
T_{312}	2.7	C) Akar (1990)
S_{312}, S_{318}	0.58	See C
BH_{312}, BH_{315}, BH_{318}	0.14	See C
BH_{313}, BH_{323}, BH_{329}	0.031	From near-field buoyant spreading analysis (Section 2.3.2.2)
S_{315}	0.41	Adapted from Holley and Jirka (1986)
T_{317}, T_{327}, T_{328}, T_{329}	0.8	D) From shoreline attached flow trajectory analysis (Section 2.3.5)
T_{318}	1.9	See D
S_{321}, S_{322}, S_{327}, S_{328}	0.15	See B
BH_{321}, BV_{321}, BH_{322}, BH_{327}, BV_{327}, BH_{328}	0.30	See B
S_{332}	0.9	E) Jones et al. (1982), and Chu and Jirka (1981)

Table 5.7(cont.)

Hydrodynamic Simulation Constants

Coefficient(s)	Value	Source(s) or Comments
C_{D13}	0.5	See E
C_{D31}	0.8	See E
C_{D32}	0.9	See E
C_{D41}	2.0	F) Simpson (1982), and Jirka and Arita (1987)
β	0.25	See F

Table 5.8

Transition Rules

TR*	Current Module	Next Module	Equation**
11	311	312	$b_{v,max} = H_a$: $y_f' = \frac{H_a \sin\sigma}{BV_{311}}$ $b_{v,max} = H_D + y \sin\theta$: $y_f' = \frac{H_D - (BV_{311}/\sin\sigma) y_v}{(BV_{311}/\sin\sigma) + \sin\theta} - y_v$
10 13 15 18 19	312 311 313 317 318	322 321 323 327 328	$\sigma \geq 45°$: $y_f' = TC_{3xx}L_m$ $\sigma < 45$: $x_f' = TC_{3xx}L_m$
12 14	311 312	313 313	$\sigma \geq 45$: $y_f' = TC_{3xxA}L_M \sin\sigma$ $\sigma < 45$: $x_f' = TC_{3xxA}L_M \cos\sigma$
16 17	314 315	341 341	$x_f' = TC_{3xx}L_M$
20	314	315	$x_f' = \frac{H_D}{BV_{314}}$
21	321	322	$b_{v,max} = H_a$: $y_f' = \frac{H_a}{BV_{321}}$ $b_{v,max} = H_D + \sin\theta$: $y_f = \frac{H_D + y_v \sin\theta}{\sin\theta + BV_{321}}$

* TR = Transition Rule
** For transition rule constants TC_{3xx}, xx stands for the transition rule (TR)

Table 5.8(cont.)

Transition Rules

TR*	Current Module	Next Module	Equation**
22 23 25	321 322 327	323 323 329	$y_f' = TC_{3xx}L_M$
24	323	341	$y_f' = TC_{3xx}L_m$
26	328	341	$x_f = \frac{7 l_m}{0.556 + 1.7 (l_m/BS)}$
27	329	341	$x_f' = TC_{3xx}L_Q$
31	331 332	341 341	$x_f' = \frac{1}{2} b_{v,z=0}$
41	341	351	$x_f = x_i + \frac{2}{3}\left(\frac{2C_{Dd}}{L_b}\right)^{\frac{1}{2}}\left\{\left(\frac{8L_y h_o}{f L_m L_Q S_i}\right)^{\frac{3}{2}} - 1\right\} b_i^{\frac{3}{2}}$

* TR = Transition Rule
** For transition rule constants TC_{3xx}, xx stands for the transition rule (TR)

Table 5.9

Transition Rule Constants

TR Constants	Value	Source(s) or Comments
TC_{312A}, TC_{314A}, TC_{316}, TC_{317}, TC_{322}, TC_{323}	3.8	A) Criteria for maximum surface jet depth as per Jirka et al. (1981)
TC_{312B}, TC_{314B}	2.7	See A
TC_{313A}, TC_{315A}	2.0	B) From Chu and Jirka (1986) trajectory data analysis (Figure 2.10)
TC_{313B}, TC_{315B}, TC_{315C}, TC_{318A}, TC_{318B}, TC_{319A}, TC_{319B}	1.0	See B
TC_{324}	3.0	Minimum penetration into channel as per Akar (1990)
TC_{325}	1.9	Fromm analysis of shoreline attaced flow trajectories (Section 2.3.5)
TC_{327}	5.0	Wake attachment criteria as per Doneker and Jirka (1990)

Chapter VI

System Verification

In this section CORMIX3 results are compared with various laboratory experiments and field studies. It is not intended as a comprehensive validation of CORMIX3 with all available data, but rather a test of the more common flow class simulations. These comparisons illustrate the flexibility of CORMIX3 to provide reasonable predictions for various prevalent flow configurations.

CORMIX3 considers many possible flow configurations. Unfortunately, studies on buoyant surface jets are very limited and laboratory experiments of many of the flow classes do not exist in a form lending themselves to quantitative comparisons. For the more basic flow configurations, sample cases have been chosen which typify the CORMIX3 predictive capabilities.

This chapter consists of two sections. Section 6.1 discusses several comparisons between laboratory experiments and CORMIX3 predictions. Section 6.2 provides comparisons between three field studies and the corresponding CORMIX3 predictions.

6.1. Comparison with Laboratory Data

The correlation of CORMIX3 predictions and laboratory experiments is divided into two subsections. The first subsection, Section 6.1.1, provides results of buoyant surface jet experiments in stagnant environments. The second subsection, Section 6.1.2, discusses the results of buoyant surface discharges into ambient crossflows.

6.1.1. Discharge into Stagnant Environments

CORMIX3 predictions of stagnant cases are compared with experiments run by Wolanski and Koh (1973) and Hayashi and Shuto (1967). The experiments of Wolanski and Koh illustrate the effect of buoyancy on the lateral spreading of a surface jet in the near-field. The Hayashi and Shuto experiment further exemplifies the predictability of the near-field flow and the unsteadiness of the region beyond the near-field.

The two tests by Wolanski and Koh depicted in Figures 6.1a and 6.1b differ only in the buoyancy of the discharge. Figure 6.1a shows a non-buoyant discharge while Figure 6.1b is for a surface jet with a density significantly less than the ambient water. It is obvious from these two experiments that buoyancy greatly increases the lateral spreading of the jet. Note the non-buoyant jet spreads at a relatively constant and weak

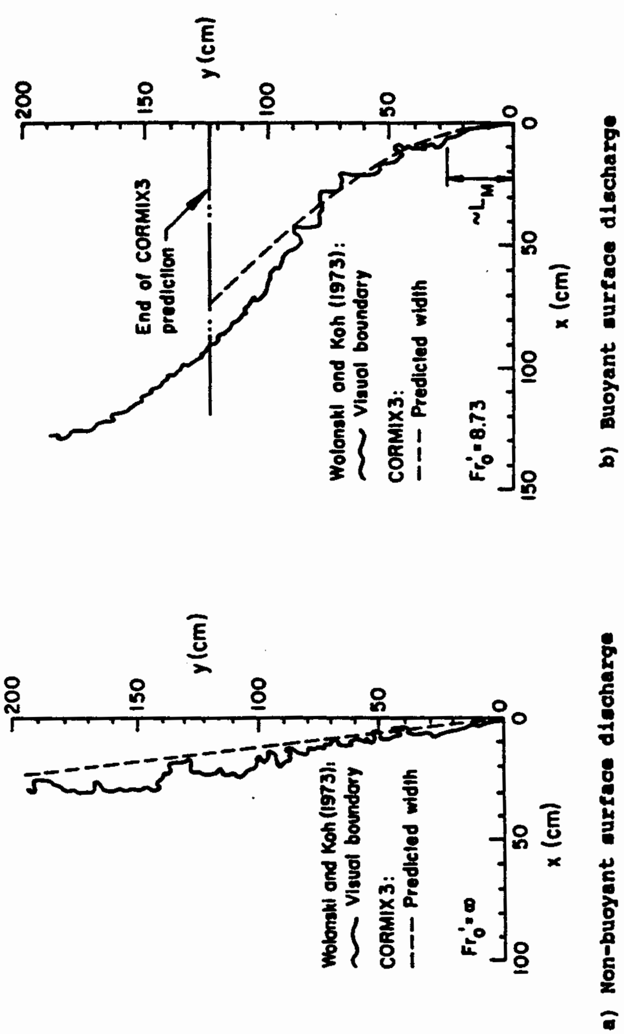


Figure 6.1 - Comparison between non-buoyant and buoyant discharges in a stagnant environment, and the corresponding CORMIX3 predictions (from Wolanski and Koh, 1973)

rate while the buoyant jet spreads at a much larger non-linear rate. In both cases, CORMIX3 predicts the width of the jet well.

Figure 6.2 shows centerline temperature measurements of the buoyant surface jet experiment by Wolanski and Koh. Note the large initial dilution near the source. This is attributed to the relatively large jet-mixing near the discharge. Further from the source, the rate of mixing decreases as the buoyant force suppresses the vertical mixing. There is good agreement between the CORMIX3 prediction and the observed centerline temperature.

The comparison to Hayashi and Shuto's experiment shown in Figure 6.3 illustrates the effect of the buoyancy-induced spreading in the near-field and provides support for predictive capabilities of CORMIX3 for this type of flow. Note that up to the transition distance y_t, a steady-state jet is formed. However, beyond this transition distance an unsteady buoyant pool of effluent is formed that continually spreads outward. CORMIX3 predicts the lateral spreading in the near-field case well, but does not deal with the unsteady far-field of the flow.

6.1.2. Buoyant Surface Discharge into a Crossflow

Comparisons of CORMIX3 predictions with laboratory experiments of surface buoyant jets in a crossflow will be divided according to three of the flow categories: free jets, shoreline attached jets, and upstream intruding plumes.

6.1.2.1. Free Jet Comparisons

Free jet comparisons are made with data from two investigators: The Delft Hydraulics Laboratory (1983) and Motz and Benedict (1970). The two Delft experiments illustrate the difference in flow trajectories and dilutions if the velocity ratio is kept constant and the discharge temperature is varied. On the other hand, the Motz and Benedict experiments both have the same discharge temperature but have different discharge velocities.

Figures 6.4a and 6.4b show the surface isotherms of the two experiments run at the Delft Hydraulics Laboratory. For both the experiments, the velocity ratio was maintained at 10. The experiment depicted in Figure 6.4a had a higher discharge temperature than the experiment in Figure 6.4b and, as a result, penetrates slightly further into the ambient current due to increased thinning of the plume.

In both cases, CORMIX3 slightly underpredicts the penetration into the crossflow. This may be caused by not accounting for the velocity profile of the ambient flow properly. Because of the no-slip condition at the basin wall, the ambient flow velocity

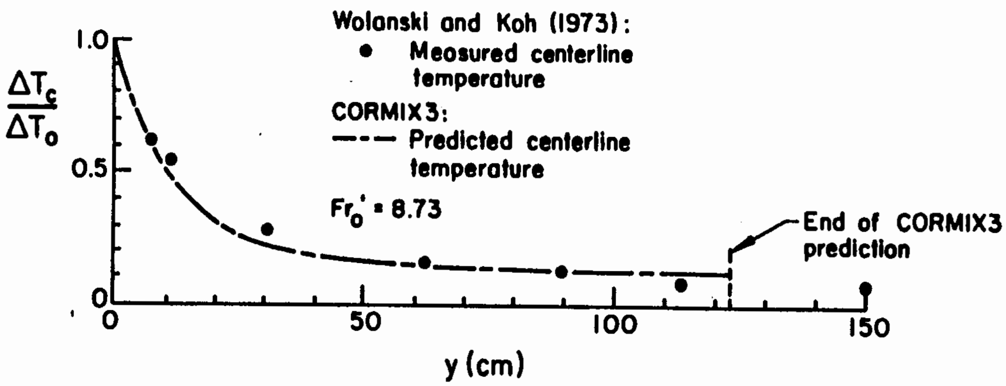


Figure 6.2 - Centerline temperature measurements for the buoyant surface discharge experiment by Wolanki and Koh (1973) shown in Figure 6.1b

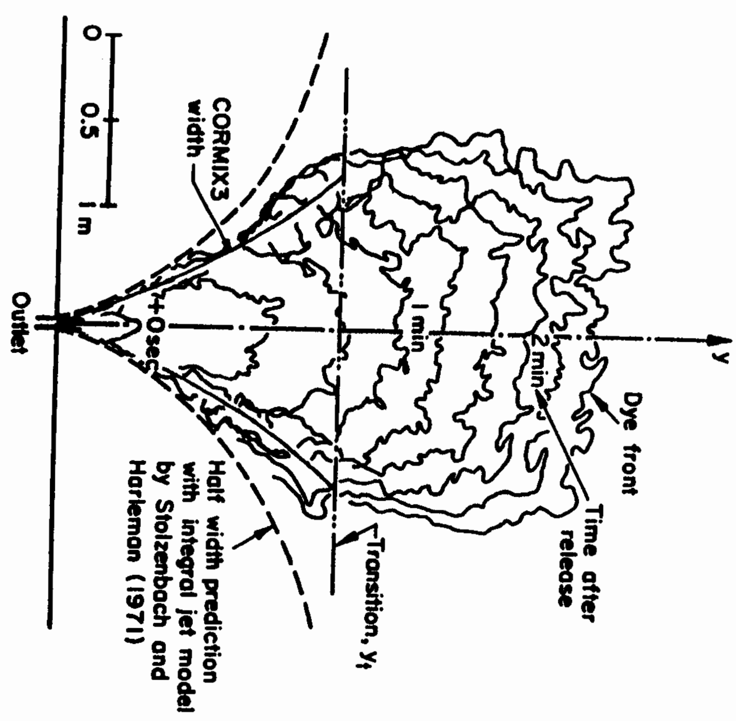


Figure 6.3 - Visual outline of a buoyant surface jet in a stagnant environment and the corresponding CORMIX3 prediction (from Hayashi and Shuto, 1967)

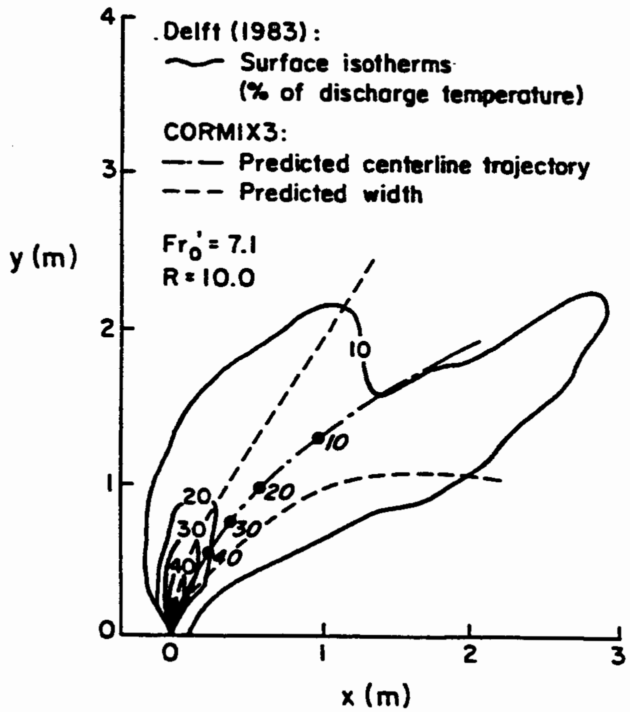


Figure 6.4a - Comparison between CORMIX3 predictions and a surface buoyant free jet experiment (from the Delft Hydraulics Laboratory, 1983)

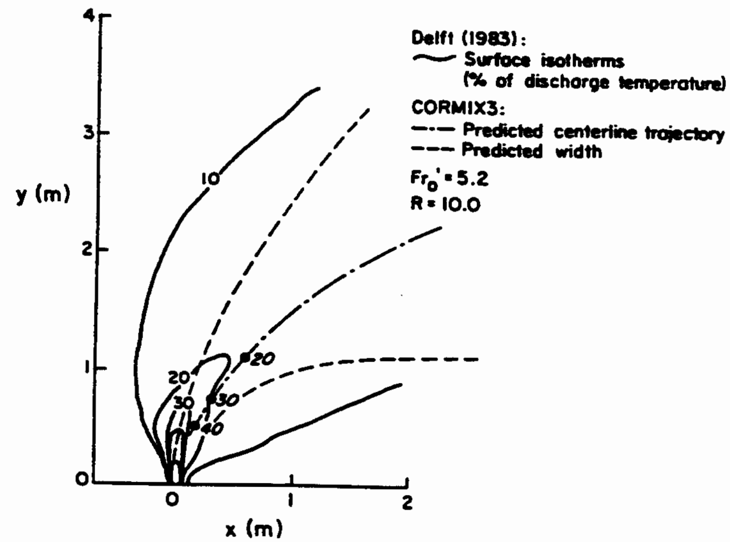


Figure 6.4b - Comparison between CORMIX3 predictions and a surface buoyant free jet experiment (from the Delft Hydraulics Laboratory, 1983)

near the wall may be significantly lower than the average ambient flow velocity. In addition, the width predictions near the source tend to be too low. However, this error becomes much less further from the source.

Figures 6.5a and 6.5b show the centerline temperature measurements for the two Delft experiments. Note that the experiment with the higher discharge buoyancy shown in Figure 6.5a has slightly lower dilution which may be attributed to increased suppression of the vertical mixing caused by the larger buoyancy force. The slight effect of increased buoyancy on both trajectory and dilution is reflected in the CORMIX3 predictions which, for both experiments, show reasonably good agreement.

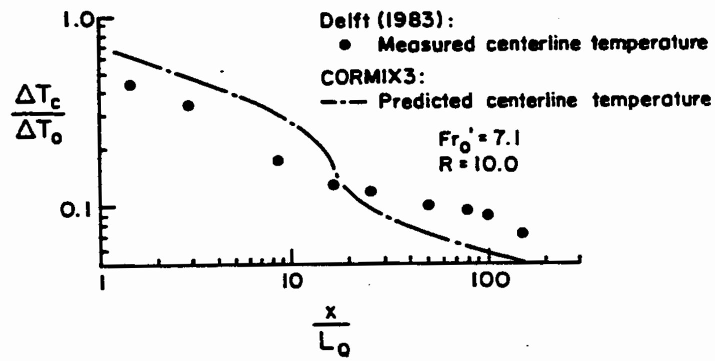
Figure 6.6 show the measured centerline trajectories for the Motz and Benedict experiments. Figures 6.7a and 6.7b depict the corresponding centerline temperature measurements. These two experiments were run with the same initial temperature difference, but with different discharge velocities. As expected, the flow with the higher discharge velocity penetrates much farther into the ambient crossflow. CORMIX3 shows good agreement with both the trajectory and temperature data.

6.1.2.2. Shoreline Attached Jet Comparisons

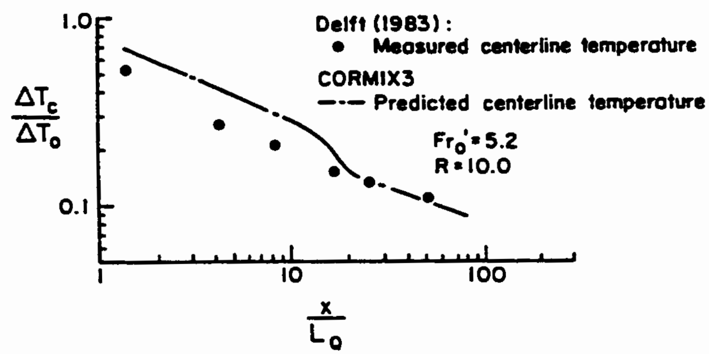
CORMIX3 predictions of shoreline attached jets are compared with experiments run by the Delft Hydraulics Laboratory (1983) and Abdelwahed and Chu (1981). The two experiments by the Delft Hydraulics Laboratory illustrate the effect of varying the ambient velocity while keeping the discharge velocity and buoyancy constant. The two Abdelwahed and Chu results are from experiments for which several flow parameters were varied.

Figures 6.8a and 6.8b show the surface isotherms of the two Delft experiments. Note the significant effect on the trajectory of the plumes caused by increasing the velocity ratio. The discharge with a higher velocity ratio shown in Figure 6.8a penetrates further into the crossflow, yet retains a zone of recirculation along the shore in the near-field. The CORMIX3 predictions for this experiment compares quite well with the experimental results. The CORMIX3 trajectory prediction for the experiment with a lower velocity ratio shown in Figure 6.8b overpredicts the lateral penetration into the crossflow. However, the lateral penetration is within approximately 30% of the measured trajectory which is reasonable considering the complexity of the flow and the range of experimental error.

Figures 6.9a and 6.9b show the centerline temperature measurements and the corresponding CORMIX3 predictions. Again discrepancies between the CORMIX3 predictions and the measured values are acceptable for such complicated flows.



a) Temperature Measurements for Figure 6.4a



a) Temperature Measurements for Figure 6.4b

Figure 6.5 - Centerline temperature measurements and the corresponding CORMIX3 predictions for the Delft free jet experiments in Figure 6.4 (from the Delft Hydraulics Laboratory, 1983)

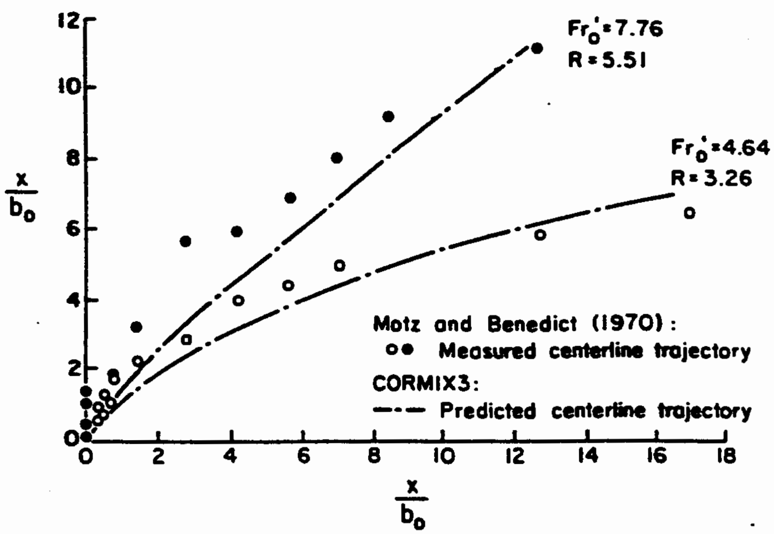
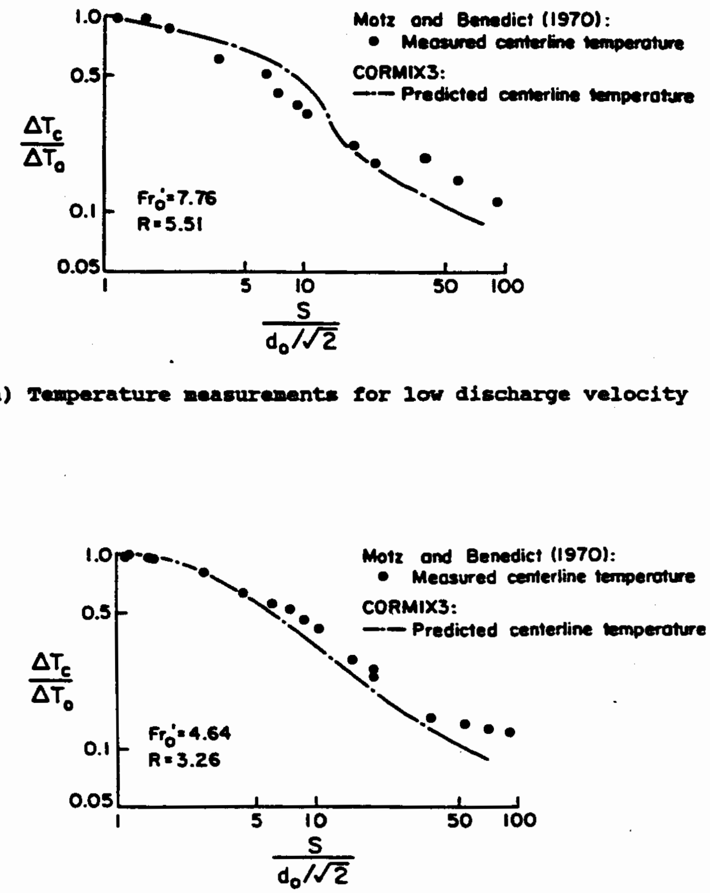


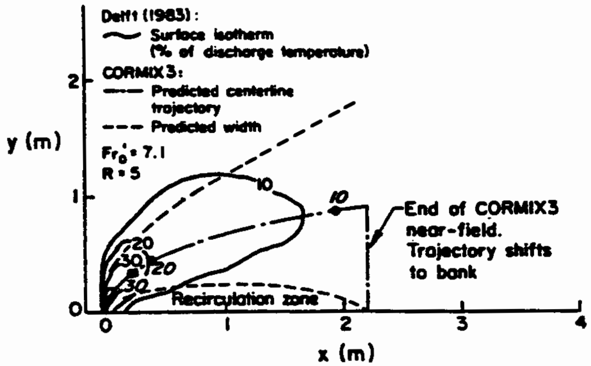
Figure 6.6 - Measured free jet trajectories and the corresponding CORMIX3 predictions (from Motz and Benedict, 1970)



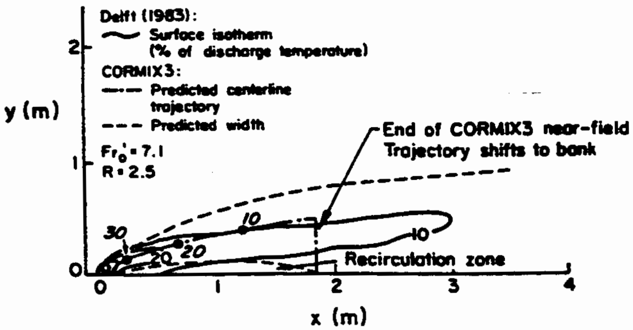
a) Temperature measurements for low discharge velocity

b) Temperature measurements for high discharge velocity

Figure 6.7 - Centerline temperature measurements and the corresponding CORMIX3 predictions for the free jet experiments in Figure 6.6 (from Motz and Benedict, 1970)

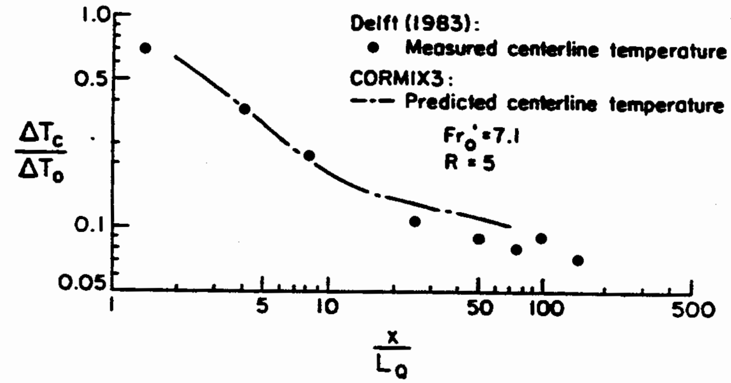


a) High velocity ratio

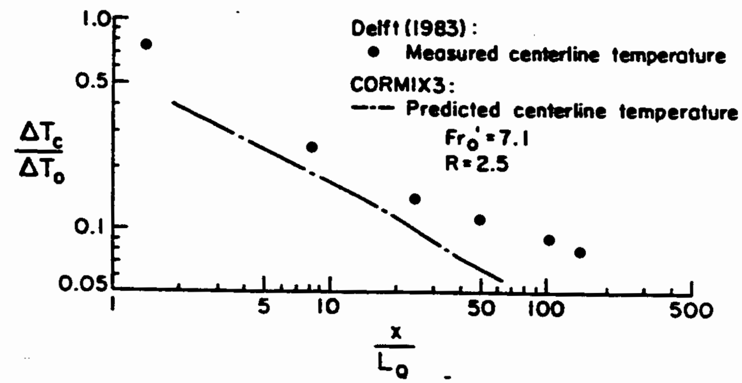


b) Low velocity ratio

Figure 6.8 - Isotherms of shoreline attached flows and the corresponding CORMIX3 predictions (from the Delft Hydraulics Laboratory, 1983)



a) Temperature measurements for the high velocity ratio experiment in Figure 6.8a



b) Temperature measurements for the low velocity ratio experiment in Figure 6.8b

Figure 6.9 - Centerline temperature measurements and the corresponding CORMIX3 predictions for the shoreline attached experiments in Figure 6.8 (from the Delft Hydraulics Laboratory, 1983)

Figures 6.10a and 6.10b present the visual outline of the Abdelwahed and Chu experiments. These experiments have different discharge velocities, initial buoyancy, and ambient current velocities. Although the interaction between these parameters is complex, one relationship can be observed. The run with the lower Froude number shown in Figure 6.10a has a larger width due to the increased lateral spreading. Note that CORMIX3 satisfactorily predicts the trajectory and width of these flows.

For all the shoreline flows, a recirculation zone exists along the downstream bank. Although CORMIX3 does not predict the width of these zones, they made be estimated using the jet width in the near-field. The recirculation zone extends the length of the near-field. This is illustrated in Figures 6.8a, 6.8b, 6.10a, and 6.10b.

6.1.2.3. Upstream-intruding Plume Comparisons

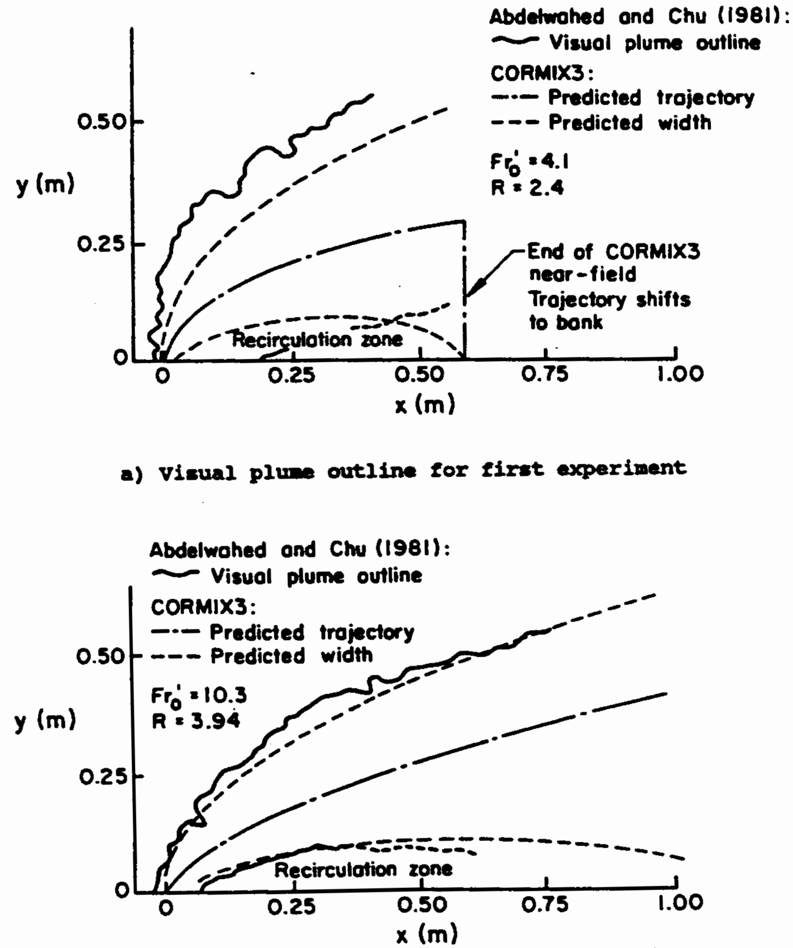
The two upstream-intruding plume experiments discussed in this section were performed by Huq (1983) and Kuhlman and Prahl (1974). Huq performed radial surface discharge experiments similar to those used by Jones et al. (1985) whose theory for upstream intruding plumes has been adopted for use in CORMIX3. Kuhlman and Prahl used a side discharge channel as is assumed for CORMIX3.

Figure 6.11 shows the visual outline and isotherms of Huq's radial discharge experiment. Since CORMIX3 uses a control volume approach in the near-field, only a few key characteristics of the flow are calculated for the near-field, including the extent of upstream intrusion and the width of the plume at the discharge. The plume outline as predicted by CORMIX3 is interpolated between these two values. As shown in Figure 6.11, these two geometric parameters can be used to adequately describe the geometry of the upstream intrusion.

The comparison with the Kuhlman and Prahl experiment is shown in Figure 6.12. CORMIX3 appears to provide reasonable predictions of the upstream intrusion. The apparent overprediction of the width is most likely due to the limited width of the laboratory receiving water which reduces the horizontal penetration of the flow. In the CORMIX3 simulation output, the user is warned that the far-field prediction may be inaccurate because of the interaction of the plume with the far bank in the near-field.

6.2. Comparisons with Field Studies

Three field studies were chosen to test CORMIX3 against actual discharge situations and to illustrate the applicability of CORMIX3 to realistic conditions. The first field study was performed at the Point Beach Nuclear Power Plant located on the Wisconsin shoreline of Lake Michigan. The second was conducted for the Palisades Nuclear Power Plant, which also discharges into Lake Michigan but from the Michigan



a) Visual plume outline for first experiment

b) Visual plume outline for second experiment

Figure 6.10 - Visual plume outlines for shoreline attached experiments and the corresponding CORMIX3 predictions (from Abdelwahed and Chu, 1981)

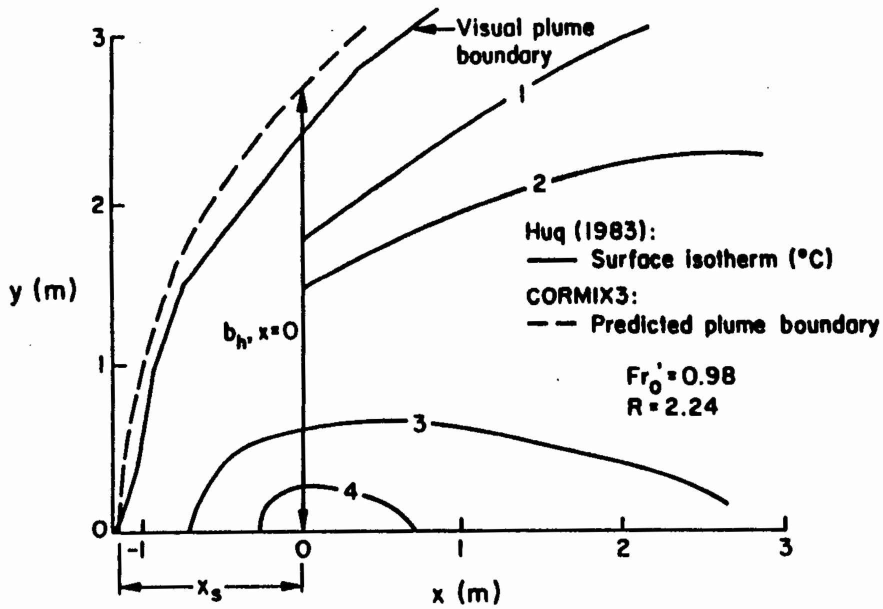


Figure 6.11 - Isotherms of an upstream intruding plume resulting from a radial discharge experiment (from Huq, 1983)

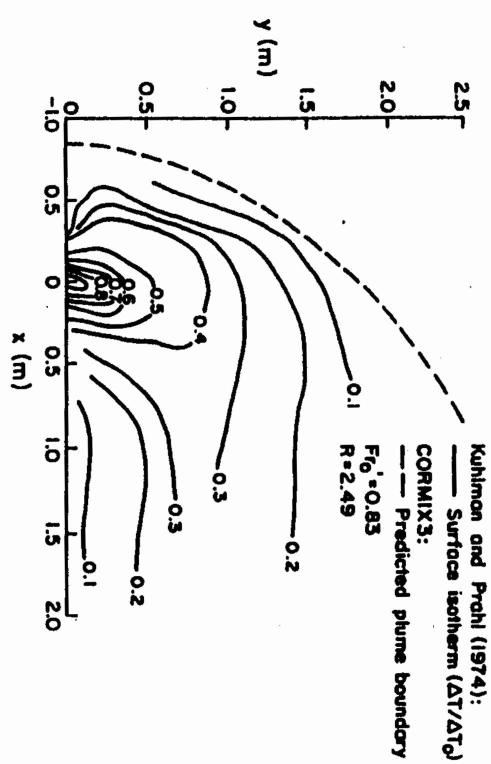


Figure 6.12 - Isotherms of an upstream intruding plume experiment resulting from a channel discharge (from Kuhlman and Prahl, 1974)

shore. The third case illustrates the use of CORMIX3 for predicting a riverine discharge by being applied to the flow of the Connecticut River into the Long Island Sound.

6.2.1. Point Beach Nuclear Power Plant

The Point Beach Nuclear Power Plant is located on the western shore of Lake Michigan. It discharges an average of 25.1 m^3/s of water used to cool condensers in the plant through one of two flumes that protrude into the lake about 55 m at angles of 60° and 120°. The heated water is discharged at an excess temperature of 8.3°C into a shallow ambient environment. The width of the discharge flumes are 10.7 m and the depth of flow in the flumes during this study was 4.0 m.

The ambient current in this study was reported as 0 m/s. However, it is rare to have completely stagnant conditions in a lake as large as Lake Michigan. By observing the isotherms shown in Figure 6.13, it appears that there might have been a small crosscurrent. Since most current meters have a minimum threshold of detection of approximately 5 cm/s, it is reasonable to suspect that a current with the magnitude of several cm/s existed. For this reason, and to get a complete prediction of near and far-field regions, a ambient current of 3 cm/s was used in the CORMIX3 prediction.

As seen in Figure 6.13, CORMIX3 shows adequate agreement with the measured field data. Due to the shallow receiving waters, the case is classified as SA2. The slight underprediction of the trajectory may be due to overestimating the ambient current. The temperature predictions are in good agreement, with the largest discrepancy between measured and predicted centerline values being about 1°C. The width prediction also predicts well the large buoyant spreading that can be observed.

6.2.2. Palisades Nuclear Power Plant

The Palisades Nuclear Power Plant is located on the eastern shore of Lake Michigan and discharges approximately 25.5 m^3/s of heated effluent into a shallow ambient environment. In this case, cooling water is discharged through a 28.3 m wide man-made channel at a depth of 2.1 m into a relatively strong ambient current of 0.7 m/s. The channel enters along a straight shoreline at an angle of approximately 90°C. The temperature excess of the effluent was measured at 9.3°C.

Figure 6.14 shows the measured isotherms of the discharge and the CORMIX3 prediction. In this case, the flow is very strongly attached to the downstream bank which reduces the dilution of the heated effluent. CORMIX3 predicts both the geometry of the flow and the surface temperature fairly well.

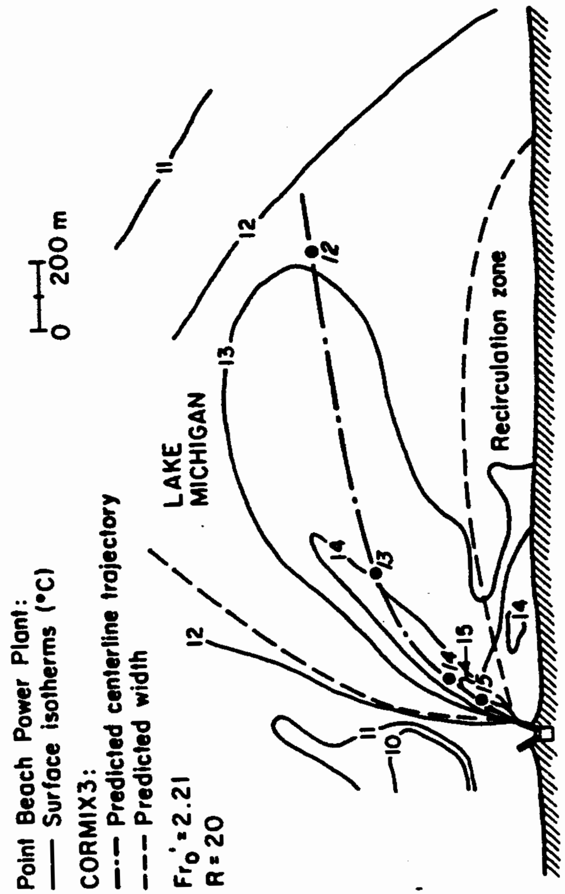


Figure 6.13 - Surface isotherms of the cooling water discharge from the Point Beach Power Plant, and the corresponding CORMIX3 predictions (from Dunn et al, 1975)

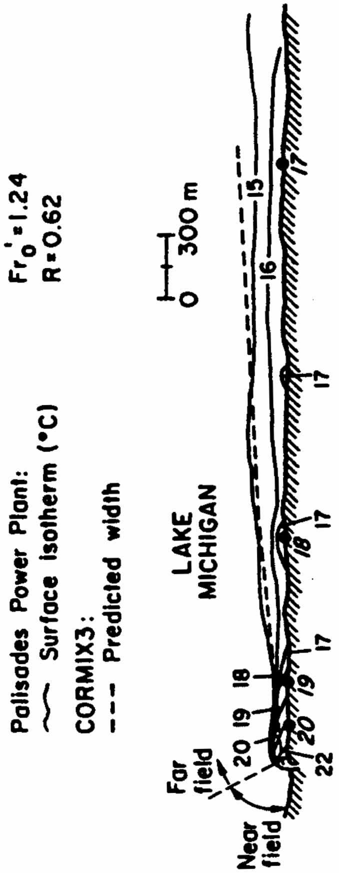


Figure 6.14 - Surface isotherms of the cooling water discharge from the Palisades Nuclear Power Plant, and the corresponding CORMIX3 predictions (from Miller and Brighouse, 1985)

6.2.3. Connecticut River

The study of the Connecticut River discharge into Long Island Sound is described by Garvine (1974). The mouth of the Connecticut River is approximately 1300 m wide and the flow during this study was about 1.8 m deep. This gives an aspect ratio much larger than that allowed by CORMIX3. Therefore, as an approximation of the situation, the discharge area and velocity was kept the same, but the width was reduced to 218.7 m and the depth was increased to 11 m. It is expected that this schematization of the discharge channel has not effected the results to any great extent since reasonable predictions were obtained.

The fresh water of the Connecticut River enters the Sound with a velocity of 0.67 m/s, approximately the same velocity as the ambient current. The discharge is positively buoyant due to the difference between the fresh water of the Connecticut River and the saline ambient environment.

Figure 6.15 shows the CORMIX3 predictions as compared to the measured values of the plume outline as reported by Jones et al. (1985). Although the model appears to slightly overpredict the spreading of the plume, the discrepancies are well within the range of error of the field measurements.

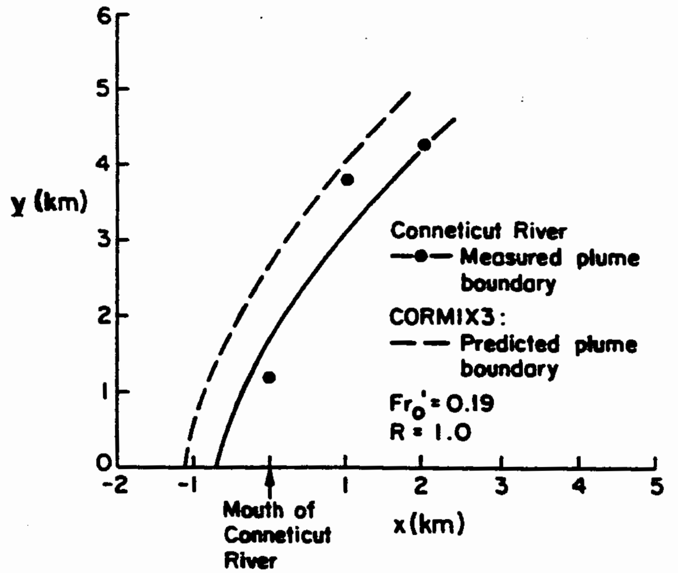


Figure 6.15 - Connecticut River plume measurements and the corresponding CORMIX3 predictions (from Jones et al., 1985)

Chapter VII

Applications of CORMIX3

The first section of this chapter, Section 7.1, illustrates the use of CORMIX3 with a hypothetical situation involving the discharge of heated effluent from a power plant into a tidal estuary. It exemplifies the schematization of the ambient and discharge conditions, specification of legal mixing zones, and interpretation of the numerical simulation results. In the second section, Section 7.2, some comments on the applications and limitations of CORMIX3 are discussed.

7.1. AB Power Plant Discharge Analysis

This example considers the effect of tidal variation on the expected trajectory, dilution, and geometric characteristics of the effluent flow. The discharger is the AB Municipal Power Plant, whose responsibility is to assure the minimization of any adverse affects arising from the warm water discharge on the estuarine environment. First, the parameters of the problem are defined in Section 7.1.1 for the two cases in consideration, low tide and high tide. In Section 7.1.2 the use of CORMIX3 to analyze these scenarios is discussed.

7.1.1. Problem Statement

The AB Power Plant discharges a constant flow of 60 m^3/s of heated water into a tidal estuary as shown in Figure 7.1. The temperature of the effluent is 26°C. The discharge channel is 10 m in width and the depth of flow at the mouth of the channel varies from 2.2 m to 2.5 m from low to high tide, respectively. A detailed cross-section of the outlet of the discharge channel is shown in Figure 7.2.

The estuary is narrow in the region where the plant is located, being only 3 km in width (see Figure 7.3). During low tide, the current reaches a maximum of 0.30 m/s in the downstream direction. However, as high tide approaches, the flow reverses and reaches a velocity of 0.025 m/s in the opposite direction (see Figure 7.1). The average depth of the estuary varies from 7.7 m during low tide to 8 m during high tide. The average ambient temperature is 10°C. A typical Manning's n roughness for the estuary is 0.02.

As seen in Figure 7.1, the intake of water for the plant is located just upstream of the discharge, and it is a primary objective of the plant operators to prevent any discharged effluent from being drawn into the intake channel. In addition, state law

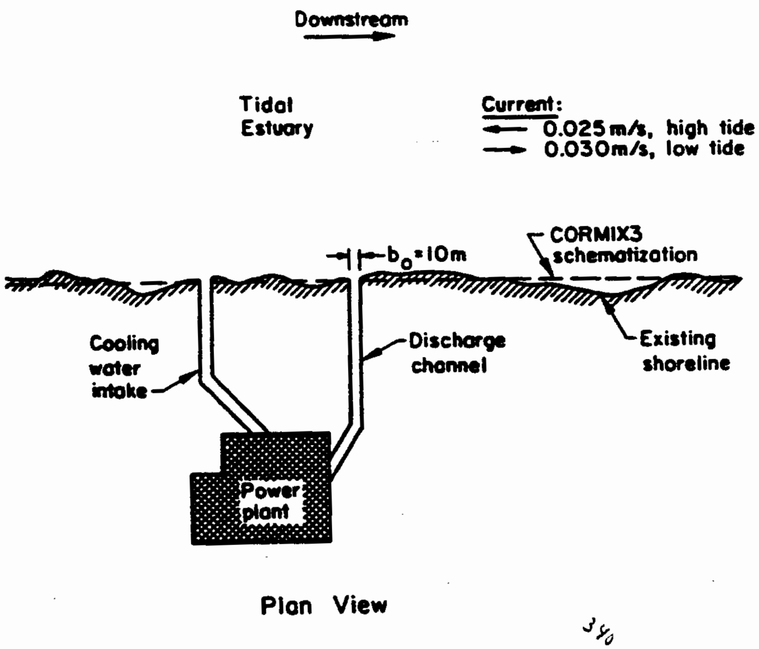


Figure 7.1 - Plan view of the design case discharge area.

requires that the temperature rise above the natural ambient temperature cannot exceed 4°C outside a radius of 300 m from the discharge point.

7.1.2. CORMIX3 Analysis

Two scenarios will be considered separately. First, low tide conditions are examined in Section 7.1.2.1, and then the high tide predictions are discussed in Section 7.1.2.2.

7.1.2.1. Low Tide Scenario

First, the ambient and discharge geometry must be simplified. Figures 7.1, 7.2, and 7.3 shows how this case can be approximated for CORMIX3 input. Note that generally, ambient conditions are highly variable and that simplification of this type may not be readily apparent. In all cases, CORMIX3 must be used iteratively, varying the ambient and discharge schematizations to observe the sensitivity to various input parameters.

Once the appropriate input is entered, CORMIX3 executes the PARAM3 and CLASS3 elements sequentially. In this case, the discharge is classified as SA2. As described in CLASS3, the SA2 flow class is fully vertically mixed in the near-field and recirculation zone exists along the downstream shoreline. In the far-field, the flow restratifies and spreads laterally due to buoyancy forces. In the HYDRO3 element, the actual mathematical simulation is performed. As shown in Figure 7.4, the recirculation region extends approximately 3.5 km downstream to the point at which the flow restratifies. A vertical downstream cross-section of the flow is shown in Figure 7.5.

In this case, the State standard of $\Delta T = 4°C$ is not met at the edge of the mixing zone. As seen in Figure 7.4, the temperature at 300 m from the source is 16°C, giving a temperature difference of 6°C. This is due to the recirculation of the heated effluent along the bank and the speed in which the ambient current carries it downstream outside the legal mixing zone. However, the flow is unlikely to interfere with the intake.

The summary element of CORMIX3, SUM3, gives suggestions on how to improve initial dilution to meet the legal mixing zone requirements. However, since there cannot be universally applicable rules for increasing dilution characteristics, the analyst must use an iterative approach to find the most effective way to optimize the design. Only by changing the parameters and observing the effects on the results can the user be assured of the correct flow simulation and optimization.

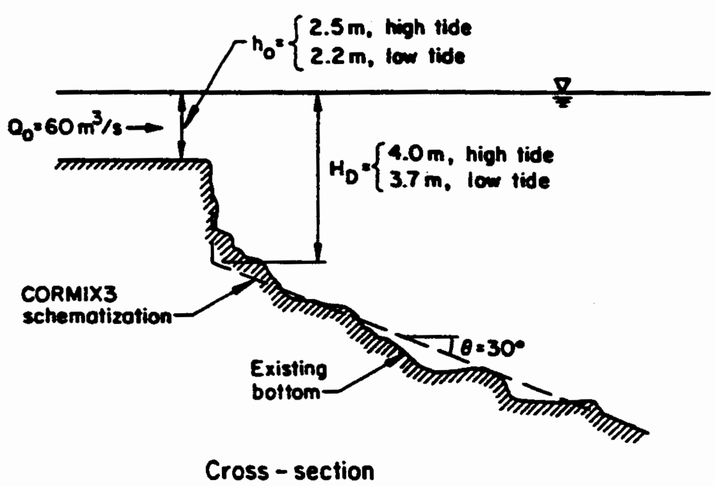


Figure 7.2 - Detailed cross-section of the discharge channel and immediate viscinity. Also includes schematization used for CORMIX3 analysis.

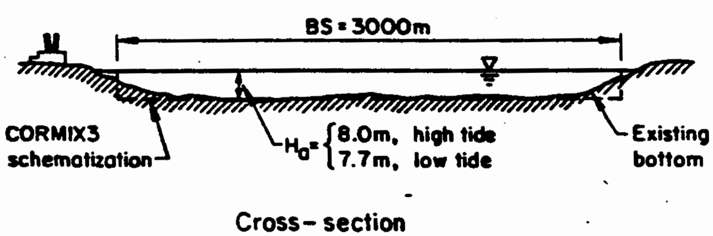


Figure 7.3 - Cross-section of the receiving tidal estuary and the schematization used for CORMIX3 analysis.

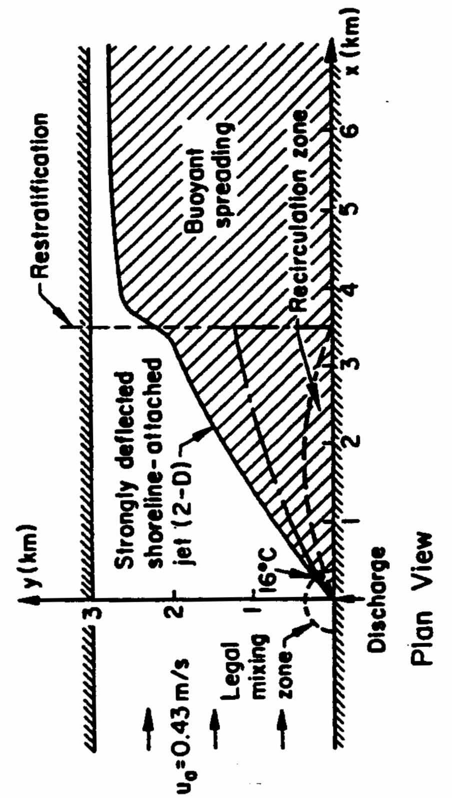


Figure 7.4 - Plan view of the CORMIX3 predictions for the low tide scenario of the design case.

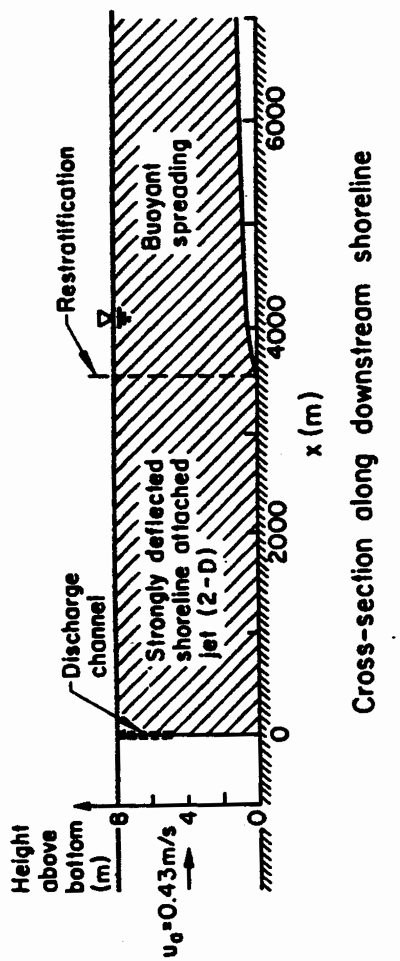


Figure 7.5 - Cross-section of the CORMIX3 predictions for the low tide scenario of the design case.

7.1.2.2. High Tide Scenario

The high tide case employs the same discharge and ambient schematization as for the low tide scenario shown in Figures 7.1, 7.2, and 7.3, but changes the ambient flow velocity, ambient depth, and discharge depth. As a result, the flow is classified as an FJ3. According to the flow description given in CLASS3, the effluent stream will interact with the bottom in the near-field, then restratify and spread laterally as it is deflected by the crossflow. The flow class FJ3 also shows no interaction with the near shoreline.

In running the hydrodynamic simulation, the analyst notes the flow interacts with the opposite bank quite strongly. This is shown in the plan view as illustrated in Figure 7.6. The crossflow is very weak, therefore the flow is only slightly deflected. Strong lateral spreading occurs in the near-field as shown in Figure 7.6. Figure 7.7 shows a cross-section along the flow trajectory centerline as obtained from the mathematical simulation. The interaction with the bottom occurs only 31 m from the discharge, and buoyant restratification occurs 218 m from the source. CORMIX3 ends the simulation at the point of interaction with the opposite bank.

For this case, the flow meets the State criteria at the edge of the legal mixing zone. As noted on Figure 7.4, the temperature difference at 300 m from the source is 13.2°C, giving a temperature rise of 3.2°C above the ambient temperature. This is below the 4°C limit set by the State. In addition, it appears that the flow is not likely to interfere with the intake.

The interaction of the plume with the opposite shoreline may be of concern, depending on the regulations that apply and the activities occurring there. For example, if a public beach were located on the opposite shore, it is likely that stricter water quality standards would have to be met, and alternative discharge designs may have to be considered.

7.1.3. Comments

The approach used to analyze this tidal situation assumes "quasi-steady" conditions exist. That is, although the ambient conditions change with time, the time-scale of the tidal variations are much larger than the time-scale of the near-field mixing processes, therefore, each scenario is assumed to be independent of the previous conditions. One should note, however, that in general this is not true, especially for far-field predictions. For further discussion on the effects of tidal variations on surface discharges, see Jirka et al. (1975).

The importance of iterative use of CORMIX3 must also be stressed. In this example, as with most discharge situations, significantly different flow patterns are predicted with changes in the ambient conditions. Since ambient and discharge conditions

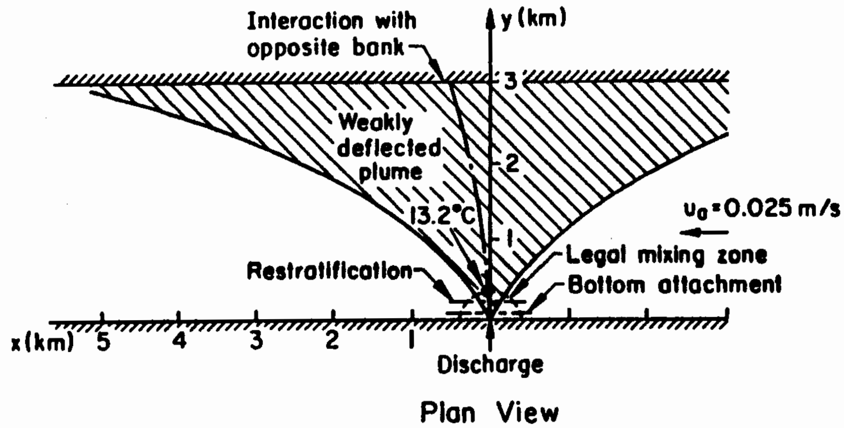


Figure 7.6 - Plan view of the CORMIX3 predictions for the high tide scenario of the design case.

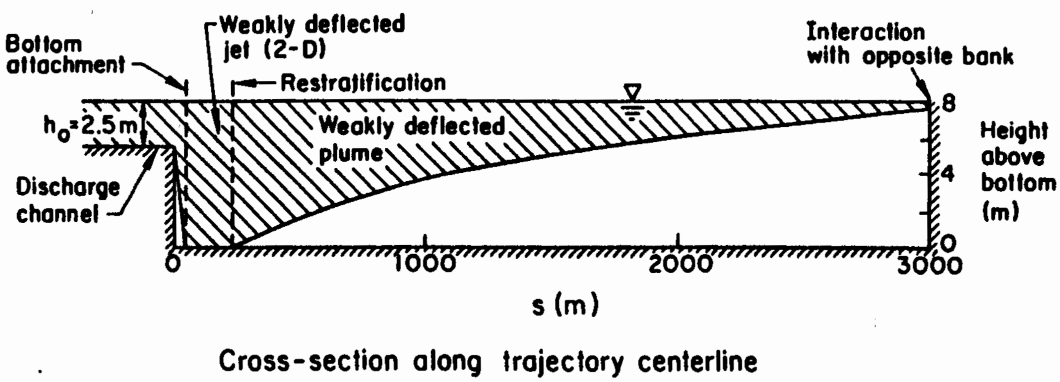


Figure 7.7 - Cross-section of the CORMIX3 predictions for the high tide scenario of the design case.

are rarely known with absolute certainty, it is imperative that when using CORMIX3 to analyze any realistic scenario, repeated runs must be made to determine the sensitivity of the predictions to varying ambient parameters.

7.2. Limitations and Applicability of CORMIX3

CORMIX3 is intended to be a robust modeling program capable of handling a wide variety of positively and neutrally buoyant surface discharges. However, some limitations still exist. For example, CORMIX3 is not intended for application to negatively buoyant discharges. Additional limitations are discussed in Section 7.2.1.

CORMIX3 may, in some instances, be adapted to unusual situations for which direct application is not intended. For example, CORMIX3 may be "tricked" to predict simple negatively buoyant discharges. This and other adaptations of CORMIX3 to atypical situations are discussed in Section 7.2.2.

7.2.1. Limitations of CORMIX3

The limitations of CORMIX3 may be categorized into three general groups: (a) flow class scenarios that cannot be quantitatively predicted, (b) odd discharge or ambient geometries that cannot be schematized, and (c) scenarios with significant ambient stratification.

(a) CORMIX3 is unable to model certain flow classes. One such situation is a discharge classified as and upstream intruding plume in a stagnant environment. The resulting flow in this case is an unsteady pool of effluent that forms at the mouth of the discharge channel. In addition, CORMIX3 can only predict the region of flow near the discharge for all stagnant cases due to the inherent unsteadiness of the far-field in such cases. Also, if strong interaction with a shoreline occurs that may severely disrupt the flow so that prediction of the flow would be difficult or impossible, the simulation will terminate at the point of this interaction.

(b) Extreme discharge and ambient geometries cannot be accounted for by CORMIX3. For example, the aspect ratio of the discharge is limited to a range of 0.05 to 2.0. Although approximations can be made for discharge channels with aspect ratios that fall outside this range such as described in Section 6.2.3 for the Connecticut River field study, caution should be used in doing so to avoid misrepresentation of the important discharge characteristics. In addition, the depth of the receiving water at the mouth of the discharge, H_D, cannot be significantly greater than the average ambient water depth, H_a.

(c) The ability of CORMIX3 to accurately predict the characteristics of a flow that falls on the borderline between two flow classes usually requires repetitive use in order to appreciate the sensitivity of the results. By examining the sensitivity to certain parameters that are might affect the flow, the user can obtain a "feel" for the true nature of the flow.

7.2.2. Application of CORMIX3 to Non-standard Situations

Three non-standard discharge situations are discussed in this section. They include the application of CORMIX3 to negatively buoyant discharges, simplification of irregular discharge geometries, and the inclusion of first-order reaction processes.

7.2.2.1. Adaptation of CORMIX3 to Negatively Buoyant Discharges Issuing at the Bottom of the Water Body

CORMIX3 may be adapted for use on non-buoyant discharges if the bottom geometry of the receiving water is simple. This implies that the discharge channel is located at the bottom of the ambient water body of which there is no significant slope or irregularities. The user must then in essence flip the picture upside down so that the bottom of the receiving water body appears to be the "surface" at which the discharge is located. By specifying a density difference equal to the existing one, but designating the discharge as less dense than the ambient, the flow may be simulated as if it were a mirror image of the actual flow.

In such cases, CORMIX3 will classify the flow and predict the resulting trajectory, dilution and geometry as if it were a positively buoyant discharge issued at the surface of the water. The user must then correctly interpret the results as the mirror image and account for any anomalies that the bottom may cause in the prediction. A similar approach is discussed by Doneker and Jirka (1990) for adapting CORMIX1 to submerged single port discharges near the surface of the receiving water.

7.2.2.2. Simplification of Irregular Discharge Geometries

CORMIX3 also allows for both rectangular discharge channels and circular (pipe) discharges assuming that the pipe is flowing full. In fact, pipe discharges may be slightly submerged, but by no more than one-half pipe diameter for them to be considered a surface discharge. If the cross-section is an odd shape (i.e.: a pipe flowing partially full), then an equivalent rectangular cross section should be used with an equivalent discharge area and width-to-depth ratio.

7.2.2.3. Adaptation to First-order Reaction Processes

The following explanation for adjusting CORMIX3 results for first-order reaction processes is adapted from Doneker and Jirka (1990).

> "CORMIX[3] assumes a conservative pollutant or tracer in the effluent. This assumption is reasonable since the emphasis of CORMIX[3] is on initial mixing mechanisms that have very short time scales (order of minutes) which are much less than the typical reaction times for growth or decay of most discharged substances, including heat.
>
> If the physical, chemical, and/or biological reaction mechanism can be represented as a first-order process with reaction time constant K_r [s^{-1}], then the user can convert the conservative pollutant concentration C predicted by CORMIX[3] in the far-field, i.e. the buoyant spreading and ambient diffusion regimes. The conversion to reacting substances yields a non-conservative concentration C_a
>
> $$C_a = C e^{-K_r x/u_a} \qquad (7.1)$$
>
> in which x/u_a represents the travel time in the far-field. This simple adaptation is acceptable if the reaction time scale, $1/K_r$, is sufficiently larger than the travel time to the end of the near-field (i.e. the hydrodynamic mixing zone), x_{HMZ}/u_a. For substances with faster reactions more detailed analyses which consider the actual travel time within the near-field have to be performed."

An alternate approach to predicting heat loss to the atmosphere for heated discharges is discussed by Jirka et al. (1975).

Chapter VIII

Conclusions and Recommendations

With growing public awareness and tighter governmental regulation of discharges into natural watercourses, it has become essential to be able to accurately predict and analyze a wide variety of possible flows. However, the broad spectrum of possible discharge configurations, ambient conditions, and resulting flow patterns has made the selection of modelling techniques difficult for practicing engineers and regulatory agencies without considerable prior experience or expertise. In addition, most currently available computer programs have limited applicability, require a high degree of familiarity with the input and limitations of the system, and offer little help to the inexperienced user. These difficulties seem to justify the development of a user friendly system capable of predicting a variety of flows for a wide range of discharge conditions.

CORMIX3 is a tool to facilitate the analysis and prediction of positively and neutrally buoyant surface discharges. It is capable of predicting the trajectory, dilution and geometry of a variety of flow patterns. It also provides qualitative descriptions of the flow that give the user a visual picture of the effluent mixing pattern, allowing for more meaningful application of the numerical results. In addition, the use of ample instructions, simple input structure, and warning messages are vital ingredients in making the system an effective means for novice users to design and analyze surface buoyant discharges.

Comparisons of CORMIX3 predictions with laboratory experiments and field studies show good to excellent agreement. In addition, CORMIX3 displays a high degree of flexibility in predicting a wide variety of flow possibilities, including various flow patterns and boundary interactions.

Overall, CORMIX3 appears to be a reliable tool for the initial analysis of buoyant surface jets. However, as more data and experience is obtained in the field, CORMIX3 will require adjustments and modifications. Improvements in the predictive capabilities can be made as further research results become available. In addition, the incorporation of high quality graphics for the display of the simulation results would make the system more attractive and useful.

REFERENCES

Akar, P. J. (1990), "CORMIX2: An Expert System for the Hydrodynamic Mixing Zone Analysis of Conventional and Toxic Multiport Diffuser Discharges," M.S. Thesis, Cornell University, Ithaca, New York.

Abdelwahed, M. S. T., and V. H. Chu (1981), "Surface Jets and Surface Plumes in Cross-flows," Technical Report No. 81-1, Fluid Mechanics Laboratory, McGill University, Montreal.

Benjamin, T. B. (1968), "Gravity Currents and Related Phenomena," Journ. of Fluid Mech., Vol. 31, No. 2.

Briggs, G. A. (1965), "A Plume Rise Model Compared with Observations," Journ. Air Pollution Control Assoc., Vol. 15, No. 9, pp. 443-438.

Brooks, N. H. (1960), "Diffusion of Sewage Effluent in an Ocean Current," Proc. Intn'l. Conf. Waste Disposal Marine Environ. 1st., Pergamon Press, Oxford, pp 246-267.

Bühler, J. and W. Hauenstein (1979), "Axisymmetric Jets in a Crossflow," IAHR Proc., Cagliari, Vol.4, pp. 257-265.

Chu, V. H., and M. S. T. Abdelwahed (1990), "Shore Attachment of Buoyant Effluent in Strong Crossflow," Journ. Hyd. Eng., ASCE, Vol. 116, No. 2, pp. 157-175.

Chu, V. H., and M. B. Goldberg (1974), "Buoyant Forced-plumes in Crossflow," Journ. Hyd. Div., ASCE, Vol. 100, No. HY9, pp. 1203-1214.

Chu, V. H., and G. H. Jirka (1986), "Chapter 25: Surface Buoyant Jets," Encyclopedia of Fluid Mechanics, Gulf Publishing Company, Houston, Texas.

Csanady, G. T. (1961), "Some Observations of Smoke Plumes," Int'l. Journ. Air Pollution, Pergamon Press, London, Vol. 4, No. 1/2, pp. 47-51.

Davidson, M. J. (1989), "The Behavior of Single and Multiple, Horizontally Discharged, Buoyant Flows in a Non-Turbulent Coflowing Ambient Fluid," Ph.D. Thesis, Department of Civil Engineering, University of Canterbury, Christchurch, New Zealand.

Delft Hydraulics Laboratory (1983), "Buoyant Surface Jets in Crossflow," Report on Experimental Investigation - S350-II.

Doneker, R. L., and G. H. Jirka (1990), "Expert System for Hydrodynamic Mixing Zone Analysis of Conventional and Toxic Submerged Single Port Discharges (CORMIX1)," Report No. EPA/600/3-90/012, U.S. Environmental Protection Agency, Env. Research Laboratory, Athens, Georgia.

Dunn, W. E., A. J. Policastro, and R. A. Paddock (1975), "Surface Thermal Plumes: Evaluation of Mathematical Models for the Near and Complete Field," Argonne National Laboratory Report ANL/WR-75-3 - Part One.

Fan, L. N. (1967), "Turbulent Buoyant Jets into Stratified or Flowing Ambient Fluids," W. M. Keck Laboratory Report KH-R-15, California Institute of Technology, Pasadena, Calif.

Fay, J. A. (1973), "Buoyant Plumes and Wakes," Ann. Rev. of Fluid Mech., Vol. 5, pp. 151-160.

Fischer, H. B., E. J. List, C. Y. Koh, J. Imberger, and N. H. Brooks (1979), Mixing in Inland and Coastal waters, Academic Press, Inc., Orlando, Florida.

Garvine, R. W. (1974), "Physical Features of the Connecticut River Outflow During High Discharge," Journ. of Geophys. Res., Vol. 79, No. 6.

Hayashi, T., and N. Shuto (1967), "Diffusion of Warm Water Jets Discharged Horizontally at Water Surface," Proc. of the 12th Conf. of the Intern'l. Assoc. for Hyd. Res., Fort Collins, Colorado, Vol. 4, pp.47-59.

Holley, E. R. and G. H. Jirka (1986), "Mixing in Rivers," Technical Report E-86-11, U.S. Army Corps of Engineers, Washington, D.C.

Hoult, D. P., and J. C. Weil (1972), "Turbulent Plume in a Laminar Crossflow," Atmospheric Environment, Pergamon Press, London, pp. 513-531, Vol. 6.

Hoult, D. P., J. A. Fay, and L. J. Forney (1969), "A Theory of Plume Rise Compared with Field Observations," Journ. Air Pollution Control Assoc., Vol. 19, No. 8, pp. 585-590.

Huq, I. P. (1983), "Experiments on Density Currents in Stratified Flowing Environments," M.S. Thesis, Cornell University, Ithaca, New York.

Jirka, G. H., G. Abraham, and D. R. F. Harleman (1975), "An Assessment of Techniques for Hydrothermal Prediction," M.I.T. - Ralph M. Parsons Laboratory for Water Resources and Hydrodynamics Report No. 203.

Jirka, G. H., E. E. Adams, and K. D. Stolzenbach (1981), "Buoyant Surface Jets," Journ. Hyd. Div., ASCE, Vol. 107, No. HY11, pp. 1467-1487.

Jirka, G. H., and P. J. Akar (1990), "CORMIX2: Expert System for Multiport Diffuser Discharges," Proc. Nat'l. Conference on Hyd. Eng., ASCE, San Diego, California.

Jirka, G. H., and M. Arita (1987), "Density Currents or Density Wedges: Boundary Layer Influence and Control Methods," Journ. of Fluid Mech., Vol 177, pp. 187-206.

Jones, J. M., G. H. Jirka, and D. A. Caughey (1985), "Numerical Techniques for Steady Two-dimensional Transcritical Stratified Flow Problems, with an Application to the Intermediate Field Dynamics of Ocean Thermal Energy Conversion Plants," Argonne National Laboratory Report No. ANL/EES-TM-271.

Knudsen, M. (1988), "Buoyant Horizontal Jets In an Ambient Flow," PhD Thesis, University of Canterbury, Christchurch, New Zealand.

Koester, G. E. (1974), "Experimental Study of Submerged Single-Port Thermal Discharges," M.S. Thesis, Department of Civil Engineering, M.I.T., Cambridge, Massachussetts.

Kuhlman, J. M., and J. M. Prahl (1974), "Laboratory Modeling of Surface Thermal Plumes," Report No. FTAS/TR-74-102, School of Engineering, Case Western Reserve University, Cleveland, Ohio.

Larsen, J., and T. Sorensen (1968), "Chapter 89 - Buoyancy Spread of Wastewater in Coastal Regions", Eleventh Conference on Coastal Engineering, London, Vol. 2.

List, E. J., and J. Imberger (1973), "Turbulent Entrainment in Buoyant Jets and Plumes," Journ. Hyd. Div., ASCE, No. HY9, pp. 1461-1474.

Miller, D. S., and B. A. Brighouse (1985), Thermal Discharges - A Guide to Power and Process Plant Cooling Water Discharges into Rivers, Lakes, and Seas, British Hydromechanics Research Association.

Moore, D. J. (1966), "Physical Aspects of Plume Models," Int'l Journ. Air and Water Pollution, Vol. 10, pp. 441-417.

Motz, L. H., and B. A. Benedict (1970), "Heated Surface Jet Discharged into a Flowing Ambient Stream," Report No. 4, Department of Environmental and Water Resources Engineering, Vanderbilt University, Nashville, Tennessee, Aug. 1970.

Morton, B. R. (1959), "Forced Plumes," Journ. of Fluid Mech., Vol. 5, No. 1, pp. 151-163.

Morton, B. R., G. I. Taylor, and J. S. Turner (1956), "Turbulent Gravitational Convection from Maintained and Instantaneous Sources," Roy. Soc. London A, Vol. 234, No. 1196, pp. 1-23.

Priestley, C. H. B. (1956), "A Working Theory of the Bent-over Plume of Hot Gas," Quart. Journ. Roy. Meteorol. Soc., Vol. 82, pp. 165-176.

Priestley, C. H. B., and F. K. Ball (1955), "Continuous Convection from an Isolated Source of Heat," Quart. Journ. Royal Meteorol. Soc., Vol. 81, No. 348, pp. 144-157.

Railston, W. (1954), Proc. Phys. Soc. B, Vol. 67, p. 42.

Rouse, H., C. S. Yih, and H. W. Humphreys (1952), "Gravitational Convection from a Boundary Layer," Tellus, Vol. 4, pp. 201-210.

Schmidt, W. (1941), "Turbulente Ausbreitung eines Stromes Erhitzer Luft," Z. agnew. Math Mech., Vol. 21, pp. 265-278, 351-363.

Scorer, R. S. (1958), Natural Aerodynamics, Pergamon Press, London.

Scorer, R. S. (1959), "Behavior of Chimney Plumes," Int'l. Journ. Air Pollution, Pergamon Press, London, Vol. 1, No. 4, pp. 198-220.

Sharp, J. J. and B. D. Vyas (1977), "The Buoyant Wall Jet", Proceeding of the Institution of Civil Engineers, Part 2, No. 63, September.

Simpson, J. E., and R. E. Britter (1979), "The Dynamics of the Head of a Gravity Current Advancing Over a Horizontal Surface," Journ. of Fluid Mech., Vol. 94, pp. 477-495.

Slawson, P. R., and G. T. Csanady (1967), "The Mean Path of Buoyant Chimney Plumes," Journ. Fluid Mech., Vol. 28, pp. 311-322.

Sutton, O. G. (1953), Micrometeorology, McGraw Hill, New York.

Tokar, J. V. (1971), "Thermal Plumes in Lakes: Compilations of Field Experience," Report ANL/ES-3, Center for Environmental Studies, Argonne National Laboratory, Argonne, Illinois.

Wolanski, E. J., and R. C. Koh (1973), "Preliminary Report on Secondary Circulation in Surface Buoyant Jets," Tech. Memo No. 73-10, W. M. Keck Laboratory of Hydraulics and Water Resources, California Institute of Technology, Pasedena, California.

Wright, S. J. (1977), "Mean Behavior of Buoyant Jets in a Crossflow," Journ. Hyd. Div., ASCE, Vol. 103, No. HY5, pp. 499-513.

Wright, S. J., and R. B. Wallace (1979), "Two Dimensional Buoyant Jets in a Stratified Fluids," Journ. Hyd. Div., ASCE, Vol. 105, No. HY11, pp. 1393-1406.

Yih, C. S. (1951), Proc. 1st U.S. Congr. Appl. Mech., p. 941.

Appendix A

Design Recommendations

A reliable environmental analysis and mixing zone prediction is possible only if each design case is evaluated through several iterations of CORMIX3. Small changes in ambient or discharge design conditions can sometimes cause drastic shifts in the applicable flow class and the size or appearance of mixing zones. Iterative use of CORMIX3 will give information on the sensitivity of predicted results on design and ambient conditions. Each predictive case should be carefully assessed as to:

- size and shape of LMZ
- conditions in the TDZ (if present)
- bottom impact of the discharge flow
- bank attachment, and other factors.

In general, iteration should be conducted in the following order:

A) Discharge geometry changes
B) Sensitivity to ambient conditions
C) Discharge flow changes (process variations)

When investigating these variations the CORMIX3 user will quickly appreciate the fact that mixing conditions at short distances (near-field) are usually quite sensitive and controllable. In contrast, mixing conditions at large distances (far-field) often show little sensitivity unless the ambient conditions change substantially or drastic process variations are introduced.

A) DISCHARGE GEOMETRY CHANGES:

Most of the following recommendations are motivated by the desire of improving conditions in the applicable mixing zones (i.e. minimizing concentrations and/or areal extent).

1) Discharge depth: Increasing the depth of the ambient water body at the discharge location discourages bottom interaction and subsequently downstream bank attachment. The effectiveness if increasing the total depth at the discharge depends on the buoyancy and the initial momentum of the discharge.

2) Aspect ratio: By decreasing the depth of the discharge channel and increasing its width, the discharge is encouraged to propagate further into the channel. This also may lead to a small increase in the initial surface area of the plume. Note, however, that CORMIX3 is limited to discharge channels with aspect ratios between 0.5 and 2.

3) Horizontal angle: The larger the angle between the downstream bank and the discharge channel centerline, the less likely shoreline attachment will occur. This may be desirable since shoreline attachment generally leads to decreased dilutions and higher pollutant concentrations along the near bank.

B) SENSITIVITY TO AMBIENT CONDITIONS:

Variations - of the order of 10 percent - of the following ambient design conditions should be considered:

- ambient velocity (or ambient flowrate)
- ambient depth (or river/tidal stage)
- ambient density structure (notably density differences)

Such variability is important for two reasons:

1) the usual uncertainty in ambient environmental data,
2) the schematization employed by CORMIX3.

Please refer to the detailed advice on the specification of environmental data that is available in program element DATIN3. In particular, note the advisory comments on stagnant ambient conditions.

C) DISCHARGE FLOW CHANGES (PROCESS VARIATIONS):

Actual process changes can result in variations of one or more of three parameters associated with the discharge: flowrate, density, or pollutant concentration. In some cases, such process changes may be difficult to achieve or too costly. Note, that "off-design" conditions in which a discharge operates below its full capacity also fall into this category.

1) Pollutant mass flux: The total pollutant mass flux is the product of discharge flow (m**3/s) times the discharge pollutant concentration (in arbitrary units). Thus, decreasing the pollutant mass flux will, in general, decrease the resulting pollutant concentration in the near-field and far-field. This occurs, of course, during off-design conditions.

2) Discharge flow: For a given pollutant mass flux, an increase in discharge flow implies an decrease in discharge pollutant concentration, and vice versa. Although, for the variety of flow classes contained in CORMIX3 there is no universal rule whether high or low volume discharges are

preferable for optimizing near-field mixing, increased discharge velocities generally increase initial mixing. Mostly, the sensitivity is small, and even more so for far-field effects.

3) Discharge density: The actual density of the discharge flow controls the buoyancy effects relative to the ambient water. Occasionally, the discharge density is controllable through the amount of process heating or cooling occurring prior to discharge. Near-field mixing may be enhanced or degraded by increasing the total density difference between discharge flow and ambient water. In most cases, however, this effect is minor.

Appendix B

Flow Class Descriptions

FLOW CLASS FJ1

This flow class exhibits no bank interaction or bottom interaction in the near-field. It is oriented at a large enough angle from the downstream shoreline to prevent shoreline attachment. The buoyancy is relatively strong and will distort the cross-section of the flow significantly near the source.

The flow consists of the following regimes:

1) Weakly deflected 3-dimensional jet: The mixing is dominated by the initial momentum, causing relatively constant spreading in both the horizontal and the vertical directions. The deflection by the ambient crossflow is relatively weak.

2) Weakly deflected plume: The flow cross-section becomes distorted by the buoyancy, resulting in thinning of the flow and increased non-linear lateral spreading. The dilution is reduced in this regime due to suppression of the vertical mixing by buoyancy forces.

3) Strongly deflected plume: The cross-section of the flow is distorted due to buoyancy-induced lateral spreading. This may result in thinning of the plume. The flow is strongly deflected by the ambient current.

4) Far-field buoyant spreading: The plume spreads laterally along the surface while being advected downstream with the ambient current. There is no net change in the centerline trajectory. The mixing rate is relatively small and the thickness may decrease in this regime. The plume may interact with the shoreline.

5) Passive ambient diffusion: The ambient turbulence becomes the predominant mixing process in this regime. The plume will grow in both the vertical and horizontal directions at a rate that is dependent on the magnitude of the ambient turbulence. The flow may interact with the bottom or the shoreline in this regime.

SPECIAL CASE: If the receiving water is stagnant, the simulation will terminate at the end of the weakly deflected plume regime (2).

FLOW CLASS FJ2

This flow exhibits no bank interaction or bottom interaction in the near-field. It is oriented at a large enough angle from the downstream shoreline to prevent shoreline attachment. The buoyancy is strong to distort the cross-section of the flow in the near-field.

The flow consists of the following regimes:

1) Weakly deflected 3-dimensional jet: The mixing is dominated by the initial momentum, causing relatively constant spreading in both the horizontal and the vertical directions. The deflection by the ambient crossflow is relatively weak.

2) Strongly deflected 3-dimensional jet: The is dominated by the initial momentum, causing relatively constant spreading in both the horizontal and vertical directions. The flow is strongly deflected by the ambient current.

3) Strongly deflected plume: The cross-section of the flow is distorted due to buoyancy-induced lateral spreading. This may result in thinning of the plume. The flow is strongly deflected by the ambient current.

4) Far-field buoyant spreading: The plume spreads laterally along the surface while being advected downstream with the ambient current. There is no net change in the centerline trajectory. The mixing rate is relatively small and the thickness may decrease in this regime. The plume may interact with the shoreline.

5) Passive ambient diffusion: The ambient turbulence becomes the predominant mixing process in this regime. The plume will grow in both the vertical and horizontal directions at a rate that is dependent on the magnitude of the ambient turbulence. The flow may interact with the bottom or the shoreline in this regime.

SPECIAL CASE: If the effluent is non-buoyant, the strongly deflected plume and far-field buoyant spreading regimes will be omitted.

FLOW CLASS FJ3

This flow exhibits no bank interaction in the near-field. However, the ambient water depth is shallow and the flow will interact with the bottom in the near-field. It is oriented at a large enough discharge angle to prevent shoreline attachment with the downstream shoreline. The buoyancy is relatively strong and will distort the cross-section of the flow significantly in the near-field.

The flow consists of the following regimes:

1) Weakly deflected 3-dimensional jet: The mixing is dominated by the initial momentum, causing relatively constant spreading in both the horizontal and the vertical directions. The deflection by the ambient crossflow is relatively weak.

2) Weakly deflected 2-dimensional jet: The flow interacts with the bottom of the receiving water body and is considered to be 2-dimensional (i.e.: fully vertically mixed). The dilution is due to momentum dominated mixing. At the end of this regime, the flow restratifies and becomes 3-dimensional in character. The deflection by the ambient current is relatively weak.

3) Weakly deflected plume: The flow cross-section becomes distorted by the buoyancy, resulting in thinning of the flow and increased non-linear lateral spreading. The dilution is reduced in this regime due to suppression of the vertical mixing by buoyant forces. The deflection by the ambient current is relatively weak.

4) Strongly deflected plume: The cross-section of the flow is distorted due to buoyancy-induced lateral spreading. This may result in thinning of the plume. The flow is strongly deflected by the ambient current.

5) Far-field buoyant spreading: The plume spreads laterally along the surface while being advected downstream with the ambient current. There is no net change in the centerline trajectory. The mixing rate is relatively small and the thickness may decrease in this regime. The plume may interact with the shoreline.

6) Passive ambient diffusion: The ambient turbulence becomes the predominant mixing process in this regime. The plume will grow in both the vertical and horizontal directions at a rate that is dependent on the magnitude of the ambient turbulence. The flow may interact with the bottom or the shoreline in this regime.

SPECIAL CASE: If the receiving water is stagnant, the simulation will terminate at the end of the weakly deflected plume regime (3).

FLOW CLASS FJ4

This flow exhibits no bank interaction in the near-field. However, the ambient water depth is shallow and the flow will interact with the bottom in the near-field. It is oriented at a large enough discharge angle to prevent shoreline attachment with the downstream shoreline. The buoyancy will distort the cross-section of the flow to some degree in the near-field.

The flow consists of the following regimes:

1) Weakly deflected 3-dimensional jet: The mixing is dominated by the initial momentum, causing relatively constant spreading in both the horizontal and the vertical directions. The deflection by the ambient crossflow is relatively weak.

2) Weakly deflected 2-dimensional jet: The flow interacts with the bottom of the receiving water body and is considered to be 2-dimensional (i.e.: fully vertically mixed). The dilution is due to momentum dominated mixing. At the end of this regime, the flow restratifies and becomes 3-dimensional in character. The deflection by the ambient current is relatively weak.

= OR =

2) Strongly deflected 3-dimensional jet: The is dominated by the initial momentum, causing relatively constant spreading in both the horizontal and vertical directions. The flow is strongly deflected by the ambient current.

3) Strongly deflected 2-dimensional jet: The flow interacts with the bottom of the receiving water body and is considered to be 2-dimensional (i.e.: fully vertically mixed). The dilution is due to momentum dominated mixing. At the end of this regime, the flow restratifies and becomes 3-dimensional in character. The flow is strongly deflected by the ambient current.

4) Strongly deflected plume: The cross-section of the flow is distorted due to buoyancy-induced lateral spreading. This may result in thinning of the plume. The flow is strongly deflected by the ambient current.

5) Far-field buoyant spreading: The plume spreads laterally along the surface while being advected downstream with the ambient current. There is no net change in the centerline trajectory. The mixing rate is relatively small and the thickness may decrease in this regime. The plume may interact with the shoreline.

6) Passive ambient diffusion: The ambient turbulence becomes the predominant mixing process in this regime. The plume will grow in both the vertical and horizontal directions at a rate that is dependent on the magnitude of the ambient turbulence. The flow may interact with the bottom or the shoreline in this regime.

SPECIAL CASE: If the effluent is non-buoyant, the strongly deflected plume and far-field buoyant spreading regimes will be omitted.

FLOW CLASS SA1

This flow is dynamically attached to the downstream bank. Along the bank is a zone of recirculating effluent which will reduce the dilution. The penetration into the crossflow is reduced due to this dynamic attachment. For this case, the flow does not interact with the bottom in the near field.

The flow consists of the following regimes:

1) Weakly deflected shoreline attached jet (3-D): The mixing is dominated by the initial momentum causing relatively constant spreading in both the horizontal and the vertical directions. The deflection by the crossflow is relatively weak. This regime tends to be very short or non-existent in shoreline attached flows.

2) Strongly deflected shoreline attached jet (3-D): The mixing in this regime is dominated by the initial momentum, causing relatively constant spreading in both the horizontal and vertical directions. The flow is strongly bent over by the ambient crossflow and is dynamically attached to the shoreline. A zone of recirculating effluent exists between the core of the flow and the shoreline.

3) Strongly deflected shoreline attached plume: The cross-section of the flow becomes distorted due to strong buoyancy-induced lateral spreading. Some of the effluent from this region is recirculated back upstream along the shoreline. However, the overall mixing rate is small and the thickness may decrease in this regime.

4) Far-field buoyant spreading: The plume spreads laterally along the surface while being advected downstream with the ambient current. There is no net change in the centerline trajectory. The mixing rate is relatively small and the thickness may decrease in this regime. The plume remains attached to the shoreline.

5) Passive ambient diffusion: The ambient turbulence becomes the predominant mixing process in this regime. The plume will grow in both the vertical and horizontal directions at a rate that is dependent on the magnitude of the ambient turbulence. The flow may interact with the bottom or the opposite shoreline in this regime.

FLOW CLASS SA2

This flow is dynamically attached to the downstream bank. Along the bank is a zone of recirculating effluent exists which reduces the dilution. The penetration into the crossflow is reduced due to this dynamic attachment. Since the discharge depth is equal or nearly equal to the depth of the receiving water at the discharge point, the flow becomes attached to the bottom. This attachment to the bottom could effectively block off the ambient current and be the cause of the attachment to the downstream shoreline.

The flow consists of the following regimes:

1) Weakly deflected shoreline attached jet (2-D): The mixing is dominated by the initial momentum causing relatively constant spreading in the horizontal direction. The deflection by the crossflow is relatively weak. This regime tends to be very short or non-existent in shoreline attached flows.

2) Strongly deflected shoreline attached jet (2-D): The mixing in this regime is dominated by the initial momentum, causing relatively constant spreading in the horizontal direction. The flow is strongly bent over and is dynamically attached to the shoreline. A zone of recirculating effluent exists between the core of the flow and the shoreline. The flow remains attached to the bottom throughout this regime.

3) Strongly deflected shoreline attached plume: The flow may lift off the bottom if it contains sufficient buoyancy. The cross-section of the flow becomes distorted due to strong buoyancy-induced lateral spreading. Some of the effluent from this region is recirculated back upstream along the shoreline. However, the overall mixing rate is small and the thickness may decrease in this regime.

4) Far-field buoyant spreading: The plume spreads laterally along the surface while being advected downstream with the ambient current. There is no net change in the centerline trajectory. The mixing rate is relatively small and the thickness may decrease in this regime. The plume remains attached to the shoreline.

5) Passive ambient diffusion: The ambient turbulence becomes the predominant mixing process in this regime. The plume will grow in both the vertical and horizontal directions at a rate that is dependent on the magnitude of the ambient turbulence. The flow may interact with the bottom or the opposite shoreline in this regime.

FLOW CLASS WJ1

Because this discharge is issued at or near parallel to the downstream bank, it will remain attached to the shoreline. The receiving water is relatively deep in the vicinity of the discharge so no interaction with the bottom will take place in the near-field. The shoreline will act as a reflective boundary to liken the flow to a jet being issued into a coflow.

The flow will consist of the following regimes:

1) Weakly deflected 3-dimensional wall jet: The mixing is dominated by the initial momentum, causing relatively constant jet-like spreading in both the horizontal and the vertical direction. The ambient current has very little effect on the flow in this regime. There is no interaction with the bottom throughout this regime.

2) Far-field buoyant spreading: Further from the source the initial momentum becomes unimportant and far-field processes take over. The plume spreads laterally along the surface while being advected downstream with the ambient current. There is no net change in the centerline trajectory. The mixing rate is relatively small and the thickness may decrease in this regime. The plume remains attached to the shoreline.

3) Passive ambient diffusion: The ambient turbulence becomes the predominant mixing process in this regime. The plume will grow in both the vertical and horizontal directions at a rate that is dependent on the magnitude of the ambient turbulence. The flow may interact with the bottom or the opposite shoreline in this regime.

FLOW CLASS WJ2

Because this discharge is issued at or near parallel to the downstream bank, it will remain attached to the shoreline. The receiving water is relatively shallow in the vicinity of the discharge and the discharge interacts with the bottom in the near-field. The shoreline will act as a reflective boundary to liken the flow to a jet being issued into a coflow.

The flow will consist of the following regimes:

1) Weakly deflected 3-dimensional wall jet: The mixing is dominated by the initial momentum, causing relatively constant jet-like spreading in both the horizontal and the vertical direction. The ambient current has very little effect on the flow in this regime. Interaction with the bottom will occur at the end of this regime.

2) Weakly deflected 2-dimensional wall jet: Once bottom interaction occurs, the flow becomes 2-dimensional in form. The mixing is dominated by the initial momentum, causing relatively constant jet-like spreading in the horizontal direction. The ambient current has very little effect on the flow in this regime. There is continuous interaction with the bottom throughout this regime.

3) Far-field buoyant spreading: Further from the source the initial momentum becomes unimportant and far-field processes take over. Liftoff of the flow from the bottom may occur due to buoyancy forces. The plume spreads laterally along the surface while being advected downstream with the ambient current. There is no net change in the centerline trajectory. The mixing rate is relatively small and the thickness may decrease in this regime. The plume remains attached to the shoreline.

4) Passive ambient diffusion: The ambient turbulence becomes the predominant mixing process in this regime. The plume will grow in both the vertical and horizontal directions at a rate that is dependent on the magnitude of the ambient turbulence. The flow may interact with the bottom or the opposite shoreline in this regime.

FLOW CLASS PL1

The discharge is issued with relatively low velocity and high buoyancy into an environment with a relatively low ambient velocity. For this reason, the effluent will spread upstream along the shoreline against the ambient current. The receiving water is relatively deep in the vicinity of the discharge so no significant interaction with the bottom takes place in the near-field.

The flow will consist of the following flow regimes:

1) Upstream intrusion: Upstream of the discharge will be a steady layer of effluent. This upstream spreading is caused by the strong buoyancy of the effluent and a relatively weak ambient crossflow. This regime ends just downstream of the discharge where far-field buoyant spreading takes over.

2) Far-field buoyant spreading: The plume spreads laterally along the surface while being advected downstream with the ambient current. The mixing rate is relatively small and the thickness may decrease in this regime. The plume remains attached to the shoreline.

3) Passive ambient diffusion: The ambient turbulence becomes the predominant mixing process in this regime. The plume will grow in both the vertical and horizontal directions at a rate that is dependent on the magnitude of the ambient turbulence. The flow may interact with the bottom or the opposite shoreline in this regime.

FLOW CLASS PL2

The discharge is issued with relatively high buoyancy into an environment with a relatively low ambient velocity. For this reason, the effluent will spread upstream along the shoreline against the ambient current. The receiving water is shallow in the vicinity of the discharge. There is unstable recirculation of the effluent in the immediate area of the discharge. However, stable restratification of the flow occurs just downstream of the source where far-field processes take over.

The flow will consist of the following flow regimes:

1) Upstream intrusion with recirculation: In the immediate vicinity of the discharge, unstable recirculation of the effluent will occur. Upstream spreading is caused by the strong buoyancy of the effluent and a relatively weak ambient crossflow. This regime ends just downstream of the discharge where the flow restatifies and far-field buoyant spreading takes over.

2) Far-field buoyant spreading: The plume spreads laterally along the surface while being advected downstream with the ambient current. The mixing rate is relatively small and the thickness may decrease in this regime. The plume remains attached to the shoreline.

3) Passive ambient diffusion: The ambient turbulence becomes the predominant mixing process in this regime. The plume will grow in both the vertical and horizontal directions at a rate that is dependent on the magnitude of the ambient turbulence. The flow may interact with the bottom or the opposite shoreline in this regime.

FLOW CLASS PL3

The discharge is issued with relatively low discharge momentum into an environment with a relatively low ambient velocity. For these reasons, far-field processes are the only important processes.

The flow will consist of the following flow regimes:

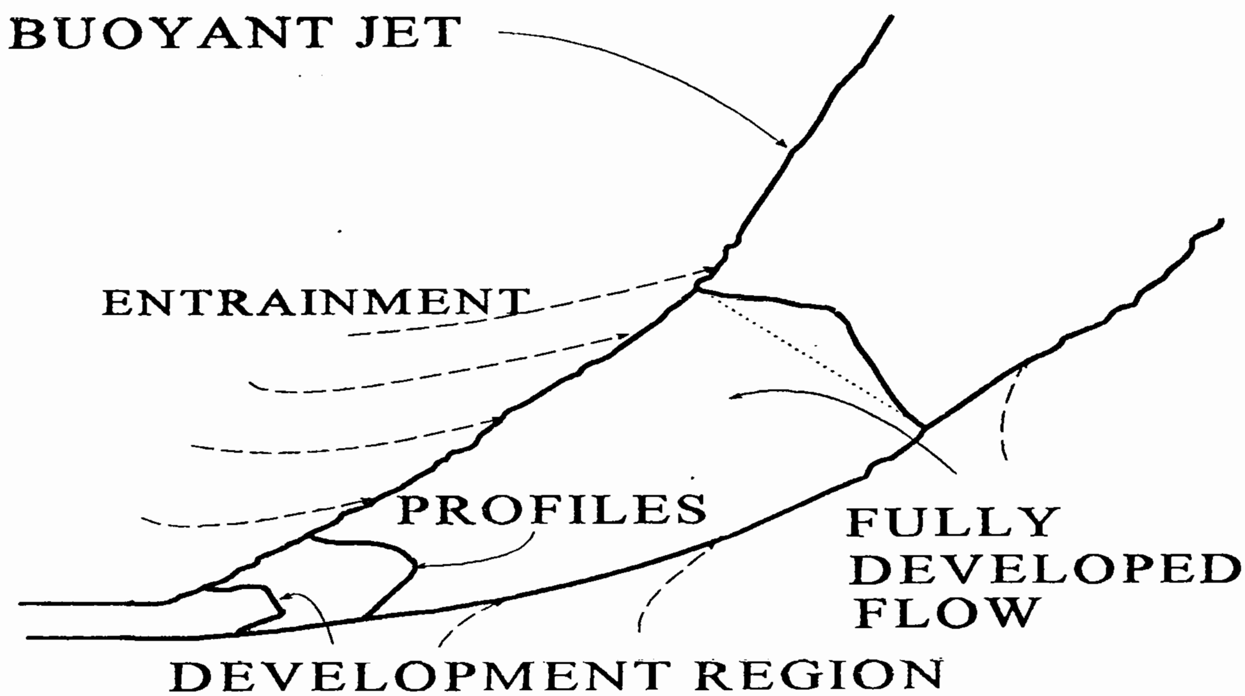
1) Far-field buoyant spreading: The plume spreads laterally along the surface while being advected downstream with the ambient current. The mixing rate is relatively small and the thickness may decrease in this regime. The plume remains attached to the shoreline.

2) Passive ambient diffusion: The ambient turbulence becomes the predominant mixing process in this regime. The plume will grow in both the vertical and horizontal directions at a rate that is dependent on the magnitude of the ambient turbulence. The flow may interact with the bottom or the opposite shoreline in this regime.

ACETATOS

INTRODUCTION

INITIAL DILUTION IS THE DILUTION WITH AMBIENT WATER ACHIEVED IN A PLUME DUE TO THE COMBINED EFFECTS OF MOMENTUM AND BUOYANCY OF THE FLUID DISCHARGED FROM AN OUTFALL, AND DUE TO AMBIENT TURBULENT MIXING IN THE VICINITY OF THE PLUME. THE RATE OF DILUTION CAN BE VERY RAPID IN THE FIRST FEW METERS FROM THE OUTFALL AND DECREASES AFTER THAT.



FAR FIELD DIFFUSION AFTER THE INITIAL MOMENTUM AND BUOYANCY HAVE DECAYED IS DUE MAINLY TO AMBIENT TURBULENCE AND IS VERY SLOW COMPARED TO THE INITIAL RATE OF DILUTION. THE TOTAL DILUTION AND AMBIENT CONCENTRATIONS, DETERMINE THE CONCENTRATIONS WITHIN THE PLUME.

IN THE U.S. CRITICAL DILUTIONS ARE SET BY REGULATORY AGENCIES (MOST OFTEN THE STATE) FOR A PARTICULAR DISCHARGE TO PROTECT THE ENVIRONMENT. THEY ARE THE MINIMUM DILUTION THAT CAN OCCUR AT A PRESCRIBED BOUNDARY. TWO BOUNDARIES ARE OFTEN SPECIFIED.

1. THE MOST RESTRICTIVE REGION IS OFTEN CALLED THE TOXIC MIXING ZONE OR ZONE OF INITIAL DILUTION, ZID. IT IS WHERE THE CRITERION MAXIMUM CONCENTRATION (CMC)MUST BE MET
2. THE SECOND IS THE REGULATORY MIXING ZONE WHERE THE CRITERION CONTINUOUS CONCENTRATION MUST BE MET (CCC).

THE CMC ZONE IS OFTEN DEFINED AS THE SHORTEST OF:

1. 10% OF THE REGULATORY MIXING ZONE
2. A DISTANCE NO FURTHER THAN 5 TIMES THE LOCAL WATER DEPTH
3. A DISTANCE OF 5O TIMES THE SQUARE ROOT OF ANY DISCHARGE AREA OR PORT.

THE DILUTION REQUIRED TO MEET THE CMC REGULATION IS OFTEN SET FROM A BIO-ASSAY.

- FOR MUNICIPAL OCEAN DISCHARGES, THE EPA HAS DEFINED THE REGULATORY MIXING ZONE AS THAT distance WHERE THE PLUME EITHER SURFACES OR REACHES A POINT OF NATURAL BUOYANCY (TRAPPED). CURRENTS THAT CAN BE USED ARE THE LOWEST 10 PERCENTILE CURRENT.

FOR INLAND WATERWAYS, THE MIXING ZONE IS OFTEN DEFINED AS A RADIUS FROM THE POINT OF DISCHARGE OR A SURFACE AREA.

IN ADDITION, RESTRICTIONS ARE OFTEN IMPOSED AS TO HOW MUCH OF A RIVER CAN BE USED BY THE PLUME: 1/4 OF THE WIDTH, 1/4 OF THE RIVER FLOW.

JETS, PLUMES, AND BUOYANT JETS.

A JET IS A DISCHARGE AT RELATIVELY HIGH VELOCITY THAT HAS SMALL BUOYANCY.

A PLUME IS A DISCHARGE WITH SMALL MOMENTUM AND LARGE BUOYANCY

A BUOYANT JET IS A DISCHARGE WITH MODERATE VELOCITY AND MODERATE BUOYANCY.

THE RATIO OF THE MOMENTUM TO THE BUOYANCY IN A DISCHARGE GIVES THE DENSIMETRIC FROUDE NUMBER.

$$F_o = \frac{U_o}{\sqrt{g D_o \frac{\Delta\rho}{\rho}}}$$

VALUES GREATER THAN 40 OR SO ARE MOMENTUM JETS. VALUES LESS THAN 2 ARE PLUMES. MOST ENVIRONMENTAL DISCHARGES FALL BETWEEN THESE.

MODELING TECHNIQUES

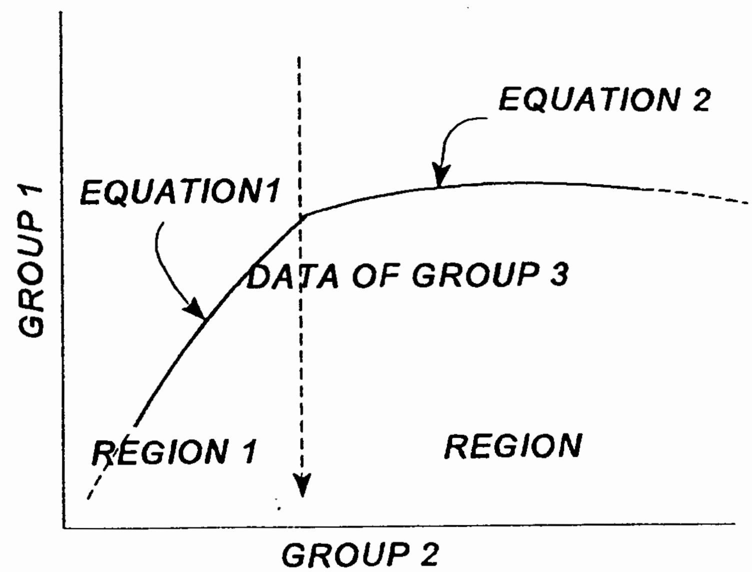
1. EXPERIMENTAL METHODS USING SCALE MODELS OF THE DISCHARGE. THESE CAN BE SIMPLE WHERE JUST THE DISCHARGE AND PLUME ARE INVESTIGATED OR COMPLEX WHERE COMPLICATED GEOMETRY IS SIMULATED. THESE ARE KNOWN AS PHYSICAL MODELS AND ARE NOT THE TOPIC OF THIS COURSE.

2. NUMERICAL MODELING IS WHERE THE DISCHARGE AND RECEIVING WATER ARE DIVIDED INTO THOUSANDS OF SMALL VOLUMES OR GRIDS EACH WITH ITS OWN FLUID PROPERTIES. COMPUTATIONAL FLUID DYNAMICS ARE USED TO SOLVE FOR THE INTERNAL GRID PROPERTIES BASED ON BOUNDARY AND INITIAL VALUES. THESE ARE HARD TO SET UP AND REQUIRE LARGE COMPUTERS... DIFFUSION COEFFICIENTS ARE OFTEN THE HARDEST PART TO MODEL.

NUMERICAL MODELS CAN BE TIME DEPENDENT AND CAN BE USED CONSIDER TIDAL EFFECTS, AMBIENT CONCENTRATION BUILT UP, AND OTHER TRANSIENTS. WITH MOST DISCHARGES, HOWEVER, THE MIXING ZONE IS REACHED WITHIN MINUTES AND TRANSIENTS ARE NOT VERY IMPORTANT. SEVERAL QUASI-STEADY STATE RUNS CAN BE MADE TO COVER THE WHOLE CYCLE.

3. INTEGRAL MODELS LOOK AT ONLY THE PLUME AND SOLVE THE EQUATIONS OF MOTION WITHIN IT. ALTHOUGH STEADY STATE, THEY HAVE BEEN FOUND TO GIVER AS GOOD OR BETTER RESULTS THAN NUMERICAL MODELS AS LONG AS COMPLICATED BOUNDARIES DON'T INFLUENCE THE PLUME. AS A RESULT THEY ARE USED MOST OF THE TIME. SEVERAL OF THE MODELS DISCUSSED IN THIS COURSE OF THIS TYPE.

4. EMPIRICAL METHODS ARE BASED ON THE RESULTS OF MANY EXPERIMENTS. MATHEMATICAL EQUATIONS ARE FIT TO THE DATE. THESE EQUATIONS DO NOT SOLVE THE CONSERVATION EQUATIONS BUY RELY ON THE EXPERIMENTS TO DO SO. TO MAKE THEM MORE GENERAL, THEY ARE USE DIMENSIONLESS GROUPS OF VARIABLES. THE EQUATIONS ARE USED TO EXTRAPOLATE AND INTERPOLATE TO DESIRED OPERATING CONDITIONS.



THE EQUATIONS GIVE GROUP 1 AS A FUNCTION OF GROUP 2 AND GROUP 3. WHICH EQUATION TO USE DEPENDS ON THE MAGNITUDE OF GROUP 2. EXTRAPOLATION BEYOND DATA IS UNPREDICTABLE.

WHY MODELING

- THE PURPOSE OF DILUTION MODELING IS TO DETERMINE WHAT DILUTIONS AND CONCENTRATIONS EXIST WITHIN THE FULL SCALE PLUME USING PREDICTIVE TECHNIQUES. MODELING CAN BE DONE BEFORE AN OUTFALL IS BUILT OR AT CONDITIONS OTHER THAN THOSE WHERE MEASUREMENTS WERE MADE. MATHEMATICAL MODELING IS MUCH CHEAPER THAN FIELD STUDIES AND COVER A MUCH WIDER RANGE OF VARIABLES.

DILUTION

- DILUTION, S, IS DEFINED AS THE TOTAL FLOW IN THE PLUME (EFFLUENT PLUS ENTRAINED FLUID) OVER THE EFFLUENT FLOW, Q/Q_o. IF THE POLLUTANT CONCENTRATION IS ZERO IN THE AMBIENT THE AVERAGE CONCENTRATION IN THE PLUME $C_i = C_o/S$. IF THE AMBIENT CONCENTRATION IS NOT ZERO:

$$C_i = \frac{C_o + C_a(S-1)}{S}$$

WHERE Co IS THE EFFLUENT CONCENTRATION, Ca IS THE AMBIENT CONCENTRATION. CENTERLINE CONCENTRATIONS ARE APPROXIMATELY TWICE THE AVERAGE VALUES. OVERLAPPING OF PLUMES FROM OTHER SOURCES CAUSE CONCENTRATIONS TO BE ADDITIVE IN THOSE AREAS WHERE PLUMES OVERLAP.

DENSITY STRATIFICATION IN THE AMBIENT IS VERY IMPORTANT. EVEN THE SLIGHTEST STRATIFICATION CAN CAUSE THE PLUME TO BE TRAPPED AND LIMIT DILUTION. THE DISTANCE TO WHERE THE PLUME TRAPS CAN CHANGE DRASTICALLY WHEN STRATIFICATION PROFILES ARE ONLY SLIGHTLY VARIED. AS A RESULT, AMBIENT STRATIFICATION SHOULD BE AS ACCURATE AS POSSIBLE.

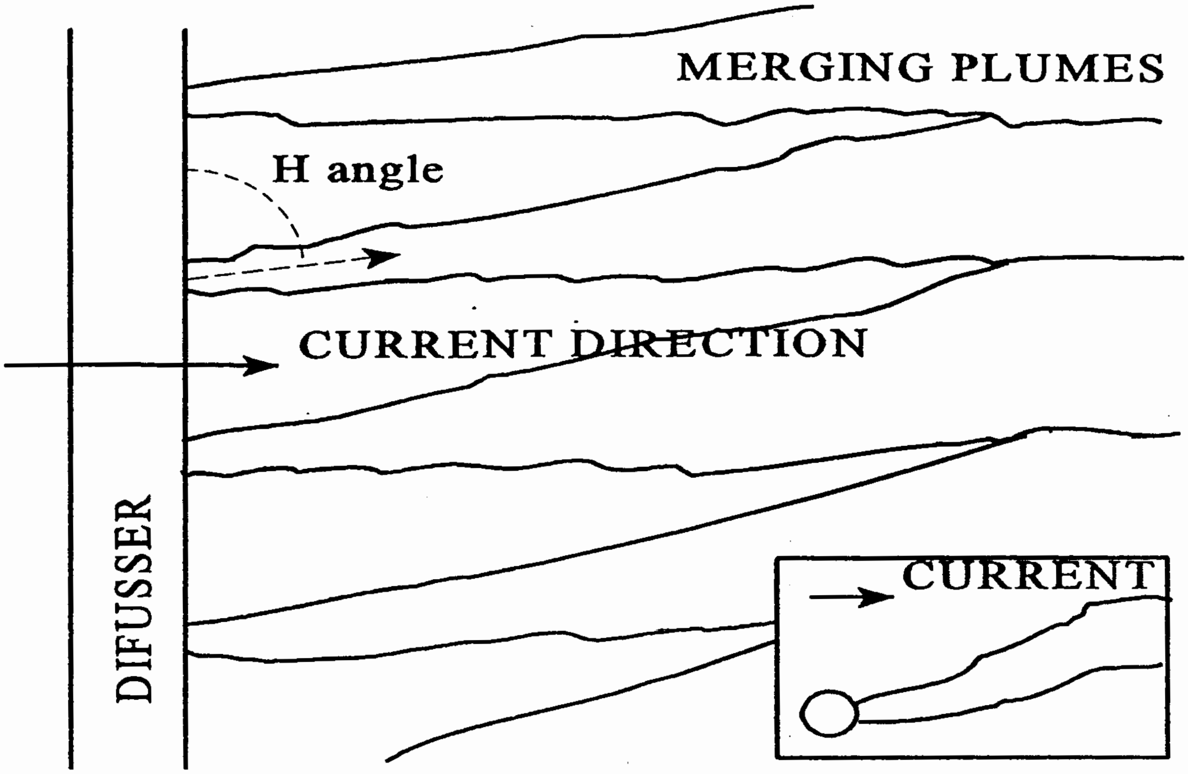
TYPES OF DISCHARGES

1. DISCHARGE CAN BE AT THE SURFACE FROM A CHANNEL OR PIPE. THE RESULTING PLUME MAY SINK BUT USUALLY STAYS AT THE SURFACE OF THE RECEIVING WATER.

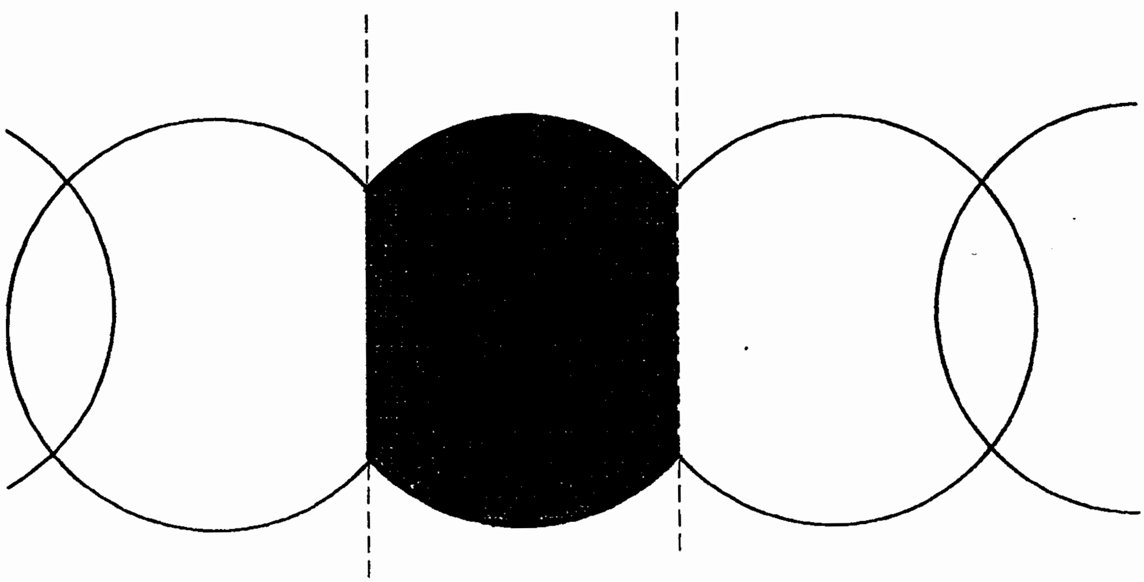
2. DISCHARGE BELOW THE SURFACE THROUGH A SINGLE ORIFICE OR MULTIPLE PORT DIFFUSER. DISCHARGE ANGLES CAN BE IN ANY DIRECTION.

SUBMERGED DISCHARGES MAY RISE TO THE SURFACE AND BECOME A SURFACE PLUME OR SPREAD UNTIL THE PLUME FILLS THE WATER COLUMN. AS LONG AS A PLUME IS SUBMERGED AND NOT TRAPPED, IT DILUTES RAPIDLY. AS A RESULT, THE DEEPER THE DISCHARGE THE BETTER.

NEIGHBORING PLUMES FROM MULTIPLE PORT DIFFUSERS MAY MERGE AND BECOME AS A LONG LINE PLUME. MERGING REDUCES THE DILUTION RATE.



LINES OF SYMMETRY



MERGING PLUMES

IN THIS CLASS WE WILL BE LOOKING AT SOME OF THE MODELS ACCEPTED BY THE U.S. EPA FOR PREDICTING THE DILUTION FROM ENVIRONMENTAL DISCHARGES. THERE ARE MANY OTHER MODELS, HOWEVER THESE ARE THE ONES USED MOST OF THE TIME FOR POINT SOURCE AND DIFFUSER DISCHARGES.

KNOWING HOW TO RUN THE MODELS AND INTERPRETING AND USING THE RESULTS ARE TWO DIFFERENT PROBLEMS ALTOGETHER. THE USER MUST BE EXPERIENCED ENOUGH TO KNOW IF THE RESULTS ARE MEANINGFUL OR NOT. MODELS WILL GIVE ANSWERS WITHOUT KNOWING WHETHER THEY VIOLATE PHYSICAL LAWS.

WE WILL BE LOOKING AT:

1. THE PLUMES INTERFACE AND THE 2 DIMENSIONAL SUBMERGED DIFFUSER MODEL, UM . (UM IS AN UPDATED UMERGE)
2. THE 3-DIMENSIONAL SUBMERGED DIFFUSER MODEL, UDKHDEN.
3. THE 3-DIMENSIONAL SURFACE PLUME MODEL, PDS.
4. THE 2-DIMENSIONAL SHALLOW RIVER MODEL, PSY.
5. AND THE EMPIRICAL CORMIX 1-2-3 MODELS THAT COVER ALL OF THE ABOVE.

WE WILL SPEND ABOUT 1/2 A DAY GOING OVER THE THEORY OF EACH, HOW TO USE IT, AND DEMONSTRATE SAMPLE PROBLEMS. THE NEXT 1/2 DAY WILL BE INDIVIDUAL HANDS ON WORK, USING THE MODEL TO SOLVE SAMPLE PROBLEMS.

WE WILL DO THIS FOR EACH OF THE MAIN MODELS. FRIDAY AFTERNOON WILL BE TIME FOR A DISCUSSION ON ACCURACY AND A QUESTION-ANSWER PERIOD (IF ANYONE CAN STICK IT OUT THAT LONG)

FUNDAMENTALS OF THE INTEGRAL METHOD

FLOW RATE =

$$Q = \int\int_{AREA} U dA$$

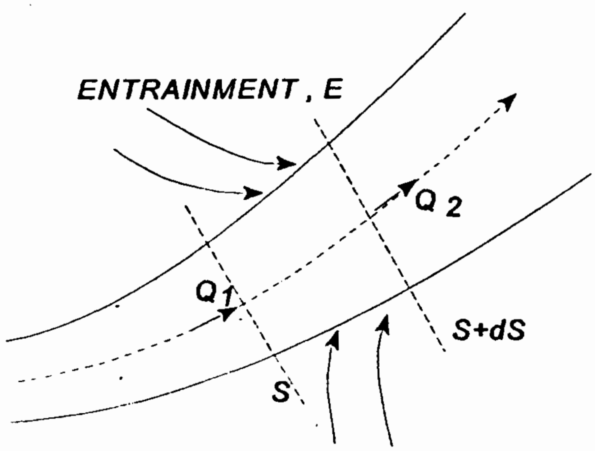
USING AN APPROPRIATE VELOCITY PROFILE AND INTEGRATING THE VOLUMETRIC FLOW RATE

$$Q = K_1 U_C B^2$$

WHERE K_1 IS A CONSTANT DEPENDING ON THE PROFILE, U_C IS THE CENTERLINE VELOCITY AND B IS THE PLUME RADIUS

$$Q_2 - Q_1 = E\, ds$$

WHERE Q_2 IS THE FLOW AT s + ds AND Q_1 IS THE FLOW AT s, E IS THE ENTRAINMENT AND ds IS THE DISTANCE BETWEEN 1 AND 2



$$E \approx \alpha B U_C$$

PLUME MOMENTUM IS GIVEN BY

$$M = \int\int_{AREA} \rho U^2 dA$$

INTEGRATING WITH THE SAME PROFILE GIVES

$$M = K_2 U_C^2 B^2$$

WHERE K_2 IS ANOTHER CONSTANT DEPENDING ON THE PROFILE

$$M_2 - M_1 = (F_B + F_D + EU_A)ds$$

WHERE M_2 IS THE MOMENTUM AT s + ds AND M_1 IS THE MOMENTUM AT s, F_B IS THE BUOYANCY FORCE DUE TO DENSITY DIFFERENCES BETWEEN PLUME AND AMBIENT, F_D IS THE DRAG FORCE AMBIENT CURRENTS IMPOSE ON PLUME, AND EU_A IS THE MOMENTUM OF THE FLUID ENTRAINED INTO THE PLUME.

WITH THE TERMS AT 1 AND ON ON THE R.H.S OF THE EQUATIONS ARE KNOWN, THE VALUES OF M AND Q AT 2 CAN BE CALCULATED AND THE FOLLOWING RESULT

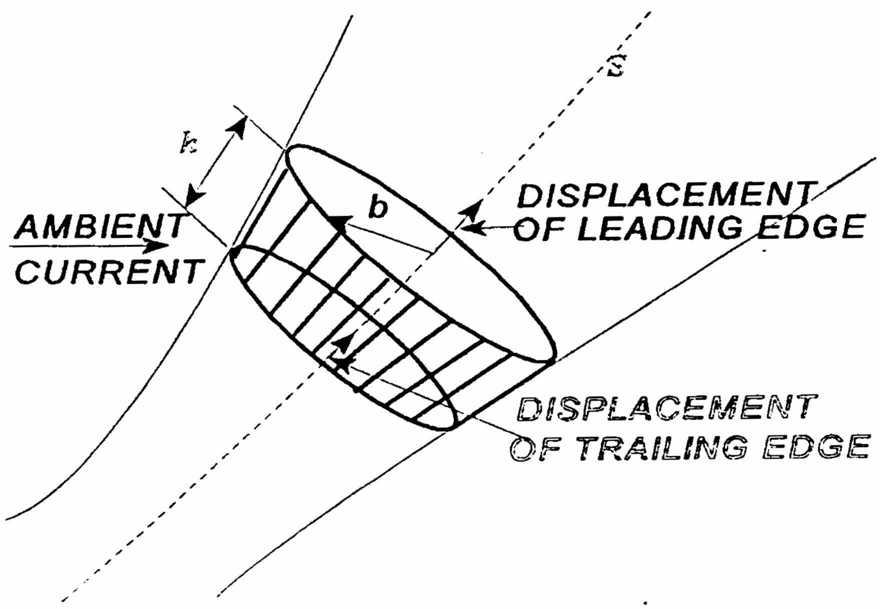
$$U_{C2} = \frac{K_1}{K_2}\frac{M_2}{Q_2}$$

$$B_2 = \sqrt{\frac{Q_2}{K_1 U_{C2}}}$$

PLUMES - <u>UM</u> MODEL

UM IS AN INTEGRAL DIFFERENTIAL MODEL BASED ON A LAGRANGIAN FORMULATION. --- IT FOLLOW A "PUFF" OF MATERIAL IN THE PLUME AS IT GROWS AND IS TRANSPORTED BY THE AMBIENT. IT CONSIDERS BOTH POSITIVE AND NEGATIVE BUOYANCY (RISING AND SINKING PLUMES)

THE SHAPE OF THE PUFF IS A SLICE THROUGH THE PLUME AS SHOWN BELOW.



THE CONSERVATION OF MASS IS EXPRESSED AS:

$$m_{t+dt} = m_t + [\rho A_p U_a + \alpha\rho(2\pi bh)\bar{V}]dt$$

where:

m_{t+dt} is the mass in slice at time t+dt
m_t is the mass in slice at time t
ρ is the density of the fluid in the plume
U_a is the ambient velocity
A_p is the projected area of the slice facing the ambient
α is an aspiration entrainment coefficient
b is the plume radius
h is the slice thickness
$\bar{V}$ is the shear velocity between plume and ambient

THE MOMENTUM (M = mV) EQUATION IS EXPRESSED AS:

$$\bar{M}_{t+dt} = \bar{M}_t + \left[\bar{U}_a \frac{dm}{dt} - m\frac{\rho_a - \rho}{\rho}\bar{g}\right]dt$$

The first term in the bracket is momentum added to the plume from entrained ambient. The second is due to plume buoyancy. The integration of these two equations is a single forward step integration

THE CONSERVATION OF ENERGY, SALINITY, AND POLLUTANT ARE EXPRESSED AS:

$$\frac{dmT}{dt} = T_a \frac{dm}{dt}$$

$$\frac{dmS}{dt} = S_a \frac{dm}{dt}$$

$$\frac{dmX}{dt} = X_a \frac{dm}{dt} - kmX$$

where T, S and X are the temperature, salinity, and species concentration respectively.

Subscript (a) refers to ambient values. The -kmX term is a first order decay term that is possible with some pollutants such as coliform.

SINCE THE MASS IN THE PLUME IS GIVEN BY $m = \rho\pi b^2 h$, THE NEW PLUME RADIUS IS GIVEN BY

$$b_{t+dt} = \sqrt{\frac{m_{t+dt}}{\pi\rho_{t+dt}h_{t+dt}}}$$

FROM THE CONSERVATION EQUATIONS, TEMPERATURE AND CONCENTRATIONS CAN BE OBTAINED FROM:

$$(mT)_{t+dt} = (mT)_t + [T_a(\rho A_p U_a + \rho A_t \bar{V})dt]$$

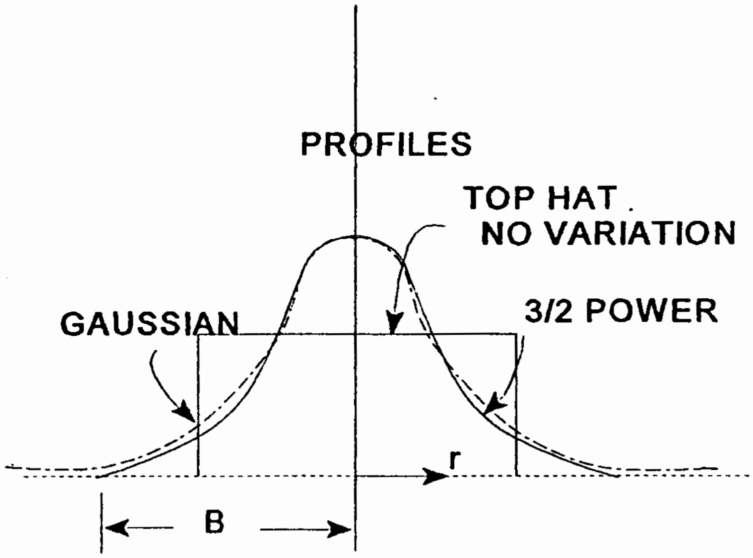
$$(mX)_{d+dt} = (mX)_t + [X_a(\rho A_p U_a + \rho A_t \bar{V}) - kmX]dt$$

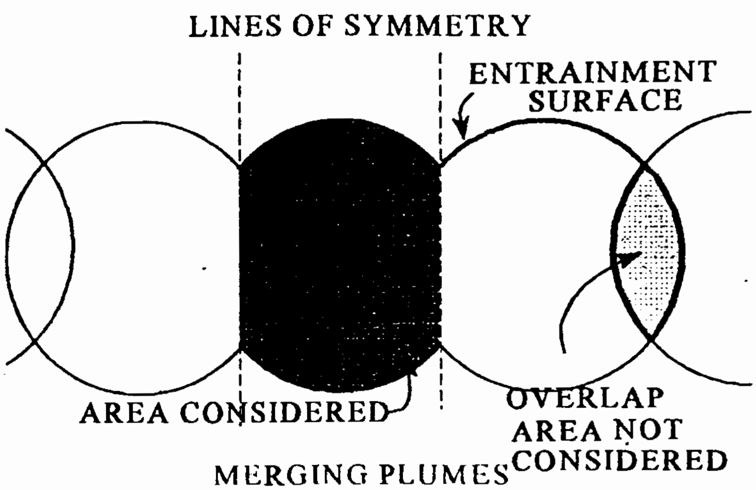
DIVIDING THESE EQUATIONS BY m_{t+dt} GIVES THE NEW TEMPERATURE AND CONCENTRATION AT t+dt.

THE NEW PLUME VECTOR VELOCITY IS OBTAINED FROM THE MOMENTUM EQUATION IN A SIMILAR WAY .

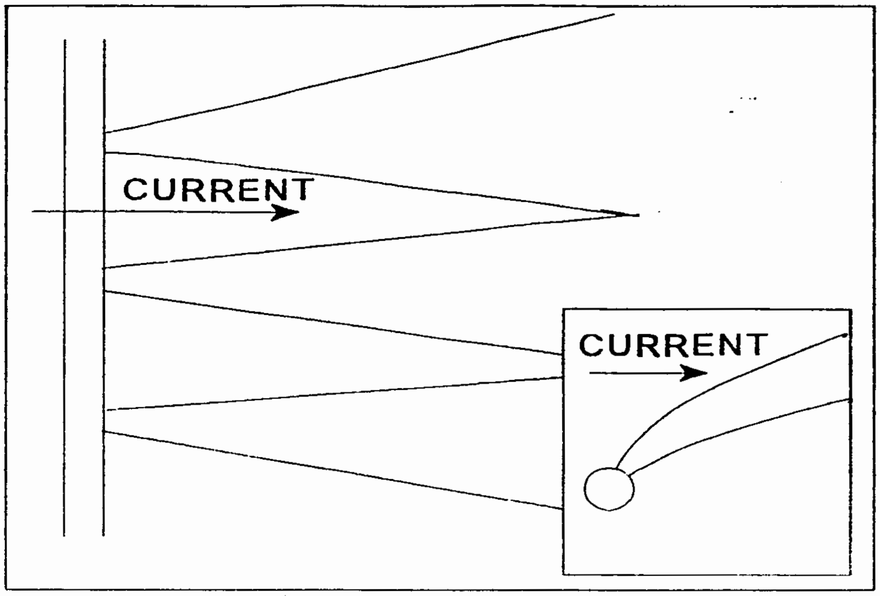
THE NEW POSITION IS OBTAINED FROM THE AVERAGE VELOCITY BETWEEN t AND t+dt TIMES dt.

PLUME MERGING IS GEOMETRIC AND EFFECTS THE ENTRAINMENT AREAS ONLY SINCE PROFILES ARE ASSUMED TO BE "TOP HAT" AS SHOWN BELOW.


PROFILES
TOP HAT
NO VARIATION
GAUSSIAN
3/2 POWER
r
B


LINES OF SYMMETRY
ENTRAINMENT
SURFACE
AREA CONSIDERED
OVERLAP
AREA NOT
CONSIDERED
MERGING PLUMES

- UM IS ONLY 2-DIMENSIONAL AND TREATS MULTIPLE PORT DIFFUSERS PERPENDICULAR TO THE AMBIENT CURRENT AS SHOWN BELOW!!



- UM INCLUDES A FIRST ORDER (VERY APPROXIMATE) FAR FIELD, AMBIENT DIFFUSION MODEL (Brooks Model) GIVEN FOR OPEN OCEAN BY:

$$S = \frac{S_i}{erf\left(\sqrt{\frac{1.5}{\left(1 + 8Kw^{1.33}\frac{t}{w^2}\right)^3 - 1}}\right)}$$

where:

erf = the standard error function

K = a dispersion coefficient

FOR RIVERS AND SMALL LAKES IT IS GIVEN BY:

$$S = \frac{S_i}{\operatorname{erf}\left(\sqrt{\frac{w^2}{16Kw^{4/3}t}}\right)}$$

WHERE S_i IS THE DILUTION AT THE BEGINNING OF THE FIELD, w IS THE WIDTH OF THE FIELD, t IS TIME AND K IS A DISPERSION COEFFICIENT.

PLUMES INPUT

THE PROGRAM "PLUMES" IS ACTUALLY AN INTERFACE USED TO SETUP AND CHECK INPUT TO THE epa PLUME PREDICTION PROGRAMS. THE OUTPUT FROM THE PLUMES PROGRAM IS IN A FORMAT THAT IS SUITABLE FOR THE UM, UDKHDEN, RSB, AND ULINE PROGRAMS. UM AND RSB CAN BE RUN DIRECTLY FROM THE PLUMES INTERFACE. TO RUN UDKHDEN, THE PLUMES OUTPUT FILE CAN BE SAVED ON A UDF.IN (UNIVERSAL DATA FILE).

THE PLUMES INTERFACE IS CONVENIENT SINCE INPUT UNITS CAN BE SELECTED AND CONSISTENCY IS CHECKED. DEPENDENT VARIABLES ARE AUTOMATICALLY CALCULATED.

THE VARIABLES ON THE PLUMES INTERFACE SPREAD SHEET ARE:

TITLE:	OPTIONAL TITLE OF PROJECT
TOT FLOW:	TOTAL DISCHARGE FLOW RATE
PORTS:	NUMBER OF PORTS IN DIFFUSER
PORT FLOW:	FLOW PER PORT
SPACING:	SPACING BETWEEN PORTS
EFFL SAL:	SALINITY OF EFFLUENT
EFFL TEMP:	TEMPERATURE OF EFFLUENT

FAR INC:	INCREMENT BETWEEN PRINTOUTS IN FAR FIELD CALCULATIONS
FAR DIS:	DISTANCE FAR FIELD CALCULATIONS ARE TO BE CONSIDERED
PORT DEP:	AVERAGE DEPTH OF PORTS FROM SURFACE
PORT DIA:	DIAMETER OF PORTS (ALL THE SAME)
PLUME DIA:	PLUME DIAMETER AT DISCHARGE (IF CONTRACTION COEFFICIENT IS DIFFERENT FROM 1.0)
TOTAL VEL:	TOTAL VELOCITY AT DISCHARGE PORT
HORIZ VEL:	HORIZONTAL COMPONENT OF VELOCITY
VERT VEL:	VERTICAL COMPONENT OF VELOCITY

ASP COEF:	ASPIRATION COEFFICIENT USED IN ENTRAINMENT (USE DEFAULT UNLESS YOU KNOW OTHERWISE)
PRINT FRQ:	THE ITERATION FREQUENCY THAT YOU WANT OUTPUT VALUES (A VALUE OF 10 GIVES LOTS OF OUTPUT. A VALUE OF 500 NOT VERY MUCH). WHAT YOU USE DEPENDS ON HOW MUCH DETAIL YOU NEED.
PORT ELV:	ELEVATION OF PORTS OFF BOTTOM
VERT ANG:	PORT DISCHARGE ANGLE RELATIVE TO THE HORIZONTAL(0 IS HORIZONTAL. 90 IS VERTICAL)(CAN VARY BETWEEN -90 AND 90)

CONT.COEF:	CONTRACTION COEFFICIENT (FRACTION OF JET DIAMETER TO PORT DIAMETER)
EFFL. DEN:	DENSITY OF EFFLUENT (AUTOMATICALLY CALCULATED IF TEMPERATURE AND SALINITY ARE INPUT)
POLL CONC:	CONCENTRATION OF POLLUTANT IN THE DISCHARGE (ANY VALUES CAN BE USED)
DECAY:	DECAY COEFFICIENT PER DAY (ZERO FOR NONE)

FROUDE: DENSIMETRIC FROUDE NUMBER DEFINED AS

$$F_o = \frac{U_o}{\sqrt{gD_o \frac{\Delta\rho}{\rho}}} = \frac{MOMENTUM}{BUOYANCY}$$

VALUES WITH RED BOXES ARE USUALLY NOT OF INTEREST EXCEPT TO MODELERS.

HOR ANG: HORIZONTAL ANGLE OF OUTFALL PIPE (90 DEGREES MEANS THE OUTFALL PIPE IS 90 TO CURRENT AND DISCHARGE IS INLINE WITH CURRENT)(IT CAN BE BETWEEN 45 AND 135 FOR UDKHDEN) UM JUST MAKES THE PORTS CLOSER TOGETHER IF THIS VALUE IS DIFFERENT FROM 90. SEE RED SPACE:

RED SPACE: EFFECTIVE SPACING IN UM MODEL IF HOR ANG IS DIFFERENT FROM 90

P AMB DEN: AMBIENT DENSITY AT PORT DEPTH

P CURRENT: AMBIENT CURRENT AT PORT DEPTH

FAR DIF: FAR FIELD DIFFUSION COEFFICIENT (USE THE DEFAULT VALUE UNLESS YOU KNOW BETTER)

^L	THIS IS THE LIST EQUATIONS OR GIVE DEFINITION OF THIS CELL. IF YOU DON'T KNOW WHAT SHOULD GO IN A GIVEN CELL, PRESS ^L AND IT WILL TELL YOU.
^B	CAUSES THE INTERFACE TO RUN THE RSB MODEL
^U	CAUSES THE INTERFACE TO RUN THE UM MODEL
^J	CAUSES THE CURSOR TO JUMP FROM ONE TYPE OF INPUT ZONE TO ANOTHER.
^W	TELLS INTERFACE TO GET AN EXISTING WORKING FILE. A TYPEOVER WINDOW IS PROVIDED FOR FILE NAME INPUT. THE EXISTING ACTIVE FILE IS STORED AND THE NEW FILE IS OPENED.

IF THE FILE DOES NOT EXIST IT IS CREATED AND FILLED WITH DEFAULT DATA.

^N	TELLS THE INTERFACE TO STORE THE PRESENT DATA IN A NEW FILE. YOU ARE PROMPTED FOR A NEW NAME AND WHICH CASES YOU WANT SAVED. (YOU CAN SEVERAL CASES UNDER ONE FILE NAME)
^C	CREATE A NEW CASE UNDER THE SAME FILE OR GO TO DIFFERENT CASE
^Z	TELLS THE INTERFACE TO USE THE IMAGE SOLUTION TO SIMULATE SHALLOW WATER. THIS OPTION IS ONLY FOR SINGLE PORTS.
^I	SHOWS THE INDEPENDENT VARIABLES SELECTED
^O	ADDS THIS VARIABLE TO THE OUTPUT TABLE

FAR VEL: AVERAGE CURRENT IN THE FAR FIELD

AMBIENT TABLE

DEPTH: IS THE DISTANCE BELOW THE SURFACE WHERE AMBIENT VALUES ARE TO BE GIVEN

CURRENT: IS THE AMBIENT CURRENT AT THIS DEPTH

DENSITY: IS THE AMBIENT DENSITY AT THIS DEPTH

SALINITY: IS THE SALINITY AT THIS DEPTH

TEMP: IS THE TEMPERATURE AT THIS DEPTH

AMB CONC? IS THE POLLUTANT CONCENTRATION AT THIS DEPTH

COMMANDS

COMMANDS WITHIN THE PLUMES INTERFACE ARE AS FOLLOWS. ^ MEANS HOLD THE CONTROL KEY DOWN

<SPACE> MOVES THE CURSOR FROM ONE CELL TO THE NEXT. THE ARROW KEYS CAN ALSO BE USED TO PLACE THE CURSOR WHERE YOU WANT IT, BUT ARROW KEYS ONLY ONE DIGIT AT A TIME IN FORWARD HORIZONTAL MOTION

<ENTER> DOES THE SAME THING AS A SPACE BAR.

^K CHANGE IN THE INPUT UNITS OF THIS CELL. SUCCESSIVE ENTRIES OF THIS COMMAND CYCLES THROUGH THE ACCEPTED UNITS.

^R	BRINGS UP THE CONFIGURATION MENU. IT ALLOWS YOU TO SELECT THE TYPE OF CONFIGURATION YOU WANT. YOUR PRESENT CONFIGURATION IS GIVEN IN THE CENTER OF THE BOTTOM LINE. i.e. ATNO0 WHICH MEANS AUTOMATIC AMBIENT FILL, TRANSMIT BROOKS EQUATION, CORMIX CATEGORIZATION NOT ACTIVE. 0 BRUNT VAISALA REVERSALS CONSIDERED.
^Y	BRINGS UP A POP-UP MENU WITH MISCELLANEOUS OTHER COMMANDS INCLUDING:

^YF	CAUSES THE AMBIENT TABLE TO AUTOMATICALLY FILL EACH CELL WITH THE VALUE ABOVE
^YI	CAUSES THE PROGRAM TO INTERPOLATE BETWEEN CELLS TO FILL THE INTERMEDIATE ONES
^YC	COPIES THE COMPLETE AMBIENT LINE
^YD	DELETES AN AMBIENT LINE
^YU	COPIES THE VALUES IN THE INPUT CELLS TO A FILE CALLED "UDF.IN" FOR USE WITH OTHER PROGRAMS SUCH AS UDKHDEN

^YZ CLEARS (ZAPS) MOST CELLS IN THE INTERFACE (DEFAULT VALUES REMAIN)

THE F1 FUNCTION KEY BRINGS UP THE POP-UP MENU WITH MAIN COMMANDS.

VARIABLES SHOWN WITH YELLOW NUMBERS ARE THOSE YOU SELECTED AS INDEPENDENT VARIABLES.

VARIABLES SHOWN WITH WHITE NUMBERS ARE DEPENDENT VARIABLES AND ARE CALCULATED BY THE INTERFACE. IF YOU TRY TO CHANGE ANY OF THE WHITE NUMBERS, THE INDEPENDENT VALUES FLASH AND YOU MUST DELETE ONE TO MAKE IT DEPENDENT.

SAMPLE PROBLEM

A DIFFUSER CONSISTING OF 285 PORTS ON ALTERNATE SIDES OF A DIFFUSER. THEY ARE SPACED 24' APART ON A SIDE (12' APART ON ALTERNATE SIDES)

TOTAL FLOW = 102 MGD (MILLION GALLONS PER DAY)
EFFLUENT TEMP = 25 C
EFFLUENT SALINITY = 0.0 PPT
FAR FIELD OF INTEREST = 2000 m
FAR FIELD INC NOT IMPORTANT, SAY 500
DEPTH OF CENTER PORT = 70 m (MLLW)
PORT DIAMETER = 8.5 cm
PRINT FREQ = 100 (GIVE OUTPUT VALUES AT DILUTIONS OF APPROXIMATELY (1,2,4,8,16,....)

PORT ELEV.. = 0.84 m
VERT ANG = HORIZONTAL = 0.0 (DEFAULT)
CONT COEF = 1.0 (DEFAULT)
POLLUT CONC. = 100
DECAY = $c = c_o e^{-kt}$ IN OUR PROBLEM IT TAKES 1 HOUR FOR 90% OF THE BACTERIA OF CONCERN TO DIE, THEREFORE t90hr = 1 USE ^K TO BRING UP t90hr.
HORIZ ANG = 90
FAR FIELD VEL = 15 cm/s

AMBIENT TABLE

DEPTH	CURR	DENS	SAL	TEMP	CONC
0.00	0		34.99	26.18	0
30.48	0		35.00	25.60	0
45.72	0		35.02	24.95	0
60.96	0		35.00	24.60	0
76.20	0		35.02	21.22	0

DETERMINE THE DILUTION AND CONCENTRATION AT THE TRAPPING LEVEL AND AT THE END OF THE FAR FIELD (2000 m)

RUN UM BY PRESSING ^U AND SEND THE OUTPUT TO THE CONSOLE

RESULTS SHOW S=84.58 AND C=1.15 AT THE TRAPPING LEVEL
S=172.3(164.2) AND C=0.485 (.508) AT THE END OF THE FAR FIELD DEPENDING ON THE 4/3 POWER LAW OR EDDY DIFFUSION MODELS

IT WOULD BE MORE CONSERVATIVE TO ASSUME ALL PORTS ARE ON ONE SIDE OF THE DIFFUSER WITH A SPACING OF 12'. MAKE THE CHANGE AND SAVE THE OUTPUT TO A FILE CALLED SAND.OUT

YOU DON'T SEE IT THIS TIME SINCE IT WAS SAVE ON FILE. YOU CAN PRINT THE SAND.OUT OR LOOK AT IT WITH A TEXT EDITOR. YOU CAN ALSO RERUN THE PROGRAM AND SELECT CONSOLE TO VIEW THE OUTPUT. THE TRAPPING LEVEL VALUES DON'T CHANGE SINCE THE PLUME STILL HAVEN'T MERGED BUT THE FAR FIELD VALUES REDUCE TO S=167(149) AND C=0.50(0.56) DUE TO A SMALLER INITIAL WASTE FIELD SIZE.

NOW LET'S MAKE FOUR CASES

1: A CURRENT OF 10 CM/S
2: A CURRENT OF 10 CM/S BUT NO DECAY
3: NO CURRENT AND NO DECAY
4: NO CURRENT AND NO DECAY BUT A FLOW OF 250 MGD.

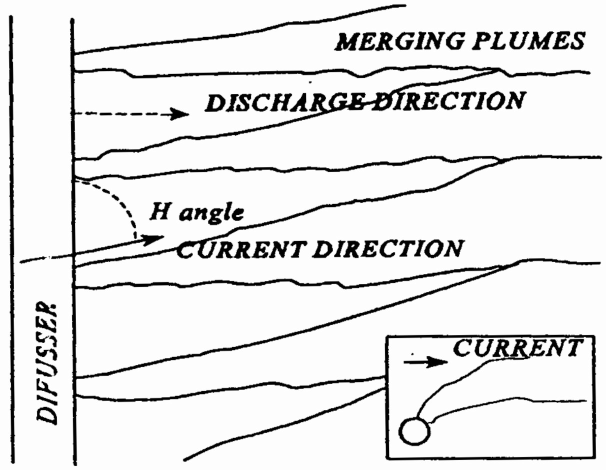
SAVE THE INPUT FOR THESE FOUR AS FILE VARY.VAR. RUN ALL CASES AND SAVE THE OUTPUT ON VAR.OUT. RUN CASE 4 AND SHOW IT ON THE SCREEN.

FIND OUT WHICH IS THE WORST CASE (LEAST DILUTION AT THE TRAPPING LEVEL AND CONCENTRATION AT THE END OF THE FAR FIELD)

TO RUN ALL CASES AT ONCE YOU MUST GO TO CASE 1 AND THEN RUN 4 CASES

ANSWER: WORST CASE IS NO CURRENT AND NO DECAY. THE EFFECT OF FLOWRATE IS MINOR
S=76, C=.81

UDKHDEN



UDKHDEN IS A 3-DIMENSIONAL, INTEGRAL PLUME MODEL THAT CALCULATES THE CHARACTERISTICS OF A LINE OF EQUALLY SPACED BUOYANT DISCHARGES INTO A FLOWING STRATIFIED AMBIENT.

THE METHOD OF SOLUTION INVOLVES 7 ORDINARY DIFFERENTIAL EQUATIONS WHICH ARE :

1. CONSERVATION OF MASS
2. CONSERVATION OF ENERGY
3. CONSERVATION OF CONCENTRATION
4. DENSITY DEFICIENCY

5.6.7. MOMENTUM EQNS IN THE X (HORIZONTAL AND NORMAL TO THE AMBIENT CURRENT), Z (VERTICAL), Y (HORIZONTAL AND PARALLEL TO AMBIENT CURRENT DIRECTION)

- EQN 4 IS USED ONLY IN ALGEBRAIC FORM. THE SIX OTHER EQUATIONS ARE WRITTEN FOR A CONTROL VOLUME WHICH IS FINITE IN A DIRECTION PERPENDICULAR TO THE JET AND INFINITESIMAL IN THE DIRECTION OF THE JET AXIS.

- DISCHARGE AT ANY ANGLE (WITHIN LIMITS) RELATIVE TO THE AMBIENT CURRENT ARE SIMULATED.

A SINGLE PORT DISCHARGE CAN BE MODELED BY SPACING THE PORTS WIDE APART SO MERGING DOES NOT OCCUR.

- IT ALLOWS FOR DEPTH VARYING AMBIENT VELOCITY, TEMPERATURE AND SALINITY OR DENSITY.

THE INPUT TO UDKHDEN IS IN THE FORMAT OF A "UNIVERSAL DATA FILE" THAT IS GENERATED BY THE THE EPA'S PLUMES MODEL.

NORMALLY YOU DO NOT NEED TO WORRY ABOUT THE FORMAT OF THE INPUT FILE. FOR COMPLETENESS, HOWEVER, IT IS GIVEN IN FOLLOWING EXAMPLE:

SAMPLE INPUT FOR UDKHDEN

```
1 1 0 0 0 0 0 0 0 0 0 0 0 0
    4.       5    0.500     12.      20.     0.10       1.0
 0.0100     90.    1000.     500.    2000.  0.000453      0.01
    4      0.0     20.  0.99827     100.      0.  0
  0.0      10.     10.     0.01   1.00756      2.
  10.      25.     10.     0.01   1.02542      2.
  11.      30.     10.     0.01   1.02542      2.
  21.      33.     10.     0.01   1.02542      2.
```

***** INPUT VARIABLES *****

*** RECORD NO. 1 *** (FIRST LINE)

1. THIS RECORD CONTAINS ANY INFORMATION THE USER WISHES TO HAVE PRINTED OUT AT THE TOP OF EACH PAGE.

THE DATA ENTERED ON THE REMAINING RECORDS ARE ENTERED IN FREE OR LIST DIRECTED FORMAT. EACH VALUE ON THE LINE MAY BE SEPARATED BY A COMMA OR SPACE.

*** RECORD NO. 2 *** (SECOND LINE)

1. INTER. IF INTER=0, ONLY ONE RUN OF THE PRESENT CASE IS TO BE MADE. IF INTER=1, ONE OR MORE RUNS USING THE INPUT DATA FOR THIS CASE ARE TO BE MADE WITH ONE OR MORE OF THE INPUT VARIABLES CHANGED. THE USER WILL BE PROMPTED AT THE END OF EACH RUN WHICH VARIABLES HE WISHES TO CHANGE.

2. IDFP. IF IDFP=1, THE INPUT FILE IS PRINTED AS PART OF THE OUTPUT. IF IDFP=0, IT IS NOT PRINTED.

THE REMAINDER OF THE VARIABLES ON THIS LINE ARE NOT USED BY UDKHDEN. ENTERING 12 ZEROS SEPARATED BY A SPACE FOLLOWING THE FIRST TWO VARIABLES WILL INSURE PROPER EXECUTION. THE VARIABLES ON THIS LINE ARE FLAGS TO THE [illegible] EPA PROGRAMS

*** RECORD NO. 3 *** (THIRD LINE)

1. QT=TOTAL DISCHARGE FLOW RATE (CUBIC METERS PER SEC)
2. NP=NUMBER OF DISCHARGE PORTS
3. PDIA=DISCHARGE PORT DIAMETER (M)
4. VANG=VERTICAL ANGLE (DEG) OF PORTS RELATIVE TO HORIZONTAL (90 DEGREES IS VERTICAL)(LIMITED TO -5 TO 130)
5. PDEP=PORT DEPTH (M) AND MUST BE GREATER THAN 0.0

THE OTHER VARIABLES ON THIS LINE ARE NOT USED BY UDKHDEN. ENTER ANY NUMBER IN THEIR PLACE.

*** RECORD NO. 4 ***

1. UW NOT USED IN THIS PROGRAM.
2. HANG=ANGLE (DEG) OF CURRENT DIRECTION WITH RESPECT TO DIFFUSER AXIS (90 DEGREES CORRESPONDS TO A CURRENT HAVING A DIRECTION PERPENDICULAR TO THE DIFFUSER AXIS, RANGE 45 - 135 DEG). FOR SINGLE PORTS. THIS ANGLE CAN BE ANYTHING.
3. SPACE=DISTANCE BETWEEN ADJACENT PORTS (M) (IF THE NUMBER OF PORTS IS 1 ON LINE 3 THE SPACE IS SET TO 1000 TO PREVENT MERGING.

THE OTHER VARIABLES ON THIS LINE ARE NOT USED BY UDKHDEN. ENTER ANY NUMBER IN THEIR PLACE.

*** RECORD NO. 5 ***

1. NPTS=NUMBER OF ELEVATION ENTRIES IN AMBIENT PROFILE TABLE (NPTS MUST BE AT LEAST 2 BUT NOT MORE THAN 30).
2. S=DISCHARGE SALINITY (PPT) IF TEMPERATURE/SALINITY OPTION USED OR DISCHARGE DENSITY (GM/CC) IF DENSITY OPTION USED.
3. T=DISCHARGE TEMPERATURE (DEG C) IF TEMPERATURE/SALINITY OPTION USED OR ZERO (0.0) IF DENSITY OPTION USED.

THE OTHER VARIABLES ON THIS LINE ARE NOT USED BY UDKHDEN. ENTER ANY NUMBER IN THEIR PLACE.

*** RECORD NO. 6 ***

THERE MUST BE NPTS (RECORD NO. 5) IMAGES OF RECORD NO. 6. ONE RECORD FOR EACH ELEVATION WHERE AMBIENT CONDITIONS ARE GIVEN. AT LEAST 2 RECORDS BUT NOT MORE THAN 30.

1. DP( )=DEPTH TO DATA POINT (M) STARTING AT THE SURFACE AND WORKING DOWN TO OR BELOW THE DISCHARGE DEPTH. THE LAST ENTRY MUST BE FOR A DEPTH EQUAL TO OR GREATER THAN THE DISCHARGE DEPTH. MUST HAVE DATA FOR DP(1)=0.0 OR COMPUTATIONAL ERRORS MAY OCCUR.
2. SA( )=AMBIENT SALINITY (PPT) AT THIS DEPTH IF THE TEMPERATURE/SALINITY OPTION IS USED OR AMBIENT DENSITY (GM/CC) IF DENSITY OPTION IS USED.
3. TA( )=AMBIENT TEMPERATURE (DEG C) AT THIS DEPTH IF TEMPERATURE/ SALINITY OPTION USED OR ZERO (0.0) IF DENSITY OPTION USED.
4. UA( )=AMBIENT VELOCITY AT THIS DEPTH (M/S)

THE OTHER VARIABLES ON THESE LINES ARE NOT USED BY UDKHDEN. ENTER ANY NUMBER IN THEIR PLACE.

REGARDING RECORD NO 5 AND 6, WHEN USING THE TEMPERATURE/SALINITY OPTION, IF FOR AN SA( ) OR S, THE TA( ) OR T IS REALLY ZERO, GIVE TA( ) OR T A SMALL VALUE LIKE 0.000001.

THE OUTPUT FROM UDKHDEN IS AS FOLLOWS

THE PROGRAM ECHOES THE INPUT VARIABLES THEN CALCULATES THE DISCHARGE FROUDE NUMBER AND PORT TO DIAMETER SPACING. IT THEN GOES INTO A SOPHISTICATED ROUTINE THAT CALCULATES THE CHARACTERISTICS OF THE PLUME IN THE ZONE OF FLOW ESTABLISHMENT NEAR THE SOURCE AND PRINTS OUT THE LENGTH OF THE DEVELOPMENT ZONE OR STARTING LENGTH. THERE IS USUALLY A SLIGHT PAUSE WHILE IT DOES THIS. THEN FOR SELECTED INTEGRATION STEPS IT PRINTS OUT THE FOLLOWING:

X - CROSS CURRENT DISTANCE
Y - DOWN CURRENT DISTANCE
Z - VERTICAL DISTANCE FROM SOURCE

THESE THREE COORDINATES GIVE THE THREE-DIMENSIONAL TRAJECTORY OF THE CENTERLINE OF THE REFERENCE PLUME

TH1 - IS THE LOCAL PLUME FLOW DIRECTION RELATIVE TO THE CURRENT WITH 90 DEGREES BEING IN THE DIRECTION OF THE CURRENT.

TH2 - IS THE LOCAL PLUME FLOW DIRECTION RELATIVE TO THE HORIZONTAL WITH 90 DEGREES BEING VERTICAL.

WIDTH - IS THE DIAMETER OF A SINGLE PLUME OR SIDE WIDTH OF MERGED PLUMES (NOT LENGTH SINCE THAT DEPENDS ON THE LENGTH OF THE DIFFUSER)

DUCL, DRHO, DCCL, AND DTCL ALL REFER TO THE RATIO OF THE LOCAL DIFFERENCE BETWEEN THE LOCAL CENTERLINE VALUE AND AMBIENT VALUE TO THAT SAME DIFFERENCE AT DISCHARGE LEVEL FOR VELOCITY, DENSITY, CONCENTRATION, AND TEMPERATURE RESPECTIVELY. FOR EXAMPLE DUCL = (Ucl - Ua) AT ELEVATION Z DIVIDED BY (Ujet - Ua) AT DISCHARGE ELEVATION WHERE U IS VELOCITY, cl IS CENTERLINE, a IS AMBIENT, AND jet IS THE DISCHARGE VALUE.

TIME - IS THE TIME OF TRAVEL FROM THE DISCHARGE TO THIS POINT IN SECONDS.

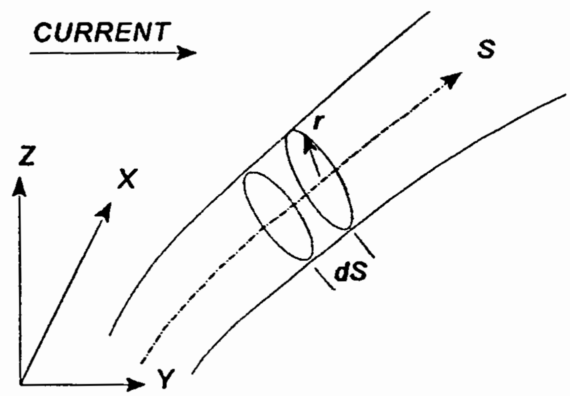
DILUTION - IS THE LOCAL AVERAGE PLUME DILUTION, Q/Qo, 1.0 BEING NO DILUTION AT ALL. IT IS ABOUT 1.93 TIMES THE CENTERLINE OR MINIMUM DILUTION FOR SINGLE PLUMES AND 1.43 TIMES THE CENTERLINE DILUTION FOR COMPLETELY MERGED PLUMES.

THE PROGRAM PRINTS OUT WHEN THE PLUMES JUST BEGIN TO MERGE, WHEN AND IF THE PLUME REACHES AN EQUILIBRIUM HEIGHT IN A STRATIFIED AMBIENT (DRHO BECOMES NEGATIVE AT THIS POINT), AND IF THE PLUME REACHES THE SURFACE.

THE PROGRAM STOPS ONCE THE PLUME REACHES THE SURFACE OR THE TRAPPING LEVEL SINCE THE MODEL IS NOT GOOD BEYOND THAT POINT.

MODEL THEORY

THE GENERAL PARTIAL DIFFERENTIAL EQUATIONS OF MOTION CAN BE INTEGRATED ACROSS THE PLUME TO YIELD THE FOLLOWING SET OF GOVERNING EQUATIONS WHEN APPLIED TO THE COORDINATE SYSTEM SHOWN IN SKETCH:



CONSERVATION OF MASS:

$$\frac{d}{ds}\int_0^\infty \bar{u}rdr = E$$

where s is the coordinate along the centerline of the plume

u = the velocity in the s direction

r = the coordinate normal to the plume centerline

E = the rate of entrainment of mass into the plume

CONSERVATION OF ENERGY

$$\frac{d}{ds}\int_0^\infty \bar{u}\bar{T}\ rdr = -\frac{dT_\infty}{ds}\int_0^\infty \bar{u}rdr$$

where T' is the temperature of the plume above the ambient.

T_∞ = the ambient temperature

CONSERVATION OF SPECIES

$$\frac{d}{ds}\int_0^\infty \bar{u}X_i r\,dr = -\frac{dX_{i\infty}}{ds}\int_0^\infty \bar{u} r\,dr$$

where X_i is the concentration of species i.

S- MOMENTUM:

$$\frac{d}{ds}\int_0^\infty \bar{u}^2 r\,dr = \bar{U}_\infty E\sin\theta_1\cos\theta_2 + \int_0^\infty g\frac{(\rho_\infty - \rho)}{\rho} r\,dr\sin\theta_2$$

CURVATURE COMPONENTS OF MOMENTUM

$$\frac{d\theta_2}{ds} = g\sin\theta_2\int_0^\infty \frac{(\rho_\infty - \rho)}{\rho} r\,dr - \frac{EU_\infty\sin\theta_1\sin\theta_2}{\int_0^\infty \bar{u}^2 r\,dr - \frac{E^2}{4}}$$

$$\frac{d\theta_1}{ds} = \frac{EU_\infty\cos\theta_1}{\cos\theta_2\left(\int_0^\infty \bar{u}^2 r\,dr - \frac{E^2}{4}\right)}$$

THE PROCEDURE IS TO ASSUME PROFILES FOR VELOCITY, TEMPERATURE AND CONCENTRATION IN TERMS OF CENTERLINE VALUES AND PLUME SIZE.

IN UDKHDEN IT WAS ASSUMED THAT:

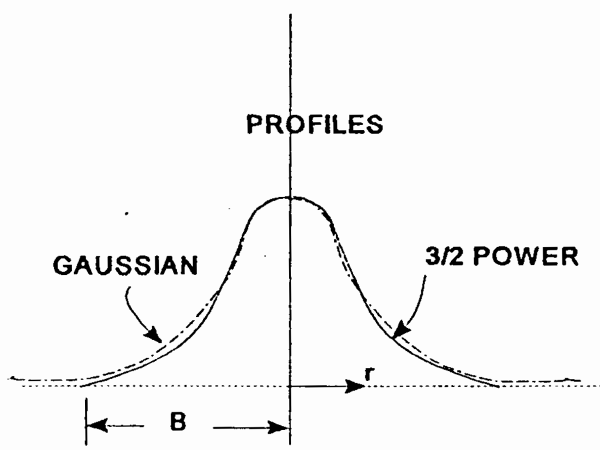
$$\bar{u} = \Delta u + U_\infty \cos\theta_2 \sin\theta_1$$

$$\Delta u = \Delta u_c \left(1 - \left(\frac{r}{b} \right)^{3/2} \right)^2$$

$$T = T_c \left(1 - \left(\frac{r}{b} \right)^{3/2} \right)^2$$

were subscript C refers to centerline values and b is the plume radius.

THESE 3/2 POWER LOW PROFILES ARE SIMILAR TO GAUSSIAN PROFILES BUT HAVE DEFINITE EDGES AS SHOWN .



THE ENTRAINMENT FUNCTION, E, IS REQUIRED FOR CLOSURE. IN UDKHDEN, *Before Merging*

$$E = \left(a_1 + \frac{a_2}{F_L}\right)\left[b|\Delta u_c - U_\infty \cos\theta_2|\left(1 - \frac{a_4 b}{L}\right) + a_3 U_\infty b \sin\theta_2\right]$$

After Merging

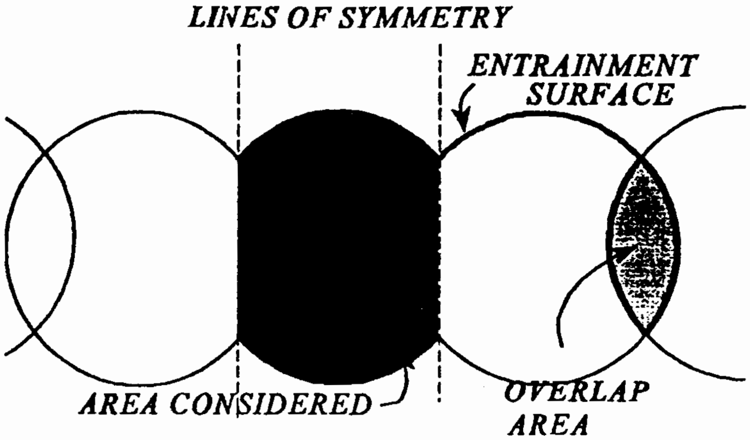
$$E = \left(a_1 + \frac{a_2}{F_L}\right)\left[b|\Delta u_c - U_\infty \cos\theta_2|\left(1 - \frac{a_4}{2}\right)\left(1 - \frac{2}{\pi}\cos^{1}\frac{L}{2}b\right) + a_3 U_\infty \frac{L}{2}\sin\theta_2\right]$$

WHERE L IS THE SPACING BETWEEN PORTS.

THE COEFFICIENTS a_i ARE DETERMINE BY EXPERIMENT.

THE EQUATIONS ARE EVALUATED FOR SMALL VALUES OF ΔS USING A FOURTH ORDER RUNGE-KUTTA SCHEME FOR ACCURACY ALONG THE TRAJECTORY. IF A REQUIRED ACCURACY CANNOT BE OBTAINED WITH A GIVEN ΔS, IT IS HALVED AND THE SOLUTION IS TIRED AGAIN. THIS IS REPEATED UNTIL PROPER ACCURACY IS ACHIEVED.

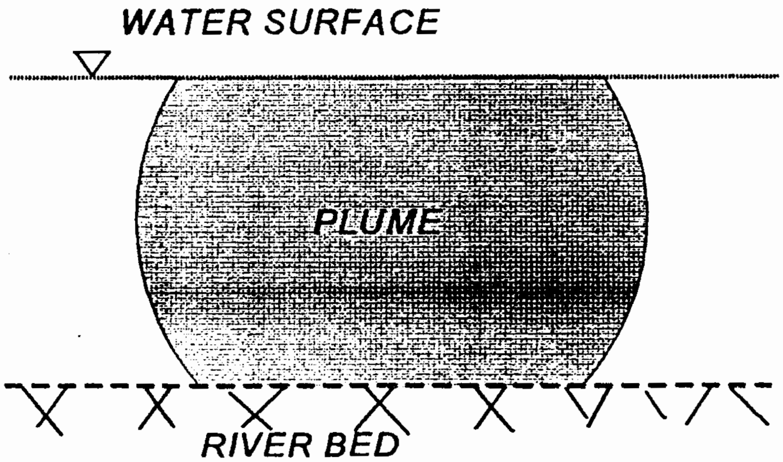
WHEN PLUMES MERGE, THEY ARE ASSUMED TO OVERLAP GRADUALLY GIVING VARIABLE PROFILES AS SHOWN BELOW AND THE ENTRAINMENT SURFACE IS VARIED ACCORDINGLY AS GIVEN ABOVE.



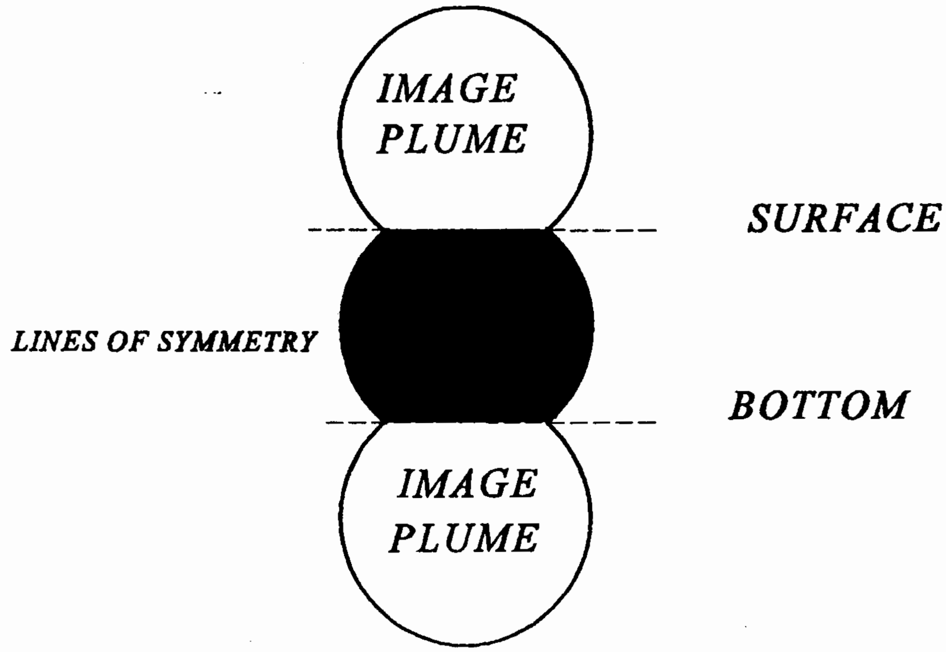
MERGING PLUMES

USING UDKHDEN IN SHALLOW WATER

MANY DISCHARGES OCCUR IN SHALLOW WATER WHERE THE PLUME FILLS THE WATER COLUMN



THIS IS VERY SIMILAR TO THE AREA CONSIDERED BY MERGING PLUMES IN UDKHDEN IF ROTATED BY 90 DEGREES!



AS A RESULT, WE CAN USE UDKHDEN TO SIMULATE A PLUME IN SHALLOW WATER. THIS IS DONE BY SETTING THE WATER DEPTH AS THE INPUT VARIABLE PORT SPACING AND SETTING THE DEPTH TO A LARGE NUMBER WHICH REPRESENT LATERAL DISTANCE.

TO INTERPRET THE RESULTS:

1. IGNORE THE VERTICAL, Z, COORDINATE SINCE IT ASSUMED THAT THE PLUME IS CENTERED IN THE RIVER.
2. WHEN THE PROGRAM INDICATES MERGING, IT HAS JUST FILLED THE WATER COLUMN.

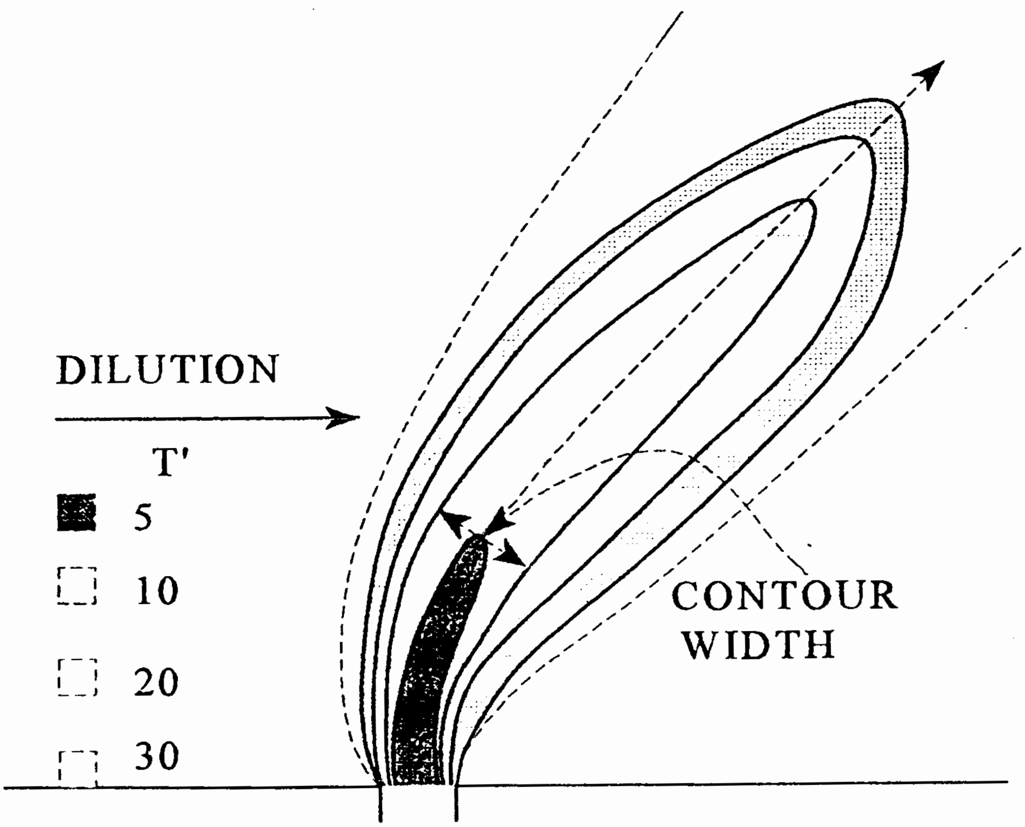
LIMITATIONS

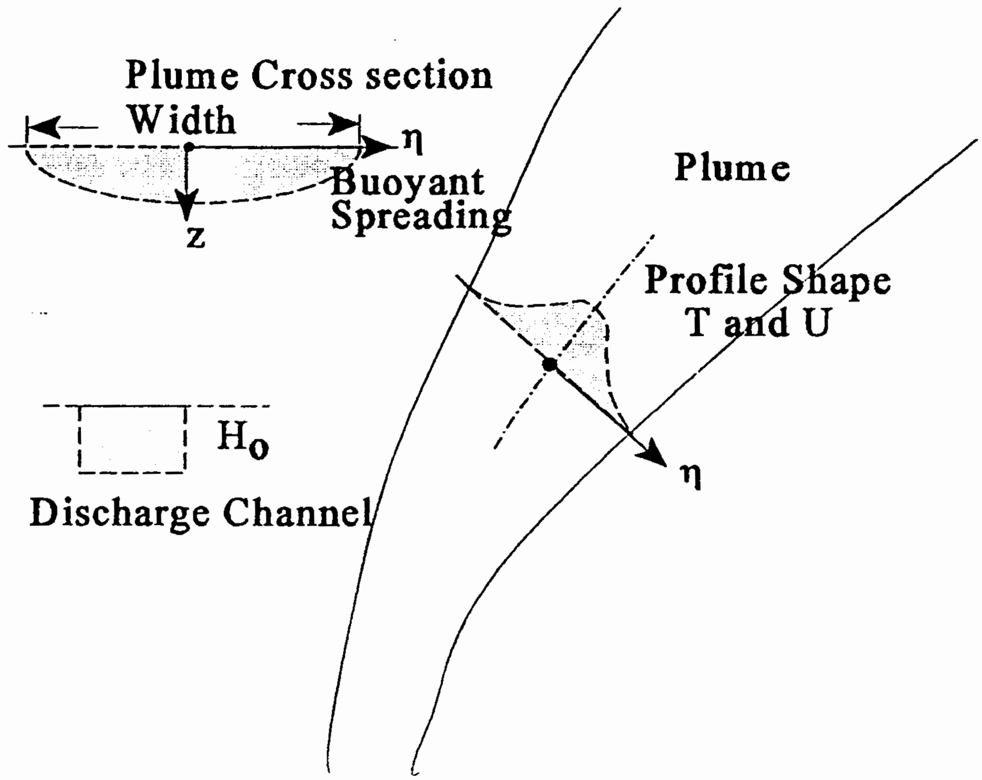
1. USE WITH SINGLE PORT DISCHARGES OR MULTIPLE PORTS WHERE PLUMES DO NOT MERGE. THE RESULTS CAN'T BE USED BEYOND WHERE THE PLUMES MERGE.
2. VERTICAL DISCHARGES IN SHALLOW WATER.
3. DISCHARGES WITH HIGH BUOYANCY MAKING THEM FLOAT. THIS IS MEASURED BY THE DISCHARGE FROUDE NUMBER PRINTED OUT BY PROGRAM. IF FROUDE NUMBER IS LESS THAN 5, TO NOT USE THE IMAGE METHOD.

$$F_o = \frac{U_o}{\sqrt{g\,D\,\Delta\rho/\rho}} = \frac{MOMENTUM}{BUOYANCY}$$

THEORETICAL ANALYSIS OF THE PDS PROGRAM

The theoretical analysis used to develop the three-dimensional surface plume program (PDS) is based on an earlier model by Prych. It was modified to its present form by Davis and was tuned to data by Shirazi, hence PDS (see *"Workbook of Thermal Plume Prediction-Vol.2-Surface Discharge," Environmental Protection Agency Report EPA-R2-72-005b, May 1984 by M.A. Shirazi and L.R. Davis).*





The method of analysis is an integral approach which assumes similarity of temperature and velocity profiles and the principle of entrainment. The profiles assumed are Gaussian such that:

$$T_r = T_c \exp(-\eta^2/B^2) \cdot \exp(-Z^2/H^2) \qquad (2)$$

$$U_r = U_c \exp(-\eta^2/B^2) \cdot \exp(-Z^2/H^2) + V\cos\theta \qquad (3)$$

where η and Z are distances perpendicular to the plume centerline in the lateral and vertical directions, respectively, T_c and U_c are the Centerline temperature and Velocity respectively and V is the ambient velocity .

With the temperature and velocity profiles assumed, the energy, volume and momentum fluxes can be integrated across the plume at any cross section leaving them in terms of centerline values and plume characteristics width, B, and Depth, H. Accordingly, the volume flux becomes

$$Q = \iint_A (U_r)\, d\eta dZ = \pi HB(\frac{U}{2} + V\cos\theta) \qquad (4)$$

where the limits of integration for V cos θ are taken as the bottom half of the region:

$$(\frac{\eta}{\sqrt{2}B})^2 + (\frac{Z}{\sqrt{2}H})^2 \leq 1 \qquad (5)$$

Solving (3) for U, yields

$$U = 2\,(\frac{Q}{\pi HB} - V\cos\theta) \qquad (6)$$

The heat flux, J,is:

$$J = \iint_A U_r T_r d\eta dZ = \frac{\pi}{2} TBH(\frac{U}{2} + V\cos\theta) = \frac{QT}{2} \quad (7)$$

The momentum flux, M, is:

$$M = \iint_A U_r^2 d\eta dZ = \pi BH(\frac{U}{2} + V\cos\theta)^2 = \frac{Q^2}{\pi BH} \quad (8)$$

The quantities dQ/ds, dT/ds, and dM/ds are calculated from conservation equations. dQ/ds is assumed to be due to contributions of jet entrainment and ambient turbulent mixing, thus:

$$\frac{dQ}{ds} = \frac{dQ}{ds}\Big|_j + \frac{dQ}{ds}\Big|_a \quad (9)$$

The jet and ambient contributions are both divided into vertical and horizontal components. The horizontal jet entrainment fluid is:

$$\frac{dQ}{ds}\Big|_{j,h} = 2E_0 \int_{-\sqrt{2}H}^{0} \Delta U dZ \quad (10)$$

where

$$\Delta U = (U^2 + V^2 \sin^2\theta)^{1/2} \exp(-Z^2/H^2) \quad (11)$$

and E_o is an entrainment coefficient. By inserting (10) into (9) we obtain:

$$\frac{dQ}{ds}\Big|_{j,h} = \sqrt{\pi} H E_o (U^2 + V^2 \sin\theta)^{1/2} \tag{12}$$

The vertical jet entrained fluid is:

$$\frac{dQ}{ds}\Big|_{j,v} = 2 \int_0^{\sqrt{2}B} E \Delta U_v \, d\eta \tag{13}$$

where $E = E_o f(Ri)$ and Ri is the local Richardson number given by:

$$R_i = \frac{\sqrt{2}}{F_o^2} \frac{H T(s,\eta,o)}{\Delta U_v^2} \tag{14}$$

The function $f(R_i)$ is a curve fit to data. It is:

$$f = [\exp(-5 R_i) - 0.0183]/0.982 \tag{15}$$

The velocity difference ΔU_v is given by:

$$\Delta U_v = [U^2 \exp(-2\eta^2/B^2) + V^2 \sin^2\theta]^{1/2} \tag{16}$$

The term T is the surface excess temperature at a distance η from the plume centerline. The value of the integral (12) is determined numerically in the program.

The effective entrainment due to ambient turbulent mixing is calculated as follows:

$$\frac{dQ}{ds}\Big|_{a,h} = 11.0\frac{H}{B}\frac{\epsilon_h}{U_o H_o} \tag{17}$$

$$\frac{dQ}{ds}\Big|_{a,v} = 11.0\frac{B}{H}\frac{\epsilon_v}{U_o H_o} f(R_i') \tag{18}$$

where ϵ_h and ϵ_v are the horizontal and vertical turbulent diffusion coefficients, respectively.

The change in heat flux along the plume due to heat exchange with the atmosphere is expressed as:

$$\frac{dJ}{ds} = -2\int_0^{\sqrt{2}B} K\, T_r d\eta = \sqrt{\pi} K T B \tag{19}$$

where K is a dimensionless heat exchange coefficient. Substituting (18) into (10) yields:

$$\frac{dT}{ds} = -\frac{T}{Q}(2\sqrt{\pi}KB + \frac{dQ}{dS}) \tag{20}$$

The conservation of momentum is applied in the s-direction and then divided into X and Y components. The net forces on the plume are balanced by the change in momentum flux. The forces considered important are (a) the internal pressure forces due to buoyancy, (b) form drag due to ambient current and (c) interfacial shear forces.

The pressure forces are found by determining the excess pressure due to buoyancy as a function of depth and then integrating the pressure over the vertical cross section of the plume. Thus, the normalized pressure force is:

$$P = \frac{1}{F_o^2} \iint_A \left(\int_{-\infty}^{Z} T_r \, dZ \right) dA = \frac{\sqrt{\pi}\, T H^2 B}{2 F_o^2} \qquad (21)$$

The form drag acting normal to the plume centerline is assumed similar to the drag on a solid body such that

$$F_D = \frac{1}{2}\sqrt{2}\, C_D H V |V| \sin^2\theta \qquad (22)$$

where C_D is a drag coefficient.

The interfacial shear forces are assumed to be similar to turbulent flow over a flat surface with a boundary layer thickness of $(2)^{1/2}H$ and a velocity equal to the vector velocity difference between the plume and ambient current. Accordingly, the X and Y components of this shear force reduce to:

$$SF_X = C_F\left(\frac{1}{R_e H}\right)^{1/4} \int_0^{\sqrt{2}B} \Delta U_v^{3/4}[V\sin^2\theta - U\cos\theta\exp(-\eta^2/B^2)]d\eta \tag{23}$$

$$SF_Y = -C_F\left(\frac{1}{R_e H}\right)^{1/4} \int_0^{\sqrt{2}B} \Delta U_v^{3/4}[V\cos\theta - U\exp(-\eta^2/B^2)]d\eta \tag{24}$$

where C_F is a friction coefficient and R_e is the jet discharge Reynolds number. The value of C_F is determined by experiment.

The change in momentum flux includes the effects of the momentum of the entrained ambient fluid, V(dQ/ds), which acts in the X-direction. Equating the forces to the change in momentum flux in the X and Y directions yields:

$$\frac{d}{ds}[(M+P)\cos\theta] = SF_X + F_D\sin\theta + VdQ/ds \tag{25}$$

$$\frac{d}{ds}[(M+P)\sin\theta] = SF_Y - F_D\cos\theta \tag{26}$$

Using equations (7) and (20) for M and P, multiplying (24) by -sin θ, (25) by cos θ and combining yields an expression for the change in flow direction,

$$\frac{d\theta}{ds} = \frac{SF_Y\cos\theta - SF_X\sin\theta - F_D V\sin\theta(dQ/ds)}{\dfrac{Q^2}{\pi BH} + \dfrac{\sqrt{\pi}}{2F_o^2}TH^2B} \tag{27}$$

Differentiating M and P, multiplying (24) by cos θ and (25) by sin θ and combining yields equation (27).

$$\frac{dH}{ds} = [SF_Y\sin\theta + SF_X\cos\theta$$

$$+ (V\cos\theta - 2Q/\pi BH)(dQ/ds) - (\sqrt{\pi}BH^2/2F_o^2)(dT/ds) \tag{28}$$

$$+ (Q^2/\pi BH - \sqrt{\pi}H^2/2F_o^2)(dB/ds)]$$

$$[\sqrt{\pi}THB/2F_o^2 - Q^2/\pi BH^2]^{-1}$$

It is noted that this expression for change in depth is undefined when the denominator is zero. Hence, results beyond this singularity are questionable.

Momentum in the lateral direction is included only indirectly through lateral spreading. It is assumed that the contributions to spreading by non buoyant horizontal jet mixing and buoyancy are independent of one another such that

$$\frac{dB}{ds} = \left(\frac{dB}{ds}\right)_{nb} + \left(\frac{dB}{ds}\right)_{b} \qquad (29)$$

where the subscripts b and nb denote buoyant and non-buoyant terms. the non-buoyant spreading is found by writing equation (27) without the buoyancy terms and assuming that

$$\left(\frac{dB/ds}{dH/ds}\right)_{nb} = (B/H)(dQ/ds)_h/(dQ/ds)_v \qquad (30)$$

where $(dQ/ds)_h$ and $(dQ/ds)_v$ are the horizontal and vertical entrainment rates.

$$\left(\frac{dB}{ds}\right)_{nb} = \frac{SF_Y \sin\theta + SF_X \cos\theta + (V\cos\theta - \frac{2Q}{\pi BH})\frac{dQ}{ds}}{-(Q^2/\pi BH)[(dQ/ds)_v/(dQ/ds)_h + 1]} \qquad (31)$$

The spreading due to buoyancy is assumed to be a function of the local excess density ratio, plume depth and aspect ratio such that

$$\left(\frac{dB}{ds}\right)_b = \sqrt{\frac{2}{\frac{B}{H}F^2 - 1}} \qquad (32)$$

It is noted that this also has a singularity. Due to B/H usually being large, this singularity is not encountered in most problems.

The preceding equations are sufficient to perform a step-wise integration along the plume. From the local conditions of the plume, dQ/ds is calculated. When this is known dT/ds, dθ/ds and dB/ds are calculated. With these known, dH/ds can be calculated. These derivatives are integrated step-wise along the plume trajectory to give local values of X, Y, T, H, B, θ, and Q. The method of integration is a very accurate Hamming's Predictor Corrector method (similar to a fourth-order Runge Kutta).

In order to start the integration within the developed zone where the above analysis is valid, starting conditions must be calculated. These are determined by a simplified analysis of the development zone and assuming that the development length is given by the equation

$$S_i = 5.4\left(\frac{A^2}{F_o}\right)^{1/3} \tag{33}$$

where A is the discharge flow area and F_o is the discharge densimentric Froude number.

COMPUTER PROGRAM

The PDS program is written in FORTRAN 77 and consists of a main program entitled PDS and five subroutines, KHPCG, AREA, FCT, RED, and OUTP.

The main program, PDS, prompts the user for input variables, initializes constants, non dimensionalizes the variables and calls subroutine KHPCG which performs the actual calculation. Subroutine KHPCG is a standard scientific subroutine which performs the stepwise integration of differential equations by the Hamming Predictor-Corrector Method.

Subroutine SIGMAT is a subroutine that calculates the density of the water as a function of temperature and salinity. It is an empirical curve fit to data.

Subroutine AREA is a step-wise integration of the area enclosed by surface isotherms. Subroutine FCT calculates the derivatives of the program variables which are used in KHPCG. Subroutine RED calculates the reduction in the vertical entrainment

coefficient as a function of local Richardson's number.

Subroutine OUTP converts the desired variables to dimensional quantities and prints them out at each integration step.

INPUT AND OUTPUT

Input and output are entered in metric units.

Input is interactive with the terminal. The user answers prompts. After a run, the user is given a chance to change any of the input variables and repeat a run. The input variables required are:

1) A run title (one line)

2) The discharge flow rate in m^3/s

3) The discharge channel width, m

4) The discharge channel depth (assumed rectangular), m

5) The ambient current, m/s

6) Discharge angle relative to the current, degrees (zero is downstream and 90 is perpendicular to current)

7) Discharge water temperature, C

8) Ambient water temperature, C

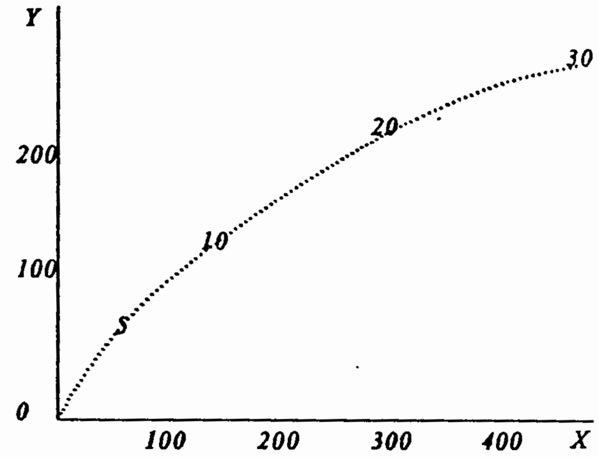
9) Salinity, ppt (both discharge and ambient assumed the same). If fresh water, use a small salinity such as 0.01.

10) Surface heat transfer rate (you are given three choices, mild, average, or severe. This has little effect on temperatures or dilution in the near field)

11) The distance downstream where simulations are to stop. (The program stops when this value is reached or you have 20 pages of output or when the excess temperature is .005 of the initial excess temperature.)

The output consists of the following:

The program plots a trajectory on the screen and shows the centerline dilution at selected points if you choose the graphics version. To clear the graphics, press the enter button.



A graphics version is being worked on that gives contours.

The output file contains:

1) X, distance downstream in the direction of current, m.

2) Y, distance perpendicular to current, m

3) Centerline excess temperature (temperature above ambient), C.

4) Time of travel to this point along centerline, sec.

5) Q/Q0, average plume dilution, total flow in plume divided by discharge flow.

6) QM/Q0, minimum dilution at plume centerline.

7) DEPTH, plume depth, m, $\sqrt{2\sigma_Z}$

8) WIDTH, plume width, m , $2\sqrt{2\sigma_B}$

9) AREA, surface area within specified excess temperature isotherms, m^2. These are given for each 1/2 degree of excess temperature. (If the simulation is terminated before a particular isotherm is reached on the plume centerline, only a partial area is given)

PLEASE NOTE: The program does not keep track of receiving water boundaries, i.e., shore or bottom. The user must check to see that the plume does not attach itself to either shore or bottom by following the trajectory, plume width and depth. Simulations beyond these attachment points are in error due to changes in entrainment.

The program has been tuned to a wide variety of data and agrees with the average of these data as outlined in the reference given at the beginning. It may not agree exactly to each specific case but should give reasonable answers if boundaries are not encountered.

The program checks its own accuracy at each integration step. If the accuracy is poor, it repeats the calculations for that step with the step size cut in half. If the accuracy is still poor, it continues to half the step size until it reaches satisfactory accuracy. If it cuts the step size in half 11 (eleven) times and still cannot achieve specified accuracy the program stops. This is usually when the user has input bad data. It also occasionally occurs for certain combinations of current, density difference, and discharge direction. When it does, try changing input variables a small amount.

RUNNING THE PROGRAM

1. Copy .EXE files on one of the source disks to one of your formatted working disks or to your hard drive.(hard drive best)

2. Store the source disks in a safe place.

3. If you are working from your hard drive, change your directory to the directory where you copied PDS.EXE and type PDS. or PDSMS if you want graphics.

   If you are working from floppies, put the working disk in your computer's A: drive and type A:PDS if you want the regular PDS or PDSMS if you want PDS with graphics.

4. Answer all questions regarding for your particular discharge and ambient conditions.

5. When the program prompts you for an output file, type in the name of an **non existing** file where you want the output to be stored. If you do not want the output stored and just want to see the results, type CON or TTY. (This does not work in graphics mode)

This sends the output to the screen. You can also send the screen output simultaneously to the printer by pressing CTRL and PRINT SCREEN at the same time before running the program.

6. At the end of the run, the program will prompt you for further input if you want to change any of the existing input variables. If so, make the changes and it will re-run the previous case with your new changes.
7. If you want a hard copy of your output when you are finished and have not done so while running, type COPY *filename* PRN where filename is the name of the output file you specified. You can also bring the file into a word processor and print it from there.

CORMIX 1-2-3

THE CORNELL MIXING ZONE SYSTEM IS A SERIES OF SOFTWARE SUBSYSTEMS FOR THE ANALYSIS, PREDICTION AND DESIGN OF DISCHARGES INTO AQUEOUS ENVIRONMENTS. IT HAS AN EXPERT SYSTEM SHELL THAT CONTROLS AND ANALYZES INPUT AND OUTPUT.

TO OBTAIN SERVICE, UPDATES AND OTHER COPIES OF THE MODEL CONTACT *KATHRYN GREEN, EPA ATHENS CENTER FOR EXPOSURE ASSESSMENT MODELING (CEAM), COLLEGE STATE ROAD, ATHENS, GEORGIA 30613.* PHONE: (404) 546-3130

THE HYDRAULIC CALCULATIONS ARE MADE USING EMPIRICAL METHODS DISCUSSED EARLIER. AS A RESULT, THE EQUATIONS OF MOTION ARE NOT SOLVED DIRECTLY BUY RELY ON EQUATIONS OBTAINED FROM EXPERIMENTS.

WHICH EQUATION TO USE IN THE HYDRAULIC MODELING SUB SYSTEM IS DETERMINE BY LENGTH SCALES DETERMINE FROM THE INPUT VARIABLES.

THE USER DECIDES WHETHER TO USE

CORMIX 1 -	FOR SINGLE SUBMERGED DISCHARGES
CORMIX 2 -	FOR MULTIPLE PORT DIFFUSERS INCLUDING UNIDIRECTIONAL STAGED, AND ALTERNATE PORT DIFFUSERS.
CORMIX 3	SURFACE DISCHAGES

AFTER THE APPROPRIATE MODEL IS SELECTED:

THE SYSTEM IS DIVIDED INTO SECTIONS:

DATIN	IN THIS SECTION INPUT VALUES ARE ENTERED AND ANALYZED FOR CONSISTANCY
PARAM	IS WHERE THE INPUT VARIABLE ARE ARRANGED INTO PARAMETERS.
CLASS	IS WHERE THE FLOW CLASS IS DETERMINED FROM THE PARAMETERS
HYDRO	IS WHERE THE HYDRAULIC CALCULATIONS ARE PERFORMED USING THE EQUATIONS DETERMINED FROM THE FLOW CLASS
SUM	IS WHERE THE OUTPUT IS GENERATED, SUMMARIZED AND REGULATIONS EVALUATED

SOME GENERAL COMMENTS

1. CORMIX IS NOT ALWAYS CONSISTANT IN PREDICTIONS WHEN GOING FROM ONE FLOW CLASS TO THE NEXT. (JUMPS)
2. VELOCITY VARIATIONS ARE NOT ALLOW EITHER IN THE HORIZONTAL OR VERTICAL DIRECTIONS.
3. DENSITY CALCULATIONS USING TEMPERATURE AND SALINITY ARE NOT MADE. EITHER USE FRESH WATER WITH TEMPERATURE OR OTHER INCLUDING SALT WATER WHERE THE DENSITY MUST BE ENTERED.
4. VERTICAL AMBIENT STRATIFICATION MUST FIT INTO FOUR TYPES. ACTUAL VARIATIONS ARE NOT USED. THIS CAUSES TRAPPING LEVELS TO BE OFF A LITTLE IN SOME AMBIENTS.

5. WHEN AN ERROR IS MADE ON INPUT, THE WHOLE SECTION MUST BE REENTERED.
6. CORMIX 2 USES AN EQUIVALENT SLOT CONCEPT FOR MULTIPLE PORT DIFFUSERS WHERE THE INLINE, ALTERNATING OR STAGED ARE SIMULATED BY A SLOT DISCHARGE THAT HAS THE SAME FLOW AND MOMENTUM AS THE MULTIPLE. THIS IS OK IF THE PORTS ARE CLOSE TOGETHER BUT IF THE PLUMES REMAIN SEPARATED FOR ANY SIGNIFICANT PART OF THE MIXING ZONE, PREDICTIONS WILL BE IN ERROR SINCE THE DILUTION RATE FOR A SLOT IS DIFFERENT THAN INDIVIDUAL PORTS.
7. RIVERS ARE SIMULATED AS RECTANGULAR.

8. THE PROGRAM AUTOMATICALLY ACCOUNTS FOR QUANDA ATTACHMENT TO THE BOTTOM
7. THE PROGRAM AUTOMATICALLY ACCOUNTS FOR SHORE ATTACHMENT ON EITHER BANK AND RECIRCULATION.
9. THE PROGRAM SIMULATES DISCHARGE AT VARIOUS VERTICAL AND HORIZONTAL ANGLES.
10. THE PROGRAM AUTOMATICALLY DETERMINES IF A DISCHARGE IS IN COMPLIANCE WITH REGULATIONS IF THEY ARE ENTERED.
11. CMC TOXIC MIXING ZONES ARE EVALUATAED BASED ON U.S.A. REGULATIONS. THEY MAY NOT APPLY TO OTHER COUNTRIES.
12. FAR FIELD CONDITIONS ARE CALCULATED.

CORMIX INPUT PROCEDURE

INPUT TO ALL THREE CODES IN CORMIX ARE SIMILAR AND ARE HANDLED INTERACTIVLY USING AN EXPERT SYSTEM ANALYZER. PROMPTS FOR INPUT AND EVALUATES ANSWERS. IT CONSISTS OF:

1. A SITE SPECIFICATIN
2. A CASE SPECIFICATION
3. AN OUTPUT FILE NAME (LIMITED TO 8 CHARACTERSS WITHOUT EXTENSION). CORMIX AUTOMATICALLY SAVES SEVERAL FILES IN THE CMXn\SIMn SUB DIRECTORY INCLUDING OUTPUT (fn.CXO), SUMMARY (fn.CXS), AND INPUT.

4. AMBIENT DATA:

A) IS IT BOUNDED OR UNBOUNDED? IF BOUNDED GIVE RECTANGULAR WIDTH AND DEPTH AND ACTUAL DEPTH OF DISCHARGE. IF THE EFFECTIVE DEPTH HAS NOT BEEN MEASURED AT THE DESIRED AMBIENT CONDITIONS, IT CAN BE APPROXIMATED FROM:
WHERE SUBSCRIPT 1 REFERS TO MEASURRED DATA.

$$H_2 = H_1\left[\frac{Q_2}{Q_1}\right]^{\frac{3}{5}}$$

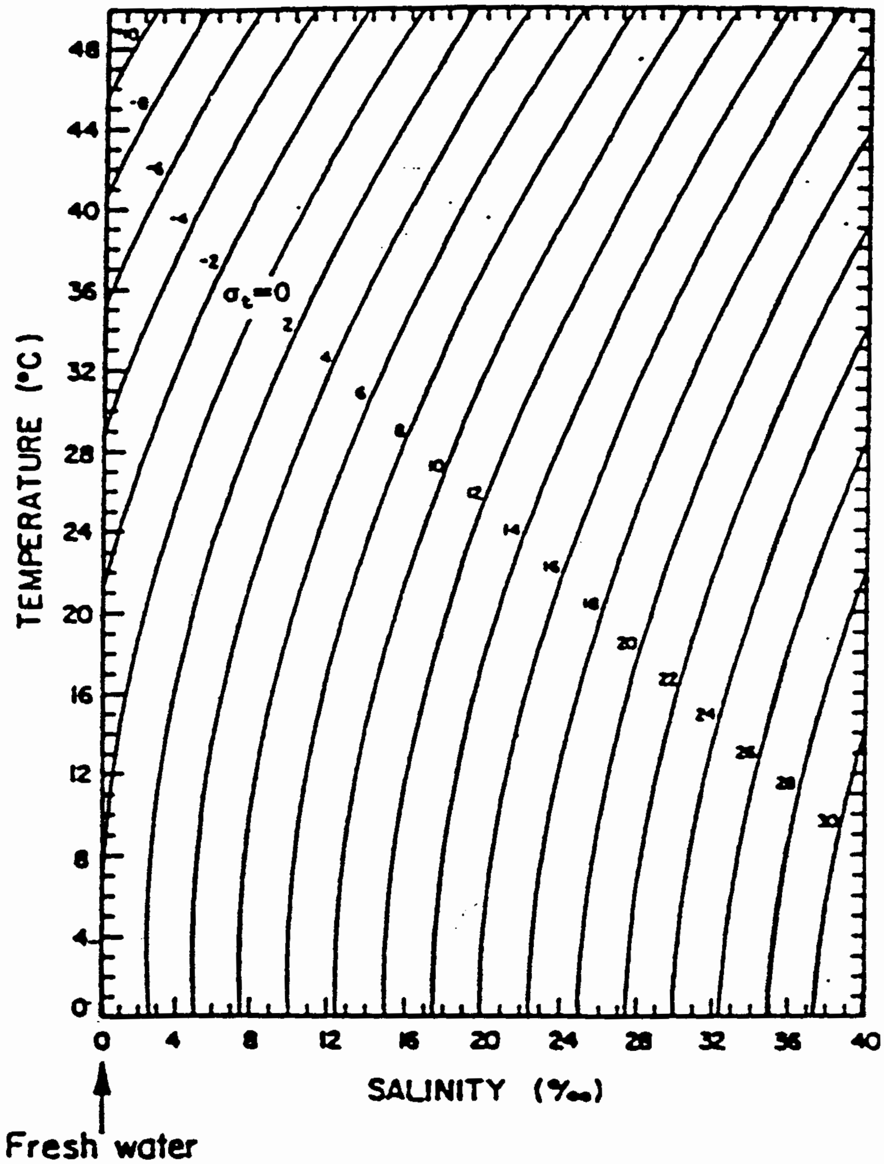
IF THE AMBIENT IS UNBOUNDED (DISCHARGE NEAR THE SHORE OF A LARGE BODY OF WATER). THE AMBIENT DEPTH IN THE AREA WHERE THE PLUME EXISTS AND THE ACUTAL DISCHARGE DEPTH.

B. MANNING'S N OR DARCY FRICTION FACTOR

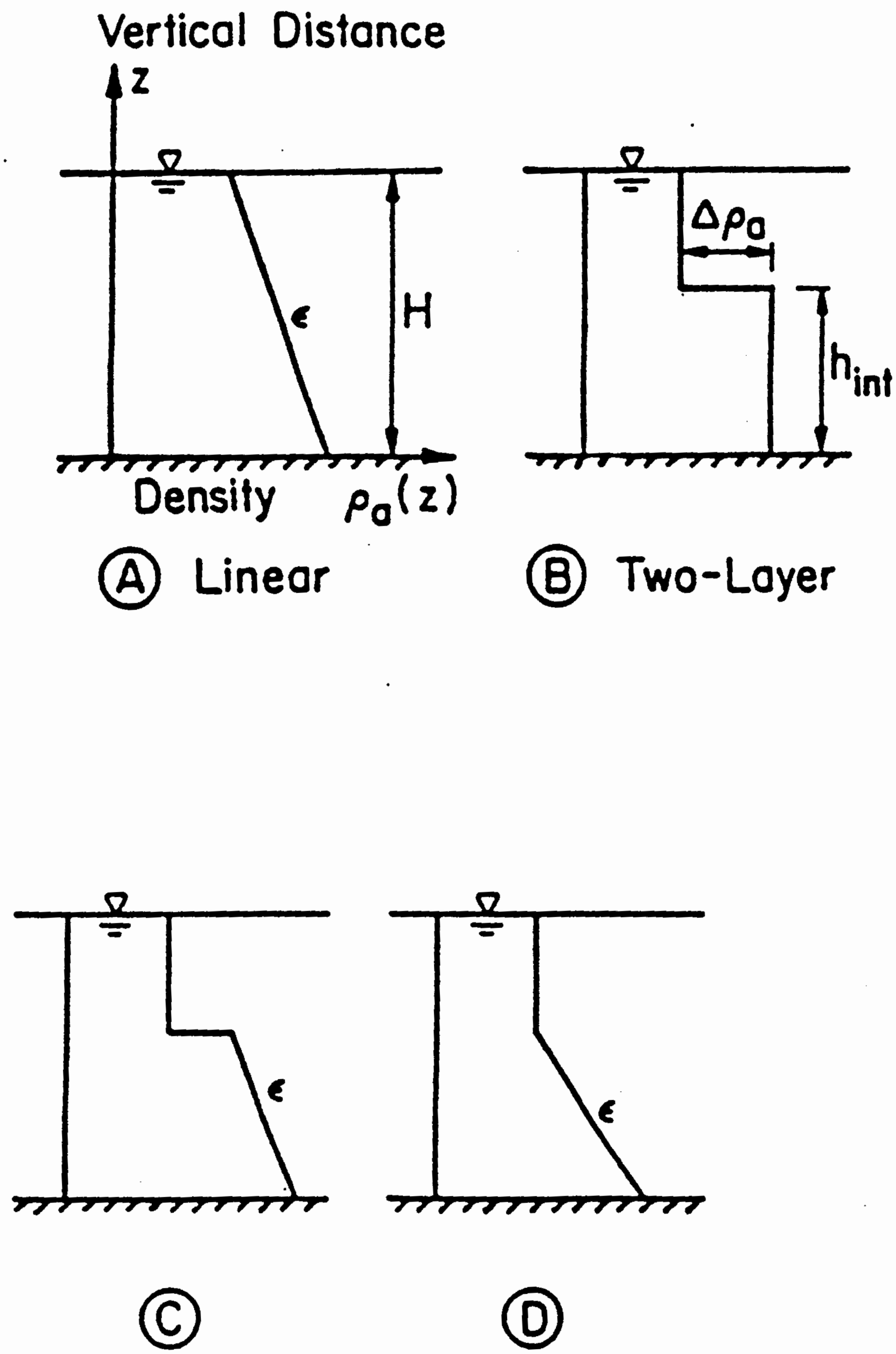
C. AMBIENT FLOW OR VELOCITY

D. AMBIENT DENSITY STRATIFICATION IF ANY. IF IT IS GIVE DENSITY OR TEMPERTURE AT POINTS TO SPECIFY ONE OF THE FOLLOWING. CORMIX 3 DOES NOT ALLOW STRATIFICATION.

E. WIND VELOCITY



Density [kg/m^3] = 1000.0 + σ_t



POSSIBLE APPROXIMATIONS FOR REPRESENTING AMBIENT DENSITY STRATIFICATION

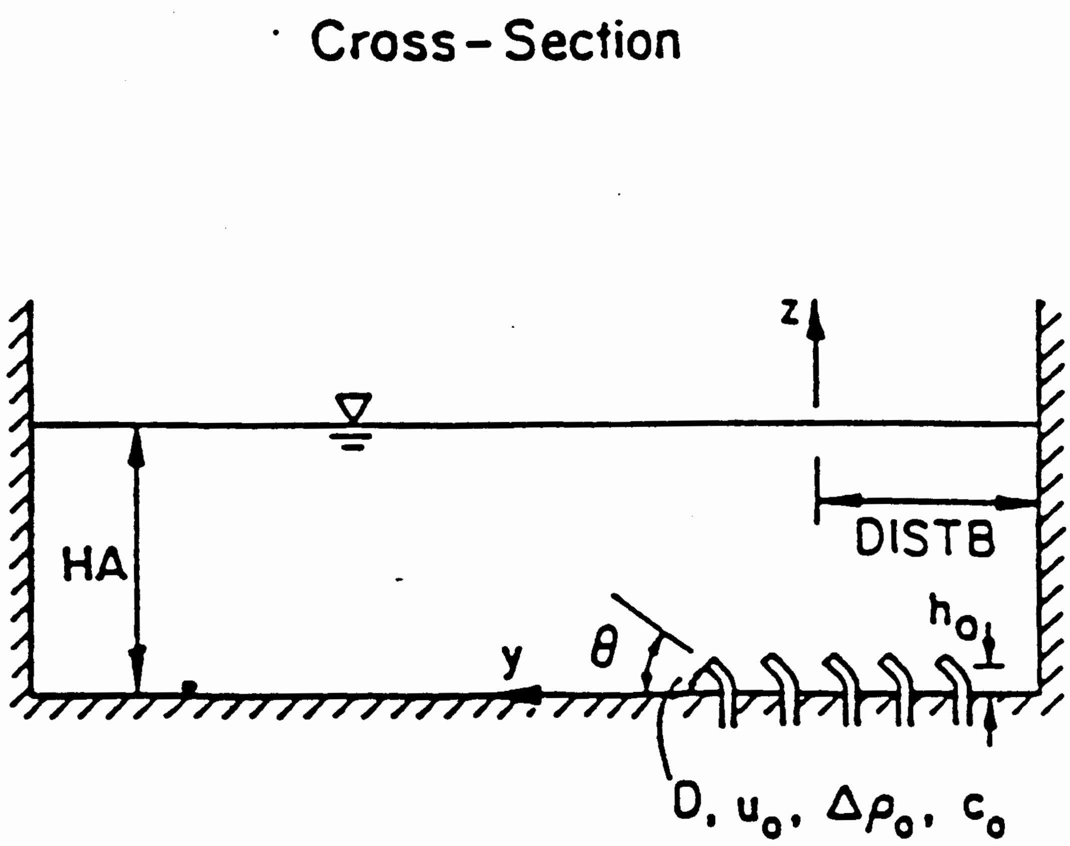
OUTFALL CONDITIONS

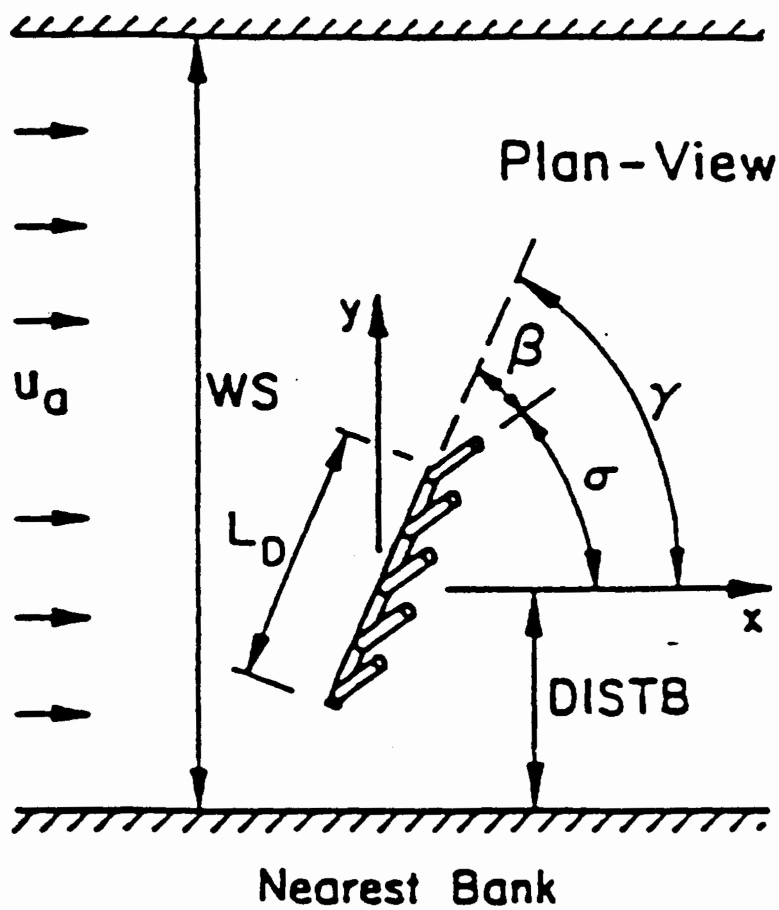
1. THE DISTANCE FROM THE OUTFALL TO NEAREST SHORE AND WHICH SHORE.
2. IF A DIFFUSER, ITS LENGTH AND DISTANCE FROM SHORE TO BOTH ENDS. (THEY MUST AGREE IF THE DIFFUSER IS AT AN ANGLE, THIS CAN BE A PAIN)
3. VERTICAL DISCHARGE ANGLE (THETA)
4. HORIZONTAL ANGLE. IF SINGLE PORT, ANGLE OF PORT RELATIVE TO CURRENT (SIGMA)

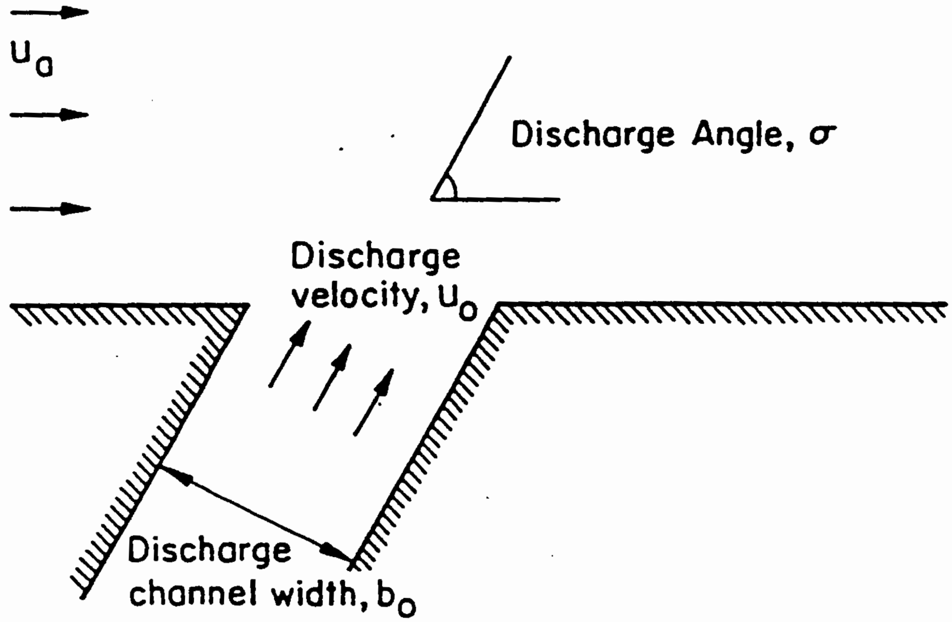
   IF DIFFUSER, THE ANGLE THE DIFFUSER MAKES WITH THE CURRENT (GAMMA) AND THE ANGLE THE PORTS MAKE WITH THE DIFFUSER (BETA)

5. IF A DIFFUSER, INDICATE WHETHER IT IS UNIDIRECTIONAL, STAGED, OR ALTERNATING.
6. DISCHARGE FLOW OR VELOCITY
7. EFFLUENT DENSITY OR TEMPERATURE
8. PORT DIAMETER AND SPACING IF DIFFUSER.

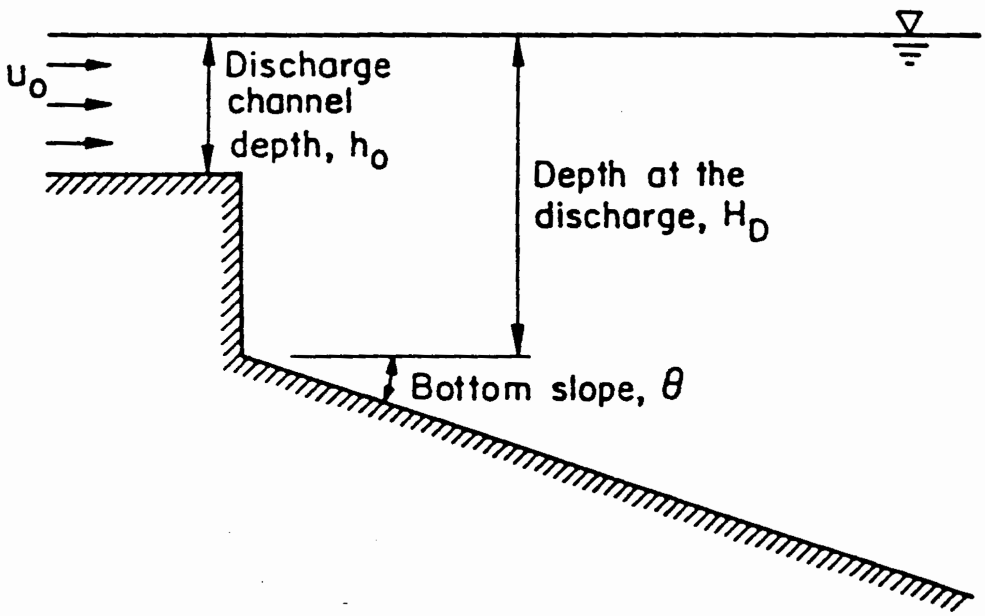
   IF A SURFACE DISCHARGE FROM A CHANNEL GIVE WIDTH, DEPTH, FLOW DIRECTION, AND PROTRUSION INTO AMBIENT WATER BODY.
9. CONCENTRATION OF POLLUTANT IN DISCHARGE. IF THERMAL DISCHARGE, EXCESS TEMPERATURE IS REQUIRED.


Cross-Section
z
HA
DISTB
h_o
y
θ
D, u_o, $\Delta\rho_o$, c_o


Plan-View
u_a
WS
y
β
γ
σ
L_D
x
DISTB
Nearest Bank



a) Plan View



b) Cross-Section

MIXING ZONE DATA

1. IF THE EFFLUENT IS TOXIC, THE CMC, MAXIMUM CONCENTRATION IN THE TOXIC MIXING ZONE (TMZ) MUST BE GIVEN.
2. IF A REGULATORY MIXING ZONE, RMZ, EXISTS, THE CCC AND SPECIFICATION OF THE ZONE MUST GE GIVEN.
3. THE REGION OF INTEREST, ROI IS THE LIMIT TO RUN THE MODEL.

A DATA CHECKLIST IS AVAILABLE FOR ALL THREE MODELS. THEY HELP TO ORGANIZE YOUR INPUT AND MAKE SURE YOU HAVE IT ALL.
AFTER EACH SECTION YOU CAN CHANGE DATA IF ERRORS ARE MADE.

OUTPUT

CORMIX PUTS OUT PAGES OF OUTPUT, SOME OF IT USEFUL, SOME OF IT NOT SO USEFUL.

THE PROGRAM GIVES A SUMMARY OF THE INPUT VARIABLES ENTERED AND GIVES DESCRIPTIVE MESSAGES AS TO THE TYPE OF PLUME AND WHAT WILL OCCUR.

IT GIVES THE LENGTH SCALES AND FLUXES IT USED TO DETERMINE THE FLOW CLASS. THESE LENGTH SCALES, FLUXES AND THE FLOW CLASS ARE OF INTEREST TO MODELERS BUT MOSE USERS DO NOT NEED TO SPEND MUCH TIME WITH THEM. AFTER A FEW RUNS, YOU CAN SKIP OVER THIS STUFF.

IT GIVES A DESCRIPTION OF THE FLOW CLASS AND THEN GOES INTO THE HYDRODYNAMICS OUTPUT.

TABLE 2 INPUT DATA CHECKLIST FOR CORMIX1

CORMIX1 – Submerged single port discharges – CORMIX1

SITE/CASE IDENTIFIER:

Site Name ________
Discharger ________
Pollutant ________
Design Case ________
DOS FILE NAME ________

Prepared by:

Date prepared:

AMBIENT DATA:

Bounded or unbounded?	________				
Channel width	________	m			
Channel depth	________	m			
Depth at discharge	________	m			
Ambient flowrate	________	m^3/s	or: Ambient velocity	________	m/s
Manning's n	________		or: Darcy-Weisbach f	________	

Density data:			Density units:	kg/m^3	
Fresh or salt water?	________		Temperature units:	°C	
Density or temp. values?	________				
<u>If uniform:</u>			<u>If stratified:</u>		
Average density/temp.	________		Density/temp. at surface	________	
			Density/temp. at bottom	________	
			Stratification type	________	
			(Pycnocline height	________	m)
			(Density/temp. jump	________	)

DISCHARGE DATA:

Nearest bank (left/right)?	________				
Distance to nearest bank	________	m			
Vertical angle (THETA)	________	°			
Horizontal angle (SIGMA)	________	°			
Port height	________	m			
Port diameter	________	m	or: Port area	________	m^2
Discharge flow rate	________	m^3/s	or: Discharge velocity	________	m/s
Discharge density	________	kg/m^3	or: Discharge temp.	________	°C
Concentration units	________				
Discharge concentration	________				

MIXING ZONE DATA:

Is effluent toxic?	________		If yes: CMC value	________	
			CCC value	________	
Is there a WQ standard for conventional pollutant?	________		If yes: value of standard	________	
Any mixing zone specified?	________		If yes: distance	________	m
			or width (% or m)	________	
			or area (% or m^2)	________	
Region of interest	________	m			
Grid intervals for display	________				

Date of data input into CORMIX1:

TABLE 3 INPUT DATA CHECKLIST FOR CORMIX2

CORMIX2 -- Submerged multiport diffuser discharges -- CORMIX2

SITE/CASE IDENTIFIER:

Site Name ______
Discharger ______
Pollutant ______
Design Case ______
DOS FILE NAME ______

Prepared by:

Date prepared:

AMBIENT DATA:

Bounded or unbounded? ______			
Channel width ______	m		
Channel depth ______	m		
Depth at discharge ______	m		
Ambient flowrate ______	m^3/s	or: Ambient velocity ______	m/s
Manning's n ______		or: Darcy-Weisbach f ______	

Density data:	Density units: kg/m^3
Fresh or salt water? ______	Temperature units: °C
Density or temp. values? ______	
If uniform:	If stratified:
Average density/temp. ______	Density/temp. at surface ______
	Density/temp. at bottom ______
	Stratification type ______
	(Pycnocline height ______ m)
	(Density/temp. jump ______)

DISCHARGE DATA:

Nearest bank (left/right)? ______		Distance to one endpoint ______	m
Diffuser length ______	m	to other endpt. ______	m
Total number of openings ______			
Port diameter ______	m	with contraction ratio ______	
Diffuser arrangement/type ______			
Alignment angle (GAMMA) ______	°	Horizontal angle (SIGMA) ______	°
Vertical angle (THETA) ______	°	Rel. orientation (BETA) ______	°
Port height ______	m		
Discharge flow rate ______	m^3/s	or: Discharge velocity ______	m/s
Discharge density ______	kg/m^3	or: Discharge temp. ______	°C
Concentration units ______			
Discharge concentration ______			

MIXING ZONE DATA:

Is effluent toxic? ______		If yes: CMC value ______	
		CCC value ______	
Is there a WQ standard for conventional pollutant? ______		If yes: value of standard ______	
Any mixing zone specified? ______		If yes: distance ______	m
		or width (% or m) ______	
		or area (% or m^2) ______	
Region of interest ______	m		
Grid intervals for display ______			

Date of data input into CORMIX2:

TABLE 4 INPUT DATA CHECKLIST FOR CORMIX3

CORMIX3 -- Buoyant surface discharges -- CORMIX3

SITE/CASE IDENTIFIER:
Site Name ________________
Discharger ________________
Pollutant ________________
Design Case ________________
DOS FILE NAME ________________

Prepared by:

Date prepared:

AMBIENT DATA:
Bounded or unbounded? ______
Channel width ______ m
Channel depth ______ m
Ambient flowrate ______ m^3/s or: Ambient velocity ______ m/s
Manning's n ______ or: Darcy-Weisbach f ______

Density data:
Fresh or salt water? ______
Density or temp. values? ______
Average density ______ kg/m^3 or: Average Temperature ______ °C

DISCHARGE DATA:
Discharge located on left/right bank?
Discharge configuration ______ (flush, protruding, or coflowing)
Horizontal angle (SIGMA) ______ ° If protruding:
Depth at discharge ______ m Distance from bank ______ m
Bottom slope (THETAB) ______ °
If rectangular cross-section: If circular cross-section:
Discharge channel width ______ m Outlet pipe diameter ______ m
Discharge channel depth ______ m
Discharge flowrate ______ m^3/s or: Discharge velocity ______ m/s
Discharge density ______ kg/m^3 or: Discharge temp. ______ °C
Concentration units ______
Discharge concentration ______

MIXING ZONE DATA:
Is effluent toxic? ______ If yes: CMC value ______
CCC value ______
Is there a WQ standard for conventional pollutant? ______ If yes: value of standard ______
Any mixing zone specified? ______ If yes: distance ______ m
or width (% or m) ______
or area (% or m^2) ______
Region of interest ______ m
Grid intervals for display ______

Date of data input into CORMIX3:

IT GIVES A SECTION ON THE MIXING ZONE, TOXIC REGION AND REGION OF INTEREST. UNFORTUNATELY OUTPUT IS REPRESENTED BY COMPUTER VARIABLES THAT THE USER MUST BE FAMILAR WITH

HYDRODYNAMIC OUTPUT IS DIVIDED INTO MODULES SUCH AS:

1. DISCHARGE MODULE
2. WEAKLY DEFLECTED WALL JET
3. STRONGLY DEFLECTED WALL JET
4. BOUNDAY IMPINGEMENT/VERTICAL MIXING
5. BUOYANT SPREADING
6. PASSIVE AMBIENT MIXING

BEFORE EACH MODULE IT DEFINES THE VARIABLES AND COORDINATE SYSTEM. IT THEN GIVES THE OUTPUT FOR THAT MODULE INCLUDING:

PLUME TRAJECTORY
DILUTION
CONCENTRATION
SIZE

IT THE CMC OR CCC VALUES ARE REACHED, IT IS REPORTED. WHEN THE MIXING ZONE EDGE IS REACHED IT IS REPORTED.

IF SUGGESTIONS ARE DESIRED REGARDING IMPROVEMENTS FOR THE DISCHARGE, CORMIX WILL GIVE THEM.

EXAMPLE CHECK LIST

SURFACE DISCHARGE FIGURES.

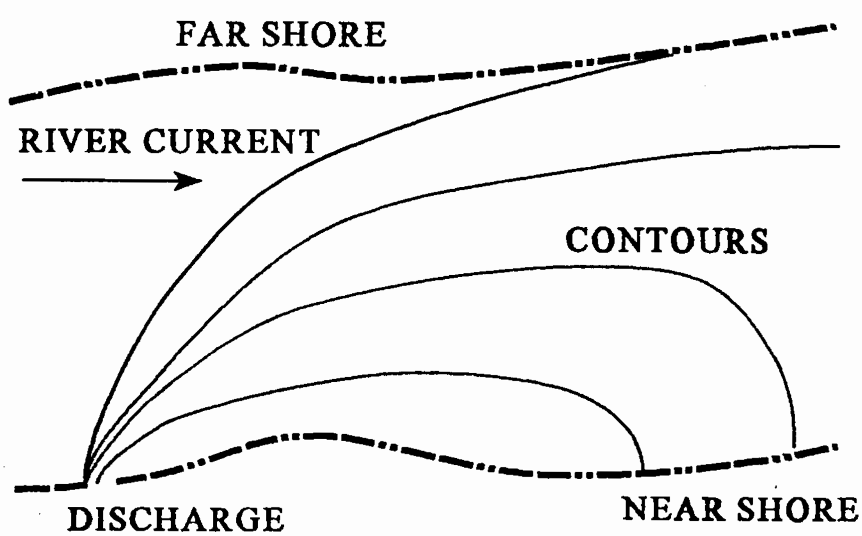
STRATIFICATION FIGURES.

DIFFUSER ANGLE DEFINITIONS

PSY - River Model

THE PSY RIVER MODEL IS FOR SHALLOW RIVERS AND SHORE ATTACHED PLUMES. IT IS A PURE DIFFUSION MODEL BASED ON AMBIENT TURBULENCE. IT DOES NOT CALCULATE THE NEAR FIELD WHERE JET ENTRAINMENT AND TURBULENCE CAUSES INITIAL DILUTION. THIS MUST BE ADDED SEPARATELY IF DESIRED.

IT DOES INCLUDE THE EFFECTS OF BOTH SHORES AND THE PLUME FILLING THE RIVER. IT ALSO INCLUDES THE EFFECTS OF RIVER BENDS ON THE DIFFUSION.



IT IS BASED ON THE THEORETICAL DEVELOPMENT BY P.P. PAILY AND W.W. SAYRE, "MODELING OF SHORE-ATTACHED THERMAL PLUMES IN RIVERS," JOURNAL OF THE HYDRAULICS DIVISION, ASCE, VOL. 104, NO HY5, MAY 1978. THE CODE WAS WRITTEN BY NICK YEN. IT IS BASED ON THE STANDARD TRANSIENT DIFFUSION EQUATION:

$$\frac{\partial C}{\partial t} = -\nabla C \cdot \bar{U} + \varepsilon_m \nabla^2 C$$

$$where\ C = \bar{C} + C$$

$$and\ U = \bar{U} + U$$

USING THE STANDARD TURBULENCE ASSUMPTIONS THESE EQUATIONS REDUCE TO:

$$\frac{\partial C}{\partial t} + u_i \frac{\partial C}{\partial x_i} = -\frac{\partial}{\partial x_i}\left(\varepsilon_i \frac{\partial C}{\partial x_i}\right)$$

WHERE ϵ_I IS THE TURBULENT DIFFUSION COEFFICIENT IN THE I^{th} DIRECTION.

WHEN DEPTH AVERAGED CONDITIONS ARE USED, THE VERTICAL DIRECTION DISAPPEARS.

IF THE THE DIMENSIONLESS DISTANCE ACROSS THE RIVER IS REPLACED BY THE DIMENSIONLESS FRACTIONAL RIVER FLOW, $p = q / Q_r$ SUCH THAT P IS 0 ON THE NEAR SHORE AND 1 AT THE FAR SHORE, THESE EQUATIONS REDUCE TO:

$$\frac{\partial C}{\partial x} = D\frac{\partial^2 C}{\partial p^2}$$

where D IS TRANSVERSE DIFFUSION COEFFICIENT.

THE SOLUTION OF THIS EQUATION USING THE APPROPRIATE BOUNDARY CONDITIONS IS:

$$C(p,X) = \int_{p\ 0}^{p\ 1} C_i(\zeta,0)\frac{1}{\sqrt{2\pi}} e^{(p\quad \zeta)^2} d\zeta$$

where C_i IS THE INITIAL SOURCE STRENGTH AT X=0 IF AN INITIAL DILUTION IS DEFINED AS $P = a\, Q_o / Q_r$ WHERE Q_o AND Q_r ARE THE DISCHARGE AND RIVER FLOW RESPECTIVELY AND THE FAR SHORE IS CONSIDERED USING AN IMAGE ANALYSIS THE FINAL SOLUTION IS:

$$\frac{C(p,x) - C_a}{C_o - C_a} = \frac{1}{a}\left\{\left[F_r\left(\frac{p+P}{\sigma_p}\right) - F_r\left(\frac{p-P}{\sigma_p}\right)\right] + \right\}$$

$$[C(p,x) - C_a]\frac{a}{C_o - C_a} = \left[F_r\left(\frac{p+P}{\sigma_p}\right) - F_r\left(\frac{p-P}{\sigma_p}\right)\right] +$$

$$\sum_{n\ 1}^{\infty}\left[F_r\left(\frac{2n+(p-P)}{\sigma_p}\right) - F_r\left(\frac{2n-(p-P)}{\sigma_p}\right)\right] +$$

$$\sum_{n\ 1}^{\infty}\left[F_r\left(\frac{2n-(p+P)}{\sigma_p}\right) - F_r\left(\frac{2n+9p+P)}{\sigma_p}\right)\right]$$

where $\sigma_p = \sqrt{2Dx}$ IS THE STANDARD DIVIATION IN p DOMAIN, C_o AND C_A ARE THE DISCHARGE AND AMBIENT CONCENTRATION. THE FUNCTION : F_r IS THE STANDARIZED MOMULATIVE NORMAL DISTRIBUTION FUNCTION CORRESPONDING TO THE PROBABILITY FUNCTION:

$$f_r(p - \zeta, x) = \frac{1}{\sigma_p}\left(\frac{1}{\sqrt{2\pi}}e^{\frac{p\ \ \zeta}{2\sigma_p}}\right)$$

THE SUMMATION TERMS COME FROM THE IMAGE SOLUTION REPRESENTING REFLECTIONS FROM THE FAR SHORE. ONLY THE FIRST 4 FOR 5 ARE NEEDED FOR REASONABLE ACCURACY.

THE EQUATIONS CAN EASILY BE SOLVED USING A COMPUTER. THE MAXIMUM CONCENTRATION AT THE NEAR SHORE IS WHERE p = 0. THE REAL PROBLEM IS FINDING THE CORRECT VALUE FOR THE TRANSVERSE DIFFUSION COEFFICIENT, D.

DIFFUSION COEFFICIENT

THE DIFFUSION COEFFICIENT USED IN THE PROGRAM IS CALUCLATED FROM:

$$D = \frac{\alpha}{0.063} \frac{n\sqrt{g}}{B^2} (\bar{d})^{5/6}$$

WHERE $\bar{d}$ IS THE AVERAGE RIVER DEPTH, B IS THE RIVER WIDTH, n IS MANNINGS n, g IS GRAVITY AND α IS A DIFFUSION COEFFICIENT GIVEN BY:

$$\alpha = 0.4\left(\frac{B}{\bar{d}}\right)^2 \left(\frac{\bar{u}}{u*}\right)^2 \left(\frac{\bar{d}}{r_c}\right)^2$$

WHERE u* IS THE SHEAR VELOCITY, $(g\bar{d}S)^{1/2}$, S IS THE ENERGY GRADIENT FROM MANNING'S EQUATION, AND r_c IS THE RIVER CURVATURE. FOR STRAIGHT RIVERS r_c IS SET TO 25 B. THE PROGRAM ALLOWS FOR DIFFERENT COEFFICIENTS FOR DIFFERENT REACHES OF THE RIVER.

PROGRAM INPUT AND OUTPUT

PROGRAM INPUT CAN BE EITHER INTERACTIVE OR BATCH FROM AN INPUT FILE. FOR THE INTERACTIVE MODE, RUN THE PROGRAM AND ANSWER QUESTIONS. INFORMATION NEEDED IS:

1. TITLE INFORMATION
2. EFFLUENT FLOW IN m^3/s
3. RIVER FLOW IN m^3/s
4. INITIAL CONCENTRTION
5. AMBIENT CONCENTRATION
6. INITIAL DILUTION (1 - 4) 1.0 BEING NO DILUTION
7. NUMBER IF SUB REACHES IN RIVER

8.... INPUT FOR EACH SUB REACH GIVING r_c, B, d, S(SLOPE), AND DISTANCE TO END OF SUB REACH in meters.

TO MAKE IN INPUT FILE:

Title Information
Q_o
Q_r
C_o
C_a
a
Number is reaches
r_c, B, d, S, Distance (m) to downstream end of subreach
(one line for each sub reach starting at discharge)

OUTPUT

OUTPUT CONSISTS OF THREE TABLES AS FOLLOWS:

DISTANCE D/S	$C_{MAX} - C_a$	DILUTION
X.	X	X
X.	X	X
X.	X	X
.	.	.
.	.	.

CONCENTRATION CONTOUR	AREA	MAX WIDTH	MAX DISTANCE
0.5	X	X	X
0.4	X	X	X
0.2	X	X	X
.	.	.	.
.	.	.	.
.	.	.	.

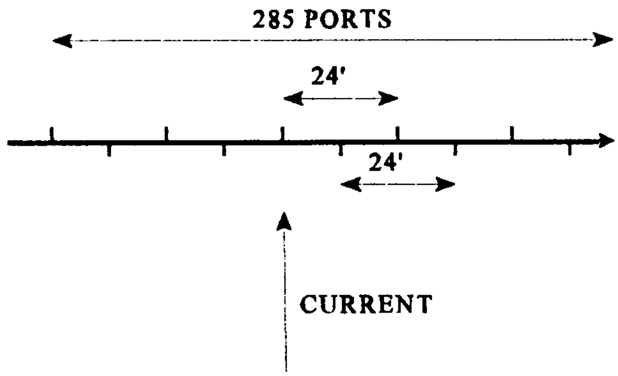
THE LAST TABLE IS OPTIONAL AND CONSISTS OF:

CONCENTRATION CONTOURS

	0.2		0.05		....	
DIST.	WIDTH	AREA	WIDTH	AREA	WIDTH	AREA
X	X	X	X	X	X	X
X	X	X	X	X	X	X
X	X	X	X	X		
X	X	X	X	X		
X	X	X				

Problem No. 1

UM, UDKHDEN, CORMIX SAMPLE PROBLEM
A DIFFUSER CONSISTING OF 285 PORTS ON ALTERNATE SIDES OF A DIFFUSER. THEY ARE SPACED 24' APART ON A SIDE (12' APART ON ALTERNATE SIDES)



TOTAL FLOW = 102 MGD (MILLION GALLONS PER DAY)
EFFLUENT TEMP = 25 C
EFFLUENT SALINITY = 0.0 PPT
DEPTH OF CENTER PORT = 70 m (MLLW)
PORT DIAMETER = 8.5 cm
PORT ELEV.. = 0.84 m
VERT ANG = HORIZONTAL = 0.0 (DEFAULT)
CONT COEF = 1.0 (DEFAULT)
POLLUT CONC. = 100
HORIZ ANG = 90
FAR FIELD VEL = 15 cm/s

AMBIENT TABLE

DEPTH m	CURR	DENS	SAL ppt	TEMP C	CONC
0.00	0		34.99	26.18	0
30.48	0		35.00	25.60	0
45.72	0		35.02	24.95	0
60.96	0		35.00	24.60	0
76.20	0		35.02	21.22	0

DETERMINE THE DILUTION AND CONCENTRATION AT THE TRAPPING LEVEL AND AT THE END OF THE FAR FIELD (2000 m)

Problem No. 2

CORMIX, UDKHDEN, UM CASE STUDY:

DISCHARGE: 0.082 m^3/s, FRESHWATER AT 25 C

RIVER: NARROW-SHALLOW , the 7Q10 Q = 8.1 m^3/s, Width 200-300 ft, The effective rectangular shape of the river is taken as 80 m wide and 0.6 m deep.

The river is fresh water with an average temperature of 20 C - non stratified.

OUTFALL: Case 1: Single port discharge 8-in(20 cm) diameter pointed downstream. Located 30 m from right bank on 0.14 m off bottom.

Case 2: Multiple port diffuser: 8 m long whose center is 70 feet (21 m) from the right bank. 7 equally spaced nozzles 2.5-in (0.0635 m) in diameter point downstream.

Regulations: RMX is 1000 ft downstream (304.8m). The required dilution is 50. The effluent is not toxic as a result no TMX (ZID) exists.

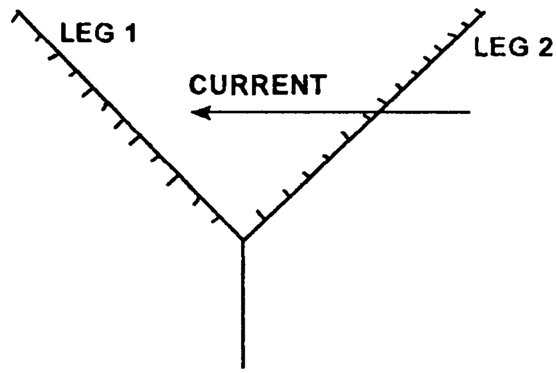
Will either of the proposed outfalls work?

Problem No. 3

UDKHDEN, UM, and CORMIX 2 CASE STUDY

A municipal sewage treatment plant discharges treated sewage into a bay. A multiple port diffuser is being studied to reduce chlorine concentrations at the edge of the mixing zone. Since the plant is existing and is planning on future expansion to accommodate population growth of the city, various flows and configurations are to be considered.

OUTFALL: The diffuser consists of a "Y" with each leg of the "Y" having ports. The angle between legs is 90 degrees and the residual current flows an angle of 45 degrees to each. Ports point downstream (at 45 degrees to the current) and up at an angle of 32 degrees with the horizontal.



For discharge rates from 0.5 to 2.0 m3/s the diffuser being considered has 50 ports spaced 2.5 m apart. The port diameter is 10 cm. (Only one leg of the "Y" is used.

For discharge rates from 2.8 to 5.0 m3/s, the diffuser considered has 100 ports spaced 2.5 m apart. The port diameter is 10 cm. (2nd leg of the y diffuser opened). The effluent is freshwater at 20C

AMBIENT The ambient is sea water with a salinity of 28 ppt. The worst case condition is when the current is low at 0.006 m/s and there is density stratification. The low tide depth is 10 m. Temperature stratification under worst conditions is measured to be.

Depth - meters	Temperature - C
0	24.9
1	22.5
2	21.5
3	20.6
4	15.0
5	13.5
6	11.4
7	10.9
9	10.8
10	[illegible]

Regulations: The minimum dilution at the edge of the mixing zone must be at least 30. The mixing zone is defined as that location where the plumes fill the water column or is trapped.

Determine the dilutions at the extreme flows for each diffuser configuration.

Determine these dilutions if the ambient is not stratified and the ambient temperature is constant at 20 C

Problem No. 4

UDKHDEN CASE STUDY

A textile plant discharges waste water into a shallow river. Conditions are as follows:

DISCHARGE: 0.044 m^3/s of fresh water at 21.5 C through a single 0.318 m diameter port. It is discharged horizontally, normal to the river current.

RIVER: 100 m wide, 3.0 m deep, 0.366 m/s average current. Average temperature = 21 C

REGULATIONS: Dilution must be at least 75 within a mixing zone of 100 meters downstream.

Use the image method to simulate discharge into a shallow river and estimate the dilution at the edge of the mixing zone. Determine where the plume fills the water column. Estimate the width of the plume 100 m downstream.

Problem No. 5

UDKHDEN, UM AND CORMIX2 CASE STUDY

A paper mill discharges colored waste water into the ocean from a submerged diffuser. The following specifications apply:

Diffuser: 4 equal ports spaced 4.6 m apart, 20 cm in diameter. They are on 50 cm risers and discharge perpendicular to the diffuser manifold pipe at a vertical angle of 40 degrees. The prevailing ocean currents are normal to the diffuser. Because of severe sand problems in the area, not all ports are open all the time. As a result,2, and 4 ports must be considered. The center of the diffuser is 7.0 meters below MLLW.

Discharge: The effluent is fresh water with a density of .9974 gm/cc. The maximum discharge rate is 13 mgd (0.56 m^3/s).

Ambient: The ocean is stratified during the winter conditions being considered as follows

Depth - m	Density gm/cc	current m/s
0.00	1.02472	0.152
1.0	1.02472	0.152
2.0	1.02503	0.152
3.0	1.02503	0.152
6.0	1.02510	0.140
7.0	1.02518	0.09

A dilution of 50 is required when the plume is trapped or reaches the surface. Determine whether this is satisfied for the three operating conditions with 2,and 4 ports open. Assume the same spacing for each.

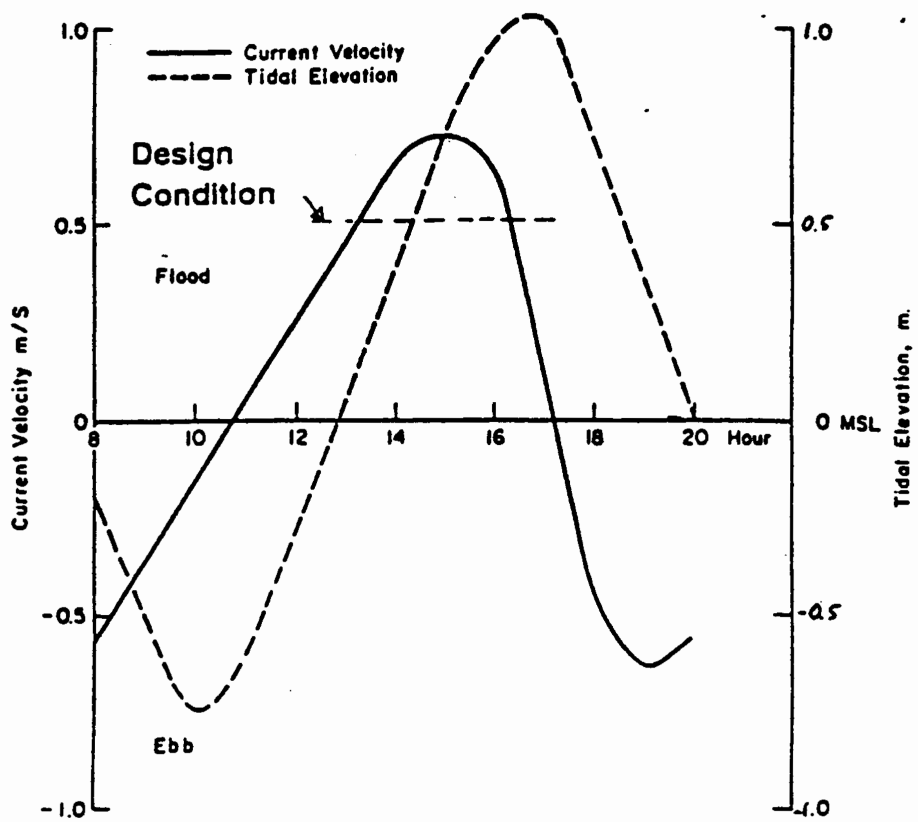
Problem No. 6

PDS **CASE STUDY:**

An industrial plant is to discharge into an estuary from a side channel giving a surface plume. Specifications are:

OUTFALL:	**2.0 m wide. Depth varies as tide from 2.0 m to 0.025 m point 90 degrees to tidal currents.**
EFFLUENT:	**2.2 m^3/s of 25 ppt salt water with an average temperature of 25 C. It contains copper at 80 ugm/L**
ESTUARY:	**6.0 m deep at MSL. Currents range from +0.7 m/s to -0.5 m/s. The depth at discharge is about 2.5 m and drops off at about 11 degrees slope. Mean salinity is 25 ppt and the temperature is 15 C. (density of 1018 kg/m^3)**
REGULATIONS:	**The regulatory mixing zone is a semi-circle of radius = 300 m. The CMC is 25 ugm/L and the CCC is 15 ugm/L**

Make several predictions during the tidal cycle to determine if this outfall will be in compliance with regulations. Check to see if the plume attaches to the shore. If so, use the image method to simulate discharge. Determine the surface area enclosed by the 2 degree isotherm for each case.

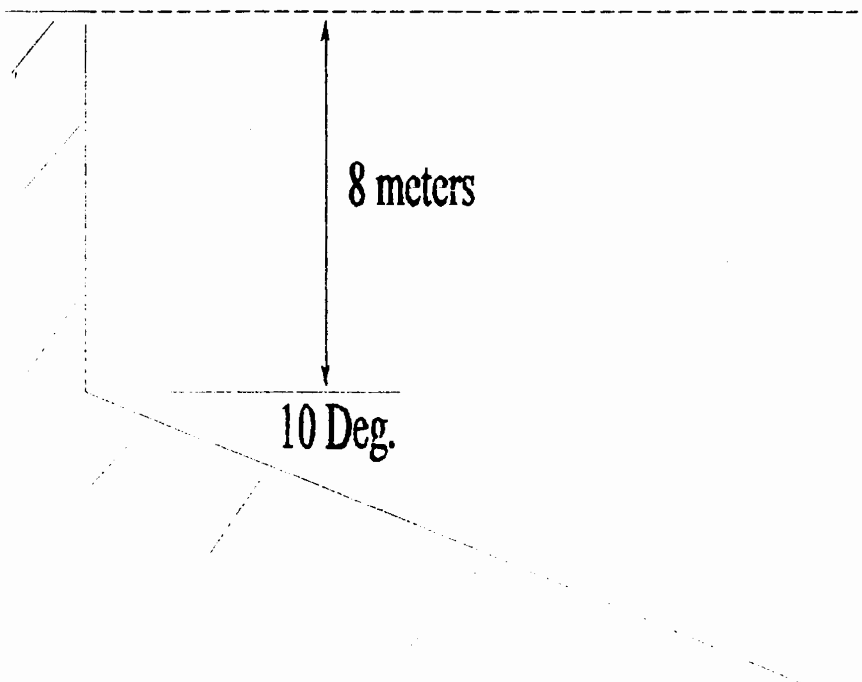


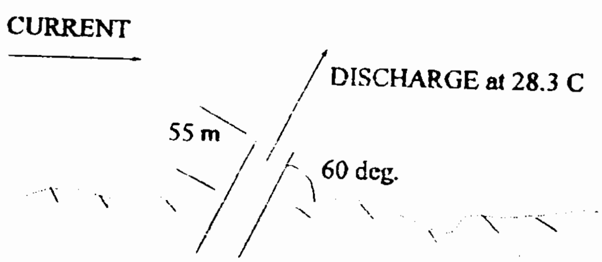
Problem No. 7

PDS CASE STUDY 2:

An electrical power plant is located on the shore of a large lake. It discharges 25.1 m^3/s of water used to cool the condensers. The outfall structure protrudes into the lake about 55 m at an angle of 60°. The heated water is discharged at an excess temperature of 8.3 C. The width of the discharge channel is 10.7 m and the average depth in the channel is 4.0 m. The ambient current is essentially zero but .03 m/s is more realistic and flows along the shore. Winds are moderate to zero under worst case conditions and the lake temperature to be considered is 20 C.

Determine the extent and surface area of the 2 degree excess temperature isotherm.





Problem No. 8

CORMIX 3 CASE STUDY:

An industrial plant is to discharge into an estuary from a side channel giving a surface plume. Specifications are:

OUTFALL: 2.0 m wide. Depth varies as tide from 2.0 m to 0.025 m pointed 90 degrees to tidal currents.

EFFLUENT: 2.2 m^3/s of fresh water with an average temperature of 20C. It contains copper at 80 ugm/L

ESTUARY: 6.0 m deep at MSL. Currents range from +0.7 m/s to -0.5 m/s. The depth at discharge is about 2.5 m and drops off at about 11 degrees slope. Mean salinity is 25 ppt and the temperature is 20 C. (density of 1018 kg/m^3)

REGULATIONS: The regulatory mixing zone is a semi-circle of radius = 300 m. The CMC is 25 ugm/L and the CCC is 15 ugm/L

Make several predictions during the tidal cycle to determine if this outfall will be in compliance with regulations.

Problem No. 9

CORMIX 3 AND PDS CASE STUDY

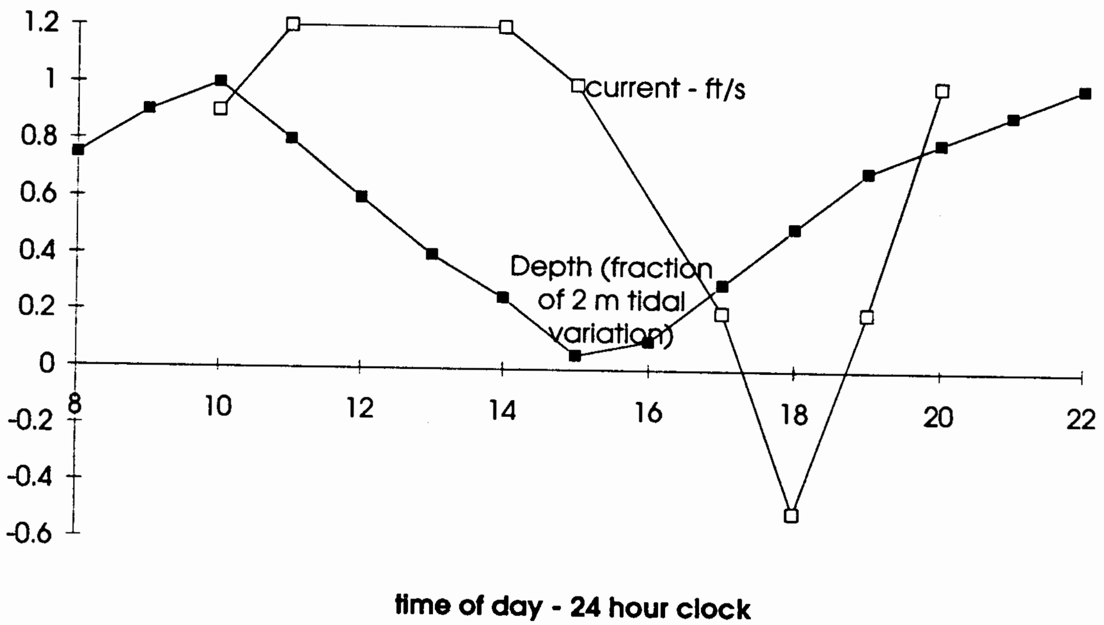
A large power plant is located in an estuary that is about 3 km wide. The plant discharges 60 m^3/s of heated 60 C water at 90 degrees to the shore through an open channel. The channel is 10 m wide and the depth varies from 2.0 to 2.5 m from low to high tide. At low tide the current reaches 0.3 m/s downstream. At high tide the flow reverses to a maximum of - 0.025 m/s. The average depth of the estuary is 8 m and has a temperature of 10 C. Regulations state that the excess surface temperature cannot exceed 4 C outside a radius of 2000 m from the discharge point. Assume that the water depth at the discharge point is 4 m and slopes at 30 degrees

Investigate the applicability of this discharge. Will it satisfy regulations? Will it hit the far shore?

Problem No. 10

CORMIX3 AND PDS **CASE STUDY**

A chemical plant discharges cooling water into a large river. The river is influenced by the tide giving a depth and velocity variations as given in the graph. The plant discharges 0.84 m^3/s of fresh water at 25.5 C through an open channel that points upstream of 100 degrees (90 being normal to shore). At low tide the discharge is through a channel 4.51 m wide and about 0.6 m deep. At high tide the flow is from a 50 m wide channel whose average depth is .2 m. The average river temperature is 13.3 C. The toxicity is such that an average dilution during the tidal cycle must be at least 3.0 at the edge of a 180 m mixing zone.



DESIGN PROBLEM

THE TREATED WATER FROM A SEWAGE TREATMENT PLANT IS TO BE DISCHARGED INTO A NARROW RIVER. THE MAXIMUM SUMMER DISCHARGE FLOW IS 0.6 M^3/S. THE 7Q10 RIVER FLOW GIVES AND AVERAGE RIVER DEPTH OF 8 METERS AND A WIDTH OF 30 METERS WITH AN AVERAGE VELOCITY OF 10 CM/S. THIS RIVER IS FRESH WATER AND HAS AN AVERAGE TEMPERATURE OF 15 C. THE EFFLUENT IS ALSO FRESH WATER WITH AN AVERAGE TEMPERATURE OF 20 C.

THE TRACER OF CONCERN IS CHLORINE USED FOR DISINFECTING THE WASTE WATER. ITS CONCENTRATION AT DISCHARGE IS 0.5 MG/L. REGULATIONS ARE THAT THE AVERAGE PLUME CONCENTRATION OF CHLORINE CANNOT BE GREATER THAN .019 MG/L WITHIN 10 METERS OF THE OUTFALL AND 0.01 MG/L WITHIN 100 METERS OF THE OUTFALL. IN ADDITION, THE DISCHARGE VELOCITY IS REQUIRED TO BE AT LEAST 3 M/S UNDER MAXIMUM SUMMER DISCHARGE RATES.

USE WHATEVER MODEL YOU THINK IS APPROPRIATE AND COME UP WITH A SUGGESTED DESIGN THAT WILL HAVE THE SMALLEST IMPACT ON THE RIVER. (YOU MAY WANT TO TRY DIFFERENT MODELS). WHEN YOU HAVE FINISHED, I'LL SHOW YOU WHAT WAS ACTUALLY BUILT.

Design Problem No. 2

A ELECTRICAL POWER PLANT DISCHARGES 10 M^3/S OF WARM WATER AT 82 C. THE AMBIENT IS LARGE LAKE AND IS SLIGHTLY STRATIFIED AS GIVEN IN THE TABLE BELOW. THE AMBIENT BOTTOM SLOPES DOWN AT ABOUT 10 DEGREES AFTER AND INITIAL 8 METER DROP.

REGULATIONS ARE SUCH THAT THE 2.0 DEGREE EXCESS TEMPERATURE OR GREATER CANNOT TAKE MORE THAN 1,000,000 M^2. IF A SUBMERGED DIFFUSER US USED, THE TEMPERATURE AT THE TRAPPING LEVEL OR SURFACE CANNOT BE GREATER THAN 1.0 DEGREE ABOVE AMBIENT .

DESIGN A DISCHARGE SYSTEM THAT WILL MEET ONE OR THE OTHER OF THE ABOVE REGULATIONS.

DEPTH	TEMPERATURE
0.0	20.0
5.0	18.0
20.0	15.0
40.0	15.0

CORMIX1 CASE STUDY: SUBMERGED SINGLE PORT DISCHARGE IN A DEEP RESERVOIR

This case study illustrates the application of CORMIX1 to the prediction of the effluent from a small manufacturing plant into a large and deep, stratified reservoir.

Problem Statement

A manufacturing plant (A-Plant) is discharging its effluent into an adjacent deep reservoir. The plant design flowrate is 3.5 mgd ($\approx$ 0.153 m^3/s). The effluent contains chlorides at a concentration of 3500 ppm, and is released at a temperature of 68° F (= 20 °C).

The existing reservoir has been formed by flooding a river valley. The reservoir length is approximately 60 miles. The water level in the reservoir is fluctuating depending on the release operation at the downstream dam with its hydropower installation. During summer conditions, the reservoir level is typically at an elevation of 710 ft above sea level. This results in a reservoir width of about 4000 ft ($\approx$ 1200 m) and a maximum depth of 310 ft ($\approx$ 95 m) at the discharge location. The mean river flow into the reservoir during the summer low-flow conditions is about 9000 cfs ($\approx$ 255 m^3/s). The typical temperature of the inflowing river water is 55 °F ($\approx$ 13 °C).

Figure 1 shows the local bathymetry (as obtained from a USGS map) in the vicinity of the proposed discharge. Since the discharge is very small relative to the reservoir size and the ambient flowrate, it is expected that mostly local conditions will be important, and not overall reservoir dimensions. (Note: Any such conjecture has to be verified against the final simulation results, and adjustments have to be made if needed.)

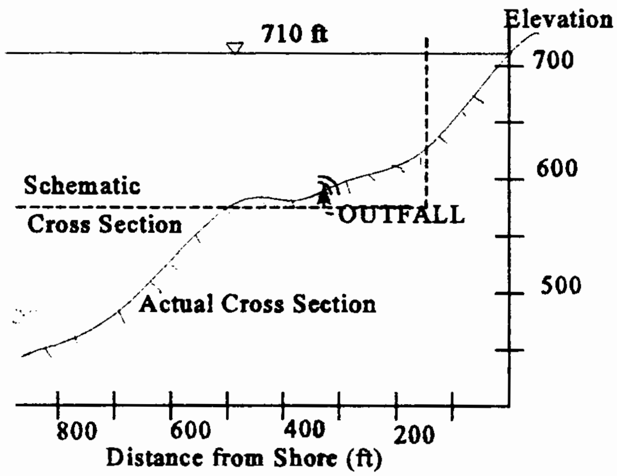


Figure 1

Temperature data as a function of depth obtained from field measurements in the center of the reservoir show a significant temperature stratification (see Figure 2), as is typical for such deep reservoirs during summer conditions. The stratification can be expected to be horizontally uniform and therefore similar conditions will hold at the discharge site. Also, the river inflow is colder than the surface layer of the stratified reservoir. Therefore, it can be expected that the river water will flow predominantly in a vertically limited layer. In this example, the layer is assumed to extend from a depth of about 30 m to about 16 m below the surface. The velocity of that flow is estimated at about 1.5 cm/s (= 0.015 m/s), given the 14 m thick layer and an about 1200 m width at that elevation. (Note: More detailed hydrodynamic investigations, using available models for stratified reservoir dynamics, can be used to obtain more precise estimates of the velocity field. Generally, however, it cannot be assumed that the velocity in stratified reservoirs is given by the simple average of the flowrate divided by the cross-sectional area.)

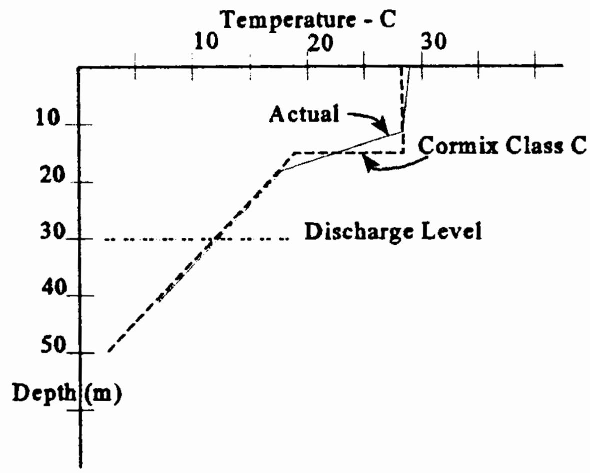


Figure 2

The proposed discharge location on the side slope of the cross-section is also shown in Figure 1: a submerged single port discharge at an elevation of 610 ft above sea level, i.e. at a local depth of 100 ft (≈ 30.5 m) below the surface, is proposed in the initial design phase. The port diameter is 10 in (= 0.254 m) and is located 2 ft (≈ 0.6 m) above the local bottom. The discharge is pointing perpendicular to the shoreline and is angled upward at 10 °.

The discharge is subject to mixing zone regulations whereby the mixing zone width is less than 10 % of the width of the water body. Furthermore, the chlorides in the effluent are considered as toxic with CMC and CCC limits of 1200 and 600, respectively.

Determine the suitability of this discharge.

PSY EXAMPLE PROBLEM

A PAPER MILL DISCHARGES WASTE WATER INTO A SHALLOW, NARROW RIVER. THE SLOPE OF THE RIVER IS 1:1000 AND THE AVERAGE DEPTH IS 1.5 m. THE RIVER FLOW DURING THE ANALYSIS PERIOD IS 53.83 m^3/s. THE CHARACTERISTICS OF THE RIVER FOR THE FIRST 330 m DOWNSTREAM ARE AS FOLLOWS:

Sub reach	Curvature - m	Width - m	Distance to end
1	0.	18.8	20.9
2	0.	12.5	64.7
3	0.	13.6	114.8
4	0.	16.7	195.2
5	300.	18.8	277.7
6	300.	11.5	294.5
7	300.	10.3	325.0
8	110.	10.4	328.9
9	15.	5.0	330.0

THE EFFLUENT DISCHARGE RATE IS 0.68 m^3/s. THE MINIMUM SHORELINE DILUTION, 300 m DOWNSTREAM MUST BE AT LEAST 15. DOES IT SATISFY THE REGULATION?

WHAT IS THE SURFACE AREA WITHIN THE C/Co = 0.2 CONTOUR?

THE INPUT TO THIS PROGRAM IS AS FOLLOWS:

Study of McKenzie River	(Title)
0.68	(Discharge rate)
53.83	(River flow)
1.0	(Concentration in effluent)
0.0	(Concentration in river)
1.0	(Initial dilution factor - no initial dilution)
9	(9 sub reaches)

```
0.     18.8  1.5   .001  20.9
.
.    (NINE ENTRIES OF radius, width, depth, slope, distance to end)
.
.
15     5.0   1.5   .001  330.0
```