Dilution Models for Effluent Discharges

4th Edition
(Visual Plumes)

by

W.E. Frick[1], P.J.W. Roberts[2], L.R. Davis[3],
J. Keyes[4], D.J. Baumgartner[5], K.P. George[6]

[1] Ecosystems Research Div., NERL, USEPA, Athens, Georgia 30605-2700
[2] Georgia Institute of Technology, Atlanta, Georgia 30332
[3] CH2M HILL, Corvallis, Oregon 97330
[4] Brown and Caldwell, Atlanta, Georgia 30346
[5] University of Arizona, Tucson, Arizona 98706
[6] Alaska Department of Environmental Conservation, Juneau, Alaska 99801

4 March 2003

Ecosystems Research Division, NERL, ORD
U.S. Environmental Protection Agency
960 College Station Road
Athens, Georgia 30605-2700

Abstract

Visual Plumes (VP), is a Windows-based computer application that supersedes the DOS PLUMES (Baumgartner, Frick, and Roberts, 1994) mixing zone modeling system. VP simulates single and merging submerged plumes in arbitrarily stratified ambient flow and buoyant surface discharges. Among its new features are graphics, time-series input files, user specified units, a conservative tidal background-pollutant build-up capability, a sensitivity analysis capability, and a multi-stressor pathogen decay model that predicts coliform mortality based on temperature, salinity, solar insolation, and water column light absorption..

VP addresses the issue of model consistency in a unique way, by including other models in its suite of models. In this way it promotes the idea that in the future modeling consistency will be achieved by recommending particular models in selected flow categories. This approach is intended to encourage the continued improvement of plume models. Consistent with this goal, VP includes the DKHW model that is based on UDKHDEN (Muellenhoff et al., 1985), the surface discharge model PDS (Davis, 1999), the three-dimensional UM3 model based on UM, and the NRFIELD model based on RSB. These models may be run consecutively and compared graphically to help verify their performance. The Brooks equations are retained to simulate far-field behavior. Finally, DOS PLUMES may be selected as one of the "models," giving full access to its capabilities.

The time-series file-linking capability provides a way to simulate outfall performance over long periods of time. Most effluent and ambient variables can be input from files that store data that change with time. This is the heart of the pollutant-buildup capability, designed for one-dimensional tidal rivers or estuaries to estimate background pollution from the source in question. It is also the basis for utilizing the Progressive Vector Diagram (PVD) approach. The time-series file linking capability is served by "summary" graphics, i.e., graphics that focus on overall performance indicators, like mixing zone dilutions or concentrations. These implementations allow plumes to be depicted in the far-field as individual plumes (for one or two runs) or as concentration "clouds" for many runs. This capability is useful for estimating beach impacts in situations exhibiting simply bathymetries.

Disclaimer

This document has been reviewed in accordance with the US Environmental Protection Agency's peer and administrative review policies and approved for publication. Mention of trade names or commercial products does not constitute endorsement or recommendation for use by the US EPA.

Table of Contents

Acknowledgments

Many individuals have contributed to the conception, development, debugging, review, modification, and encouragement of this work. We would like to gratefully acknowledge some of them:

Rosemarie Russo	Harvey Holm	Gabrielle Puz
Craig Barber	Hening Huang	Diana Robles
Jon Bergengren	Carlos Irizarry	Lew Rossman
Sandy Bird	Sam Karickhoff	Tim Rowan
Hira Biswas	Russell Kinerson	Henry Salas
Maynard Brandsma	Bob Languell	Robson Sarmento
Dave Brown	Astrid Larsen	Jenny Scifres
Tom Cavinder	Joseph Lee	Anne Sigleo
Wen-Li Chiang	Winston Lung	Mills Soldate
Curtis Dalton	Brandy Manders	Frank Stancil
Debra Denton	Steve McCutcheon	Robyn Stuber
Joseph DiLorenzo	Frank Meriwether	Patricia Tapp
Bill Driskell	Susanne Metzgar	Bruce Titus
Larry Fradkin	Bill Mills	Gil Veith
Norm Glenn	Phil Mineart	Jim Weaver
David Hericks	Bruce Nairn	John Yearsley
Mike Heyl	Terry Oda	David Young
Thomas Hogbom		

1   Introduction

Visual Plumes is a Windows-based mixing zone modeling application designed to replace the DOS-based PLUMES program (Baumgartner, Frick, and Roberts, 1994). Like PLUMES, VP supports initial dilution models that simulate single and merging submerged plumes in arbitrarily stratified ambient flow. Predictions include dilution, rise, diameter, and other plume variables. The Brooks algorithm is retained for predicting far-field centerline dilution and waste field width. New features include the surface discharge model (PDS), the multi-stressor bacterial decay model (based on Mancini, 1978), graphics output, time-series input, a sensitivity analysis capability, user-specified units, and a conservative tidal background pollutant build-up capability.

VP differs greatly from PLUMES in terms of operating system, model enhancements, model additions, organization, and appearance. The single-port Windows version of the UM model is now a fully three-dimensional flow model. It is renamed UM3 to emphasize this change and to differentiate it from the previous version. UDKHDEN, also a three-dimensional model, was one of the models in EPA's earlier guidance (Muellenhoff et al., 1985) that is reintroduced under the name DKHW. This addition illustrates a commitment to a comprehensive modeling platform that will foster scientific competition by encouraging modelers to continue to improve their applications.

Some parts of PLUMES have been brought in basically unchanged. The RSB model is one of these although its new name, NRFIELD and FRFIELD, sets the stage for introducing planned changes. For the many users of PLUMES who wish to support earlier projects or take advantage of some of its features, like computing length-scale, similarity parameters, and related input variables not explicitly supported by VP, VP is backward compatible. DOS PLUMES is one of the "models" supported VP. When selected, VP prepares the necessary PLUMES input file and displays the output, interpreting some of it graphically. For porting projects to VP, it also reads PLUMES input files (files with the *VAR* extension).

Like DOS PLUMES, VP allows the user to run many cases, however, multiple cases are easier to set up and to compare. Determining model sensitivity to various input parameters is facilitated. The ability to run different models, such as UM3 and DKHW, side by side and compare the results in graphical form, should facilitate model comparison. The ability to link in and graph verification data from files rounds out the ability to compare models.

Perhaps no other capability sets VP apart from PLUMES more than its ability to link in time-series files. This capability provides a way to simulate outfall performance over a long period of time and, thereby, over many environmental scenarios. Most effluent and ambient variables, such as effluent discharge rate and current direction, can be read from files containing values that change with time over different time intervals. Thus, a 24-hour diurnal flow file, cycled repeatedly, might be combined with a current-meter data set thousands of records long. This is the heart of the pollutant-buildup capability, the ability in one-dimensional tidal rivers or estuaries to estimate background pollution from the source in question. The time-series file linking capability is served by "summary" graphics, i.e., graphics panels that focus on overall performance indicators, like mixing zone dilutions or concentrations. The use of time-series files does imply the preparation of the necessary data in ASCII form, as described herein.

For all its changes, VP is still a transition product. First, VP only begins to take advantage of the object-oriented programming paradigm offered by the Delphi language in which it was

developed. Ideally, there will be time in the future to fully take advantage of this modeling paradigm, which would help enhance the interface and solve some of VP's current problems with maintainability. Second, VP is only a prototype for a comprehensive modeling platform that could ultimately support diverse mixing zone models, while also including a protocol designed to satisfy applicable regulations. When specified, the protocol would automatically run the currently recognized official models. Thus VP would foster scientific competition while supporting modeling consistency. Third, the tidal pollutant buildup capability is only a step in the direction of a fully three-dimensional mixing zone analysis package of the future.

1.1   General Overview of the Interface

The VP user interface is organized into five tabs: *Diffuser, Ambient, Special Settings, Text Output*, and *Graphics*. For setup and input, several Windows controls and components, such as tables, pull-down and pop-up menus, buttons, and lists are provided. Numerical input is dominated by two input tables, defining the diffuser characteristics and flow conditions and the ambient conditions. Other information is input in a memo box, a number of control panels, lists, and buttons, and, various edit boxes, lists, file dialogs, and radio buttons on the *Special Settings* tab.

A context-sensitive help system allows one to right-click on any component on the screen, or use the help menu. Many help topics contain hypertext links; text displayed in green may be clicked to display further information on the indicated item.

To reduce redundancy, several input interpretation techniques have been written into VP to make input requirements contingent on actual availability of data. In many applications, input tables must be completely filled in with data, whether the data are redundant or not. In VP, data need not be entered into the input tables when their existence is not implied. For example, if a measurement program sampled current speed at 10 and 30 meter depth and temperature at the surface and 25m, then four rows of input cells are needed to hold the data. However, VP requires the user to enter current speed only on the 10 and 30m rows, not on the surface and 25m rows. The exception is in the diffuser table where all required columns must have a value in the first row, which is called the base case.

To prepare VP to run the user must define the base case and complete at least one ambient profile in the table on the *Ambient* tab. Model selection and case specific information determines which columns require input; columns labeled *n/r* are not required by the specified configuration or target model. For more than one run, or rows, only cell values that are different from the base case need be entered. If a cell is empty, its value is inherited from the previous row. The runtime mode is determined by the setting of the Case selection radio button panel; choices are individual cases, all cases in sequence (running all ambient files or parsing the case range appended to the ambient file name), or all possible combinations of cases.

The organization of the data on different tabs emphasizes that VP diffuser and ambient input data are maintained in separate files with a *db* extension. With PLUMES it was often difficult to tell how individual cases varied from each other. VP alleviates that problem by file separation, which avoids repetition of ambient data. The advantage of storing the data in separate files is that ambient data files can be used for other projects.

VP supports user-specified units. On both *Diffuser* and *Ambient* tabs, the user can click on the row above the input table to select units from a list of up to five choices revealed on a pop-up list. Unless the *Units conversion* radio button is set to *label only*, the data in the affected columns are automatically updated to convert to the new unit. In addition, some of the columns are multi-use columns. For example, the salinity column can be changed to a density column by simply selecting a density unit from the list of unit options.

The *Special Settings* tab provides a choice of output variables and access to other controls, parameters, and options. The *Text Ouput* and *Graphical Output* tabs display the output. Graphics can be customized by double-clicking in the margins of each panel. Other options are provided on the left side of the graphing panels, including the Verify button that opens a verification file dialog box. Many VP settings are stored in the project file with the *lst* extension.

1.2   Models Supported by the Visual Plumes Platform

There are presently five recommended models in VP: DKHW, NRFIELD/FRFIELD, UM3, PDSW, and DOS PLUMES. These and the Brooks far-field algorithm and an experimental vector model are briefly described below.

UM3

UM3 is an acronym for the three-dimensional Updated Merge (UM) model for simulating single and multi-port submerged discharges. The model is coded in Delphi Pascal, the language of Visual Plumes.

UM3 is a Lagrangian model that features the projected-area-entrainment (PAE) hypothesis (Winiarski and Frick, 1976; Frick, 1984). This established hypothesis (Rawn, Bowerman, and Brooks, 1960) quantifies forced entrainment, the rate at which mass is incorporated into the plume in the presence of current. In UM3 it is assumed that the plume is in steady state; in the Lagrangian formulation this implies that successive elements follow the same trajectory (Baumgartner et al., 1994). The plume envelope remains invariant while elements moving through it change their shape and position with time. However, ambient and discharge 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 at maximum rise.

To make UM three-dimensional, the PAE forced entrainment hypothesis has been generalized to include an entrainment term corresponding to the third-dimension: a cross-current term. As a result, single-port plumes are simulated as truly three-dimensional entities. Merged plumes are simulated less rigorously by distributing the cross-current entrainment over all plumes. Dilution from diffusers oriented parallel to the current is estimated by limiting the effective spacing to correspond to a cross-diffuser flow angle of 20 degrees.

The runtime and display performance of UM3 has been improved by better controlling the simulation time step. In addition to being controlled by the amount of entrainment, the time step is now also sensitive to the amount of trajectory curvature. In some cases, this sensitivity to curvature actually reduces the number of time steps needed to produce a simulation because the sensitivity to entrainment can be reduced.

Due to the fact that UM3 is coded in Delphi Pascal, the native language of VP, UM3 is fully integrated with VP's background build-up capability. Given that a time-series record for tidal flow in a one-dimensional channel can be provided, VP can estimate the buildup of background concentration resulting from the repeated passage of a given fetch of water past the discharge.

DKHW

DKHW is an acronym for the Davis, Kannberg, Hirst model for Windows. Like UM3, DKHW is also a three-dimensional plume model that also applies to single and multi-port submerged discharges. Unlike UM3, DKHW is a Fortran-based executable that is called by VP on demand. This method of implementation plus a more detailed near-field theory carries a penalty in the form of generally greater execution time.

Within VP, DKHW runs from a DOS SHELL evidenced by a DOS window that appears when it is run. Depending on the operating system, one may need to close the DOS window after DKHW is finished running. The word "finished" appears in the window's title bar to indicate that DKHW is done, at which time the window may be closed.

DKHW is based on UDKHG and UDKHDEN described in *Fundamentals of Environmental Discharge Modeling* (Davis, 1999). It uses the Eulerian integral method to solve the equations of motion for plume trajectory, size, concentration and temperature. In this approach distance is the independent variable, whereas in the Lagrangian formulation time is the independent variable.

DKHW provides detailed calculations in both the Zone of Flow Establishment (ZFE) and in the fully developed zone, and considers gradual merging of neighboring plumes. This ability to model the near field in great detail is currently receiving renewed interest because salmonids are very sensitive to elevated temperature.

DKHW is presently limited to positively buoyant plumes.

PDSW

PDSW is the VP name for the PDSWIN executable model, an acronym for the Prych, Davis, Shirazi model for Windows, which has been modified to be compatible with VP. PDSWIN is a version of the PDS surface discharge program also described in *Fundamentals of Environmental Discharge Modeling* (Davis, 1999). PDS is a three-dimensional plume model that applies to discharges to water bodies from tributary channels, such as cooling tower discharge canals. Like DKHW, PDSWIN is a Fortran-based executable that is called by VP on demand.

PDSWIN provides simulations for temperature and dilution over a wide range of discharge conditions. It was used to develop the nomograms in Shirazi and Davis (1972). PDS is an Eulerian integral flux model for the surface discharge of buoyant water into a moving ambient body of water that includes the effects of surface heat transfer. The plume is assumed to remain at the surface with buoyancy causing it to rise and spread in all directions. The initial discharge momentum causes the plume to penetrate the ambient at the same time that the current bends the plume in the direction of flow. Discharge is assumed to be from a rectangular conduit into a large body of water. PDSWIN calculates plume trajectory, average and centerline dilution, plume width and depth and centerline excess temperature. It also calculates the areas within selected isotherms. In addition to VP output, additional output data are available in the PDS.OUT file in the default directory using any

compatible text editor. The user must monitor boundaries. Calculations beyond the point where the plume hits a boundary are questionable. Plume attachment at the near shore can be simulated using the image method in which the discharge flow and width are doubled and only one half of the resulting plume is considered. As with DKHW, the DOS window may have to be closed when PDSWIN is finished.

NRFIELD

NRFIELD (RSB), as its entry on the Model menu suggests, is the successor to the PLUMES RSB model. NRFIELD is an empirical model for multiport diffusers based on the experimental studies on multiport diffusers in stratified currents described in Roberts, Snyder, and Baumgartner (1989, a, b, c) and subsequent experimental works. NRFIELD is based on experiments using T-risers, each having two ports, so at least four ports must be specified for it to apply. An important assumption is that the diffuser may be represented by a line source. This assumption may have important implications on small mixing zones, in which the plumes may not have merged.

FRFIELD

The FRFIELD model estimates the long-term distribution of pollutants in the vicinity of the outfall. This models is based on the two-dimensional "visitation-frequency" model is not currently operational.

DOS PLUMES (DP)

DOS PLUMES, formerly called PLUMES, is the direct predecessor of VP. The PLUMES users' guide (Baumgartner, Frick, and Roberts, 1994) is available on the VP compact disk and the program may run independently of VP. It is linked to VP for two main reasons. First, many individuals have used PLUMES and have developed project files that they wish to import into VP, and, second, PLUMES has some unique capabilities that may be useful to the VP user. For example, the DOS application provides an easy way to develop the numerical identity between related variables, like between port effluent speed and direction and port effluent vector velocity components. The ability to process and display PLUMES output also allows one to see how modeling improvements have changed the predictions.

3-D, single-port vector model

This is an experimental model that is not currently recommended for official use.

Brooks far-field algorithm

This "model" is functionally different from the foregoing models in that it is not listed on the *Models* menu. Instead, it is specified by checking the *Brooks far-field solution* option on the *Model Configuration* checklist on the *Diffuser* tab. Briefly, the algorithm is a simple dispersion calculation that is a function of travel time and initial waste-field width.

Compared to the PLUMES version, the Brooks far-field algorithm has been considerably improved. In addition to having better control over output variables, the algorithm, through the VP time-series capability can now simulate time-dependent behavior. Thus, diel and other cycles can now be simulated. This is very important for estimating the effect of highly variable mechanisms such as bacterial decay, which depends greatly on the variable intensity of ultra-violet radiation.

2 General Introduction to Visual Plumes (VP)

VP has been designed to be similar to standard WINDOWS applications featuring menu bars, dialogue windows, check boxes, tabs, panels, and other components. The more important of these visual features are briefly described below.

2.1 Menus

File Menu (and file naming conventions)

The *File* menu (Figure 2.1) allows the user to open an existing project, add an ambient file to the project, create a new project, save project files under a new project name, import DOS PLUMES VAR files, import UDF (ASCII) files, recall a previous project, print the display, setup a printer, and exit VP. The open, create, and other commands invoke a standard file directory dialogue window. Ambient files may also be added to the Ambient file list by right-clicking in the file list window to reveal the corresponding pop-up menu. The print command prints the graphical image of the selected tab. For complete text output, an output file may be specified on the *Special Settings* tab, or text can be selected and copied from the text tab and pasted into a text editor.

Figure 2.1 File menu

When a new project is created, a *How to Proceed...* window (Figure 2.2) appears to allow the user to specify the intended target model for the application, and, whether or not to retain graphics from the previous project. By clicking successively on each model, the table headers can be seen to change in the background. This reflects the requirements of the different models. Once the *Continue* button is pressed, VP completes the creation process by giving the project a unique default name, which the user can change with the *Save project as* command. The project file, associated with the *Diffuser* tab, is given the extension *vpp.db*. An example of a default project file name is *VP plume 1.vpp.db*, where *VP plume 1* is the default project name.

To complete the project file-creation process, VP automatically creates two additional files.

The additional file appears on the *Ambient* tab, its name consists of the project name followed by a *yyy.db* extension, where yyy is a numerical code like 001, 002, etc., a three-digit number which may be used as a template for additional ambient files identified by different numerical constants (001, 002, etc.)
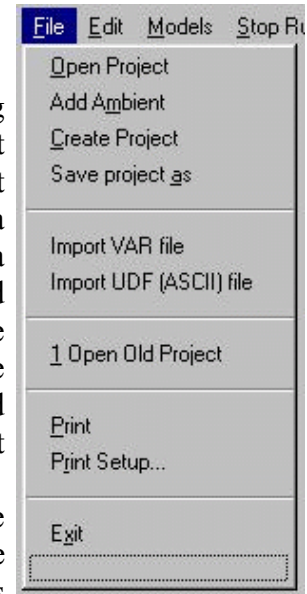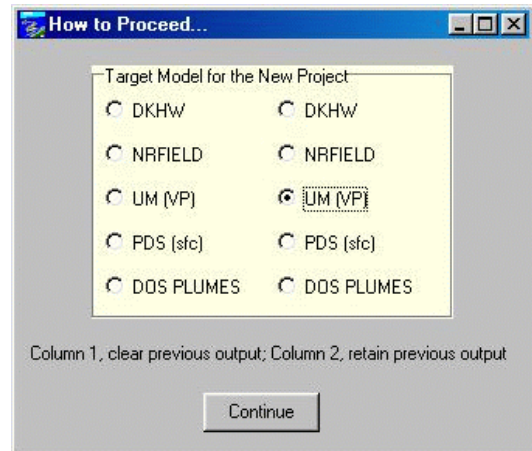

Figure 2.2 Specifying a target model

2.1

The second additional file is an ASCII files that bears the project name followed by the *lst* extension. This file lists project properties for future reference. The list file is read the next time the project is opened, using it to re-establish most of the settings existing in the previous session. The file may be edited in any common word processor although this is not recommended. Manual editing changes must follow the established format or the results may be unpredictable.

Edit Menu

A limited *Edit* menu allows the user to reset headers and substitute backup files to replace unwanted changes made since the beginning of the project session. The latter capability is useful when data corruption is suspected or has occurred.

Models Menu

The *Models* menu allows the user to select and run the desired model. The models are described briefly in Section 1.2 above. Notice that at the menu level most models are accessible by typing the underlined character or hot key. The hot key can be used without clicking the *Models* menu. For example, UM3 can be run with a control-U, (^U) keystroke, DKHW with ^K, etc.. The key sequence is not case sensitive. In addition, on both the *Diffuser* and *Ambient* tabs there is a small blue and yellow icon of a plume. Once a particular model has been selected, a click of the icon runs the identified model. The *Models* menu offers a way to change the original target model.

Stop-Run Menu

The *Stop Run* menu option allows the resident native models, namely UM3 and 3-D, to be terminated during execution. With VP it is easy to set up numerous runs that can take a long time to complete. The *Stop-Run* option will terminate a long chain of runs. The other models run externally and may be interrupted by closing their windows.

Help Menu

The *Help* menu provides a fairly traditional help capability, including *contents*, *search*, and *about* options. Help topics frequently contain green hyperlinks that may be click to cross-reference related topics.

Speed bar buttons

A speedbar provides menu shortcut buttons for some of the more important menu commands. As with other components, when the cursor is moved over the buttons, their function is displayed on the status line at the bottom of the VP window.
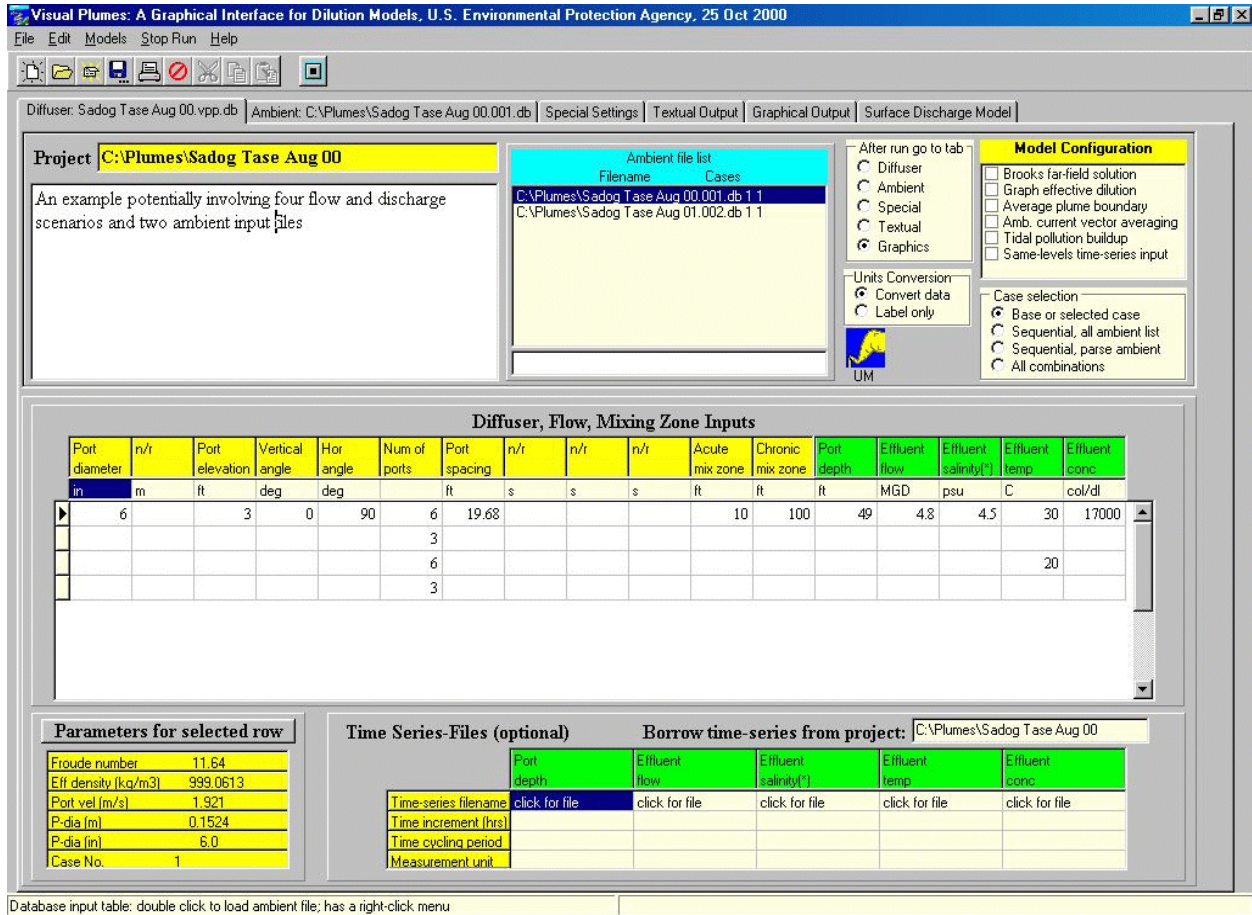
Figure 2.3 The VP interface with the *Diffuser* tab selected.

2.2   Tabs

Prominent VP features are tabs located below the speed-bar buttons that are stacked over each other so that only one is visible at a time. They are the *Diffuser*, *Ambient*, *Special Settings*, *Text Output*, and *Graphical Output* tabs. Clicking on any tab places that tab page on top. The *Diffuser* and *Ambient* tabs append the names of the open files to the tab name. Figure 2.3 shows the interface when the *Diffuser* tab is clicked. This is the default tab when VP is started.

The tabs' components are generally formatted by color. White background means that the component can be modified by directly typing, or entering, data and information into the given cells. A beige background means that the component's contents are manipulated by clicking on it. In some cases, the beige components have pop-up menus activated by right-clicking on the component. The contents on the window of each tab are discussed below.

2.2.1 Diffuser tab

The *Diffuser* tab displays the name of the current project in the yellow box above the *Project* memo box. There are several panels on the *Diffuser* tab that contain various components. From the top left to the bottom right these include:

Figure 2.4 Ambient file list pop-up menu.

1. *Project memo box*: This yellow box displays the name of the current project. Below it is a memo box that may be used to define a project by name and to write specific notes about the project. This is a limited-capability editing window. Scrolling is supported but space is limited and text after the fifth blank line is ignored.

2. *Ambient file list*: This list box displays the name of ambient files that have been previously created and selected for use with the project. In addition to the *add-ambient* command available on the file menu, a right-click pop-up menu (Figure 2.4) provides facilities for editing the file list. For more information, see the description at the end of this subsection.

Figure 2.5 Units conversion pop-up menu

3. *After run go to tab* radio group: This group specifies a favorite tab to move to after a model has been run.

4. *Units conversion* radio group: Occasionally one may wish to change the units label without also converting the values in the column. This option allows the units conversion pop-up menus (Figure 2.5) to be used to change only the unit label.

5. *Model configuration* checklist: This is a checklist for changing fundamentally the way input is read, models are set up to run, output is presented, and other options. For example, if *Average plume boundary* is checked, VP graphs an internal boundary at which the plume dilution equals the average plume-element dilution.

6. *Case selection* radio button group: This group of push buttons allows the user to set the run mode. For example, individual or multiple cases can be selected for subsequent execution.

Figure 2.6 Diffuser table pop-up menu.

7. *Diffuser, Flow, Mixing Zone Inputs* panel: The heart of this panel is the diffuser table that provides space for inputting diffuser specifications and flow conditions. Cells are provided for inputting data. A right-click on the diffuser table reveals the pop-up menu shown in Figure 2.6, which provides additional facilities for editing the table.
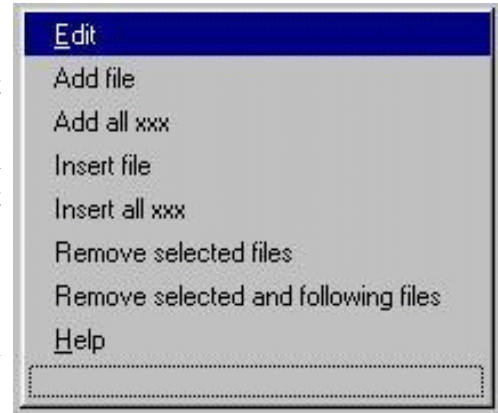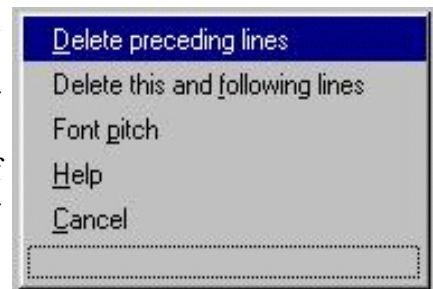
8.	Unit conversion pop-up menus: These pop-up menus allow one to change the units or labels associated with the cell. A click on the unit label reveals the pop-up menus. An example of a menu is given in Figure 2.5.

9.	*Parameters for selected row* panel: The button by that name, when clicked, displays additional information about a specified selected case. The selected case is identified by the ‣ character in the left hand margin of the diffuser table. The case is changed by simply clicking on the row.

10.	Finally, the optional *Time series files* panel and table provides linkage information and values which, when time-series files have been prepared, is linked by clicking on the top cells in the grid. This grid is accompanied by a *Borrow time-series from project* edit box which allows other projects to be specified from which to input time-series data.

The diffuser input table

This is the large input table in the middle of the tab designed to display diffuser and effluent flow data. A convenient property of the table is the way it corresponds to desired information. Except for the first row, or base case, data need only be entered when a particular diffuser or flow characteristic is to be changed for that row, or case. Blank cells either indicate that the column is not required (labeled *n/r*) or the value is the same as the one in the previous row. This implies that all required columns in the first row, or base case, must be specified.

These general rules are modified to accommodate time-series file data, explained in more detail in Section 5.

Multiple Ambient Files

Ambient files associated with a project are shown in the *Ambient file* lists, one on the diffuser tab and the other on the ambient tab. Both lists can be clicked to change the ambient file displayed on the ambient file tab and table. However, the master list is on the diffuser tab; only it can be manipulated to add, insert, or remove files from the list.

Additional ambient files can be added to the master list by right clicking on the list and selecting the *Add-file* option from the pop-up menu (see Figure 2.4) and selecting the name of the desired file from the dialogue window. Again, to display a particular ambient file, or table, simply click on a particular ambient file name on the list and the identified file is put on the *Diffuser* and *Ambient* tabs. At the same time, the file name appears in the edit box below the list where it can be edited.

The purpose of the edit box is to make it possible to append case numbers that specify the range of cases to which the ambient file is to apply. For example, if there are five different cases and two ambient tables named ambient_file.001.db and ambient_file.002.db, and one wishes the first file to be used for Cases 1 through 4 and the second for Case 5, the ambient table names can be modified in the edit box to show ambient_file.001.db 1 4, and ambient_file.002.db 5 5. The numbers must be in ascending order.

After the ambient list has been edited to include the case ranges, there is another way to place the corresponding files on the *Diffuser* and *Ambient* tabs. To see which ambient file is associated

with a specific row or case in the table, double click on a row and the corresponding ambient file is shown on the *Diffuser* and *Ambient* tabs.

Case selection

In the *Case selection* box, the *Base or selected case* option refers to running a specific case, or row, of diffuser data. A click on a cell in the diffuser table moves the triangular arrow (▸) to the left of the table to the specified row and gives that case has focus. If it is the first row, then the base case has focus. When the required columns are filled in, at least in the first row, a model consistent with the required columns is ready to be run. (The exception to cells requiring input is when one or more of the flow variables is linked to a time-series file, in that case the column should be blank because the variable is read from a file at run time; this is discussed further in Section 5.) Lines below the base case (first row) need only contain values that are different from the base case.

When one of the *Case selection* sequential buttons is selected, VP runs the input table from the top to the bottom of the diffuser table, either systematically running through the ambient-file list or parsing the row numbers appended to the filenames in the list. If parsing is specified but no numbers are appended to the file names, VP runs all cases with only the file specified. When the *All combinations* option is selected, VP mixes variables in all possible combinations and runs them creating a complete matrix of input conditions. This includes all ambient tables associated with the project. (Note: the *All combinations* option may result in a very large number of runs, the number depending upon the number of cases, or rows, specified and the number of variables, or columns, containing more than one value).

Units conversion

The *Units conversion* radio group options sets one of two modes. In the default *Convert data* mode VP will automatically convert a column of values from one set of units to another whenever a units pop-up menu (Figure 2.5) is used to change units. For example, a temperature of 32 F becomes 0 C if the initial unit was degrees Fahrenheit and its value was 32 and the Celsius unit is selected from the pop-up menu. The column unit header is changed accordingly. Try this.

In the *Label only* mode, only the label is changed. This is convenient when one inadvertently entered data with a set of units in mind that differ from the indicated set of units. This capability is also useful when information on units is lost after a power outage. However, usually VP is able to correctly reestablish the units after a crash as it stores information on units not only in the diffuser and ambient *db* files but also in the project list file. Sometimes, depending on the situation, restoring the backup files may be the best way to reestablish a pre-existing file configuration.

Model configuration

The *Model configuration* checklist has several check boxes for configuring VP. A check indicates that the option is operational. The checklist is used to fundamentally change the way VP and the models, particularly UM3, perform at run time. For example, if the *Brooks far field solution* selection is checked, VP will add the far-field algorithm solution to the initial dilution prediction.

Only on the *diln* (dilution) graphic panel, the *Graph effective dilution* selection will substitute an alternative dilution measure, <u>the ratio of the effluent concentration to the plume concentration</u>, for the mass (or, approximately, the volume) dilution. The effective dilution differs from the mass dilution when the ambient receiving water contains quantities of the pollutant in question, i.e., background pollution. The entrainment of polluted ambient fluid reduces the apparent dilution because the ambient entrained fluid effectively adds to the plume's pollutant burden.

The *Average plume boundary* selection graphs an internal plume boundary corresponding to the average dilution. The value of this capability is that the concentration at this boundary corresponds to the average dilution output by UM3 and DKHW, both of which adopt the 3/2 power profile to describe the profile of concentration across the plume cross-section. This is helpful for determining mixing zone boundary concentrations where its specification is critical. For example, if the plume approaches the mixing zone boundary at an angle, the criterion concentration may not be exceeded until this internal boundary crosses the mixing zone.

The *Amb current vector averaging* selection specifies how the ambient table, specifically, the current variables, are interpreted. When checked, interpolation at a given depth is based on vector averaging, otherwise, directions are averaged. For example, suppose the plume element is halfway between two depths at which the corresponding directions are northward (0 deg) and southward (180 deg) and at which the current speed has equal magnitudes. Vector averaging will result in a zero current while angular averaging will result in a current directed in the eastward direction. In an estuarine or riverine settings, vector averaging is recommended.

UM3 only: the *Tidal pollution buildup* selection invokes VP's pollution buildup capability. This capability is intended to be used in estuarine discharge situations where the effect of tides is to cause the receiving water to pass repeatedly over the discharge point. When checked, every time a parcel of receiving water passes over the discharge point, effluent is added to it. If the freshwater flow is small, this process can lead to the repeated loading of a particular parcel of receiving water, before it finally passes downstream permanently, out of the influence of the discharge. Combined with the *Amb current vector averaging* selection, it allows UM3 to estimate (in a conservative fashion) the background concentration of the receiving water, assuming that the source is responsible for the presence of pollution in the receiving water.

<u>Useful parameters and conversions</u>

At the lower left corner of the diffuser table is the *Parameters for selected row panel* that gives the densimetric Froude number and other project parameters. In addition to the densimetric Froude number, the effluent density, and the port discharge velocity, it gives the value for the

selected cell, one in the primary unit (like meters) and the other in the selected unit (like feet). The last value is the case number for the specified row. The top three are only updated when the button is pressed. The value of this panel is limited by the need to update variables which can only be done by going from column to column, not from row to row. For more information, see Section 4.1 and Section 6.1.

Time-series input

Near the bottom right of the *Diffuser* tab is the *Time Series File (optional)* panel. When there is no time-series data, or it is not an issue, this panel can be ignored. For time-series analysis, however, this is the panel where the appropriate time-series files are linked to VP. These files must have been previously created and have the correct file name. The creation of these files is discussed in Section 5. Files other than those named using the default names can be specified by using the *Borrow time series from project* dialog box.

For example, assuming that the time-series data on effluent flow has been stored in a file with the correct file name, a click on the *click for file* cell causes the corresponding file to be linked to VP. This action causes the words *click for file* to be replaced by the file name, followed by time increment, cycle period, and measurement units in the cells below it. These values are read from the first line of the time-series file. Any value that was in the effluent flow cell of the diffuser table is replaced by a blank. The reason for this is that VP is set to take flow data from the time-series file and not from the diffuser table, which represents steady state input. The same procedure holds for other time-series variables for which there are files.

The time-series borrow edit box specifies an alternative project that is checked to establish a file if a time-series filename made up of the project name and the appropriate extension is not found.

2.2.2   Ambient tab

The input table of the *Ambient* tab is similar to the input table the *Diffuser* tab. Figure 2.7 is a sample of the *Ambient* tab window. Components include:
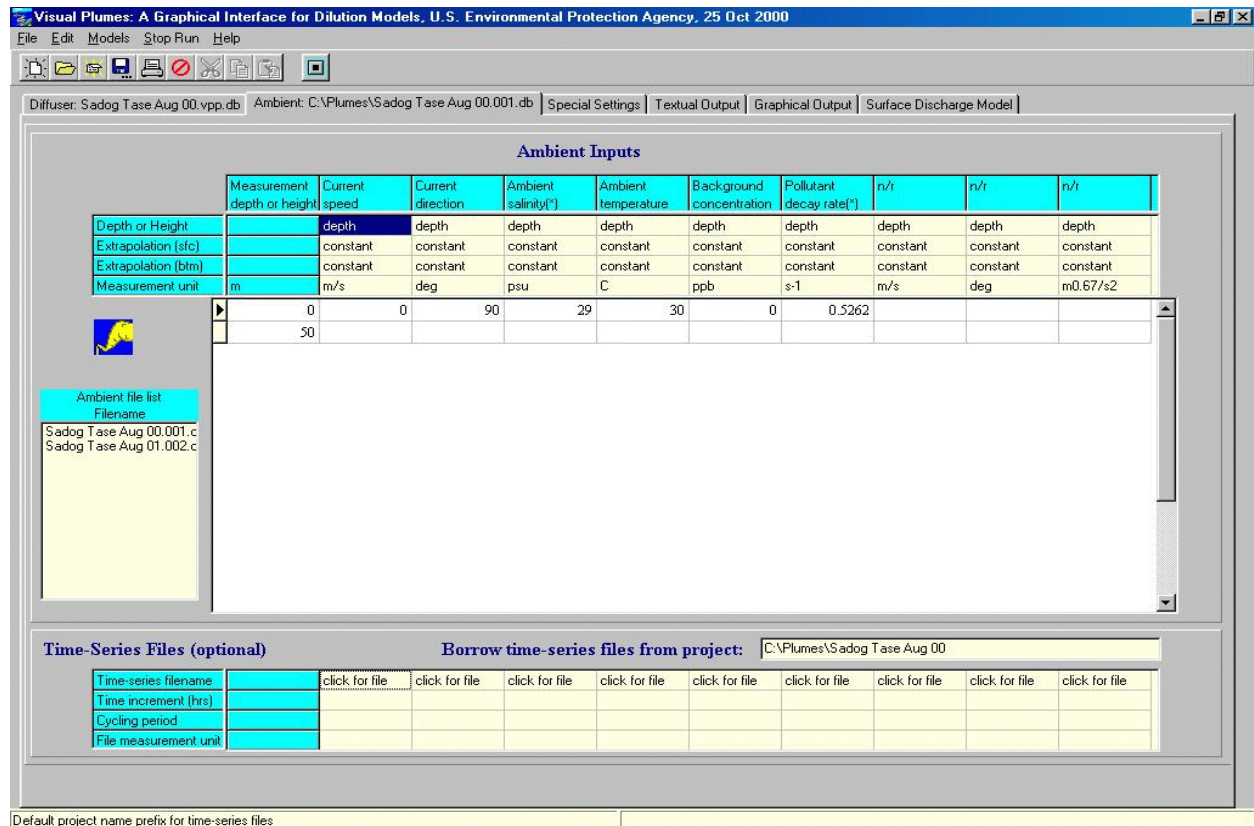
Figure 2.7 The ambient tab

1.   The *Ambient-Inputs* panel includes the ambient input table, for entering ambient data at various depths (or heights). Above the data table is an expanded selection table (or header array) for specifying depth or height mode, interpolation options, and units.

2.   An *Ambient file list* that lists existing ambient files that were added previously; like the list box on the *Diffuser* tab, a click on a filename puts the corresponding data on the ambient tab. Unlike the one on the diffuser tab, this list cannot be used to add or delete files.

3.   The *Time-series files* panel for linking time-series ambient data files to VP, serving the same function as the corresponding panel on the *Diffuser* tab (see Time-series input, above).

<u>The ambient input table</u>

This is the large input table in the middle of the tab designed to display ambient water column data. A convenient property of the table is the way it corresponds to available data, that is, data need only be entered at depths at which it is available. Blank cells indicate that data are not available at those particular depths or are not required (*n/r* columns). The first line need not correspond to the water surface, nor must the last line equal or extend below the port depth. The data correspond to actual depths at which measurements were made or inferred. With the exception of at least one value to define a given ambient property, the cells are best left blank when there is no data. The input values are all actual measured, or, assumed, values. Unlike the diffuser table base case, the first row of the ambient table may have empty cells.

2.9

There are a couple of exceptions, or recommendations. It is recommended that a zero depth row is specified. Also, to force VP to extrapolate to the bottom, a line specifying the bottom depth is recommended, even if there are no other data on that line.

VP has an interpolation and extrapolation capability that allows an uncluttered display of the data. VP's acceptance of blank cells is intended to help give an immediate conception of the coverage and completeness of the actual data.

These general rules are modified to accommodate time-series file data, explained in more detail in Section 5.

The header array

VP consults the header array to determine the proper way to interpolate or extrapolate the data in the ambient data table. For example, if at any time during the simulation the plume element is below the deepest indicated water column depth (input in the first column or *Measurement depth or height* column), VP consults the header array to determine how to extrapolate the ambient data table to provide the correct ambient value at any depth.

The *Depth or Height* row of the header array allows one to toggle between depth or height modes of data input. Depth mode means measurements were taken at distances below the surface. Height mode means measurements were taken at distances above the bottom.

The *Extrapolation (sfc)* row of the header array cycles between the *constant, extrapolated, and linear_to_zero* modes of surface extrapolation. Its setting determines how VP will calculate temperature or other variables between the surface and the first data value. In the constant mode the value of the property is equal to the first cell value in the column. If there is more than one value specified in the column, the values in the surface layer are extrapolated from these.

The *Extrapolation (btm)* row of the header array works similarly for depths between the last cell value in the column and the bottom.

The *Measurement unit* row of the header array functions like the units row on the *Diffuser* tab. A click on it pops up a list of units from which to choose.

These first three specifications are available for all variables in the header array except the *water column*, i.e., the first column. The latter values in the first column are always specified, either as depths or heights. To change the units in this column, one must first double click on the cell to put it in the change mode. These are advanced concepts treated in more detail in Section 5.

Run-time efficiency

For computational economy it is desirable to smooth the ambient data to avoid many small vertical increments in the ambient file. While this is not a requirement, the presence of many vertical depths will cause VP to more frequently define bracketing depths. This is a complicated process, involving the Borland Database Engine, that is necessary to create a filled, internal array of values; running it at every time step can mean a significant difference in performance when many cases are run.

Creating additional ambient tables

Frequently a mixing zone analysis involves several or many ambient scenarios. Unless time-series files are linked to VP, each scenario is expressed in its own ambient table and associated file. When a new project is created the ambient table will be blank, except for an assumed value of 0 in the depth column. As ambient data is stored in a database (direct access) file, the data are stored as soon as they are entered. The file name is determined by tacitly accepting the given filename or giving the file a new name using the *Save ambient file as* command on the *File* menu. In this way many ambient files may be created, each one serving as a template for the next.

To facilitate linking many ambient files to other projects it is useful to use the ambient file naming convention consisting of the appropriate project name, followed by a numeric sequence (001, 002,...), followed by the extension *db*, e.g. *VP ambient.003.db*. When using the *Save ambient file as* command this convention is offered by default and, upon acceptance or change, are added to the ambient file list. Files named in this way may be added to another VP project en masse using the *Add all xxx* command from the pop-up menu shown in Figure 2.4.

Limit on restoring initial ambient table data

CAUTION: As for the diffuser tab, The *Edit* menu has a *Substitute ambient backup file* command for restoring the ambient table to its original state, values that existed when the project was first opened. For ambient files, this only works for the file at the top of the ambient file list. This means, effectively, that unless you make copies of the other ambient input files, any changes you make are permanent.

Time-series ambient input files

The time-series panel on the *Ambient* tab is similar to the one on the *Diffuser* tab. The ambient time-series files must have been created beforehand as discussed below. They are selected by clicking on the *click for file* cell, as described above for the diffuser tab. The time-series-files array is covered in more detail in the section entitled "Time related data, time-series files."

2.2.3 Special Settings tab

The *Special Settings* tab, Figure 2.8, controls text and graphical output format and other functions. There are five panels on the tab:
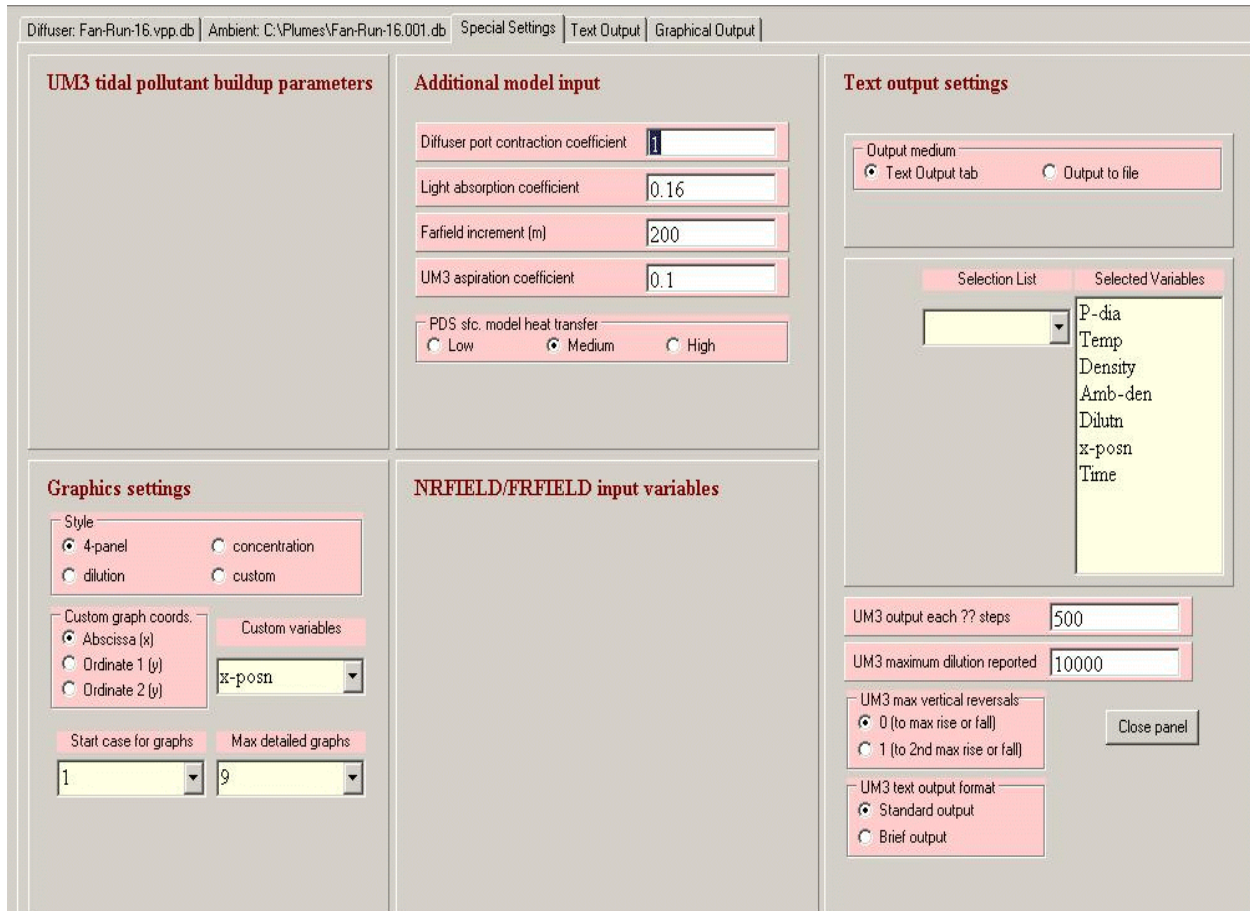
Figure 2.8 *Special Settings* tab with *UM3 tidal pollutant buildup parameters* components hidden (not required because UM3 is not the currently selected model).

UM3 tidal pollutant buildup parameters panel

The components on this panel are revealed when the *Tidal pollution buildup* option is checked on *Model Configuration* panel. An example is shown in Figure 2.9. The edit boxes must be completed to use the tidal pollutant-buildup capability. Briefly, in tidal rivers, this capability makes it possible to estimate the time-varying background pollution concentration that can cause the effective dilution of the outfall to decrease. This option is currently limited to the UM3 model.
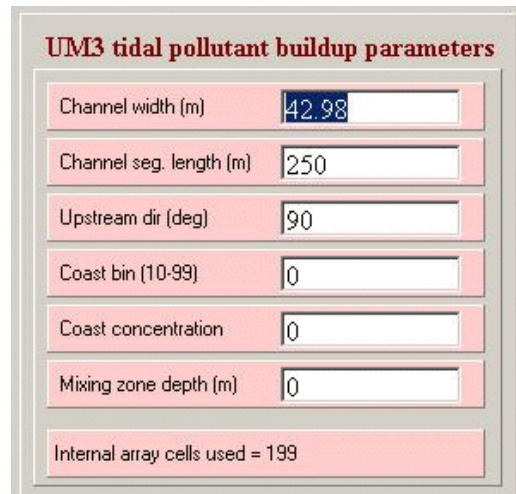
Figure 2.9 Tidal buildup parameters.

Tidal pollutant buildup details

The first two edit boxes on the *UM3 tidal pollutant buildup parameters* panel are labeled *Channel width* and *Channel seg. length*. The product of the channel width and the sum of the port depth and port elevation should represent the cross-sectional area of the channel. The channel segment length, on the other hand, is arbitrarily assigned. Its length, however, determines the convergence of the computed background concentration on an optimally high value. VP provides for a maximum of 200 internal array elements to store the accumulated concentration in cells that previously passed over the discharge point. If the segment length is large, only a few of these cells will be used because the tidal excursion will be small relative to the size of the storage cells. This will result in poor spatial resolution (input effluent will be averaged over large volumes). On the other hand, if the segment length is too small, 200 array elements will not be sufficient to store the information and the array limits will be exceeded. If this happens UM3 may stop running and a larger length will need to be specified. Some experimentation will be necessary to determine a segment length that is near the smallest value allowed. At the end of the run, VP outputs the number of internal array cells used. The object of the optimization procedure is to maximize this value near 200, for example, 199 would be the most optimum value.

The *Upstream dir*ection edit box is used in conjunction with the *Amb current vector averaging* setting (on the *Model configuration* panel) to help VP determine whether inputted directions are net upstream or downstream. Here the value of vector averaging should be apparent. For example, if flows at two depths are upstream and downstream respectively, directional averaging (the default setting when ambient current vector averaging is not checked) would indicate a cross-stream component to the direction!

The *Coast bin (10-99)* edit box identifies the position of the coastal bin along the one-dimensional tidal channel in which the buildup of background pollution is occurring. During the simulation, on the flood, any bin that moves upstream through the coast bin has its concentration set to the value specified in the *Coast concentration* edit box. The discharge is always initially assumed to be located in bin 100. The coastline bin number is determined by equating the downstream volume to the product of the segment volume and the number of bins between the outfall and the coastline. For example, if the total depth of the channel at the point of discharge is 10m, the channel width is 100m, and the segment length is 100m, then each bin has a volume of $100,000m^3$. If the reference volume of the estuary below the point of discharge is $5,000,000m^3$ then the coast bin value would be 50.

The *Coastal concentration* edit box is used to specify the background pollutant concentration in the ocean, bay, or estuary to which the tidal channel connects.

The *Mixing zone depth* edit box identifies a mixing zone "ceiling" at which an output may be specified, i.e., a depth of interest in the water column at which estimates of concentration are desired. When this criterion is reached, UM3 outputs a message to that effect.

Additional model input panel

The *Diffuser port contraction coefficient* edit box is provided to specify the value of the contraction coefficient for the discharge ports. The discharge coefficient of sharp-edged ports (a

cylindrical hole in the diffuser pipe wall) is about 0.61. For bell-shaped ports (flaring inward into the direction of flow), a value of 1.0 is usually used.

The *Light absorption coefficient* edit box is provided for specifying the light absorption coefficient for the Mancini bacteria model. This is the coefficient k_e found in Mancini, 1978, that describes water clarity. The default value is 0.16.

The *Farfield increment* controls the number of lines output by the Brooks far-field algorithm. A small value produces more lines and graphic output than a large value. The value specified depends on the size of the zone of mixing, but, typically might be anywhere from 100 to 1000m.

The *UM3 aspiration coefficient* specifies the rate at which ambient fluid is entrained into the plume. The default value of 0.1 is an average value that, historically speaking, was rarely changed by users of DOS PLUMES. A large value causes more rapid plume spreading and affects other characteristics, like plume rise.

The *PDS sfc. heat transfer* radio group specifies the surface heat transfer rate used when running the PDS surface discharge model. The available options are low, medium, and high corresponding to low wind and high humidity conditions, "average" conditions, and to windy and dry conditions respectively.

Text output settings panel

The *Output medium* radio group is used to specify whether output appears on the text output tab or is written to a file (the *Output to file* option). When the file option is selected, the *Output file* selection list box appears. Clicking on it causes the *Text Output File* dialog window to appear that allows an input file to be selected or input. The default name is *VP.txt*.

The *Selection List* drop-down list may be clicked to add or remove variables from the output variables list shown in the *Selected Variables* list box. A click on a variable name acts like a toggle, adding or removing a variable from the text output tab the next time a model is run. To add or remove a variable, click the arrow on the *Selection List* list box to get the drop-down list and then click on the variable you want included or removed from the output.

The *UM3 output each ?? steps* edit box is used to control the frequency of UM3 output. For example, a value of 100 causes UM3 to display output after 100 program steps, or full cycles. This does not affect output when special criteria are met, like when the trapping level is reached.

The *Maximum dilution reported* edit box serves to limit the maximum number of UM3 program steps. It should be specified to be greater than the maximum dilution of concern, furthermore, it should be large if model is configured to compute a far-field solution. It limits computation when conditions do not inhibit plume behavior, for example, a plume in deep, unstratified water.

The *UM3 max vertical reversals* radio group controls where the UM3 simulation terminates. With the *to max rise or fall* option UM3 terminates upon reaching maximum rise or maximum fall, depending on input conditions. With the *to 2nd max rise or fall* UM3 will run to the next maximum rise or fall. This option is important when a plume still has great potential for rising or falling upon reaching the first extremum. For example, a dense discharge discharged upward has not completed the initial dilution process at maximum rise, as it will reverse its upward motion and accelerate downward

The *UM3 text output format* radio group allows a selection of standard or brief output. Brief output suppresses the writing of headers and input conditions.

The *Close panel* button hides the *text output settings* panel. This button serves to hide the panel when it is made visible after pressing the *Output options* button on the *Text output* tab.

Graphics settings panel

The *Style* radio group simply selects the graphic panel that is displayed whenever one clicks on the graphical output tab. It determines the "default" graphic on the graphics tab. This is basically the same *Style* radio group that appears near the left margin of the graphics tab, except that the labels are spelled out.

The *Custom graph coords.* radio group and the *Custom variables* drop-down list work in tandem to specify the output variables plotted on the *custom* graphic. After one selects *Abscissa (x)*, the drop-down list is used to specify the corresponding variable to be plotted on the x-axis the next time the model is run. The same is then done for the left y-axis by first selecting *Ordinate 1(y)* followed by specifying the corresponding variable from the drop-down list. The procedure is repeated for the right y-axis, first selecting *Ordinate 2(y)* followed by selecting the corresponding variable.

When running multiple cases it is usually desirable to suppress all plotting except the runs of interest. The *Start case for graphs* edit box is used to specify the first case that will be plotted. The *Max detailed graphs* edit box is used to specify how many additional cases are plotted starting with the one specified above. These specifications apply to the 4-panel graph only.

NRFIELD/FRFIELD input variables panel

This panel is unfinished.

2.2.4   Text Output tab

This tab presents model output in text form, an example is shown on Figure 2.10. Every time a model is run, each case, or row, from the *Diffuser* tab is given a case number starting with 1 and running consecutively through the last case. In configurations where only one case is selected (base or selected case) there will be output from only one case, unless time-series files are linked into VP, in which case there may be many. The *Model Configuration* panel and *Special Settings* tab influence the text output significantly.

Several points are noteworthy. The complete set of input values (from the *Diffuser* tab) are only given for the first case, Case 1. Subsequent cases show just the input values that are different from the previous case. When multiple cases are run they are labeled sequentially (in Figure 2.10 two individual runs were run, consequently, they are both Case 1). For time-series runs, VP assigns case numbers as it reads input values from the files and runs the corresponding conditions. It displays the values read from the time-series files in the run header. Depending on the model used, VP flags where plumes merge, are trapped, reach the surface, and reach the chronic and acute mixing zone limits. The output can be saved to a file (specified on the *Special Settings* tab) or printed using the print option on the *File* menu. When subsequent runs are made the case count is

reset to one and text output for these runs is appended to what is already in the text window. If previous output is no longer wanted, the *Clear Display* button may be clicked before making additional runs.
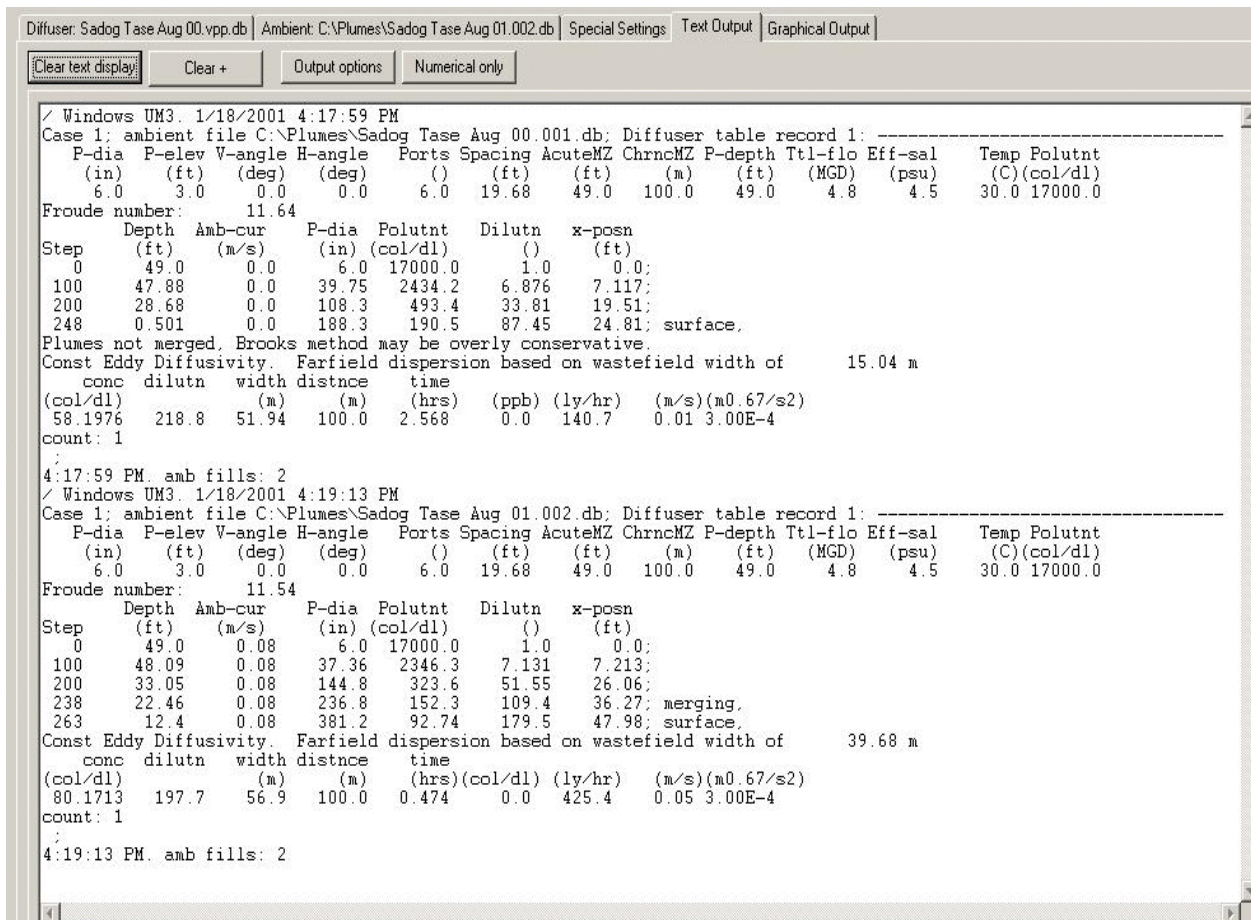
Figure 2.10 *Text Output* tab.

Users familiar with UDKHDEN and PDS standard output will notice that the VP output for these two models is abbreviated. However, the complete output is saved in its original format in the default directory. For DKHW, it is saved as DKHW.out. For PDSWin, it is PDS.out. These files include output for all cases simulated during the run, they are deleted and re-written each time the models are run. If this output is to be saved the output files should be renamed before the models are run again. This is of particular interest to users interested in surface thermal plumes since PDS prints out the area within surface isotherms in the PDS.out file.

2.2.5   Graphical Output tab

This tab displays output in graphic form. A sample of the 4-panel style of graphical output is given in Figure 2.11. The *Style* radio group on the *Graphical Output* or *Special Settings* tab is used to select the style. There are four different styles of graphs:
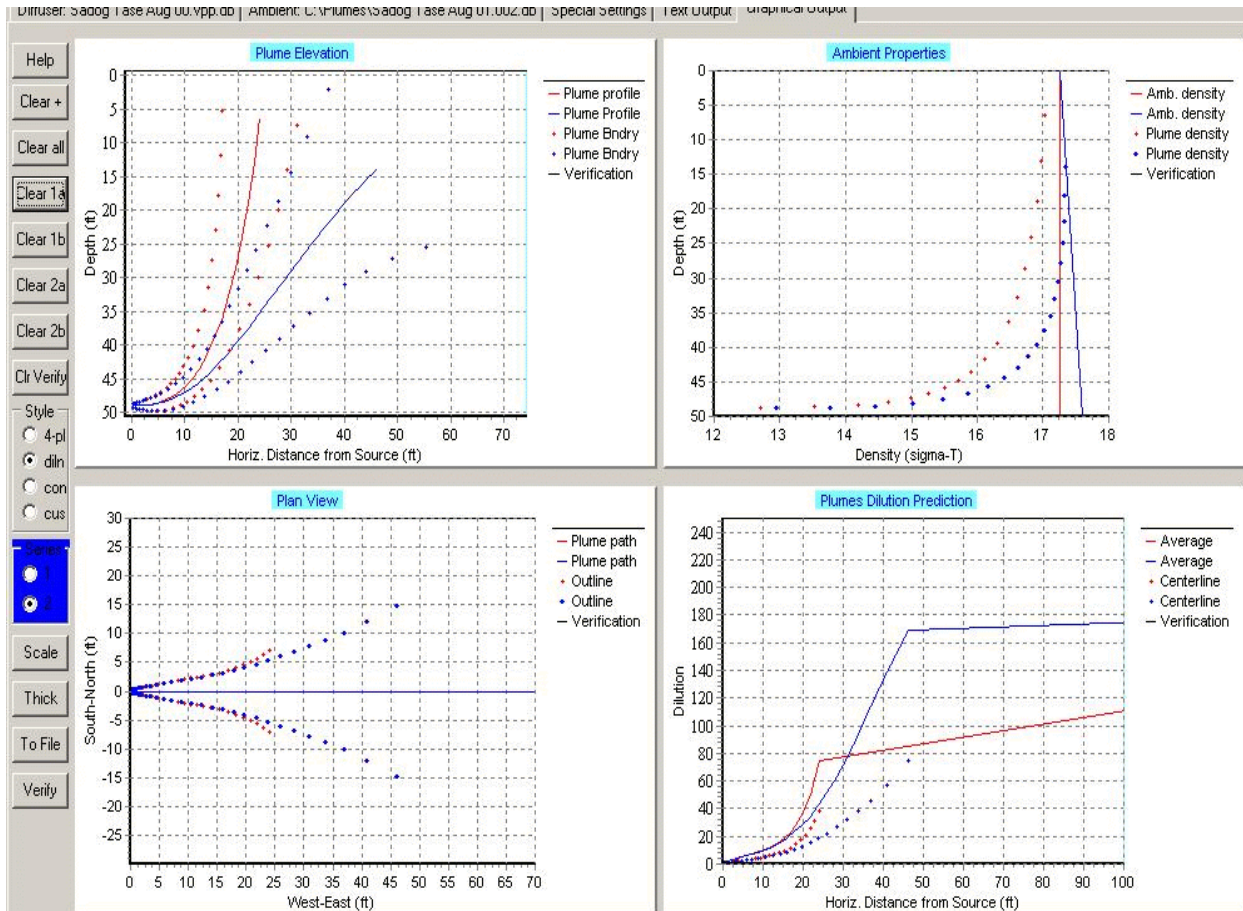
Figure 2.11 *Graphical Output* tab.

4-panel graphics

This graphic depicts plume and ambient properties of the chosen cases. The four graphs include, from left to right and top to bottom: (a) an elevation view looking horizontally through the water column, (b) a plot showing the density of the plume and ambient as a function of depth (this is helpful to observe trapping effects), (c) a plan view showing single plumes in the near field (before merging) and the merged-plume width or waste-field width in the far field, and (d) a plot of centerline and average dilution versus distance from the discharge (radial distance, or, more precisely, distance along the length of the projection of the plume trajectory on a level surface). The solid lines in the plan and elevation views plot the plume centerline, the dots either plot the "absolute" plume-ambient boundary or the average concentration boundary when *Average plume boundary* is checked on the *Model configuration* checklist.

Dilution graphic

2.17

This is a summary graphic of end-point dilution. End-point dilutions correspond to predicted dilutions at specific criteria, notably at maximum rise, upon hitting the surface, and at the mixing zone boundaries. The triangles represent the dilution at the acute and chronic mixing zones. The rectangles represent the dilution at a hydrodynamic criterion, usually maximum rise or plume surfacing, whichever comes first. The mass or volume dilution is plotted unless *Graph effective dilution* is checked on the *Model configuration* checklist and the effective dilution is plotted instead. The former dilution describes the hydrodynamic mixing efficiency of the plume, the latter includes the pollutant added to the plume by entrainment and describes the reduction in concentration achieved within the plume. The two dilutions are equivalent when the background concentration is zero.

Concentration graphic

The concentration graphic is the inverse of the effective dilution graphic. It always depicts pollutant conditions at end points (maximum rise, surface hit, mixing zone boundaries).

Custom graphic

This may be a summary graphic if the x-axis variable is Casecount or a detailed graphic if the x-axis variable is any other variable besides Casecount. To select the custom graphic variables, see the *Graphics settings* panel description in Section 2.2.3.

Series radio group

Color facilitates comparing model predictions. Two graph series may be selected, Series 1 or Series 2, corresponding to red or blue. Depending on the selection, values are plotted in one or the other. The act of specifying a series automatically returns the user to the *Diffuser* tab since this is the most common place from which model runs are initiated.

Verify button

Verification data in x-y format can be superimposed on Visual Plumes predictions on the Graphics tab. Clicking the "Verify" button opens a file dialogue window for locating and opening previously created ASCII input files. An example of file input data excerpted from the Fan-Run-16 project verification file, Fan16.txt, is:

side view
  0.0001 1.0145
  0.0068 1.0157
  0.0149 1.0158

  0.0197 1.0161
  0.0264 1.0159
density profile
 17.3 0.0
 25.2 1.0

Blank lines will cause a space between data (lifted pen). The key words (side, profile, path, dilution, effdilution, concentration, and generic) shift plotting to the corresponding graphic panel, namely, the elevation, density profile, plan view, and dilution on the four-panel graphs, and dilution, concentration, and the generic custom panels. Units should correspond to the ones chosen in Visual Plumes.

Other controls

The remaining buttons give the user the ability to clear the screen, get help, select other graphic images (styles), or to apply VP's automatic scaling function.

The *Clear all* button clears all graphics. The *Clear 1a* and *Clear 1b* buttons clear the centerline graph and plume boundary graph of Series 1 (red) respectively. The corresponding buttons clear Series 2 (blue).

The *Clear +* button clears all graphics and text.

The scaling function (*Scale* button), applied individually to each style, is handy for establishing graphic scales that capture all of the plotted points. It works best on single runs.

The *To File* button creates bitmap files of all seven graphics using the project name followed by a graphic identifier and finally the bmp extension to build the bitmap filenames. The graphic identifiers are traj, prof, path, dil4, dil, con, and gen for the trajectory, water-column profile, path, 4-panel dilution, summary dilution, summary concentration, and custom graphics respectively. The bitmaps can be brought into reports and other applications and do not include the surrounding Windows material.

As they are bitmaps, the graphics files that are created when the *To File* button is clicked are memory intensive. Each file requires about 2Mbytes of memory. Consequently, it is recommended that these files are handled promptly, either deleted or saved on another medium. Programs such as Microsoft Paint may be used to convert these files to more condensed formats.

The bitmap files of the 4-panel graphics are re-scaled by VP before they are saved. This procedure improves the quality of the 4-panel graphics.

Customizing graphics

Other means are available to customize the appearance of the graphs. For example, the position of the graph may be altered within a particular window (panning) by making a right click

with the mouse in that window, holding down the button and dragging the mouse. One can also left double click on the margins of a graph to display a window that contains various formatting components and options. These options include changing the titles, setting the coordinate limits, setting the background color, etc..

Zoom options are also available. A left click on the mouse inside the graph followed by holding down the button and dragging the mouse creates a fence or rubber band that establishes new zoom limits. Dragging from lower right to upper left returns the settings to the original values. As (some of) the settings are maintained by VP in the project list file, it is a good idea to return to the preferred appearance before exiting VP.

As with other components, a right click on a graph brings up a help screen.

Printing graphics

Graphs can be printed on a graphics printer by selecting *Print* from the *File* menu. They can be copied by using the Alt-Print-Screen key sequence and pasted into a graphics package or word processor of choice. A control-V, (^V), key sequence is the quick way to paste. Of course, the graphics bitmap files (see the section on other-controls above) can also be processed and printed.

2.3   Model-specific panels and components

Hidden components

The user should be aware that some of VP's components are configuration dependent, which means that they are not always visible. For example, the components on the *UM3 tidal pollutant buildup parameters* panel on the *Special Settings* tab are not visible unless the *Tidal pollution buildup* option is checked on the *Model Configuration* checklist on the diffuser tab. The same is true for the *Output file* edit box on the *Text output settings* panel on the *Special Settings* tab, which is only visible when the *Output to file* radio button is selected.

Changing labels

The labels on the diffuser and ambient tables are similar, changing depending on the target model which is identified is identified under the model logo on the diffuser tab (PDS, UM3, etc.).

Other components change depending on context. Noteworthy is the save-file-as command on the *File* menu, which changes from *Save project as* to *Save ambient file as* depending on whether the command is issued from the diffuser or ambient tabs respectively.

Displaying the Froude number and other parameters

Some components require user intervention before they are completed. For example, the densimetric Foude number for a particular case is not displayed correctly until the case is established by clicking on the appropriate line in the diffuser table and until pressing the *Parameters for selected row* button that also serves as a label for the parameters panel.

2.20

3   Entering Data

When creating a project for the first time the *How to proceed* window appears to allow a target model to be specified. This is done so VP can identify the variables, or columns, required by the target model. If one clicks on different models in succession, the column labels can be seen to change as some of the columns go from showing a label, like *Time increment*, to *n/r* for not required. Once a model is accepted, the required columns can still change. For example, entering a value greater than one for the number of ports will change the port spacing column label from n/r to *Port spacing*. The model configuration options can have a similar effect. Notice that the column labels may change again if subsequently models other than the initial target model are run.

General data input

General Windows editing conventions apply to VP. This includes selecting and editing text and numeric values. In the numeric tables some database program conventions apply as well. Fine editing can be achieved by first selecting a cell and then clicking a second time to highlight only the numeric portion. The cursor can then be used move about the value to edit it.

Experimentation is encouraged. With the cursor in a table row, the Ctl-Delete key sequence will delete a full row of data. The Insert key will insert an empty row. Cutting and pasting in the input tables is not presently supported.

3.1   Diffuser Tab

General considerations

In VP input focuses first on the diffuser and flow table (or, simply, the diffuser table) on the *Diffuser, Flow, Mixing Zone Inputs* panel. However, it is useful make notes in the *Project* memo box and it is generally important to check the proper selections on the *Model Configuration* checklist, the *Case selection* radio group, and other settings on the diffuser tab.

Once a model is specified it is recommended that the units are selected. The default units are mainly SI (generally MKS) units. They may be changed by clicking on the unit and selecting the desired unit from the pop-up list of available units.

VP cannot run any model until all required values for the base case, consisting of one completed row (the top one) in the diffuser table, are provided. If all required values are not defined, an error message will appear. The mouse and arrow and tab keys may be used to navigate the diffuser table. Values may be typed in immediately upon entering a cell, if a value already exists it will be replaced by the new one. See the tutorials for more editing instructions.

Additional runs (on subsequent rows) inherit all the base case information except values that are specifically changed in subsequent rows (except that the sequence is different when the *All*

combinations option is selected). Pressing the down-arrow key when the cursor is located in the bottom row of the diffuser table creates a new row.

3.1.1 Diffuser table variables

The variables listed in the diffuser and flow table are as follows:

1. *Port diameter*: this is the port discharge diameter (abbreviated P. dia. on the *Text Output* tab). If PDS is the target model it is labeled *Conduit width*. On the *Text Output* tab, *P. dia.* should be interpreted to refer to the plume diameter, which is equal to the port diameter at the origin.
2. *Conduit depth*: this variable is required by PDS which assumes a rectangular surface discharge. The column is not required by the other models.
3. *Port elevation*: this is the vertical distance between the port and the bottom of the water body. This variable does not have a major effect on model prediction and can sometimes be changed to avoid related output messages.
4. *Vertical angle*: the discharge angle relative to the horizontal with zero being horizontal, 90 being vertically upward, and -90 being vertically downward.
5. *Horizontal angle*: the horizontal port discharge angle relative to the x-coordinate. Assuming that the default units (deg) are used, zero is in the direction of the x-coordinate (flow towards the east) and 90 in the direction of the y-coordinate (flow towards the north). This default coordinate system is essentially the scientific convention in which eastward corresponds to the x-coordinate direction and angles increase counter-clockwise. An optional unit is labeled *N deg* and conforms to the surveying standard in which north is zero and angles increase clockwise. ***For PDS the horizontal angle is the discharge angle relative to the x-coordinate.***
6. *Num of ports*: the total number of equally spaced ports on the diffuser. If there are multiple ports, they are all assumed to be on one side of the diffuser. Modeling opposing directions generally requires at least two separate runs. For T-shaped risers with two ports on each riser, a conservative approximation is to assume all the ports are on one side of the diffuser with the spacing equal to half the spacing between risers. (When running NRFIELD, which is based on T-riser experiment, VP makes the necessary adjustments to this convention.)
7. *Port spacing*: the space between ports. This is assumed to be the same for all ports. This variable is not required unless the number of ports is two or more. NRFIELD requires at least four ports. See Item 6, above.
8. *Start time*: the starting time for a time-series run. (If there are linked-in time-series files, time = 0 is assumed to be the common origin for all files and runs.)
9. *Ending time*: the ending time for a time-series run.
10. *Time increment*: the time increment for a time-series run.
11. *Acute mix zone*: the distance to the acute mixing zone limit or CMC. Output is flagged when this distance is reached. When the *Brooks far-field solution* option is checked, the output intermediate output distances are specified on the *Special Settings* tab.
12. *Chronic mix zone*: the distance to the chronic mixing zone limit or CCC. Output is flagged when this distance is reached or when the plume reaches the surface. For PDS this label is

changed to *Region of interest*, and represents the distance along the plume centerline where simulations are to stop.

13. *Port depth*: the distance from the surface to the port centerline. (This variable is optionally a time-series variable.)

14. *Effluent flow*: the total volumetric flow from all ports. (This variable is optionally a time-series variable.)

15. *Effluent salinity (density)*: either the effluent salinity or density depending on the units selected. Note that the column label changes when units are changed from salinity to density. Also, when converting from salinity to density units, or from density to salinity, the temperature must be defined. (This variable is optionally a time-series variable.)

16. *Effluent temperature*: the effluent temperature. If salinity is selected in 15, above, VP calculates the effluent density from the salinity and temperature. (This variable is optionally a time-series variable.)

17. *Effluent conc*: the concentration of the pollutant of concern, or tracer, in the effluent. (This variable is optionally a time-series variable.)

3.2   Ambient Tab

Concerning data input to the ambient table, it is emphasized again that only the first column in this table must contain values in each indicated row, which must not number less than two. All the other required columns must contain at least one value (to define a value for that variable), however, it is recommended that these values only be specified at the depths at which they were measured. Cells in the column corresponding to depths at which the given variable was not measured should be left blank. If no measurements are available the data may be synthesized. In that case, if the variable is to be held constant, only a single value need be input in the column, which can be in any row with a specified depth.

All variables on the ambient table with the exception of the first column are optionally time-series variables.

3.2.1   Ambient table variables

The variables listed in the ambient table are as follows:

1. *Measurement depth or height*: This column holds the data at which water column measurements are available, generally depth below the surface but they can also be heights above the port. If data are measured at only one depth, a second depth must still be specified. If data are synthesized, i.e., data are assumed or inferred but not actually measured, at least two depths must be specified in the column, even though the other columns may contain only one value. A depth specifying the bottom will assure that ambient density is plotted to that depth on the density stratification profile graphic and that extrapolation, if ordered, is performed. An arbitrary number of rows may be specified but VP runs more slowly the more depths are used and more than 20 depths are not recommended due to the size of the internal ambient array.

2. *Current speed*: the speed of the current at the specified depth.
3. *Current direction:* the direction of the current at the specified depth. For angle conventions see the discussion of the horizontal angle in the *Diffuser table variables* section above.
4. *Ambient salinity (density)*: either the effluent salinity or density depending on the units selected. Note that the column label changes when units are changed from salinity to density. Also, when converting from salinity to density units, or from density to salinity, the temperature must be defined.
5. *Ambient temperature*: the ambient temperature. If salinity is selected in 4, above, VP calculates the effluent density from the salinity and temperature.
6. *Background concentration*: the concentration of the pollutant of concern, or tracer, in the ambient water column at depth.
7. *Pollutant decay rate (solar radn)*: Like salinity and density, this is a multi-purpose column. Decay rate is indicated on the header when the top three units (per sec, s^{-1}; per day, d^{-1}; or T90hr) are selected. As an alternative, a selection of the ly/hr (langleys/hr) unit invokes the Mancini model (1978) for calculating decay rate and changes the header to *Pollutant solar radn*. The model is a four stressor model including salinity, temperature, water column light absorption, and solar radiation.
8. *Far-field current speed*: the average speed of the current over the time period required for the plume element to travel from source to receptor site. If the receptor site is nearby, the value of this variable can be often assumed to be the same as *Current speed*, **however, it should not be zero as this will imply infinite travel time and cause an error**.
9. *Far-field current direction:* the direction of the current over the time period required for the plume element to travel from source to receptor site. For angle conventions see the discussion of the horizontal angle in the *Diffuser table variables* section above.
10. *Far-field diffusion coeff*: This is the diffusion coefficient used by the Brooks far-field algorithm. A generally conservative default value is considered to be $0.0003 m^{2/3}s^2$. Two solutions are offered, the 4/3 Power Law and the Constant Eddy Diffusion algorithm, the former for open water and a more restrictive one for inland channelized waters. The 4/3 Power Law solution is output when the *4/3Eddy* variable is placed on the *Selected Variables* list on the *Special Settings* tab, otherwise the more conservative solution is output. The coefficient affects only horizontal dispersion, vertical dispersion is assumed to be much smaller.

3.2.2   Creating Ambient tables

For creating additional ambient tables refer to the sub-section titled *Creating additional ambient tables* in Section 2.2.2.

When multiple ambient files are linked into a project, as is apparent by the presence of multiple filenames in the ambient file list maintained on the diffuser and ambient tabs, it is useful to remember how to access the other files. Clicking on the desired file name in one of the lists closes the current ambient file and opens the newly selected one. When the *Base or selected case* radio button is pressed, the model will run that case with the ambient conditions contained in the current ambient database file.

3.3 Database Files

Data in the diffuser and ambient tables are stored in database binary format. The files, bearing the *db* extension, are compatible with Paradox and other database applications, in which the archetypal files were set up. If they are manipulated outside of VP, one should be aware that each closed db file is headed by several rows of data that represent units and other information. If these are changed inadvertently or are lost in a computer crash there generally will not be loss of data. Information on units is also stored in the project list file (bearing the *lst* extension) that VP maintains upon exiting a project, so that, even after a crash, VP can usually re-establish the correct database table information. The signs for this problem are usually clear, an abnormal termination and a re-establishment of the default units, however, bugs may exist that would obscure the problem. If VP infers the wrong units the *Label only* radio button may be selected to correct unit labels without affecting the values in the column. Once completed, the *Convert data* radio button should be re-selected. The *Reset diffuser headers* and *Reset ambient headers* comments are available on the *Edit* menu for the rare times that data corruption has occurred and diffuser and ambient labels can be re-established.

The same comments apply to the ambient table.

3.4 Files and Filename Conventions

Project files

A Visual Plumes project consists of a single diffuser database file (a binary file with the *db* extension), one or more ambient database files (also with the db extension), and, a "list" file bearing a *lst* extension. If a list file for a diffuser project file is not found, VP creates one. The list file stores information on the way the project was last modified, for example, many customized graphics settings are stored in the list file.

Optionally, a VP project may have one or more time-series files associated with it. These files have variable extensions associated with them than indicate the kind of data that they store, for example, file TreamentPlant.flo would be a proper name for a time-series file containing flow data (see the section entitled "Example time-series filenames" below).

VP also creates a file called *VPsetup*. This file is used to store the name and path of the last project accessed by the user.

Filename conventions

The diffuser database filename consists of project name followed by the extension .vpp.db. Given the filename *VP plume 1.vpp.db*, *VP plume 1* is the project name, *vpp* is an abbreviation for Visual Plumes Project, and *db* is a standard extension for a database file.

In completing the project file-creation process, VP automatically creates two additional files. The first appears on the *Ambient* tab consisting of the project name followed *yyy.db*, where yyy is a unique numerical code like 001, 002, etc.. This file may be used as a template for additional ambient files identified by different numerical constants.

The numerals may be used by VP to place a series of ambient files on the *Ambient list* tables. By right-clicking on the ambient file list, VP is able to automatically attach ambient files using yyy as a counter. The same project name is generally recommended for both ambient and diffuser time-series. However, sometimes it is handy to reuse files from other projects and VP does allow ambient files from other projects to be listed.

The time-series filename suffix (or extension) is unique to each variable. Given that the numeric extension yyy is *001*, project time series files might have might be named as follows:

Example time-series filenames

Project name.dep	Port depth
Project name.flo	Effluent flow
Project name.sal	Effluent salinity
Project name.tem	Effluent temperature
Project name.pol	Effluent pollution concentration
Project name.001.spd	Current speed at the diffuser
Project name.001.dir	Current direction
Project name.001.sal	Ambient salinity
Project name.001.tem	Ambient temperature
Project name.001.pol	Background (ambient pollutant) concentration
Project name.001.rat	Pollutant decay rate (or solar radiation)
Project name.001.far	Far-field current speed
Project name.001.fad	Far-field current direction
Project name.001.dis	Far-field diffusion coefficient

4   Introductory Tutorial

4.1   Important mixing zone modeling terms and concepts

Outfalls and diffusers, plumes and jets

Waste water is discharged through a myriad of outfall structures that can differ in many respects. However, all waste streams that are discharged from outfalls share this basic property: in operation all form a single or multiple streams that can be distinguished from the ambient receiving water by some physical properties, notably by a velocity that differs from the ambient current and by the concentration of one or several properties, except in the degenerative case in which outfall velocity and current velocity and concentration of properties are perfectly matched.

Schematic views of outfall and diffusers

Figure 10a.
The outfall tunnel is bored through bedrock, and the diffuser section of the outfall is located along the final 1.25 miles of the structure.
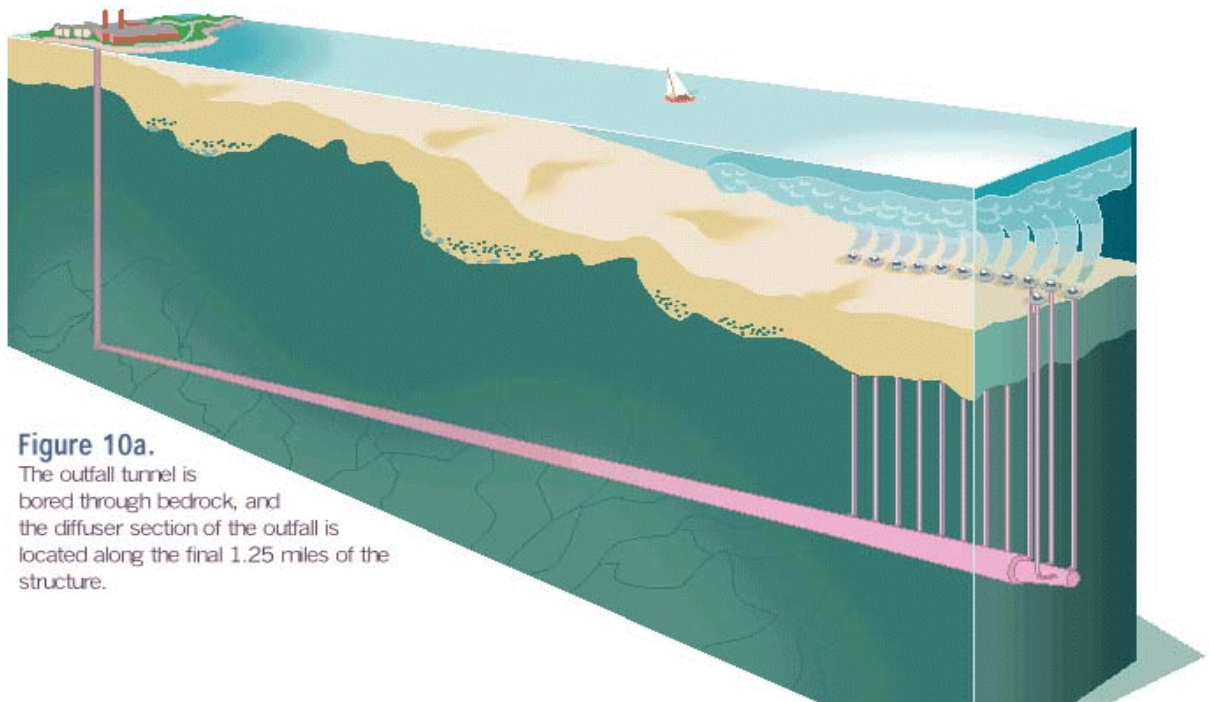
 Figure 4.1 A schematic of the Boston outfall tunnel beneath Massachusetts Bay. (Massachusetts Water Resources Authority [MWRA], 2000)

A general appreciation of an outfall can be obtained from schematic drawings of the recently completed Boston outfall. This outfall consists of a 15km tunnel bored through bedrock (Figure 4.1) designed to carry treated waste water ranging up to $44m^3s^{-1}$ (1000MGD) from land-based collection and treatment facilities to a distant location in Massachusetts Bay. The outfall tunnel terminates in a diffuser section consisting of over 50 risers, or vertical shafts (Figure 4.2). Finally, each vertical

shaft is capped by a turret that has eight discharge ports spaced evenly around its circumference (Fig. 4.3). When operating as intended, the fluid discharging from any given port forms a jet or plume that mixes with the ambient water, yielding the dilution that the outfall is designed to facilitate.

The building block of a mixing zone analysis

A plume from one of the single ports in Figure 4.3 may then be seen to be a representative example of an effluent discharge. While the port is round in cross section, other discharge cross sections could be rectangular, like the cooling canal of a power plant, or ragged, like a natural stream discharging to a larger water body. In fact, if the densimetric Froude number of one of the plumes in Figure 4.3 were less than unity (< 1.0), then the fluid would be so buoyant that it would rise to the top of the discharge port and have an elliptical cross section. In this case ambient water may actually flow into the outfall under the plume, causing an effect in sea water known as saltwater intrusion, which can cause various problems in outfalls. Rubber flex valves that open up with increased flow have been developed to increase the Froude number by reducing the port cross-sectional area in low flow situations. They also prevent the backflow of saltwater by acting as a check-valve.

A vertical discharge with Froude number less than one may briefly decrease in radius. In weak current the plume would have an hour glass shape.

As can be seen in Figure 4.1, multiple plumes are subject to complex behavior; they interact, or merge. Still, a single port (stream, conduit, seep, etc.) and its plume are the building blocks of any mixing zone analysis.

Entrainment and dilution

The process of mixing ambient fluid into the plume is called entrainment. The dilution process and entrainment are essentially synonymous. An understanding
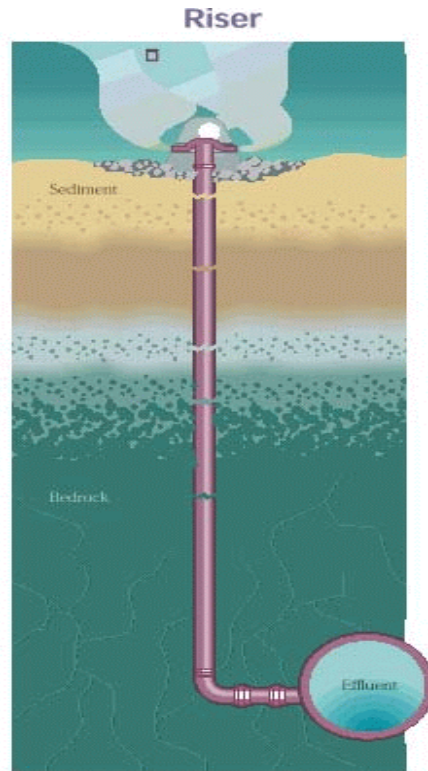

Figure 10b. The 55 risers carry effluent from the deep rock outfall tunnel up to the diffuser heads at the seafloor.

Figure 4.2 Outfall riser (MWRA, 2000)

Figure 10c. The diffuser heads each disperse the effluent through eight ports.

Figure 4.3 Turret with multiple ports (MWRA, 2000)

4.2

for the driving force for this mechanism may be obtained by studying the turbulence material on the Visual Plumes CD.

Pure jets are concentrated streams of fluid flow that possess no buoyancy, i.e., their density is the same as that of the ambient fluid. They are momentum jets. Pure plumes, on the other hand, initially possess only buoyancy. Most waste water discharges possess both momentum and buoyancy. Movement through the water column, often in combination density stratification and sometimes the with nascent density effect, can lead to plume buoyancy in portions of the plume trajectory where no buoyancy exists at the source.

Effective dilution

Outfalls rarely carry pure product. Generally a wastewater stream consists primarily of water, the carrier fluid, which contains undesirable constituents, the pollutants. Traditionally, the dilution process focuses on the mixing of the carrier fluid with the ambient fluid, usually water mixing with water. However, in recent years it has been recognized that what is most important is what concentrations of pollutants exist in the receiving water after the dilution process has terminated. If both effluent and ambient waters contain pollutants the dilution ability is reduced until it becomes zero when the ambient water pollutant level is at or greater than the water quality standard. This concern is at the heart of the Total Maximum Daily Loading (TMDL) determinations. When modeling a discharge to a body of water one also would want to consider other pollutant discharges and possibly mixing zones in that water body, whether or not the other mixing zones coincide with the mixing zone being modeled.

The effective dilution (Baumgartner, Frick, and Roberts, 1994) addressed this concern by introducing the concept of effective dilution, which measures not the dilution of the carrier fluid but the dilution of the pollutant. The effective dilution is the ratio of the effluent concentration to the concentration of the plume at the point of concern, like the mixing zone boundary. The effective dilution implies a rigorous, total mass balance of the pollutant, providing that the background pollution concentration in the receiving water is accurately described (refer to the next section on how modeling assumptions affect the estimation of effective dilution). It may be possible that plume models, such as UM3, in conjunction with field data that accurately describe background concentrations in the vicinity of the outfall, may be used to satisfy some or many of the requirements of the simpler TMDLs.

The plume element

One may imagine that a control volume, or plume element, can be established that wraps around and cuts through the plume, defined by the plume surface and by two cross-sectional slices through the plume. If plumes were cylindrical in shape the plume element might be compared to the shape of a tuna can. However, generally plumes bend in the current or from the effect of buoyancy and they tend to grow in diameter. Therefore, a more general conception of the plume element is a wedge-shaped portion of a bent cone.

For mathematical modeling purposes the length of the plume element is small but the diameter can grow to be comparable to the depth of the water column. The plume element is a sort of mathematical hybrid. This has important implications on estimating effective dilution when the

background concentration is stratified, which might occur if the background is produced by other sources in the area. Some plume models, like UM3, assume that the concentration of the entrained fluid at any point along the trajectory is defined at the depth of the center of the round cross-section of the plume element. However, often most of the entrainment occurs along the upstream surface of the plume element, which will generally be at different depths. Sometimes, the only way to accurately estimate effective dilution is to alter the background concentration profiles to compensate for knowledge about where the entrainment surfaces are in the water column.

4.1.1   Properties that affect entrainment

The plume issuing from any given port in Figure 4.3 may be considered to be a prototype of all effluent discharges. Even though this one port is only one of hundreds belonging to the outfall diffuser, and many diffusers are simply straight pipes with holes drilled in the sides, and outfalls are natural channels, the behavior and properties of its plume are representative of this collection of plumes and other discharges.

Diffuser properties that affect entrainment

Figures 4.1 to 4.3 suggest that specific diffuser and ambient properties act to enhance or inhibit dilution. The diameter of the port affects the effluent velocity and the surface to volume ratio of any isolated plume. Similarly, the orientation of the port modifies the plume trajectory which may enhance dilution by lengthening the travel distance. The depth of discharge will have similar effect. Other less obvious factors contribute to enhancing entrainment, including the spacing between ports. The larger the spacing the greater the path over which plumes do not interact, increasing the surface area over which clean water can cross the plume surface.

Some port properties, notably the horizontal angle, affect plume behavior and entrainment by changing the orientation of the plume to the ambient conditions.

Ambient properties that affect entrainment

Just as diffuser properties affect entrainment, it is easy to see that ambient properties do also. High current speeds deliver more ambient fluid to the surface of the plume and can act to increase the shear between plume and ambient fluid that contributes to the production of turbulence that will directly affect dilution. The current direction, like the port orientation, can affect the path of the plume through the ambient fluid and hence the exposure the plume has to the ambient fluid.

While current can be seen to act directly on the plume surface, thereby enhancing entrainment, the effects of density stratification are less obvious. Often jets and plumes will have or will acquire a vertical component of motion. For example, a buoyant plume discharged horizontally will begin to accelerate vertically and bend upward. The density of the plume element will frequently differ from the ambient fluid at that depth. The entrainment process will tend to equilibrate the densities. If the ambient fluid is not density stratified the plume will asymptotically approach the density of the ambient fluid, without ever quite attaining the same value. Hence, in

unstratified fluid the plume will tend to rise or sink, depending on its density, until it reaches the surface or the bottom.

If the ambient fluid is density stratified, then the plume element can come to density equilibrium at some intermediate depth in the water column, not at the surface or the bottom. This is called the trapping or trap level. For example, a buoyant plume discharged near the bottom will generally rise and gradually approach the density of the receiving water. Suppose it has acquired an average density of ρ at some depth z, then, since the fluid is density stratified, it can be seen that at a shallower depth, to which the plume element is rising, the density will be less than ρ. Consequently, the plume can pass through a trapping level and change its buoyancy, going from positively to negatively buoyant. The plume will then sometimes undergo a wavelike (Brunt-Vaisala) oscillation, passing repeatedly through the trapping level.

Salinity and temperature are important state variables (state referring to the condition of the fluid, as in equation of state that determines the density of water). Thus salinity and temperature are two important variables that affect plume buoyancy. The non-linearity of the equation of state can cause unusual density effects, including double diffusion and nascent density effects. Fresh water near the freezing temperature and salt water brines are two fluids that can reverse their buoyancy during the entrainment process. For this reason, it is sometimes important that models using linear equations of state are not used for some plume applications.

The level of turbulence in the ambient can also enhance entrainment, especially after the self-generated turbulence within the plume has dissipated due to loss of plume kinetic energy (or high velocity) through the entrainment process.

Effluent properties that affect entrainment

Effluent properties that affect entrainment include salinity and density, as these determine the density of the plume element. Another obvious effluent property that affect entrainment is the flow rate itself. Generally speaking, the more fluid is forced through a given port the less diluted the effluent will become. However, unusual combinations of flow, current, and density stratification can sometimes cause conditions that will cause higher flow plumes to penetrate into unstratified layers where dilution will proceed without hindrance.

4.1.2   Properties that affect effective dilution

Some properties are not important to the hydrodynamics of the plumes and the dynamics of mixing and dilution per se, but still affect the concentration of the pollutant, in other words, they help determine the effective dilution. It is useful to remember that the effective dilution is the ratio of the effluent concentration to the plume concentration, whether average or centerline values, at a point of interest, often a point on the mixing zone boundary. The effective dilution is a true indicator of pollution concentration in terms relative to the effluent concentration.

Effluent and ambient pollutant concentration

The most obvious property that controls the final concentration in the plume element is the effluent concentration. Using UM3 as an example, the plume element concentration is a linear

function of the effluent concentration. An increase in the latter results in a proportional increase in the plume element concentration.

It may be less obvious that the ambient concentration affects the effective dilution. The entrainment process mixes ambient carrier fluid into the plume. With the carrier fluid comes any pollutant mass that is contained within it. In the final analysis, entrainment of polluted ambient fluid is like an additional contribution of pollutant to the effluent flow. Its effect on determining plume pollutant concentration is not as simple. The ambient concentration acts more like a floor for plume concentration. Thus, if the effluent concentration is 100ppm and the ambient concentration is 1ppm, no matter how much the carrier fluid is diluted by ambient fluid, the concentration in the plume will never fall below 1ppm and a limit on the effective dilution will be 100.

Mixing zone boundaries and other variables

The distance to the mixing zone boundary will also modify the effective dilution. Variables such as start time used with time-series input affects it by determining the inputs at the specific times of interest.

Finally, the pollutant decay rate and far-field current help determine the effective dilution. The decay of non-conservative pollutants occurs independently of the hydrodynamic mixing processes in the plume. Pollutant decay is a function of time. Changing the units on the pollutant decay column to langleys per hour causes VP to calculate decay rate based on solar insolation, light absorption, salinity, and temperature.

The far-field current affects plume concentration indirectly. Mixing zone modeling focuses inherently on concentrations measured or predicted at distances measured relative to their orientation to the source. These distances and the far-field current are used to estimate the travel time between source and receptor site.

4.1.3   The dilution and ambient tables as property repositories

Most of the diffuser, effluent, and ambient properties that affect entrainment and effective dilution have a column devoted to them in either the diffuser or ambient tables. These tables are the primary place for inputting the data that are essential to simulate the plumes that factor into a mixing zone analysis.

4.2   The One-port example

This example demonstrates how to set up Visual Plumes and to use it to analyze a single port plume simulation. It is highly recommended that the user runs VP while reading this tutorial.

4.2.1   Starting and exploring VP

<u>Starting VP</u>

The Visual CD has a subdirectory called "Visual Plumes (VP) Setup" that contains an executable program called setup.exe that can be run to install VP. The default options, that are recommended, will establish the software in the target subdirectory called c:\plumes (or other drive or directory). Of course, the default name, Walter Frick, and company, USEPA, should be replaced by your own name and business. *If case of problems send email to* frick.walter@epa.gov. For more, see the subsection entitled "What happens when things go wrong?" below.

After setup is complete, an examination of the plumes subdirectory will reveal several applications programs including **Plumes.exe. This is Visual Plumes which can be renamed if desired to VP or anything else**. Other files include a number of prepared examples, this single-port example is not one of them. Double clicking Plumes.exe executes VP. An introductory message appears that gives some information, most importantly, it identifies the Windows settings for which VP is designed.

Because the file Vpsetup is not one of the installed files, VP is unable to establish a previous project, therefore it creates one. The next action is to reveal the *How to proceed* window (Fig. 2.2) which is used to define the target model, the default is UM3. As the window sometimes appears between projects, there are two columns from which to choose the target model. If chosen from the first column, a click will clear any graphics and text output that is left over from a previous run. In this way output from separate projects can be discarded or retained.

Note that by clicking on the different models before exiting the *How to proceed* window will show that the selection affects the headers on the diffuser table. This happens because different models require different inputs.

<u>Tab identifiers</u>

A glance at the diffuser tab at the top of the screen should show the name of the file "VP plume 0.vpp.db", Figure 4.4. This is the default name that VP creates the first time a project is started. If a file by that name existed, VP would increment the zero to one, and so forth, until a unique project name is found. The extension vpp identifies the file as a VP diffuser project file and the extension db is a reminder that this is a binary database file. Notice that the project path and filename also appear in the *Project* box, without the extensions, and in the time-series "borrow" boxes on the diffuser and ambient tabs.

| Diffuser: VP plume 0.vpp.db | Ambient: c:\plumes\VP plume 0.001.db | Special Settings | Text Output | Graphical Output | Surface Discharge Model |

Figure 4.4 The VP tabs. Each entire tab page is often itself referred to as a tab.

Similarly, the ambient tab should show the name of the file "VP plume 0.001.db". This is also a default name. The three-letter numeric sequence is useful when several ambient files belong to the same project. Notice that the ambient file name also appears in the *Ambient file list* edit box, where, eventually, it could be one of many filenames.

<u>Changing the project name</u>

It is a good idea to give projects meaningful names. This can be done by picking the *Save project as* command off the *File* menu and typing the project name "One port" into the edit line of the *Save Project File as* dialog window that appears. The original project files will not be erased in the renaming. They can be discarded through the quitting-time *Pre-exit: File Editing Dialog Window* or from a directory management application, like Windows Explorer.

<u>Exploring Visual Plumes</u>

Before entering data it is instructive to look over the interface and to click on the separate menus and tabs. Except for default values, the VP tabs are effectively blank.

On the diffuser tab, it is instructive to note the "configuration" radio-button groups and checklist at the upper right quadrant of the diffuser tab. Click on the appropriate items such that VP will jump to the Graphical Output tab after running one of the models, the *Units Conversion* selection is *Convert data*, no special options are checked, and *Case selection* radio group shows that VP is set up to run the base or a selected case when a model is picked to run.

The model icon is the same for each model; the text below it shows the target model that is selected. The default model is UM3. The *Parameters for selected row* panel in the lower left hand quadrant states there is one port and identifies the case number (1). The referenced case is identified by a triangle (▸) to the left of the first row in the diffuser table. This first row, row 1, is the *base case*. Close examination reveals that the label *Parameters for selected row* is on a button. Provided the first row is filled in, a press of the button will calculate three important parameters, the (densimetric) Froude number, effluent density, and port velocity.

The *Special Settings* tab is also important. Similar to the columns on the diffuser and ambient tabs that show "n/r" when a variable is not needed by the target model, some of the information on this tab is hidden to reduce confusion. In particular, the *UM3 tidal pollutant buildup parameters* panel is blank. Its components only appear when the *Tidal pollution buildup* option is checked on the diffuser tab's *Model Configuration* checklist. Similarly, the dialog box that appears when the *Output to file* radio button is pressed is hidden when output is to the console (in other words, to the *Text Output* tab).

Also important is the *Selected Variables* list, which shows the variables that are currently on the "output-variables" list. The drop-down list immediately to its left may be used to place variables on the list or to remove them. The default list includes the "P-dia", which should be interpreted to be the port diameter at the point of origin and the plume diameter at subsequent simulation intervals. Finally, the default value of the *Diffuser port contraction coefficient* edit box is 1; this corresponds to a discharge that is shaped by a bell-shaped orifice so that the port diameter closely approximates the initial plume diameter. There is no *vena contracta* effect. A value of 0.61 represents a discharge from a knife-edged port in which the vena contracta effect is pronounced; the minimum area of the plume cross section is 0.61 times the area of the port.

The remaining tabs are basically blank. It may be worthwhile to push the *Style* radio buttons on the *Graphical Output* tab to get an idea for the graphics that are available. The *cus* abbreviation identifies the panel that can be customized on the *Special Settings* tab, in a manner similar to adding variables to the output variables list.

<u>Context-sensitive help</u>

   Most components on the interface have help topics associated with them which are revealed by placing the cursor over the component and clicking the right mouse button. The status line at the bottom of the application is also a useful source of information. The context-sensitive help is not fully updated as of this writing and may differ somewhat from the information provided herein. The help topic in Figure 4.5 is obtained by right clicking on the ambient table, selecting *Help* from the popup menu, and clicking on the *Far-field diffusion coefficient* hypertext topic.

```
wispdelphi                                              _ □ ✕

File  Edit  Bookmark  Options  Help

Help Topics │   Back    │   Print   │   <<   │   >>   │

Far-field diffusion coefficient


  ❑    A coefficient used in the far-field algorithm.


More: This coefficient is used in Brooks far-field equation which predicts effective
dilution in the far-field plume.

α =∈o/b4/3

where

α= the far-field diffusion coefficient

b= width of the plume field at the end of initial dilution

t= the time of travel from the point of the end of initial dilution to the point of interest

      For coastal areas of known high energy dissipation features, or in many
geographical areas at certain times of the year, α may have a value as high as
0.0005 m 2/3/sec.  In less turbulent situations α may be as low as 0.0001
m2/3/sec, thus the user has many options to employ in generating more or less
conservative estimates of far-field dilution.  Small values of α yield the most
conservative estimates of far-field dilution.
```

Figure 4.5 Help topic one the far-field diffusion coefficient.

<u>Exiting VP</u>

   Before beginning the one-port example problem, it is instructive to exit and re-enter VP. One exits by choosing the *Exit* command from the *File* menu (or the Windows **X**). Before shutting down VP displays the *Pre-exit: File Editing Dialogue Window*. Choosing either the *Close* or *Cancel* buttons terminate VP. The <enter> keystroke (carriage return) does the same thing. The purpose of the dialogue window is to provide an opportunity to dispose of unwanted files. This can be done by

selecting files for disposal followed by pressing the <delete> key. The *Files of type* pull-down list provides additional extension filters to help select different files for disposal. The dialogue window is a convenient way to dispose of the default files that were created before they were renamed.

At of this writing, a renaming of the *Open* button to *Close* has been unsuccessful. Similarly, the *Cancel* remains even though its function also closes the application.

NOTE: VP Version 1.0 and above has an *Exit without saving* option.

4.2.2   Problem description

This example might represent a screening, or preliminary, test used to determine whether or not the discharge is likely to exceed water quality criteria. Based on the findings a decision to conduct further analysis may be made.

Effluent variables

The example for this tutorial is a single-port discharge to coastal marine receiving water. This is a relatively small discharge of only 0.05 MGD (million gallons per day). The depth of the discharge below Mean Lower Low Water (MLLW) is 49m. The influence of tides is ignored. The effluent is "fresh" at a temperature of 25C. The pollutant is conservative, which means it does not decay with time. It could be a metal, in any case, it is discharged at a concentration of 100ppm. Note that these parameters are shown below the green headers on the diffuser table.

Diffuser and mixing zone variables

The outfall consists of a pipe that terminates at a pipe-end port 5cm, or 0.05m, in diameter. The port is directed upward slightly at an angle of 20deg, perhaps to prevent scouring of the bottom. The pipe points towards the north, in the direction of the prevailing current, and is one meter above the bottom at the orifice. (The height above the bottom is a minor variable that is often estimated.) As it is a simple pipe opening, there is only one port.

Two mixing zones are established for the pollutant, one at 10m and the other at 100m, corresponding to the boundaries where acute and chronic criteria met, respectively. The mixing zone is defined as a circle around the end of the outfall pipe. Most of the input is shown in Figure 4.6.

Ambient conditions

As is explained above, this is a discharge to marine water. It is presumed that conditions of concern are well represented by average conditions (one scenario). The current is in the northward direction, which means that the flow is towards the north. The current speed, possibly representing the 10th percentile current, is 10cm/sec. The salinity is typical of ocean water at 33psu (psu is an abbreviation for Practical Salinity Units which is essentially identical to parts per thousand, or o/oo). The ambient temperature is 15C and the background pollutant concentration is zero.

Figure 4.6 Project diffuser table with port depth and effluent concentration still undefined.

4.2.3 Inputting the data

<u>The diffuser table</u>

For input parameters given in the primary units (mostly SI, or MKS, units) in the problem description above, Figure 4.6 shows diffuser (yellow, left) and flow input (green, right). The numerical values are simply typed into the cells using the <tab> key to move from cell to cell. Columns showing the "n/r" label are left blank.

Figure 4.7 Bearing units.

The primary unit for horizontal angle is degrees, using the scientific convention in which zero degrees is to the east and the angle increases in the counterclockwise sense, thus north is 90deg. Clicking on the abbreviation "deg" in the *Hor angle* column reveals four options for angular units: *deg, rad, N-deg, N-rad*, corresponding to degrees (scientific), radians (scientific), degrees (bearing, surveying convention, measured clockwise from the north), and radians (bearing). Selecting the *N-deg* unit causes VP to convert to the surveying convention, as shown in Figure 4.7.

The foregoing unit conversion exercise demonstrates the way units are selected in VP. To continue, return the horizontal angle unit back to its original value and change the effluent flow and effluent concentration units to MGD and ppm respectively. After entering the appropriate values from the problem description above the diffuser table should look like the one in Figure 4.8. As far as the diffuser table is concerned, the "base case" is completely defined.

Figure 4.8 Completed diffuser table with user-specified units MGD and ppm.

<u>Documenting the project</u>

It is usually a good idea to write meaningful notes in the project memo edit box found in the upper left quadrant of the diffuser tab. Figure 4.9 shows an example.

<u>The ambient table</u>

The ambient table requires only one change of units in the concentration column (to ppm), to be compatible with the diffuser table. The completed table is given in Figure 4.10. Note that two depths must be specified even though the second row contains no information except an arbitrary depth. The second row is prepared as soon as one tabs past the last column in the table. The extrapolation parameters indicate that the ambient values are constant in the water column. However, even if the extrapolation parameters were changed by clicking on them, the fact that there is only one row data would be interpreted by VP to indicate that the ambient values are constant.

Project C:\Plumes\One port

Project "C:\Plumes\One port" memo
A single port, ocean discharge
no ambient stratification
UM3 was specified as the initial model
units on concentration changed to ppm and flow to MGD
Compatible models: UM3, DKHW, Brooks far-field solution
Incompatible models: PDS is for surface discharges,
NRFIELD requires at least 4 ports
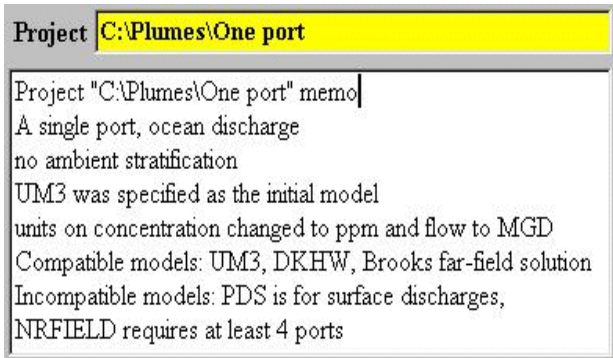
Figure 4.9 Project notes.

Ambient Inputs

	Measurement depth or height	Current speed	Current direction	Ambient salinity	Ambient temperature	Background concentration	Pollutant decay rate(°)	n/r	n/r	Far-field diffusion coeff
Depth or Height		depth	depth	depth	depth	depth	depth	depth	depth	depth
Extrapolation (sfc)		constant	constant	constant	constant	constant	constant	constant	constant	constant
Extrapolation (btm)		constant	constant	constant	constant	constant	constant	constant	constant	constant
Measurement unit	m	m/s	deg	psu	C	ppm	s-1	m/s	deg	m0.67/s2
	0	0.1	90	33	15	0	0			0.0003
	50									

Ambient file list
Filename
One-Port.001.db 1 1

Figure 4.10 The completed ambient table for the one-port project.

The arbitrary depth of 50m could have been set to any other value, however, a depth greater than or equal to the port depth is strongly recommended to force VP to extrapolate to that depth (when extrapolation for one or more columns is on). Also, the ambient values could have been input on the second line, leaving the first line blank except for the surface depth (which does not have to be zero).

Running the models

An initial analysis is ready to be performed. A click on the model logo will cause the target model UM3 to be run. Alternatively, the ^U (<Ctrl-U> key sequence) short cut may be used to start UM3. Finally, it may be selected from the *Models* menu. The four-panel graphical output is shown in Figure 4.11 after the *Scale* button to the left of the graphics panel was pushed.

The numerical output on the *Text Output* tab should look something like that in Figure 4.12. There is a variety of information given but one prediction that might be extracted from the text output is that the predicted dilution at the acute mixing zone boundary is 785.5 to one. If the corresponding pollutant concentration of 0.124ppm easily satisfies all applicable water quality
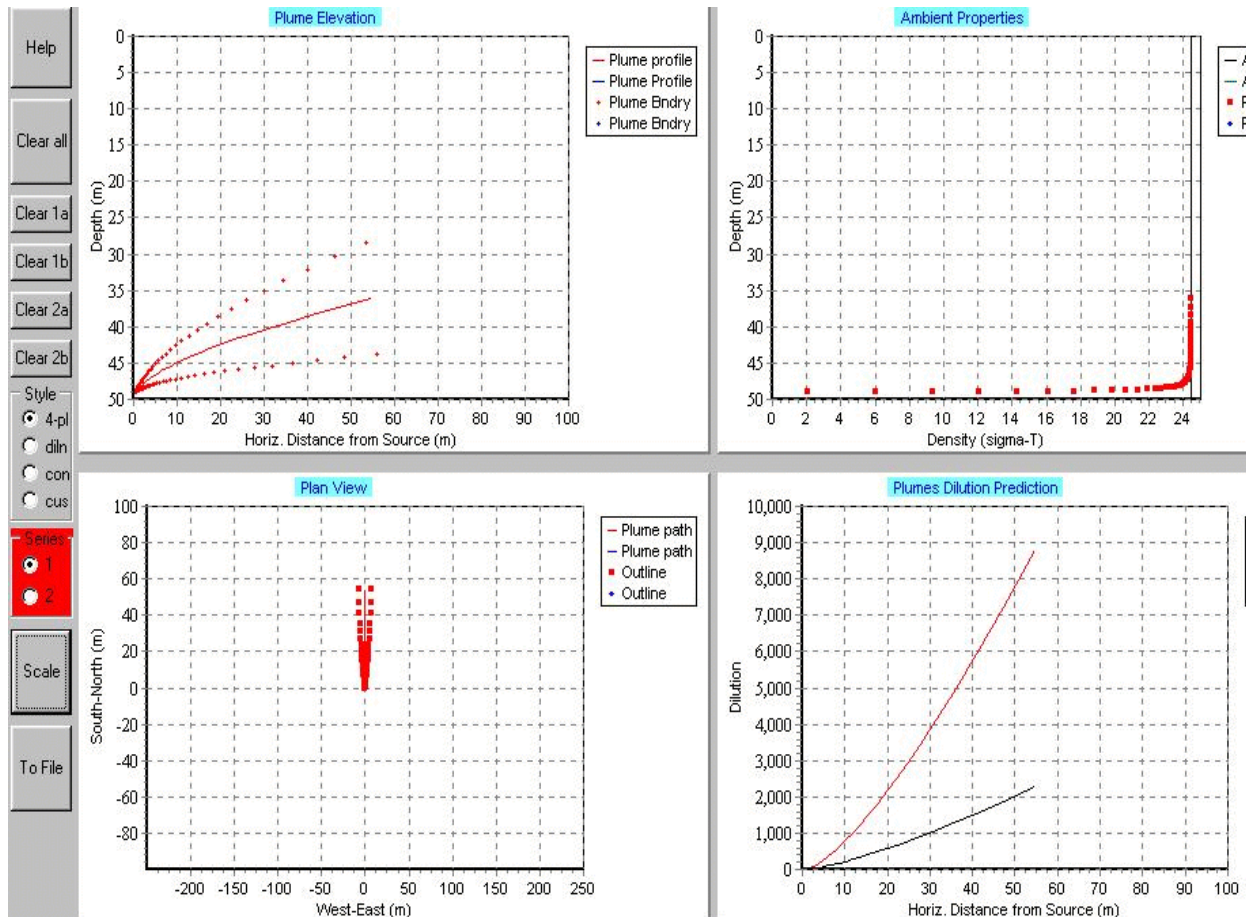

Figure 4.11 First graphic adjusted by pressing the *Scale* button on the *Graphical Output* tab.

criteria, the analysis might be done.

It is worth noting that the *max dilution reached* message is triggered by the fact that the value in the *Maximum dilution reported* edit box on the *Special Settings* tab is set by default to 10000. As soon as UM3 predicts a higher dilution the run is terminated. In reality, a truly buoyant plume, one that is not subject to nascent density effects, will rise to the surface. This can be done by setting a higher dilution limit, say 100000. The effect of this change is explored below.

```
/ Windows UM3. 11/14/00 10:24:27 AM
Case 1; ambient file C:\Plumes\One port.001.db; Diffuser table record 1: ----------------------
   P-dia  P-elev V-angle H-angle   Ports AcuteMZ ChrncMZ P-depth Ttl-flo Eff-sal   Temp Polutnt
    (m)     (m)   (deg)   (deg)     ()     (m)     (m)     (m)    (MGD)   (psu)     (C)  (ppm)
   0.05     1.0   20.0    90.0     1.0    10.0   100.0    49.0    0.05    0.0      25.0  100.0
Froude number:      9.62
          Depth  Amb-cur    P-dia  Polutnt   Dilutn   x-posn   y-posn
 Step      (m)    (m/s)      (m)    (ppm)      ()      (m)      (m)
   0      49.0     0.1      0.05    100.0     1.0      0.0      0.0;
 100     48.75     0.1     0.276     13.8     7.077    0.0      0.596;
 200     47.94     0.1     1.026     1.905   51.11     0.0      2.032;
 300     46.14     0.1     3.08      0.263   370.1     0.0      6.212;
 338     44.92     0.1     4.558     0.124   785.5     0.0     10.05; acute zone,
 400     41.72     0.1     8.538     0.0363 2681.2     0.0     23.27;
 467     35.26     0.1    16.69      0.00963 10104.9   0.0     60.45; max dilution reached,
   .
10:24:27 AM. amb fills: 2
```

Figure 4.12 Initial UM3 prediction displayed on the *Text Output* tab.

What happens when things go wrong?

The last thing the Visual Plumes programmers want to happen is for things to go wrong. But, the reality is that it is difficult to build a large application like VP without inadvertently programming bugs. Some problems are system specific and are virtually impossible to anticipate. Annoying problems may be reported. In an email communication to the email address given above it is usually helpful to attach the input files with which the problem is associated.

A common problem with programs that shuttle information between files, like changing projects, is proper re-initialization of all variables. Before giving up on a problem totally, it sometimes works to simply exit and restart VP. Moving between tabs may also be helpful. A known bug in VP results in the *Style* radio group indicating one style of graphics while VP actually displays another. Pushing a different radio button helps to synchronize these two components.

4.2.4   Modifying the project

Among the most common changes made to VP output are adding variables to the text output and customizing the graphics.

Adding and removing output table variables

Due to the fact that both effluent and current are directed northward, this problem is a two-dimensional problem in the y-z plane (in VP the x-axis points towards the east). This is the reason

why all values in the *x-posn* column in Figure 4.12 are 0.0. For this project the x-axis position of the plume element is uneventful and it may as well be removed from the output table. This may accomplished by clicking on the *Selection List* pull-down list on the *Special Settings* tab, Figure 4.13, and finding and clicking the *x-posn* selection. The selection is removed from the *Selected Variables* list. In the same way the centerline dilution (*CL-diln*) might be added to the list.

Far-field variables

The plume in the simulation listed in Figure 4.12 is halted prematurely by the maximum dilution criterion, consequently, the chronic mixing zone is never reached. In anticipation of running the simulation into the far field, and, due to the fact that the outfall is to the open ocean, this is a good time to add the *4/3Eddy* variable to the output list. If this variable is not added to the list the more conservative constant eddy-diffusivity approach is assumed to apply.

To obtain far-field output the *Brooks far-field solution* option must be checked on the *Model Configuration* radio panel. Doing so changes the remaining column headers on the ambient tab from "n/r" to *Far-field current speed and Far-field current direct(ion)*. As it takes a relatively short amount of time (about 1000sec) for the plume element to move from source to chronic mixing zone boundary, the current is assumed to be uniform. To be conservative, a diffusion coefficient of 0.0003 may be assumed. Making these changes (inputting 0.1, 90, and 0.0003 in the last three columns) and rerunning UM3 yields the text output shown in Figure 4.14.

A discussion of the far-field diffusion coefficient is presented in Figure 4.5.

Figure 4.13 Adding and removing variables from the output table.

Analyzing the revised output

The new output reveals some interesting facts. Most importantly, the plume element reaches the chronic mixing zone before it hits the surface, consequently, even though the Brooks far-field algorithm has been linked in, it is never used. For the same reason, the values in the *Polutnt* and *4/3Eddy* columns are identical, suggesting that the former variable could have been removed from the output table.
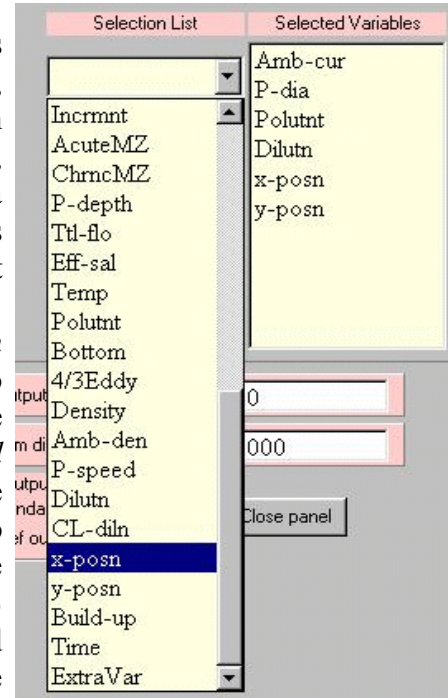
4.15

```
/ Windows UM3. 11/14/00 12:14:32 PM
Case 1; ambient file C:\Plumes\One port.001.db; Diffuser table record 1: ----------------------
   P-dia  P-elev V-angle H-angle  Ports AcuteMZ ChrncMZ P-depth Ttl-flo Eff-sal   Temp Polutnt
    (m)     (m)   (deg)   (deg)     ()     (m)     (m)     (m)   (MGD)   (psu)     (C)  (ppm)
   0.05    1.0    20.0    90.0     1.0    10.0   100.0    49.0    0.05    0.0     25.0  100.0
Froude number:          9.62
       Amb-cur    P-dia  Polutnt  4/3Eddy  Dilutn  CL-diln   y-posn
Step   (m/s)      (m)    (ppm)    (ppm)     ()       ()       (m)
   0     0.1      0.05   100.0    100.0     1.0      1.0      0.0;
 100     0.1      0.276   13.8     13.8     7.077    3.027    0.596;
 200     0.1      1.026    1.905    1.905  51.11    17.85     2.032;
 300     0.1      3.08     0.263    0.263 370.1    108.1      6.212;
 338     0.1      4.558    0.124    0.124 785.5    220.2     10.05; acute zone,
 400     0.1      8.538    0.0363   0.0363 2681.2   721.7    23.27;
 500     0.1     23.18     0.00501  0.00501 19423.9 5078.3   97.68;
 502     0.1     23.64     0.00482  0.00482 20208.6 5281.7  100.6; chronic zone,
 549     0.1     37.71     0.0019   0.0019 51255.9 13314.6  200.5; surface,
Outside chronic zone
```

Figure 4.14 Text output with plume hitting the surface.

Another interesting comparison is between the *Dilutn* and *Cl-diln* columns (the dilution and centerline dilution columns). The centerline is substantially less diluted than is the plume element on the average. However, the average dilution, as it is associated with the total mass of the plume element, is a primary UM3 variable. The centerline dilution is estimated based on assumed concentration and velocity profiles. This relationship can be summarized by a peak-to-mean ratio between average and centerline values. These dilutions are graphed in the lower right-hand panel of Figure 4.11. Peak-to-mean ratios are still the subject of research in the plume modeling community (Frick et al., 2000).

Far-field output revisited

For the sake of this demonstration, let it be assumed that there is a second pollutant that is discharged at 200ppm and subject to acute and chronic mixing zone boundaries of 50 and 500m respectively. A new project could be started or the changes could be treated as a separate case. If the latter, it is only necessary to make a second row on the diffuser table and to input the new values in the respective columns, as shown if Figure 4.15. Before running the second case press the clear buttons on the text and graphics tabs. Also, be sure that the row-marker triangle appears to the left

Diffuser, Flow, Mixing Zone Inputs																
Port diameter	n/r	Port elevation	Vertical angle	Hor angle	Num of ports	n/r	n/r	n/r	n/r	Acute mix zone	Chronic mix zone	Port depth	Effluent flow	Effluent salinity(*)	Effluent temp	Effluent conc
m	m	m	deg	deg		m	s	s	s	m	m	m	MGD	psu	C	ppm
0.05			1	20	90	1					10	100	49	0.05	0	25	100
											50	500					200

Figure 4.15 Two concentrations, two sets of mixing zones, two cases.

of the second line of data. If not, move the cursor out of the row, click somewhere, and click on the row a second time.

The output for Case 2 is shown in Figure 4.16. (Notice that Case 2 is associated with row 2 in Figure 4.15, but in Figure 4.16 it is labeled Case 1. When a new run sequence is started, by running a model, VP always resets the label to Case 1.) Due to the fact that the chronic mixing zone boundary is now outside of the initial dilution zone, the Brooks far-field algorithm output is appended to the former output. The dilution at the chronic mixing zone boundary is almost 65000 to one. This is characteristic of plumes in deep water where there is nothing to stop a plume from rising and achieving it maximum dilution potential.

The far-field interval in Figure 4.16 implies that the *Farfield increment (m)* edit box was set to 50m on the *Special settings* tab.

```
/ Windows UM3. 11/14/00 1:02:47 PM
Case 1; ambient file C:\Plumes\One port.001.db; Diffuser table record 2: -----------------------
    P-dia  P-elev V-angle H-angle   Ports AcuteMZ ChrncMZ P-depth Ttl-flo Eff-sal   Temp Polutnt
     (m)     (m)   (deg)   (deg)     ()      (m)     (m)     (m)    (MGD)  (psu)     (C)   (ppm)
    0.05     1.0   20.0    90.0      1.0    50.0   500.0    49.0    0.05    0.0     25.0  200.0
Froude number:        9.62
         Amb-cur   P-dia  Polutnt  4/3Eddy   Dilutn  CL-diln   y-posn
Step     (m/s)     (m)    (ppm)    (ppm)     ()       ()        (m)
   0      0.1      0.05   200.0    200.0     1.0      1.0       0.0;
 100      0.1      0.276   27.61    27.61    7.077    3.027     0.596;
 200      0.1      1.026    3.811    3.811   51.11   17.85      2.032;
 300      0.1      3.08     0.526    0.526   370.1   108.1      6.212;
 400      0.1      8.538    0.0726   0.0726  2681.2  721.7     23.27;
 454      0.1     14.66     0.0249   0.0249  7811.4  2062.4    50.1; acute zone,
 500      0.1     23.18     0.01     0.01    19423.9 5078.3    97.68;
 549      0.1     37.71     0.0038   0.0038  51255.9 13314.6  200.5; surface,
4/3 Power Law.  Farfield dispersion based on wastefield width of    37.71 m
   conc   dilutn   width distnce    time
   (ppm)           (m)    (m)       (hrs)   (ppm)   (s-1)   (m/s)(m0.67/s2)
2.08E-5 51367.2   43.85  250.0     0.138    0.0     0.0     0.1 3.00E-4
1.39E-6 52843.0   50.35  300.0     0.276    0.0     0.0     0.1 3.00E-4
4.95E-7 55515.4   57.14  350.0     0.415    0.0     0.0     0.1 3.00E-4
2.68E-7 58606.1   64.21  400.0     0.554    0.0     0.0     0.1 3.00E-4
1.74E-7 61798.5   71.55  450.0     0.693    0.0     0.0     0.1 3.00E-4
1.25E-7 64966.0   79.15  500.0     0.832    0.0     0.0     0.1 3.00E-4
```

Figure 4.16 Case 2 with far-field output

Customizing graphics

A crude way to customize the graphics, or, to make the plots fit the graphics panels, is to use the *Scale* button on the *Graphical Output* tab. For single runs this often frames the graphics well, however, the scaling algorithm does not work as well when multiple plots must be scaled.

The graphic panels may be customized individually by double-clicking in the margins of each graphic (outside the plot area). Figure 4.17 shows one of the *Graphics control* windows, for the four-panel dilution graph, that can be accessed in this way. This control is set to customize the horizontal axis. The values next to the *Minimum* and *Maximum* edit boxes are extreme values encountered during the plotting process. They serve as guides for framing the graphic but are not reliable indicators when many runs are graphed.

It is important to refer to the *Auto Scale* check box. If it is checked the inputted minimum and maximum values will not be accepted. To manually change the settings, the *Auto Scale* check box should not be checked. Visual Plumes always attempts to reset *Auto Scale* to off at the beginning of each project.

While for finished products the graphics may be highly customized, the VP list files only store selected settings, for example, minimum, maximum, and increment for each coordinate axis, and, legend on or off. Consequently, such customized graphics should be saved in some manner before exiting the project. The *To File* button may be used to create bitmap files of the graphics panels. The bitmap files are each over two megabytes and should be converted to more compact formats or deleted from the directories promptly. A customized version of the graphical output is shown in Figure 4.18.
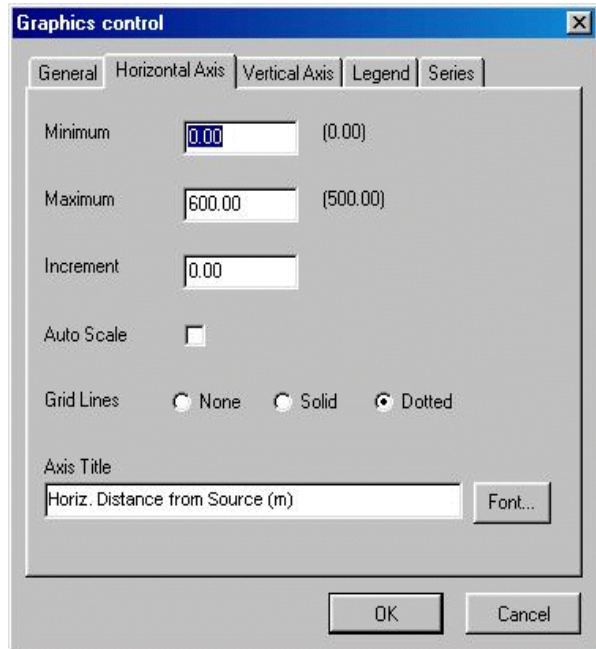

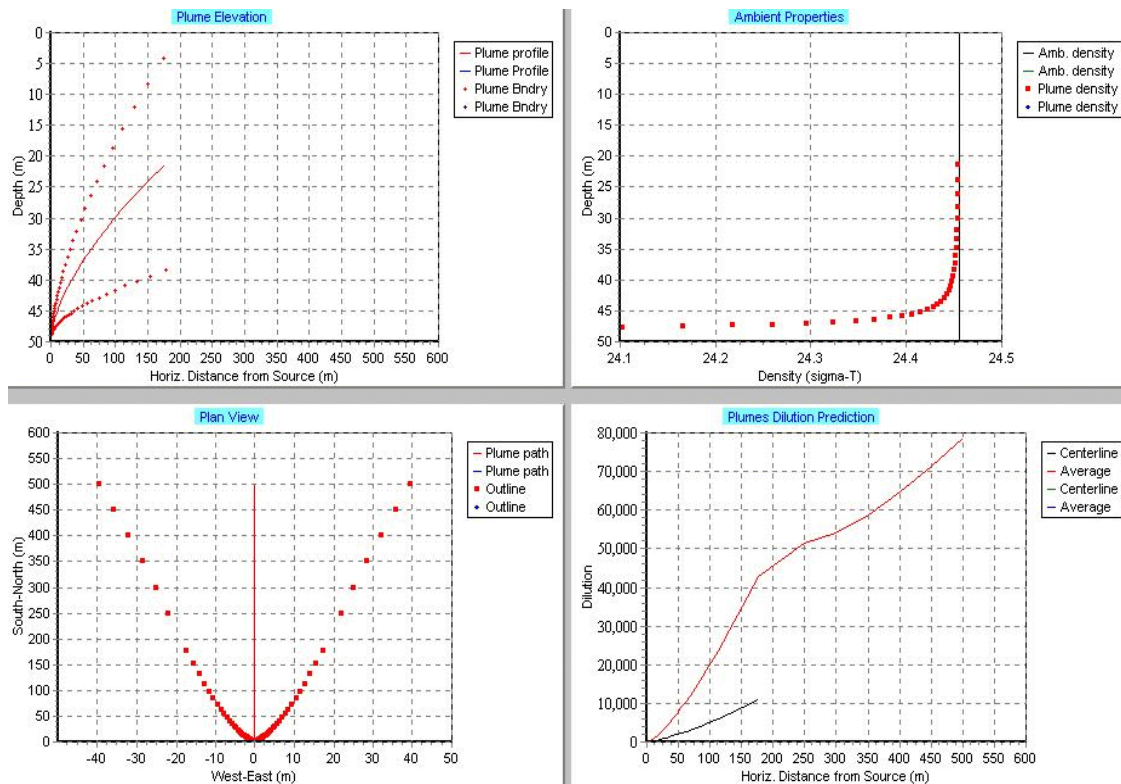**Figure 4.17** Graphics control for the four-panel dilution graph frame.

Figure 4.18 Customized graphics output.

4.18

4.2.5 Adding another ambient scenario

The example above shows the great dilution potential of plumes discharged to deep, unstratified receiving water. This modification to the project illustrates the profound effect of density stratification on dilution while providing an opportunity to develop multiple ambient scenarios.

Creating a second ambient input file and table

To start this exercise it is important that the ambient tab is the active tab. Click on the *File* menu and select *Save ambient file as* command. (The *Add existing ambient file* command is reserved for adding existing files and cannot be used to create new ambient files. If the ambient tab is not the active tab, VP will assume that an entirely new project is to be created. One should make sure that the *File* menu option is not *File project as*.)

Once the *Save ambient file as* command is issued the *Save as* dialogue window should appear with the default filename *One port.002.db*. This ambient filename can be changed to any filename consistent with the ambient file naming convention, however, if the default name is accepted it establishes a sequence of names that is easily manipulated in the *Ambient file list* on the diffuser tab. The disadvantage is that one would have to remember what ambient scenario is stored in what could be project having many ambient files.

For the purposes of this tutorial, the default name is accepted. VP then adds the new file to the *Ambient file list*, closes the previous file *One port.001.db*, and puts *One port.002.db* on the ambient tab. Thus, the way the *Save ambient file as* command works is that it closes the existing ambient file after making a copy of it and giving the copy a new name.

A density stratified scenario

To add realism to this example and to illustrate some of VP's more general capabilities, a fairly elaborate ambient scenario is proposed. To get to the input that looks like that in Figure 4.19, the lines could be erased (using the <Ctrl-delete> key) and input started over. The following alternative way affords opportunities to practice editing skills.

First, click on any cell in the first (0m) row of the ambient table and press the <Insert> key. This will insert a new first line. Enter a 0 in the first column to establish the row (otherwise it will disappear if you move off the row). Move to the second row, repeat the procedure and insert another line and enter a depth of 3m. Now move to the temperature column and enter the value 20C. Next, move to the first column, third row and change the value to 20m. Move to the 50m row and insert another row, making its depth 40m and its salinity 33.1. Still in the salinity column, move the cursor up to the *Extrapolation (btm)* row and click until it shows *extrapolated*. Finally, delete the decay rate value (0) from the third row and re-input it on the first. To delete the value from the cell, click on it a second time to select only the value and then press the <delete> key. The ambient tab should now look something like Figure 4.19.

	Measurement depth or height	Current speed	Current direction	Ambient salinity(*)	Ambient temperature	Background concentration	Pollutant decay rate(*)	Far-field current speed	Far-field current direct	Far-field diffusion coeff
Depth or Height		depth	depth	depth	depth	depth	depth	depth	depth	depth
Extrapolation (sfc)		constant	constant	constant	constant	constant	constant	constant	constant	constant
Extrapolation (btm)		extrapolated	constant	extrapolated	constant	constant	constant	constant	constant	constant
Measurement unit	m	m/s	deg	psu	C	ppm	s-1	m/s	deg	m0.67/s2
	0							0		
	3				20					
	20	0.1	90	33	15	100		0.1	90	0.0003
	40			33.1						
	50									

Ambient file list
Filename
One port.001.db 1 1
One port.002.db 1 1

Figure 4.19 The density-stratified scenario: *One port.002.db.*

This ambient table emphasizes that not all cells need to be filled, in fact, an important attribute of VP is to avoid redundant input. This is not so much a convenience to the user as it is a way to be able to examine the table to gain an appreciation for the data that are available and to make comparisons with similar tables easier.

Before continuing, if desired, any text and graphics that have accumulated might be cleared. One way to clear the board is to exit and restarting VP; if that path is chosen the new scenario can be re-established by clicking on the *One port.002.db* filename in either *Ambient file list*. Once again, VP is able to recover the changes from a previous session from the project's List file, but, fine changes to the graphics panels (like customized labels and titles) are not currently stored for future reference.

CAUTION: It is a good idea to check the name of the ambient input file recorded at the top of the text output for each run to assure the right ambient input is being used.

To appreciate the effect that the new input and extrapolation setting have on the output, go to the *Special Settings* tab and add the ambient salinity (*Amb-sal*) and temperature (*Amb-tem)* to the *Selected Variables* list. This will cause these variables to be output variables and will show how they change as the plume element rises through the water column.

Customized graphics

One of the graphics styles available is the custom graph accessed by clicking the *cus* radio button on the *Style* cluster on the *Graphical Output* tab. The abscissa (x-axis) and ordinate (y-axis) of graph panel can be specified on the *Graphics settings* panel on the *Special Settings* tab. To set the variables, push the *Abscissa (x)* and *Ordinate 1(y)* radio buttons, each time selecting the desired coordinate variable from the *Custom variables* pull-down list. In this case time (*Time*) and pollutant concentration (*Polutnt)* are selected for plotting. The time is the time required for a given plume to travel from the port to any point along the plume trajectory. (The second ordinate is not presently fully implemented).

If the *Casecount* variable is selected for the x-axis, VP graphs "summary" statistics. For example, if the corresponding y-axis variable is *Poltnt* (average pollutant concentration in the

plume) then the predicted concentration at maximum rise will be plotted for every case in the run. Maximum rise could be either at some intermediate level in the water column or when the plume hits the surface.

Comparing models and interpreting the results

In the *One port* project memo it is stated that UM3, DKHW, and the Brooks far-field algorithm are appropriate for this problem. PDS would not be used with this input because it is a surface discharge model, and, NRFIELD would not be used because it is an empirical model for line sources, requiring at least four ports (two T-risers) to be applicable. This example provides an opportunity to compare the UM3 and DKHW initial dilution models. Some of the model predictions are shown on the 4-panel graph in Figure 4.20, UM3 output in red (the trajectory with the greater rise and slightly lower average dilution) and DKHW in blue. Note that the legend for the elevation view has been turned off on the *Special Settings* tab.
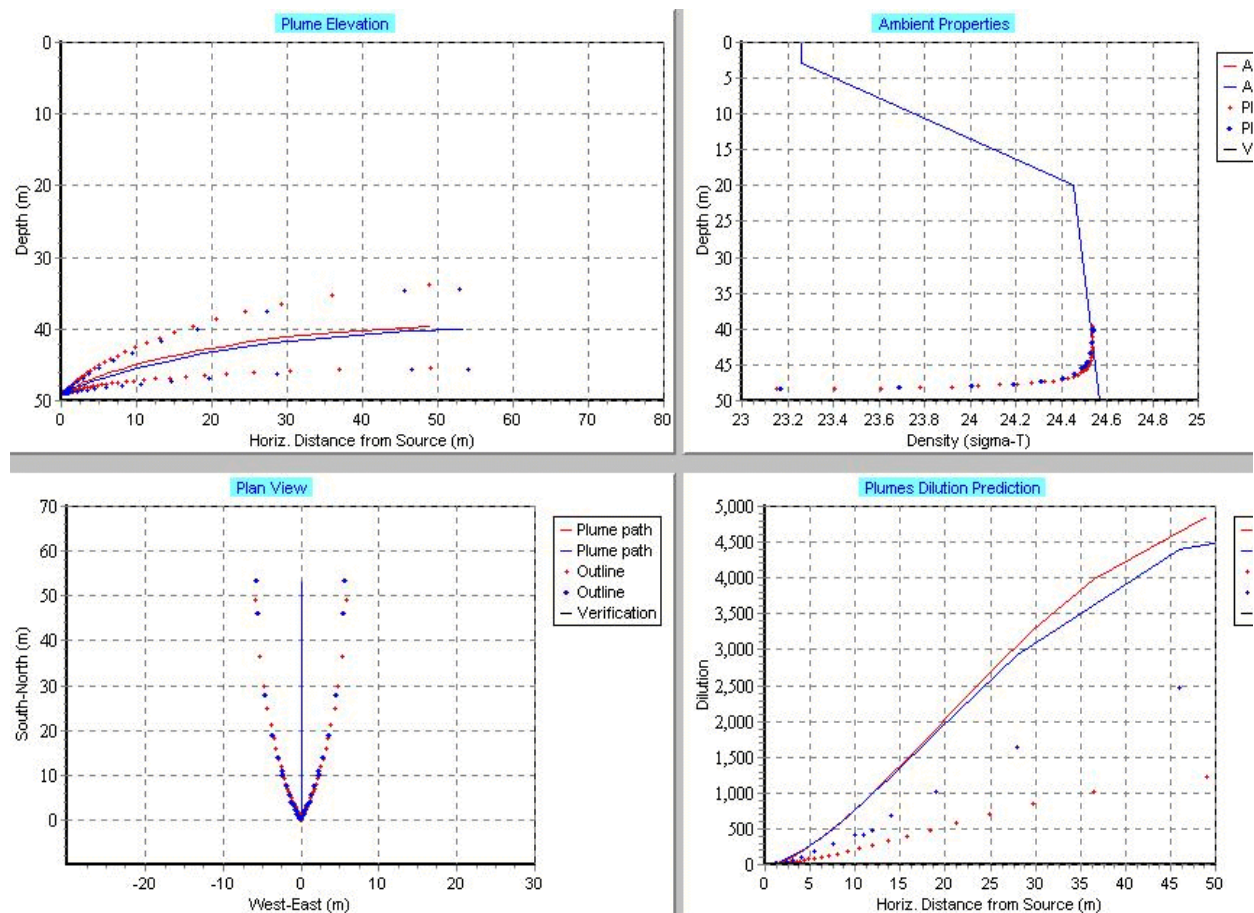

Figure 4.20 One port output for the density stratified ambient scenario.

From the predictions it is apparent that, compared to the unstratified case in Figure 4.16, the plumes do not reach the surface and the dilution attained is much smaller. The average dilution is

less than 5000 compared to over 50000 previously. Even relatively weak stratification has a profound effect on dilution. Stronger density stratification would further reduce dilution. The solid lines in the *Ambient Properties* panel plots the stratification in sigma-T units (for example, $24\sigma_T$ is equivalent to 1024kg-m³). It can be seen how the plume element changes its buoyancy (compare point density to ambient density at the same depth) as it rises through the trapping level, the point at which the average plume density and ambient density are equivalent. As the plume element possesses upward momentum at this point it passes through this level and becomes negatively buoyant. The negative buoyancy above the trapping level decelerates the plume element, reducing the vertical velocity of the element until the vertical velocity reverses and the plume falls back towards the trapping level.

Under ideal conditions a plume will oscillate about a varying trapping level at the so-called Brunt-Väisala frequency as a wave form. Check this by pressing the *1 (to 2nd max rise or fall)* radio button on the *UM3 max vertical reversals* radio group and rerunning UM3. The simulation passes through maximum rise and is continued to the next trapping level. Subsequent trapping levels vary slightly because entrainment continues to change the average density of the element.

Extrapolation

In setting up the ambient table in Figure 4.19, the salinity column *Extrapolation (btm)* cell was toggled to the *extrapolated* setting. In Figure 4.20 the effect of this setting is reflected by the fact that the density continues to increase (is stable) below the 40m depth. Effectively, the density is extrapolated from the values at the 20 and 40m levels. On the other hand, shallower than 3m the density is constant because surface extrapolation is not specified.

Zooming graphics

Any of the graphics can be dynamically zoomed. Using the mouse, click and drag an imaginary box, corresponding to the rectangular area which is to be included, from upper left to lower right. A "rubber-band" box will appear that shows the zoom area. Upon releasing the mouse button the graph is re-scaled. The initial scale can be restored by creating a box in the opposite sense, in other words, by starting from the lower right-hand corner and releasing the mouse button at any upper left-hand corner. Figure 4.21 shows the results of a dynamic zoom on the *Ambient Properties* panel. Magnified in this way, the **Figure 4.21** Density modeling details. predicted depths of the trapping levels can be read more easily, varying from about 42m for UM3 to about 44m for DKHW.

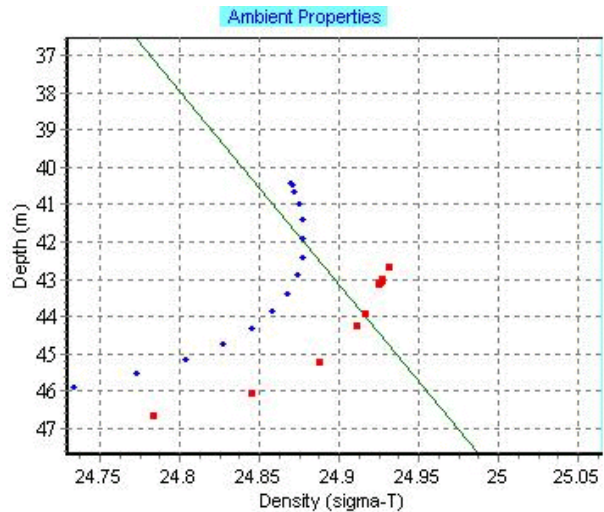Another significant difference in the predictions is of the centerline dilution (*Plumes Dilution Prediction* panel in Figure 4.20). While the average dilutions compare favorably, the centerline dilutions differ by about a factor of two. As both models are fundamentally predicting average dilutions, the reason for the difference is in the way the centerline dilutions is calculated. Both assume a 3/2-power profile but integrate in a different way to estimate the peak-to-mean ratio between centerline and average plume concentrations. The results are shown.

Recent experiments by Roberts (2000) suggest that the peak-to-mean ratio calculated by UM3 may be substantially too high. This problem is likely to be the focus of future research.

Custom graphics

The custom graphic set up in the *Customized graphics* section above is given in Figure 4.22. The rapid decay of pollutant concentration due to entrainment as a function of time is apparent.
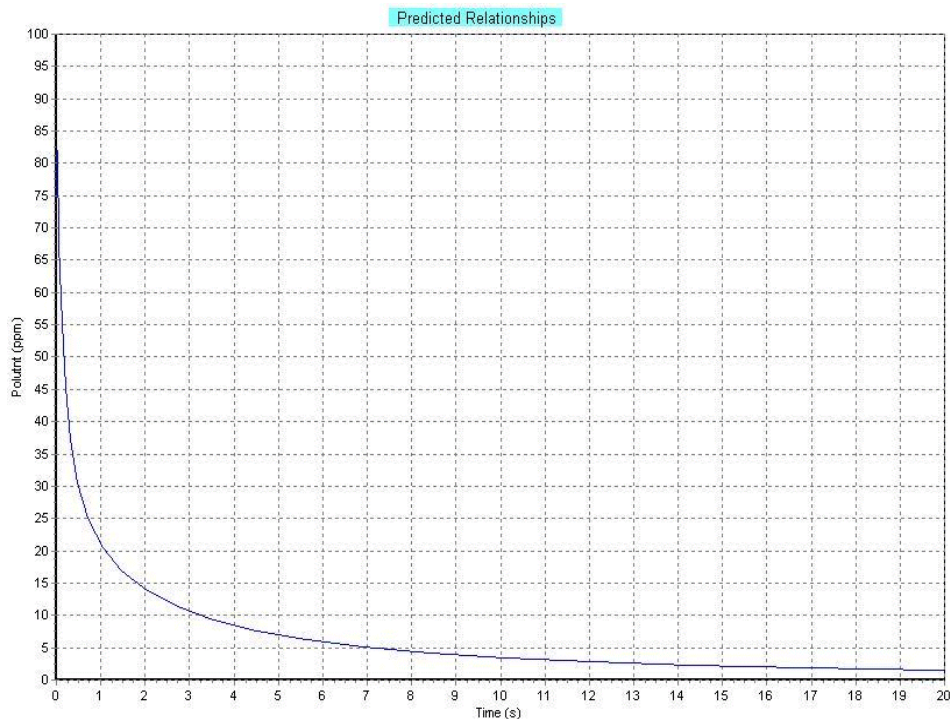

Figure 4.22 Custom graphic showing pollutant concentration plotted versus time.

Tips to remember

When the *Case selection Base or selected case* radio button is selected, it is important to remember to click on the desired row, or case, before running the models. The case-indicator triangle (▸) should be pointing to the appropriate case before running the models. The correct Case

Number should show in the *Parameters for selected row* panel in the lower left quadrant of the diffuser tab. Similarly, when multiple ambient files are displayed in the *Ambient file list*, be sure to select the desired ambient file (or scenario) before running the models. The correct filename should be displayed on the protruding tab at the top of the ambient tab.

On the ambient tab, if any extrapolations are indicated, as they are for salinity in the *One port.002.db* file, then a 0 depth and a depth below the depth of the discharge port should be specified. This will assure that any ambient pairs of data are properly extrapolated and that the density profile is plotted fully on the *Ambient Properties* graphics panel.

Partnership with the user

If one simply reads this tutorial, the One-port example may appear to have been developed quickly and smoothly. In reality, the act of creating the tutorial encountered difficulties that were generally overcome by drawing on experience, but, in a couple of instances minor bugs were revealed that had to be resolved. Not to resolve them would risk adding to the frustration and annoyance that the user may feel when things do not work as advertised or the procedure is not intuitive.

There is no question in the minds of the authors that, as with its predecessor, Visual Plumes will give the best results if its application is partnership between the developers and users. The user does not share the experience of the developers, but, the reverse is true also. Until models are perfect, and this development project shows that this one is far from that, the best that can be hoped for is that there exist a partnership between developers and users that will draw on the talents of both to make an imperfect tool serve the needs of the user.

Figure 4.23 Multi-port diffuser.

4.3   Multi-port discharge

Specifying multi-port diffusers

The single-port example above can easily be changed to a multi-port example by specifying more than one port in the *Num of ports* column of the diffuser tab, Figure 4.23. Once so specified, the *Port spacing* label appears in the column to the right of the *Num of ports* column and a value must now be specified. A two-port diffuser, as shown, would have significant end effects and would represent a conservative analysis.

Some assumptions

The assumptions for multi-port plume prediction vary somewhat with model. The UM3 model treats multi-port plume prediction the same as single-port plume prediction up to the point merging. One important assumption is that all plumes in a given linear diffuser, with the exception

of orientation, are identical. In contrast, the NRFIELD (RSB) model is based on the assumption that multi-port plumes can often be assumed to be emitted from line sources.

With all three VP models it is assumed that the ports point perpendicular to the axis of the diffuser, therefore, staged diffusers, which have ports pointing at an acute angle to the diffuser, can only be modeled by making additional assumptions.

Merging

Merging theory is covered in some detail in Baumgartner, Frick, and Roberts (1994). UM3 uses the concepts of effective spacing and reflection from a plane to help estimate the effects of merging. Effective spacing is an estimate of the true spacing between plumes at any point. With the generalization of UM3 to three dimensions, additional assumptions regarding the merging of plumes are made. Most importantly, when the angle between the current and the instantaneous plume element direction is less than 20deg, the effective spacing is no longer trigonometrically reduced.

An important simplification involving cross-diffuser merging

Diffusers with ports on both sides of the diffuser pipe discharge plumes into opposite directions, creating co-flowing and counter-flowing plumes. Counter-flowing plumes are discharged up-current and will generally rise and bend back into the direction from whence they came, eventually merging with the co-flowing plumes that are discharged on the opposite side of the pipe in direction of the current. This is called cross-diffuser merging.

One way to model cross-diffuser merging is to model the counter-flowing plumes first and to use the output from them to establish background conditions, especially pollutant concentrations, for subsequent co-flowing plume runs. This can be a tedious undertaking.

An alternative approach, that represents a major simplification and that appears to be fairly accurate based on the results of informal modeling trials, is to treat the diffuser as if all ports are on one side with half the spacing. In the context of merging plumes, this approach works well when the distances of interest are somewhat beyond the point of merging. Thus, there is only a limited range of distances, beyond the point of merging and this point where the assumption is unimportant, where the modeling results will suffer some level of uncertainty.

5   Advanced Considerations

5.1   Time Related Data, Time-Series Files

The value of time-series files

For major diffusers or special studies, there may be a need for detailed analyses to adequately describe performance under a range of conditions. In such cases, it is often useful to run the plume models with measurement data taken specifically to predict the likely performance of the outfall under a range of conditions actually recorded during a monitoring study. VP addresses this need by allowing the use of time-series input from files, such as from monitoring reports, in lieu of data specified in the input tables.

Once data are formatted in the prescribed fashion including the information echoed under the time-series filename row of eligible variables, viz., *Port depth, Effluent flow*, etc. on the diffuser table, and *Current speed, Current direction*, etc., on the ambient table, VP can read the time-series files in place of the input table and provide the corresponding prediction and output. In this way, VP can analyze literally thousands of cases in a systematic fashion. The additional time and effort (which depends on the amount of data manipulation required to get the time-series into VP compatible format) may then be recovered from not having to create the numerous ambient input files that would otherwise be required.

VP can cycle through the time-series files repeatedly to simulate initial and far-field dilution. For example, a time-series effluent flow file that records only the diurnal cycle of a wastewater treatment plant may be combined with a very long current meter file thousands of records long. When VP reaches the end of the short effluent flow file it will simply reset it and read it again, as many times as necessary. In addition, VP will synchronize time-series files of unequal time increments. Each file must be set up to start at the same time (time "zero") even though the user has control over when the simulation begins through the *Start time* column on the *Diffuser* tab. The ending time and time increment determine how many cases are analyzed.

A tutorial gives a complete time-series analysis example to help illustrate the main points of this capability.

Linking time-series files

The *Diffuser* and *Ambient* tabs both can accommodate time-series files. The linkage is accomplished in the *Time-series files (optional)* panels, Figure 5.1, after the time-series files are created, as is explained subsequently. Given the associated files exist, they may be linked in by clicking on the cell that displays the *click for file* prompt in the appropriate column. When this prompt is visible it indicates that no file is presently linked to this column, the default configuration. However, the replacement of the prompt by a filename is not a guarantee that the corresponding file is linked in if the file is not in the indicated directory. Clicking a file off and re-linking it will establish whether or not the time-series file is actually available.

Given that "TS series.vpp.db" is the name of the project file listed on the *Diffuser* tab, the default name for the "borrow project" will be the same and will appear in the edit box after the *Borrow time-series from project* label. If another project is to be referenced, it can be made the borrow project by clicking the *Borrow time-series from project* edit box to pop-up a directory dialogue. Once the correct references are established and the file "TS series.vpp.flo" exists, a click on the *click for file* cell in the *Effluent flow* column will open the time-series file.

Preparing diffuser table time-series files

Time-series files must be created external to VP using an appropriate ASCII text editor (which can be a word processor that can save files in ASCII format). Each time-series file is headed by a line of information that allows the data in the file to be properly interpreted, followed by a
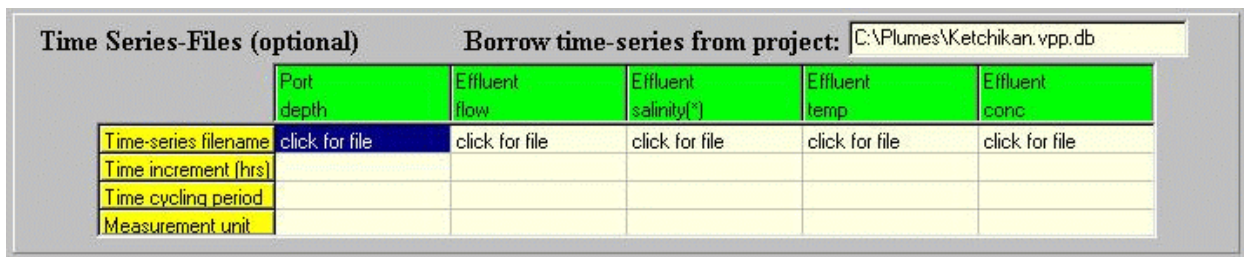
Time Series-Files (optional)		Borrow time-series from project: C:\Plumes\Ketchikan.vpp.db			
	Port depth	Effluent flow	Effluent salinity(*)	Effluent temp	Effluent conc
Time-series filename	click for file	click for file	click for file	click for file	click for file
Time increment (hrs)					
Time cycling period					
Measurement unit					

Figure 5.1 Window for linking effluent (diffuser table) time-series files to Visual Plumes.

listing of the data. Given a project name of "TS series", the following data is an example of an effluent flow time-series file:

1.0 5 MGD TS series.flo
2
3.2
5
4.5
3.5

The first line is interpreted to mean that the time-series increment is 1.0 hour, there are five periods, and the unit of the values in the file is MGD. When it is clicked, the filename replaces the *click for file* prompt and the time-series increment and period values are displayed in the cells immediately below the filename. The name of the file on the header line in the sample file above is not required, read, or used by VP. It's presence is recommended for documentation purposes.

The data above are flows at hour zero (2MGD), hour zero plus one (3.2MGD), hour zero plus two (5MGD), through hour five (3.5MGD). If hour zero corresponds to midnight, the data would represent flows from midnight to one, one to two, etc.. If more than five hours were simulated, the data would be repeated, five to six a.m. would be 2MGD, etc.. The value in the *Start time* column will determine whether the first datum used is 2, 3.2, 5, or some other value.

CAUTION: Upon linking a time-series file, i.e., upon clicking on the *click for file* prompt, VP clears existing data in the corresponding column of the diffuser table, here the *Effluent flow* column, because it is superseded by the time-series file data. If there is important data in this column

pertinent to another part of the study, it might be best to create another project before linking in time-series files.

Time-series file extensions

The file extensions for the time-series (tan) flow variables are:

dep: port depth,
flo: effluent flow
sal: effluent salinity
tem: effluent temperature
pol: effluent pollutant.

Ambient Time-Series Files

The ambient time-series files are more complicated because the information is not scalar by nature. A time-series effluent flow variable (one of those under the tan headers on the diffuser tab) is described by a single number at each measurement time. Ambient data, on the other hand, are only represented by a single value if the ambient property is constant with depth. The ambient variables may require multiple inputs for each measurement time, corresponding to the available depths at which measurements are recorded. Consequently, the ambient time-series file header array contains additional information defining the depths of the subsequent columns of numbers. For example, consider the time-series input file "TS series.001.spd" for current speed:

1.00 6.00 m/s 0.0 constant extra m 10 15 20 25 30 TS series.001.spd
0.15 0.16 0.16 0.16 0.20
0.15 0.16 0.17 0.18 0.19
0.14 0.13 0.12 0.11 0.10
0.10 0.20 0.30 0.40 0.50
0.20 0.20 0.20 0.20 0.20
0.21 0.21 0.22 0.23 0.24

As before, the header line provides information on measurement units, depths, and other pertinent parameters. The values of the first line have the following meanings:

1.00	time increments (hr),
6.00	Total period (hr),
m/s	units of measurement (m/s).
0.0	depth or height (depth = 0.0; height would be indicated by a positive value defining the distance between the port and the surface),
constant	surface extrapolation information (constant in this case, other options are extra(polated) or lin(ear-to-zero)
extra	bottom extrapolation information (same options as above)
m	the units of the depths at which the measurements were made (m),

5.3

10...   the depths (10, 15, 20, 25 and 30m in the example; must correspond to the number of columns),

TS series...   the file name (the filename and other appended text is optional).

Both time increments and total period must be expressed in hours.

Following the header line is a matrix of currents with rows representing consecutive evenly spaced times and columns representing the different depths. For example, the current during the sixth hour at 25 meters depth is 0.23 m/s. See the previous section for a discussion of the time convention.

A detailed discussion of depth and height is given in the following section. Briefly, if measurements are recorded depth-wise a value of 0.0 is indicated after m/s in the header line. If measurements are recorded height-wise, the height of the measurement datum (for example, Mean Lower Low Water, or MLLW) above port depth will be entered.

The file extensions are:

spd: ambient current
dir: ambient current direction relative to the x-coordinate
sal: ambient salinity
tem: ambient temperature
pol: ambient pollutant concentration
rat: ambient pollutant decay rate (or solar insolation)
far: ambient far-field current
dis: the ambient far-field dispersion coefficient.

When an ambient time-series file is linked to VP the corresponding column in the ambient table receives the depths in the time-series file: 10, 15, 20, 25, and 30m. This information is used by VP to interpolate and extrapolate the times series data. *Caution*: as with the diffuser table, any existing values in the column are overwritten. When a time-series file is linked in, the values in the corresponding column in the ambient table are depths and the unit of the column will be a depth unit.

5.2   Depth vs. Height

CAUTION: This section describing the height mode describes capabilities of VP that exist but have not been independently tested.
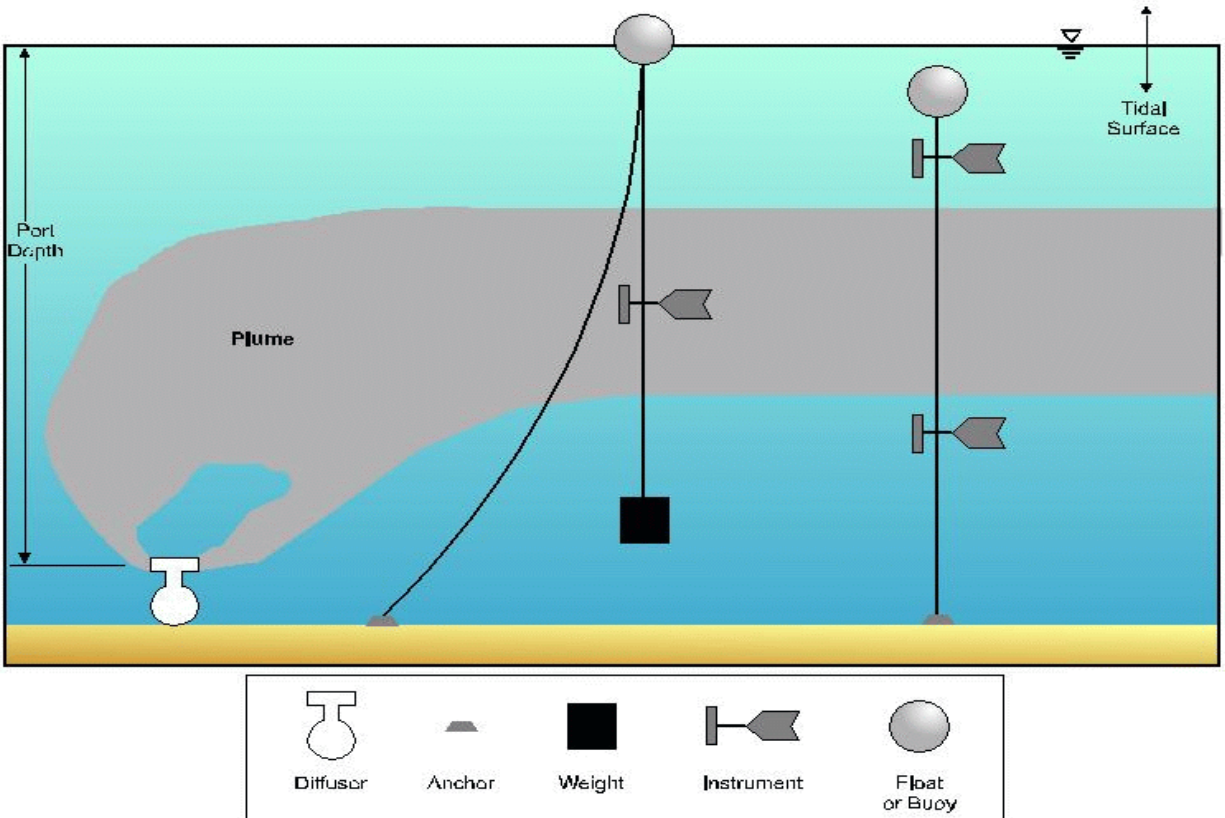
Figure 5.2 Two measurement configurations comparing height and depth-based measurement scenarios.

The *Diffuser* tab allows for a *Port depth* time-series input file. This is to accommodate variable water body depth arising from river stage variations, reservoir and lakes levels, or tides. Variable water-column depths pose programming problems when data are used that were collected from an instrument platform moored to the bottom, because its depth will vary with time. This problem, illustrated in Figure 5.2, can be overcome by allowing for both depth-based and height-based ambient time-series input files.

Figure 5.2 depicts a diffuser running perpendicular to the plane of the paper with current from left to right. A variable tidal surface is shown, which could just as well be a changing stage of river flow or surface elevation of a reservoir. Two measurement strategies are depicted, one in which an instrument is suspended from a buoy and a second one in which instruments are moored to the bottom, the mooring held more or less vertical by a float.

It should be apparent that in the buoyed case the depth of the instrument stays constant even as the depth of the water column changes. In the moored case, however, the depth of the instrument is variable, depending on the water column depth. In order to maintain a level of simplicity in programming, VP uses the same algorithm to interpolate both depth and height based ambient data. For this to work, certain rules for creating height based input files must be observed. For example, given the current speed file *TS series.001.spd*, recorded on the header line of the ambient time-series file, the value of 0.0 following the file units of m/s must be replaced by the height of water above port depth corresponding to the appropriate datum, such as MLLW or Mean Water Level (MWL).

5.5

In the Sand Island example in a following tutorial, it might be 69.8m. One could also think of this to be the depth to port level measured from the water datum.

An ambient file similar to the depth file above but using height and a diffuser at a MLLW depth of 33 m, might be as follows (changed parameters in italics):

1.00 6.00 m/s *33.0* constant extra m *23 18 13 8 3* TS series.001.spd
0.15 0.16 0.16 0.16 0.20
0.15 0.16 0.17 0.18 0.19
0.14 0.13 0.12 0.11 0.10
0.10 0.20 0.30 0.40 0.50
0.20 0.20 0.20 0.20 0.20
0.21 0.21 0.22 0.23 0.24

Once the height-based ambient data mode is established, VP will take the base case port depth and use it to compute the corresponding depths at which the height-based data were obtained. In other words, at run time VP will offset the interpolation depth for height-based ambient time-series files. In this way the same interpolation algorithm used for depth specific ambient files can be adapted to height-specific files.

6   Additional Applications

This section describes several sample problems that demonstrate the use of VP. These problems are included in the built-in help system as additional tutorials and predate the tutorial given in Section 4. Some of the material complements the tutorial in Section 4, but some is redundant. An effort is made to suggest the possibly redundant nature of passages with appropriate headings. For example, the section below on editing data has been covered to some extent in the tutorial in Section 4. As VP has evolved substantially over its development cycle, in some cases the material may not reflect the most recent changes and improvements to VP.

The sample problems given here are available with the VP setup CD and are ready to run.

6.1   Oil well problem

The following problem is based on cases where offshore drilling rigs are discharging effluent as a by-product of the drilling exploration process. The industry wants to run several cases to determine the effects created by varying the effluent density and flow. The problem provides an opportunity to compare the single-port models, UM3 and DKHW.

This project is on the Visual Plumes CD as a prepared example (Oil wells.vpp.db), however, it is described here as if it were developed from scratch. It is up to you whether you want to start from scratch or not.

Editing data

Some of the fundamentals of editing data are presented in Section 4. The hints here recapitulate and expand on the topic.

To overwrite a value a existing value, enter the cell and type the new value before doing anything else. Editing an existing value requires first selecting the cell and pressing <Enter> to put the cell in edit mode. The standard editing keys, like <Home> or <End>, may then be used to move about the cell to change or add characters. After first exiting the cell editing mode, for example, by clicking on another cell, the entire row may be deleted by holding down the <Control> key while pressing the <Delete> key.

Editing data with popup menus

When the cursor is in the diffuser or ambient tables, a right-click of the mouse brings up a popup menu, which includes options for the wholesale deletion of rows of data (Fig. 6.1). Another option is *Font pitch*. This is a useful option when input values are too large for the boundaries of the cell. However, notice that values can usually be examined in their entirety in the *Parameters for selected row* list on the *Diffuser* tab by selecting the cell to be examined.
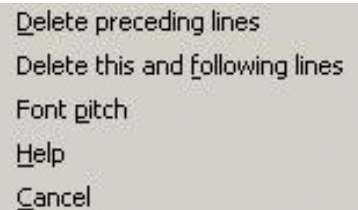

Delete preceding lines
Delete this and following lines
Font pitch
Help
Cancel

Figure 6.1 Popup menu, diffuser and ambient tables.

Units conversion and the *Label only* option

The first column in the diffuser table is for the port diameter, which is 11.75 in. Before entering the value one would change the unit by clicking on it.

The second column is not required (it is reserved for the conduit depth used in PDS) but has the units of meters. The fact that the column is not required may be ignored to illustrate the *Label only* option. For the sake of practice, without first changing the unit, enter

Figure 6.2 Units options.

a value of 11.75 into this empty cell and then imagine that you filled in an entire column of values before realizing that the units do not match the data. To change meters to inches without affecting the inputted values in the column, click the *Label only* option on the *Units Conversion* panel (Fig. 6.2). Then click on the units cell in the column with the newly input value and select the *in* unit. In response to this command only the unit label changes, as desired.

After the label has been corrected, be sure to return the *Units Conversion* panel back to the *Convert data* mode. When done the test value may be deleted.

Oil well diffuser and effluent input conditions

This is a multi-run problem (effluent flow and effluent temperature vary). The following values describe the produced water discharges.

- Port diameter   11.75in
- Port elevation   50m
- Vertical angle   90deg
- Horizontal angle   90deg
- Number of ports   1
- Acute mix zone   10m
- Chronic mix zone   100m
- Port depth   50m
- Effluent flow   10000, 6000, 2000bbl/day
- Effluent salinity   31psu
- Effluent temperature   16.46, 38.49, 74.96C
- Effluent concentration   100ppm

The base case should look like Figure 6.3. Remember, to change the units (flow and concentration) before entering the rest of the data.

Port diameter	n/r	Port elevation	Vertical angle	Hor angle	Num of ports	n/r	n/r	n/r	n/r	Acute mix zone	Chronic mix zone	Port depth	Effluent flow	Effluent salinity(*)	Effluent temp	Effluent conc
m	m	m	deg	deg		s	s	s		m	m	m	bbl/d	psu	C	ppm
0.29845		50	90	90	1					10	100	50	10000	31	16.46	100

Figure 6.3 Oil well project base case for given input conditions.

<u>Adding other cases</u>

Again, the purpose of the study is to determine the effects of flow rate and densities (effectively, salinity and temperature) on dilutions. To obtain a permit to build or operate an outfall, it is necessary to establish limits on flows, if any, that will meet water quality criteria. To determine the dilution, or concentration, of the different effluent flows at varying densities, the user might set up various cases representing the range of conditions likely to be encountered. In this problem there are three effluent flows, each with three different densities, nine cases in all. The two columns that will need additional input values representing are the effluent flow and density columns, both found in the green section of the diffuser table.

The easiest way to add these values is to click on the column that requires additional input, using the down-arrow key to create a new line. The first column chosen is the *Effluent temp* column (it could just as well be the *Effluent flow* column). By changing the temperature of the effluent the implied density changes as well, salinity held constant. The following values will be input into the effluent temperature column going from top to bottom: ***16.46, 38.489, 74.961***, repeated two more times.

Finally, under the effluent flow column, go down to the fourth line and type in ***6000*** (bbl/day). VP interprets this to mean that the first three cases have the same flow. Beginning with the fourth case, it will use all of the input values from the base case except the effluent flow and temperature. Repeat the procedure on the seventh line, entering ***2000***. There should be a total of nine lines in the diffuser table. Clicking on the last line after scrolling the window will show Case 9 in the *Parameters for selected row* panel. Don't panic when the base case is no longer visible, it is still there, however, when moving from row to row, press the *Parameters for selected row* button to update the variables in the panel.

<u>Useful parameters</u>

The *Parameters for selected row* panel shows useful information. The densimetric Froude number is a key similarity parameter that gives information about the relative importance of buoyancy and momentum on plume rise. A low value, especially one less than unity indicates the plume is dominated by buoyancy, a large value, commonly 10 to 100, indicates momentum is dominant. A value of unity represents conditions near the point at which horizontally discharged effluents begin to no longer fill the nozzle. The buoyancy is so great that the effluent squeezes through the upper portion of the orifice leaving space underneath the exiting flow for ambient inflow. This process can radically alter the performance of the diffuser. Flex valves offer a way to increase the Froude number.

The panel also reports effluent density and velocity. These are frequently of interest. The latter is sometimes an indicator of internal diffuser hydraulics. Low pipe velocities can lead to deposition within the pipe which will then alter the distribution of port velocity in multi-port diffusers.

As one uses the tab key to move from column to column in the diffuser table, the values in the panel change. This offers a way to see the value converted into the primary units, if the column's

unit is something other than the primary unit. This is useful because the base case may be out of view.

Salinity, temperature, and nascent density

If you wonder where the temperature used in this problem came from, the answer is from density. In the original problem, neither effluent salinity or temperature were provided. This is a problem from the viewpoint of the nascent density effect because buoyant brines can become negatively buoyant as they rise through the water column and entrain ambient water. While the non-linearities in the equation of state may not be important in some cases, it is always best to specify both salinity and temperature. Therefore, supposing it is known that temperature is the variable determinant of density, VP input is based on the presumption that a guess of the value of salinity will probably lead to better results than using a linear equation of state. (A similar argument holds if salinity is the variable determinant of density.) The linear equation of state essentially assumes that the mixing process results in densities that fall on the straight line that can be constructed between the ambient and effluent densities on a density diagram. This is not generally a good assumption.

This problem is intended to be instructive. No

In this problem the density for the base case was $22.6\sigma_T$, or 1022.6kgm^{-3}. This is easily established by clicking on the units (remember to revert to the *Convert data* option on the *Units Conversion* radio group, Figure 6.2). The salinity was not given but it is known that it is in the neighborhood of 31psu. (It could be 33psu, in which cases the temperatures would need to be readjusted to produce the desired densities.) To establish the original densities for the other cases, it would be necessary to first input 31 (psu) in the first three cells in the column to force VP to convert that cell to a density value. The corresponding densities are $15.6\sigma_T$ and $-4.4\sigma_T$. In converting, VP may report $-4.40004\sigma_T$ for $-4.4\sigma_T$. These conversion errors occur because VP uses a Newton-Raphson method to solve the highly non-linear equation of state when converting from temperature to density.

Notice the process of changing salinity units causes the column header to change to density if a density unit is selected.

Substitute diffuser backup file

If you entered a number of 31psu values in the salinity column to see the conversion to density, you may now wish to erase them. The normal way to do this would be to erase each cell individually.

In this case, if you started this problem from the prepared case (i.e., you did not enter all the input values from scratch), and, you have not otherwise modified the file since it was opened then VP still has the original backup file for reference. To re-establish the backup file one may use the *Substitute diffuser backup file* from the *Edit* menu.

Multiple runs

To set VP up for multiple runs, the *Sequential, parse ambient* case selection radio button in the *Case selection* panel should be highlighted. This selection will run the cases from the top to the bottom on the *Diffuser* tab, but will only do so with the ambient table indicated on the ambient tab. (If there were case number ranges after the filename, it would also select the appropriate ambient files for those cases, after parsing or extracting that information from the filename.) The *Model configuration* checklist also should be checked as appropriate. For this problem requiring estimates at the mixing zone boundary set at 100m, the *Brooks far-field solution* option should be selected. The *Graph effective dilution* option may also be checked. The *After run go to* radio group options may checked and changed, for example, to *Graphics*. The completed diffuser tab should look something like Figure 6.4.

Port diameter	n/r	Port elevation	Vertical angle	Hor angle	Num of ports	n/r	n/r	n/r	n/r	Acute mix zone	Chronic mix zone	Port depth	Effluent flow	Effluent salinity(°)	Effluent temp	Effluent conc
in	m	m	deg	deg	ports	m	min	hr	hr	m	m	m	bbl/d	psu	C	ppm
11.75		50	90	90	1					50	200	50	10000	31.0007	16.46	100
															38.489	
															74.961	
													6000		16.46	
															38.489	
															74.961	
													2000		16.46	
															38.489	

Figure 6.4 Completed *Diffuser* tab for the *Oil well* problem.

Oil well ambient input conditions

The discussion about the linear and non-linear equations of states given in the section *Salinity, temperature, and nascent density* is not repeated here. Once again, the ambient temperature is estimated simply to define the problem in terms of the more accurate non-linear equation of state.

The corresponding ambient input values are given below. Once again, remember, to change the units (salinity/density column and concentration) before entering the data.

- Measurement depth or height   0, 50, 100m
- Current speed   0.115m/s
- Current direction   90deg (same as port discharge)

- Ambient density   25.1, 25.6, 26.1sigmaT (σ_T) at depth
- Ambient temperature   15C
- Background concentration   0 ppm
- Pollutant decay rate   0s-1
- Far-field current speed   0.115m/s
- Far-field current direction   90deg
- Far-field diffusion coefficient   0.000462m0.67/s2, $m^{2/3}s^{-2}$ in more elegant format

       The discharges are discharged about from mid-depth from platforms in water of about 100m depth, so the input depths are realistic.

       From the value of zero in the first column, first row, it is apparent that the values are depths. Ambient data was apparently collected at depths of 50m and 100m. The bottom density value would not be required if the salinity *Extrapolation (btm)* is set to *Extrapolated*. The completed ambient table should look like Figure 6.5.

Ambient Inputs

	Measurement depth or height	Current speed	Current direction	Ambient density(°)	Ambient temperature	Background concentration	Pollutant decay rate(°)	Far-field current speed	Far-field current direct	Far-field diffusion coeff
Depth or Height		depth	depth	depth	depth	depth	depth	depth	depth	depth
Extrapolation (sfc)		constant	constant	constant	constant	constant	constant	constant	constant	constant
Extrapolation (btm)		constant	constant	constant	constant	constant	constant	constant	constant	constant
Measurement unit	m	m/s	deg	sigmaT	C	ppm	s-1	m/s	deg	m0.67/s2
	0	0.115	90	25.1	15	0	0	0.115	90	0.000462
	50			25.6						
	100			26.1						

Ambient file list
Filename
Oil wells.001.db

Figure 6.5 Completed *ambient* table for the *Oil well* problem.

Completing the *Special Settings* tab

       The *Custom graph coords.* radio button cluster on the *Graphics settings* panel allows one to specify the coordinates of the *custom* graph. By defining the abscissa and the ordinate, the x and y-axis, respectively, a customized graph can be created. For the sake of practice, the plume diameter versus distance along the plume might be selected. Click on the *Abscissa (x)* radio button and select *y-posn* (both the discharge and current are in the y-direction, 90 degrees from the x-direction). Then click on the *Ordinate 1 (x)* button and select P-dia.

       To avoid cluttering the *4-panel* graphic, the starting case and maximum number of graphs to be plotted may be specified as shown in the *Start case for graphs* and *Max detailed graphs* edit

boxes (starting case 1 and number of cases 3). The completed *Special Settings* tab should look something like Figure 6.6.

Running the models

The models appropriate to this problem are UM3 and DKHW because they are the only submerged, single-port discharge models in VP. Figure 6.7 shows that UM3 is the target model, hence the square blue icon with the yellow plume on the diffuser or ambient tabs might be used to run that model. Alternatively, the hot key, <Ctrl U>, could be used to run UM3. The *4-panel* style graphics are shown in Figure 6.7. The display has been customized according to Section 2.2.5.
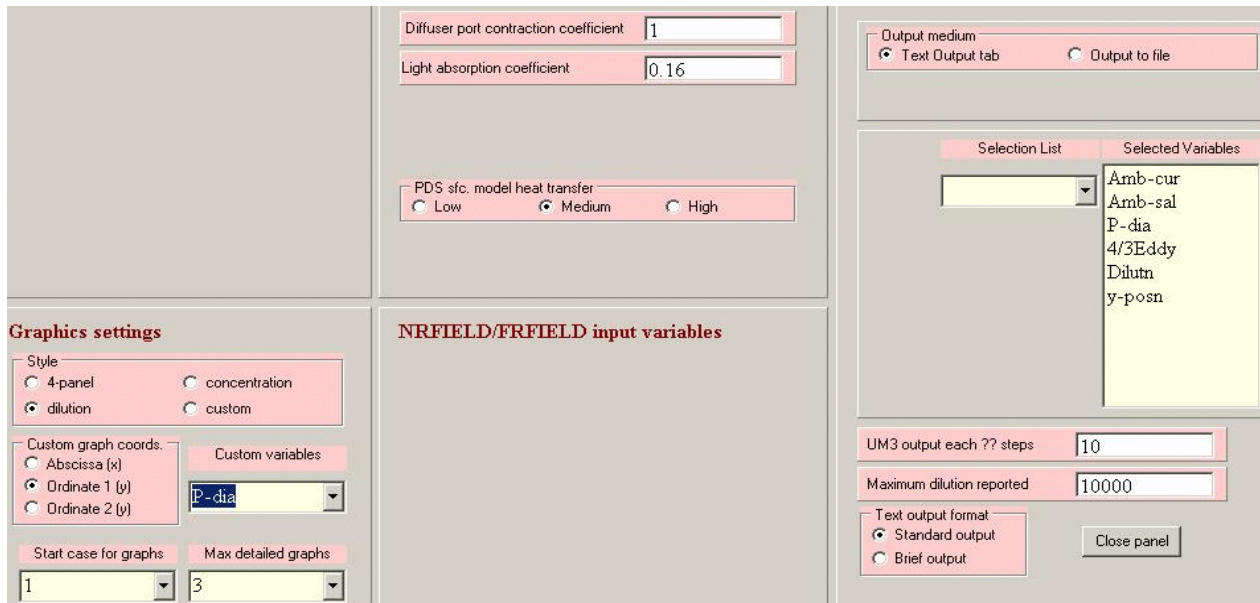
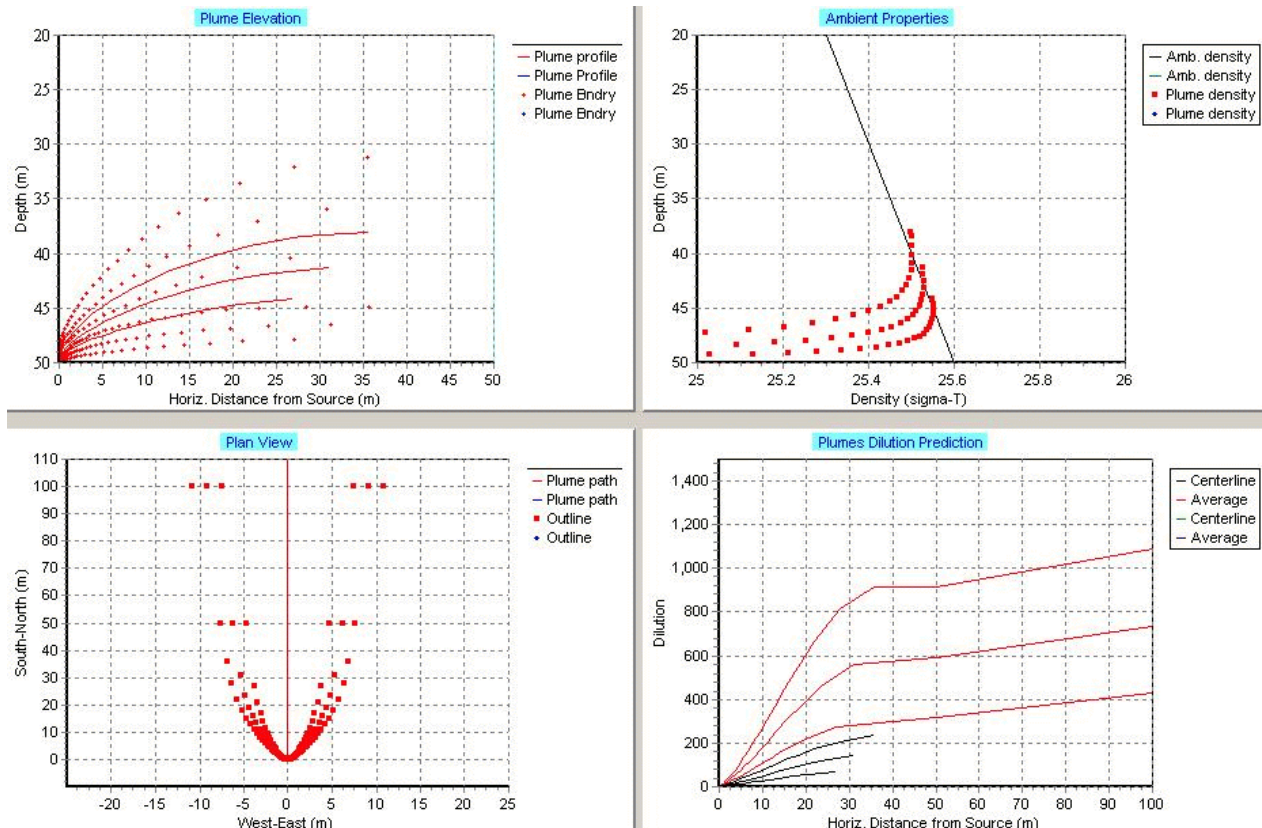**Figure 6.6** Completed *Special Settings* tab for the *Oil well* problem.

Figure 6.7 *4-panel* style graphics for the *Oil well* problem. UM3 model predictions, cases 1-3.

Summary graphics

To continue with the comparison, the DKHW model should be run next. First go to the graphics tab and select *2* (blue) from the *Series* radio cluster. Next select (run) *DKHW* from the *Models* menu. The DOS window will appear which will automatically close depending on Windows system settings, or, **it may have to be closed manually after the word "finished" appears in the DOS window's title bar**. Because the manner of the window closing is unknown, VP displays the message shown in Figure 6.8 as a reminder to close the DOS window after the

Figure 6.8 DOS close message.

finish message appears. The summary dilution graphic is shown in Figure 6.9. Note that the mixing zone values (triangles) have been cleared off the display with the *Clear 1b* and *Clear 2b* buttons.
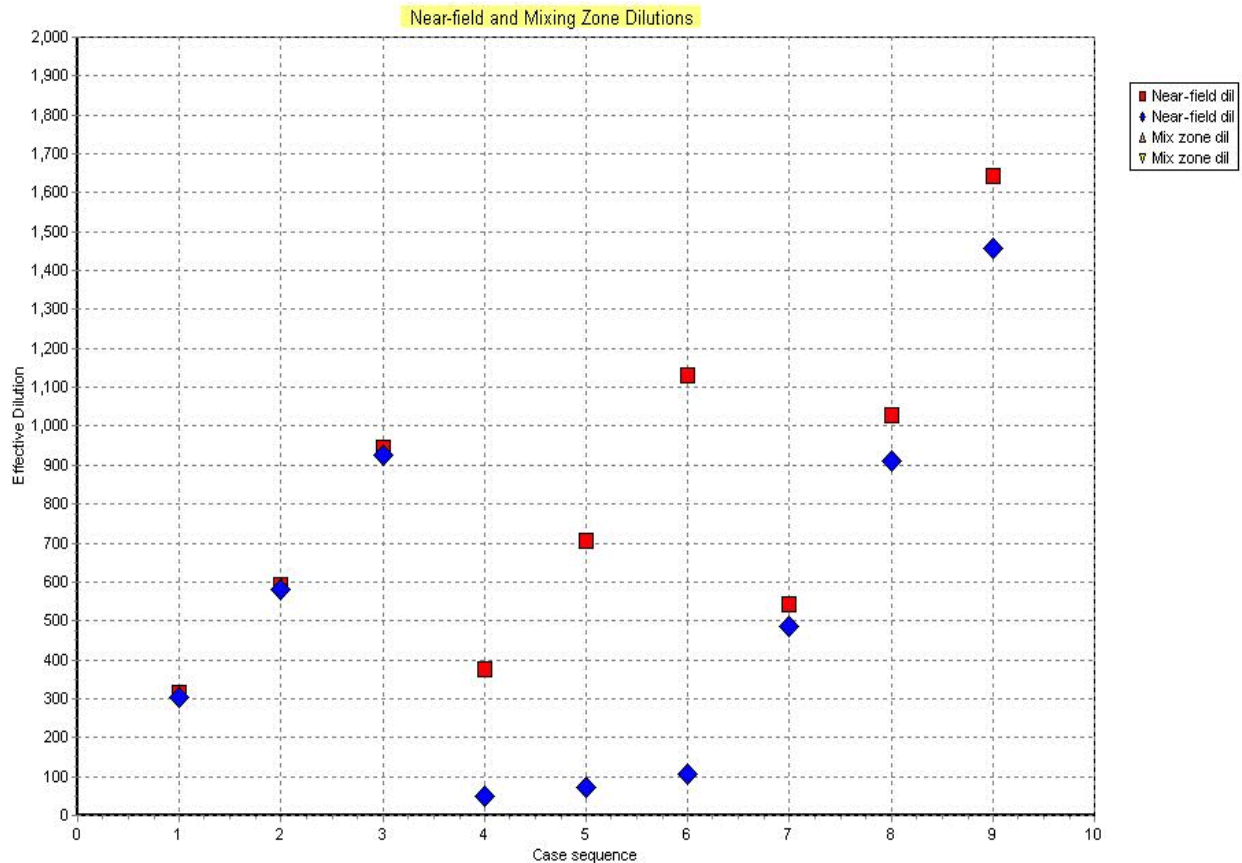
6.8

Figure 6.9 A comparison of UM3 (red squares) and DKHW (blue diamonds).

Figure 6.9 demonstrates the potential value of Visual Plumes as a way to compare different models to lay the foundation for model improvement. The substantial differences in runs 3-6 are current topics of investigation. From continuity it might be concluded that the UM3 predictions are more correct for the three middle runs than is DKHW. But, to make a value judgement is not the point of exercise, it is entirely possible that in some range of conditions continuity would favor the DKHW model. Furthermore, DKHW may generally give better estimates of centerline dilutions, compared to UM3.

The important point is that VP, as a platform for competing models, can facilitate the process of modeling development by identifying areas of disagreement. In the future, it is foreseen that this capability will be further enhanced after the capability to display verification data together with predictions is added.

Custom graph

To complete the discussion, note that the custom graph shows a zigzag curve of plume diameter at maximum rise. The units are in inches, consistent with the units on the diffuser tab. To display the graphic in meter units, it would be necessary to change the *Port diameter* units on the diffuser tab to meters.

6.9

In general, UM3 displays the most complete graphics. For example, there is no zigzag curve for DKHW. The reasons are twofold, first, UM3 was essentially co-developed with VP whereas DKHW is an independent application, and second, in some cases the independent models must be modified to output the variables required by VP to plot specific graphics.

6.2   The Tidal Pollution Background Buildup Capability

As large populations live in coastal areas and many industries are located there, discharges to estuaries and tidally influenced rivers and channels are not uncommon. Such waters are significantly influenced by the tides, causing fluctuations in the velocity of the channel and even periodically reversing the sense of flow, from downstream to upstream. If there are flow reversals, then, even in the absence of diffusion, the conditions exist to cause a buildup of pollution from a particular discharge in the receiving water. During periods of flow reversal, the receiving water loaded with pollutants from the source in question can travel upstream. On returning on the ebb tide, the polluted volume of water passes over the discharge a second time, or even multiple times if the freshwater discharge is relatively small compared to the tidal flow. Thus, the discharge participates in raising the ambient pollution background, over and above what might otherwise be there due to the presence of other nearby sources or to natural occurrence or distant sources.

General theory

If these waters are reasonably one-dimensional in character, VP has a simple way for estimating the buildup of background pollution that is self generated. The general approach is fairly simple, ignoring the influence of most of the channel details, at least upstream and downstream, and dispersion on distribution of pollutants. It depends only on knowing the cross-sectional area of the channel at the point of discharge and the water velocity in the channel as a function of time. Consequently, it must be used with the time-series capability. At the source a representative reach of channel is defined which should be small to prevent the discharge over a period of time from being distributed over a volume of ambient water that is unrealistically large. That would tend to underestimate the level of background. On the other hand, for the sake of memory (currently 200 bins of storage), the fetch length must be large enough to avoid overflowing the upstream end of the array, since information of that pollution would be lost. VP reports when the upper limit of the array is exceeded, in which case the fetch length should be increased. If the fetch length is not increased, one symptom of an inadequate fetch length is the maximum reported concentrations begin to decrease after peaking (loss of pollutant). This might happen after building up to a maximum sometime in after the first week or so of the simulation.

The terms bin, cell, and fetch length are used synonymously. The cell represents one in a series of ambient water volumes that are continuously connected to define the tidal estuary.

Mass conservation only

The model is mass conservative, however, an important underlying assumption is that no redistribution of carrier water from the source is necessary. In other words, all volume contributed by the source is maintained in the fetch element to which it was discharged whenever that element

passes over the point of discharge until that element finally ceases to pass over the discharge due to the presence of freshwater flow or until the fetch element reaches the coastline and fails to return on subsequent tide cycles. That is to say unnatural water elevations can be implied by this approach that would be eliminated if the discharged water were routed by a hydrodynamic model. As a result, comparison between measured and observed concentrations at any given time may be adversely affected. However, over a long period of time, considered statistically, the approach is expected to give realistic results. In other words, while a modeled high concentration may not be observed at the stated time, such a measurement is likely if flow conditions continue to follow the same pattern.

More assumptions

This approach is most convenient when the source is isolated from other sources, so that "natural" background is easily defined, and when the source is sufficiently far upstream so that a fetch element does not reach the mouth of the channel during a single tidal cycle. If the natural background is known, it may be entered as a time-series file, unless it is zero or invariant in time, in which case it may be entered as constant (at least at any given depth).

Setup

The mouth of the channel must be established by specifying its bin (array element) at the start of the simulation, actually, time zero, when the source is assumed to discharge to element 100. This is done by estimating the volume of water existing between the source and the channel mouth. Dividing this volume by the fetch element volume results in the number of bins that must be subtracted from 100 to establish the initial location of the channel mouth. If this number is greater than 100 and it is known that a fetch element can reach the mouth within one tidal excursion, then the length of the fetch element should be increased to reduce the number of elements to under 100. However, if that is not the case, maximum resolution should be pursued with the first restriction in mind, in other words, the upstream distance should be sufficient to avoid the loss of information about the content of the fetch at that terminus.

If tidal velocities are not specifically recorded and available, they can sometimes be estimated or synthesized from knowledge of maximum current speeds under known flow conditions. A simple harmonic program might then be used to generate a time-series current record, including the effect of freshwater drift in the calculation.

Also, since current velocity in VP is a vector quantity, both speed and direction must be provided. Speed must always reported as positive because direction determines the sense of flow. If current velocity is actually recorded, any direction in a given 180 degree sector will represent upstream flow and the remaining sector will represent downstream flow. (Water movement is reported in the direction toward which it flows, opposite the convention used for winds.) On the Settings tab the user specifies the central upstream direction. For example, for western coastline and a channel oriented west to east, 90 degrees would represent the central upstream direction.

6.3   Advanced application: time-series files

The following problem has two parts. Part one involves a conservative pollutant, meaning that it does not decay over time. Part two involves a non-conservative pollutant that does decay with time. This problem is based on a publicly owned sewage treatment work (POTW) on the East Coast. The discharge is seaward of tidewater, but consists mainly of freshwater, in other words it discharges into a tidal river. The effluent has secondary treatment. It should be noted that in the second part the effluent concentration of bacteria of 1,000,000 colonies/100 ml, which is used in this problem, is not realistic for secondary treated waste. A high value is chosen to demonstrate the interplay of the tidal background pollutant, the pathogen model, and the total concentration. The POTW analyst wants to run a number of cases to determine the concentration of both the conservative and non-conservative pollutants over time. In order to meet regulatory requirements, the POTW must demonstrate it can meet water quality criteria to be issued a discharge permit not requiring additional treatment.

The projects are bundled with the VP software and are called MD_metal and MD pathogen respectively.

6.3.1   Conservative pollutant

Diffuser table input

To set up VP to show all column headers that are relevant to a time-series application, the *Tidal pollution buildup* option should be checked on the *Model Configuration* checklist.
The diffuser and flow input values are given as follows:

- Port diameter   0.1m
- Port elevation   0.2m
- Vertical angle   90deg
- Horizontal angle   90deg
- Number of ports   12
- Port spacing   0.455m
- Starting time   1s
- Ending time   720 hours
- Time increment   0.5hr
- Acute mix zone   10m
- Chronic mix zone   50m
- Port depth   5.79m
- Effluent flow leave this column blank. The effluent flow is changing over time, therefore, a time-series file will represent the values.
- Effluent salinity   0psu
- Effluent temperature   28.42C
- Effluent concentration   65ppm

Creating diffuser table time-series files

6.12

Time-series files are created in spreadsheets or word processing programs and are brought into VP as ASCII files. In other words, when you save the file in a word processing program using *save as* option, choose the *Save as an ASCII file delimited text* option. Save the file to a name that has the same name as the project, and a suffix the same as the type of data file. For example, in this case, the time-series file is for effluent flow. The extension for flow is *flo*, therefore, the full name should be *MD_metal..flo*.

There is a standard format that must be followed when creating time-series files. The first line is always a header line. On the header line, the first number is the time increment used for this time-series file. This will be dictated by the measurement time interval of the data provided by the POTW. The time increment is assumed to be 1.0 hour. The second value on the header line is the number of records in the file. Here a daily record is assumed to be available, hence 24 records. The next value on the header line is the unit of measurement, literally the unit as it is spelled on the appropriate unit popup list, m3/s. Following the unit is the optional name of the project followed by its extension.

The next line should contain the flow at the start time (in other words, starting at time zero) and proceed with flows on each line representing the flow at the increasing time increments. For example, the first five lines (the header line and four data lines) should look like the following list.

A fragment of the effluent flow input file

1.0 24.000 m3/s MD_metal.flo
0.155
0.170
0.194
0.225....

There should be 20 additional lines of flow input representing the flow over 24 hours. If the model is run for a time that is longer than the number of records present in the time-series, the records recycle, starting over at the beginning. In this case because the number of records is equal to a 24-hour period, the 25th hour period (the next 24-hour period) starts back at the beginning record.

After saving the effluent flow file as an ASCII file with the name given above it is ready to link into the project. To do so click on the *click for file* in the cell under the *Effluent flow* label in the *Time Series Files (optional)*. Be sure that the *borrow* references the right project path and name, e.g., *c:\plumes\MD_metal*. Click on the *borrow* edit box to navigate to the right project if necessary.
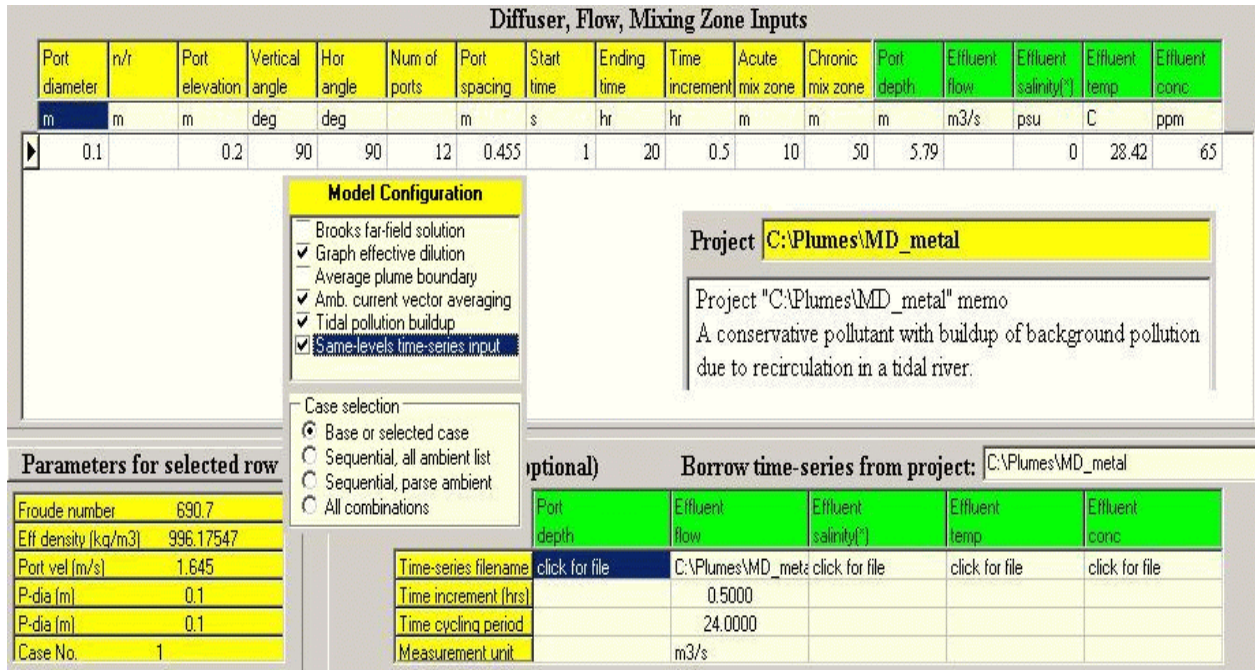
Diffuser, Flow, Mixing Zone Inputs

Port diameter	n/r	Port elevation	Vertical angle	Hor angle	Num of ports	Port spacing	Start time	Ending time	Time increment	Acute mix zone	Chronic mix zone	Port depth	Effluent flow	Effluent salinity(°)	Effluent temp	Effluent conc
m	m	m	deg	deg		m	s	hr	hr	m	m	m	m3/s	psu	C	ppm
0.1		0.2	90	90	12	0.455	1	20	0.5	10	50	5.79		0	28.42	65

Model Configuration

- ☐ Brooks far-field solution
- ☑ Graph effective dilution
- ☐ Average plume boundary
- ☑ Amb. current vector averaging
- ☑ Tidal pollution buildup
- ☑ Same-levels time-series input

Case selection
- ⦿ Base or selected case
- ◯ Sequential, all ambient list
- ◯ Sequential, parse ambient
- ◯ All combinations

Project C:\Plumes\MD_metal

Project "C:\Plumes\MD_metal" memo
A conservative pollutant with buildup of background pollution due to recirculation in a tidal river.

Parameters for selected row

Froude number	690.7
Eff density (kg/m3)	996.17547
Port vel (m/s)	1.645
P-dia (m)	0.1
P-dia (m)	0.1
Case No.	1

Borrow time-series from project: C:\Plumes\MD_metal

	Port depth	Effluent flow	Effluent salinity(°)	Effluent temp	Effluent conc
Time-series filename	click for file	C:\Plumes\MD_meta	click for file	click for file	click for file
Time increment (hrs)		0.5000			
Time cycling period		24.0000			
Measurement unit		m3/s			

Figure 6.10 A composite of diffuser tab components showing input values, memo notes, model configuration settings, borrow window reference, and flow file linkage.

The diffuser tab should look something like Figure 6.10.

The *Tidal pollution buildup* option

The *Tidal pollution buildup* option must be checked to prepare VP for running the pollutant buildup capability. The *Graph effective dilution* should also be checked because it is anticipated that the background pollution concentration will not be zero, therefore, the effective dilution is most meaningful if the focus is concentration, and not simply volume dilution. The buildup capability is only valid for receiving water that is substantially one-dimensional in character, hence the *Amb current vector averaging* option should also be checked. Vector averaging is appropriate for this kind of current.

Ambient table input

Input values into the *Diffuser* and *Ambient* tabs. The values used in this example problem are:

- Measurement depth or height   0 and 5.79m
- Current speed   Leave blank, input from time-series file
- Current direction   Leave blank, input from time-series file
- Ambient density   0psu
- Ambient temperature   28.4C

Ambient Inputs

	Measurement depth or height	Current speed	Current direction	Ambient salinity	Ambient temperature	Background concentration	Pollutant decay rate(*)	n/r
Depth or Height		depth	depth	depth	depth	depth	depth	dept
Extrapolation (sfc)		constant	constant	constant	constant	constant	constant	cons
Extrapolation (btm)		constant	constant	constant	constant	constant	constant	cons
Measurement unit	m	m/s	deg	psu	C	col/dl	s-1	m/s
	0				0	28.4	0	0	
	5.79								

Figure 6.11 Ambient table input, current speed and direction columns are blank pending time-series file linking..

- Background concentration   0ppm
- Pollutant decay rate   0s-1

       The ambient table should look something like Figure 6.11. Notice that the current speed and current direction columns are left blank pending time-series files being linked in.

Creating ambient time-series files

       The time-series file may be created in the same manner as above and saved as an ASCII file. The naming of ambient time-series files is a little different than it is for the diffuser time-series. The base name for time-series files is the ambient table file name in which the *db* extension is replaced by a three-letter extension specifying the variable contained in the file. See section 3.4, *Files and Filename Conventions*. The names are *MD_metal.001.spd* and *MD_metal.001.dir*.

       The Ambient time-series files have a standard format that is slightly different than that of the Diffuser time-series files. The first line is similar with the first value being the time increments, followed by the number of records in the file, followed by the literal name of the unit. After that, the Ambient time-series files differ from those of the *Diffuser* tab. Differences include: the measurement depth or height, extrapolation information both at the top of the water column and the bottom of the water column, the units of the depths at which the measurements were made, the depth in those units, and the file name. If the value is 0 after the units, the data is assumed by VP to be depth-based

(referenced from the surface), a positive value would indicate that the input is height-based (referenced from port depth). For example, the first six lines of this current speed time-series file would appear as follows:

0.5 1920 m/s 0 constant constant m 0 5.79 MD_metal.001.spd
0.1659 0.1659
0.3155 0.3155
0.4462 0.4462
0.5498 0.5498
0.6196 0.6196....

Do not add blank lines at the end of the file.

As you might have noted from the first line of this time-series, there are 1920 records (or rows of data) in this file. There are that many different current records. Tidal currents cycle approximately over a lunar month (although the lunar-solar cycle is much longer, approximately 18.6 years). The value of 1920 derives from the longest period in the synthetic record. To use the time-series file that has already been created for these different currents, click on borrow edit box and navigate to the appropriate time-series reference file, here *MD_metal.001.db*. The time-series file for the current direction is set up the same way. Both are linked in clicking on the corresponding *click for file* cell.

The completed ambient tab should look something like Figure 6.12.

Ambient Inputs

	Measurement depth or height	Current speed	Current direction	Ambient salinity	Ambient temperature	Background concentration	Pollutant decay rate(*)	n/r	n/r	n/r
Depth or Height		depth	depth	depth	depth	depth	depth	depth	depth	depth
Extrapolation (sfc)		constant	constant	constant	constant	constant	constant	constant	constant	constant
Extrapolation (btm)		constant	constant	constant	constant	constant	constant	constant	constant	constant
Measurement unit	m	m	m	psu	C	col/dl	s-1	m/s	deg	m0.67/s2
		0	0	0	0	28.4	0	0			
		5.79	5.79	5.79							

Ambient file list
Filename
MD_metal.001.db 1 1

Time-Series Files (optional)   Borrow time-series files from project: C:\Plumes\MD_metal.001

Time-series filename		C:\Plumes\MD_	C:\Plumes\MD_	click for file	click for file	click for file	click for file	click for file	click for file	click for file
Time increment (hrs)		0.5000	0.5000							
Cycling period		1920.0000	1920.0000							
File measurement unit		m/s	deg							

Figure 6.12 Foreshortened ambient table showing current speed and direction time-series files linked.

Special settings

When the *Tidal pollution buildup* option is checked, the *UM3 tidal pollutant buildup parameters* panel components becomes visible on the *Special Settings* tab, as shown in Figure 6.13. The important variable determined by the actual channel is the channel width. The product of the channel width and the depth of the channel should equal the cross-sectional area of the tidal channel. Based on this area and the average depth of the channel (5.79m) the channel width is computed to be 42.98m.

The upstream direction is also fixed by the geometry of the channel. In this case the freshwater flow at the diffuser is approximately from east to west, hence the upstream direction is 90deg. This variable should be determined by the position of the current meter which, ideally, is located near the diffuser.

Figure 6.13 The *UM3 tidal pollutant buildup parameters* panel.
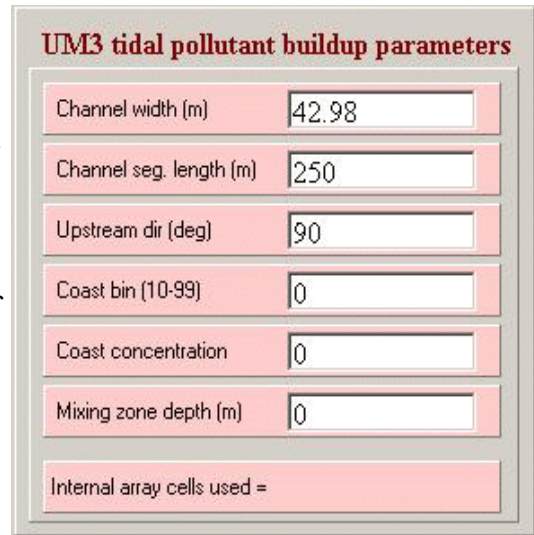
The discharge is located at bin 100, there are 100 bins upstream and 100 bins downstream of the source. In this example the discharge is about 20 miles upstream, more than the distance of a single tidal excursion, hence the coast bin is set to zero. In general, the position of the coastal bin is computed by dividing the mean downstream channel volume by the mean volume of a single channel segment. The volume of the channel segment is the product of the cross-sectional area and the channel segment length. If the downstream volume is less than 100 segment volumes, then the coast bin is identified correspondingly. For example, if the downstream volume is 50 times the channel segment volume, then the coast bin is 50.

The concentration at the mouth of the channel may be specified in the *Coast concentration* edit box.

Comments on optimizing UM3
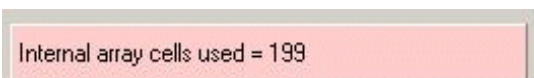
The most tedious part of using the Tidal-buildup-of-background-pollution capability is determining an optimum segment length, which is done by varying the channel segment length. The key point to remember is that for the highest resolution the number of cells used should approach the number of cells available, viz. 200. This is a trial and error process. VP provides feedback on the number of array storage cells actually used in the simulation. A large value, but one less than 200, 199 would be optimum, indicates that no values were ever discarded at the upstream end. This is the most mass-conservative approach. However, for many hours of simulation, the lowest effective dilution will not always be associated with the largest number of cells. This is due to the fact that the cell that contains the highest background may never again cross the diffuser at a time that corresponds to the simulation time of any subsequent run.

In this example, a process of trial and error

Figure 6.14 Optimum tidal resolution.

result in the maximum number of cells, 199, as shown in Figure 6.14, indicating the optimum resolution.

Tidal buildup capability caveats

In the big picture, this capability should not be seen to be a substitute for actual, measured background pollutant data. Rather, this capability is useful when measurements are unavailable or for planning purposes. The pollutant-buildup capability does not include dispersive and other effects that will influence pollutant concentrations in tidal rivers.

The custom graph

For a look at the synthesized tidal current speed the *Amb-cur* can be specified as the ordinate on the *Graphics settings* panel on the *Special Settings* tab.

A UM3 simulation

Figure 6.15 shows the results of the UM3 simulation. The effect of pollutant buildup is apparent. During the first couple of days dilutions show a regular pattern of high dilutions during times of maximum currents and relatively low dilutions during times of low currents. However, as background concentration increase, the effective dilution shows a steadily decreasing trend. However, after about 10 days (500 half hour periods) the effective dilutions show an increasing tendency.
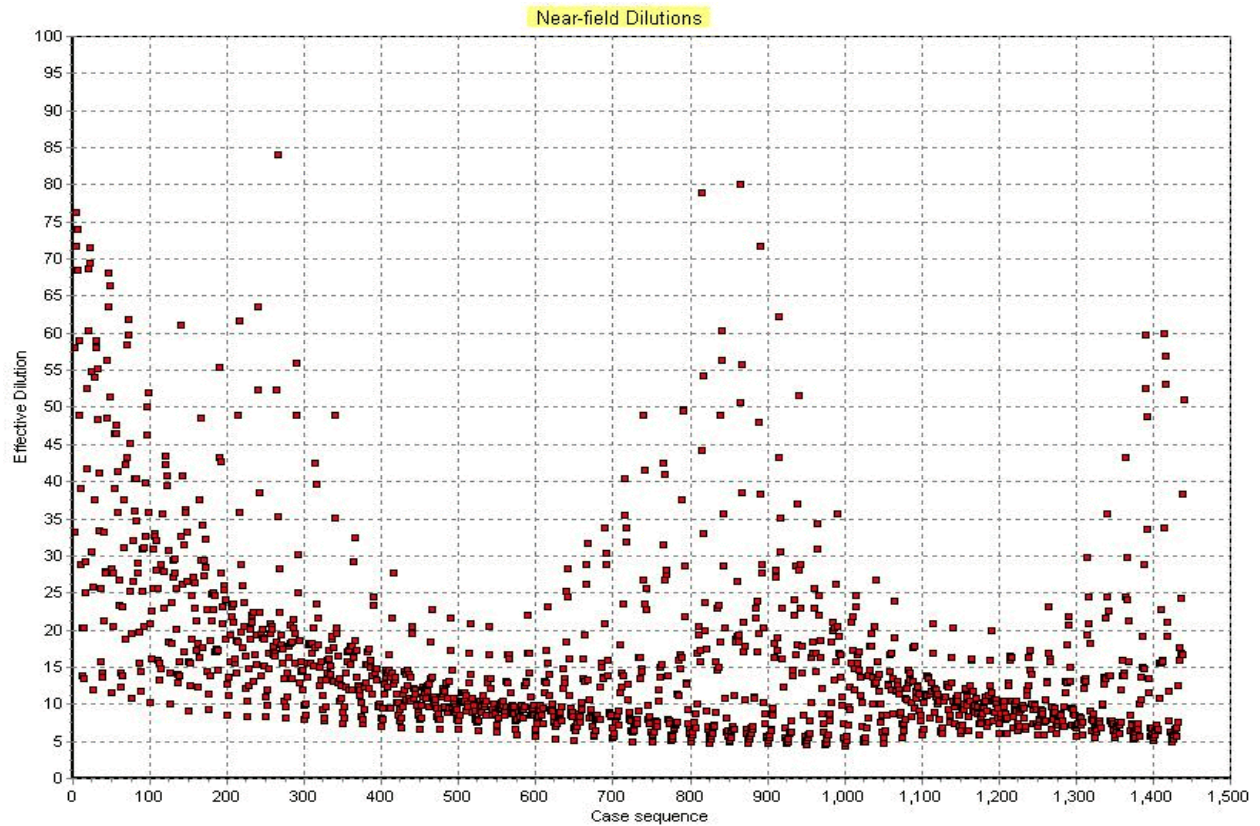
Figure 6.15 Month-long simulation showing the effect of tidal recirculation and background pollution buildup on effective dilution. Note customized graph title (which is not maintained in the list file).

The reason for the increasing tendency in effective dilutions in the middle of the period in Figure 6.15 can be understood by studying the current speed plotted on the custom graph, Figure 6.16.
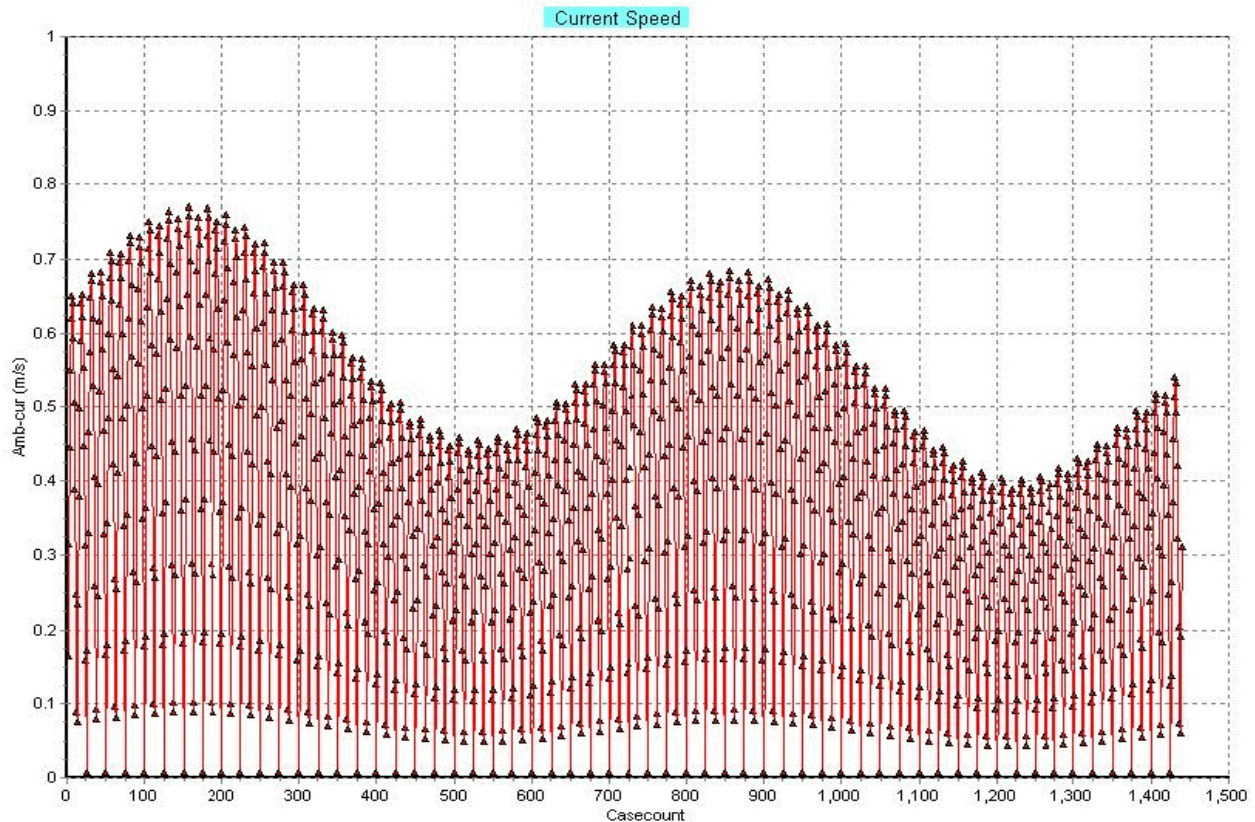
Figure 6.16 A thirty day record of the synthesized tidal current speed in the channel.

Figure 6.16 shows the ambient current responding to daily ebb and flood tides in addition to monthly spring and neap tides. The tendency for effective dilutions to increase beginning about hour 300 (12.5 days, or at case 600) is associated with the transition from neap to flood tides. The steady freshwater flow is sufficiently low that increasing flood tides temporarily increase the effective dilutions.

However, the minimum effective dilutions occur around hour 500 (case 1000) as each successive tide becomes weaker.

6.3.2 A pathogen

The difference between a non-conservative pollutant and a conservative pollutant is the decay time, represented in VP by the *Pollutant decay rate* column on the ambient tab. Conservative pollutants do not decay with time whereas non-conservative pollutants do. Pathogens are an example of a non-conservative pollutant, which may decrease, or occasionally increase, with time. To create a new project that contains many of the values from a project that has previously been created, open the previously created project, which in this case will be the project created for the above exercise. If starting from scratch, immediately after opening the *MD_metal* project, rename it *MD_pathogen*

6.20

using the *Save project as* command. Alternatively, the project is on the Visual Plumes CD and may be opened directly.

Changes to the diffuser tab

A couple of minor changes to the diffuser tab are to the project memo and to the *Effluent conc* column. The effluent concentration a million col/100ml. In general, this column might be the primary candidate for linking to a times-series file, projects would then be associated primarily with the pollutant of concern. This is reflected by the modifiers *metal* and *pathogen* for these two related projects.

Effluent flow borrow files on the diffuser tab

While there are reasons to separate pollutant constituents under different projects, many of the time-series files are likely to be shared by projects. Such files should not be duplicated under new names simply because the new project will search for time-series files using the new project name as a reference. This is the purpose of the *Borrow time-series from project* edit box, to specify a project from which to borrow time-series files. When UM3 is run, VP will establish the time-series files to link to the project. If a time-series file for the identified variable is found under the project reference, that file is linked in. For the time-series file that name would be *MD_pathogen.flo*. However, if *MD_pathogen.flo* is not found, then VP will try to find a time-series file referenced to the borrow project, i.e., it will search for *MD_metal.flo*.and link it to VP if the file exists.
The relevant portions of the diffuser tab are shown in Figure 6.17.

The *Pollutant decay rate* time-series file

The hardest part of the project is to create a time-series file for the *Pollutant decay rate* column. This has been done; the file *MD_pathogen.001.rat* is included on the Visual Plumes CD and should be ready to link to the project. The beginning contents of the file with annotations, added here but not found in the file, in italics are listed below:
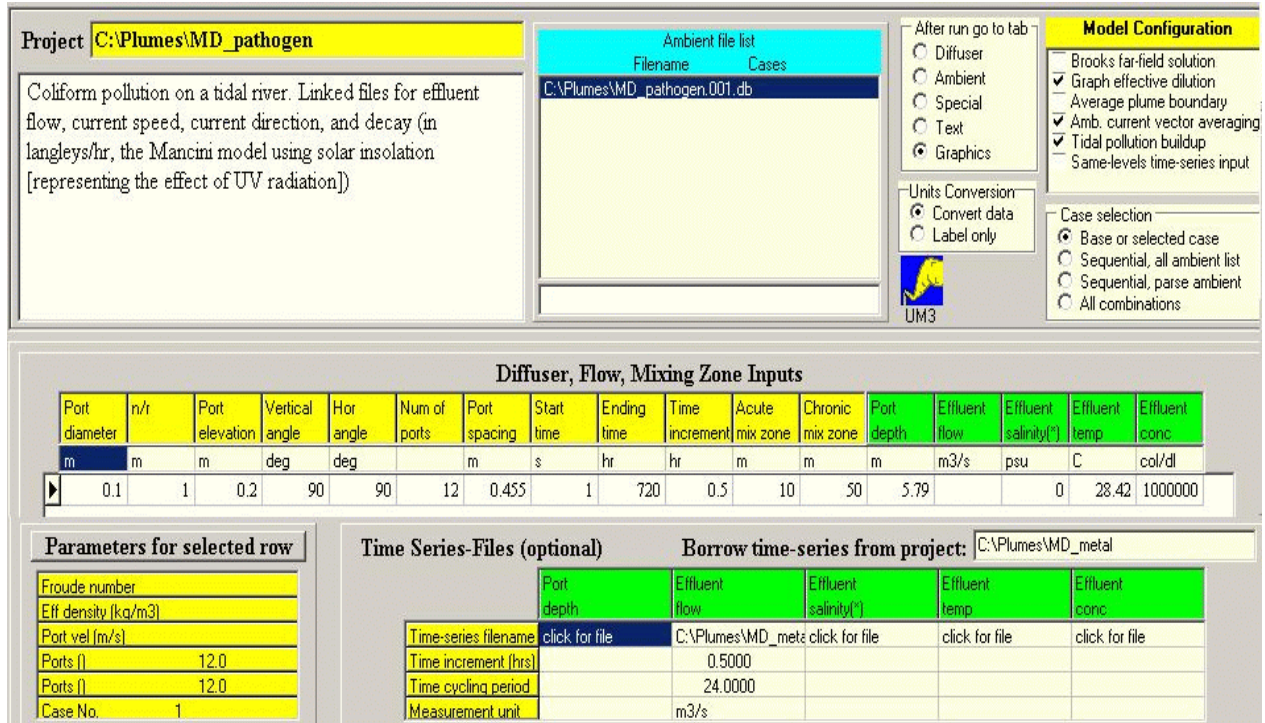
Figure 6.17 Foreshortened diffuser tab for the *MD_pathogen* project showing flow file linked in.

Solar insolation time-series file

0.5 476 ly/hr 0 constant constant m 0 5.79 MD_pathogen.001.rat
0   0
....   (six hours of no solar insolation)
0   0
1.76   0.18
5.34   0.53
11.7   1.17
20.0   2.00
29.6   2.96
40.0   4.00
50.4   5.04
60.0   6.00
68.2   6.82

      Consistent with the VP implementation of the Mancini model (Mancini, 1978), the data represent solar insolation given in langleys/hour.

      Important note: The actual solar insolation at depth is function of the *Light absorption coefficient* on the *Special Settings* tab, for a which a default value of 0.16 is given (see Figure 6.18). The second column in the time series is not required.

To show how solar intensity varies with time it is useful to add the *Decay* variable to the custom graph, as shown in Figure 6.18.

UM3 output

The UM3 effective dilution output for the pathogen problem is shown in Figure 6.19. Again, the graphical appearance of the output reflects customized changes to the axes ranges. The corresponding solar insolation is shown if Figure 6.20; it shows a basic pattern of five sunny days followed by five very cloudy ones.
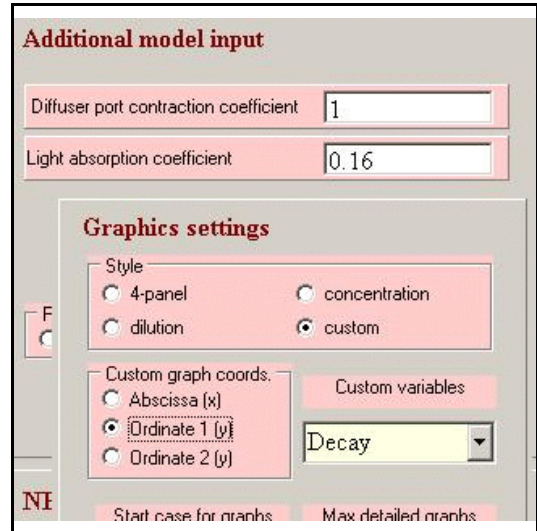
Figure 6.18 Composite graph of relevant parameters on the *Special Settings* tab.
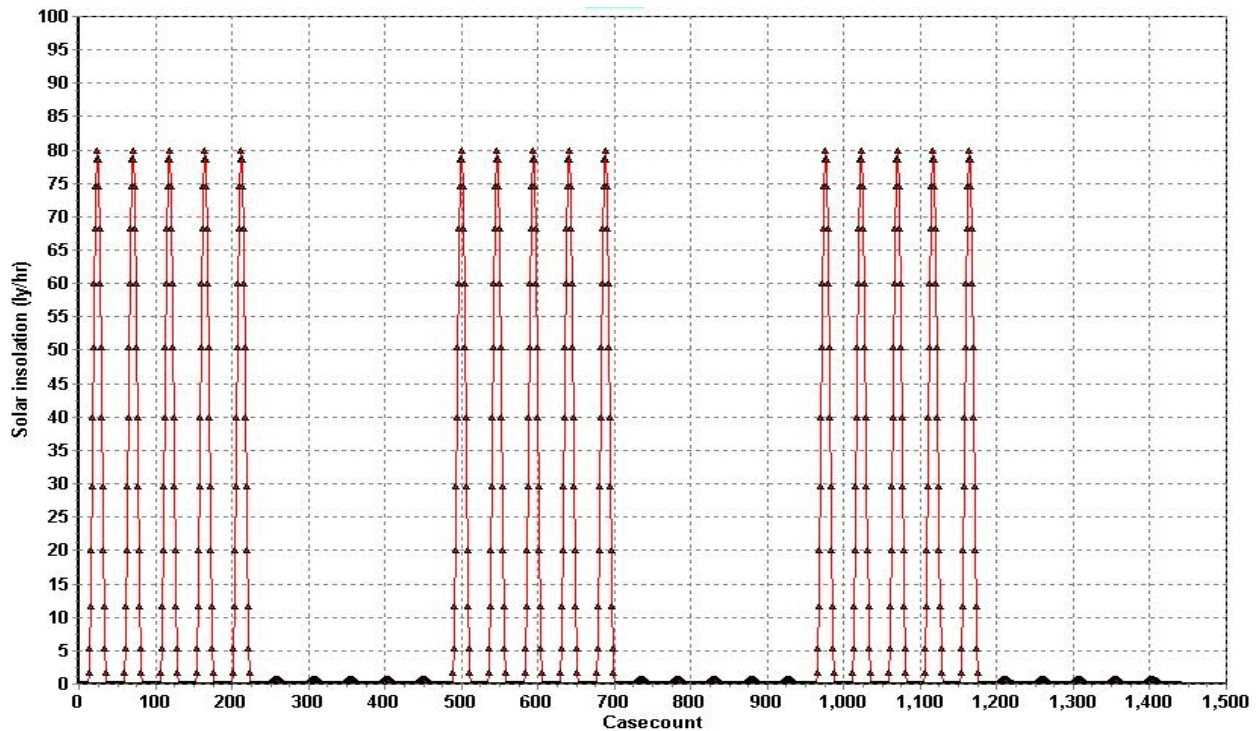
Figure 6.20 Daily solar insolation pattern. Note customized labels that are not permanent.

Figure 6.19 Effective dilution predictions for the *MB_pathogen* project.

Analysis

The effective dilution pattern shown in Figure 6.19 is not as distinctive as is the one for the conservative pollutant (Fig. 6.15). The lack of pattern is due to the fact that the pollutant is

6.23

constantly decaying in the background, thus the buildup of pathogen is not very great. The effect of the long-term buildup of background pathogen is barely discernible except during cloudy periods

6.4 Application of the surface discharge model, PDS

6.4.1 General comments regarding the PDS (PDSwin.exe) model

The PDS model uses only a subset of Visual Plumes input variables. As with the other models, most of these must be defined on the *Diffuser* and *Ambient* tabs while the remainder are on the *Special Settings* tab.

PDS assumes a shoreline, cross-sectionally rectangular surface discharge. The variables defining the discharge include Conduit Width, Conduit Depth, Horizontal Angle, and Region of Interest (ROI).

6.4.2 Sample PDS problem

Input conditions

This problem investigates a thermal plume generated by cooling water discharged from a power plant into a large lake. Discharge is from a channel perpendicular to the lake's shore. A parameter of interest is the distance along the plume centerline to the 2.0 deg C excess isotherm and the total surface area within it. A large region of interest (ROI) of about 8000m may be tried to capture the 2 degree excess temperature isotherm.

Input conditions for the problem are as follows:

- Discharge channel width: 30 ft
- Discharge channel water depth: 4 ft
- Total discharge rate: 230 MGD
- Discharge temperature: 27 C
- Discharge salinity: fresh water, use   0.001 psu
- Discharge angle: 90 degrees (in the y-direction)
- Ambient temperature: 13 C
- Ambient salinity: 0.001 psu
- Ambient current: 0.005 m/s
- Current direction: parallel to shore (zero degrees)
- Surface heat transfer: Low for conservative case

As PDS is a surface discharge model the depth of discharge is zero. The depth on the first row of the ambient table should also be zero. As always, a second ambient depth is required, assume it is 10m. The rest of the second row should be left blank since PDS assumes well mixed ambient conditions.

Go to the *Special Settings* tab. For a conservative result *PDS sfc model heat transfer* parameter may be set to low. On the *Graphics settings* panel, set the custom graph to display plume temperature versus distance, *y-posn* and *Temp* for the abscissa and ordinate respectively.

The diffuser and ambient tabs are shown in Figures 6.21 and 6.22. The relevant input parameter settings on the *Special Settings* tab are shown Figure 6.23.

Figure 6.21 A foreshortened diffuser tab showing *A PDS run* project input.

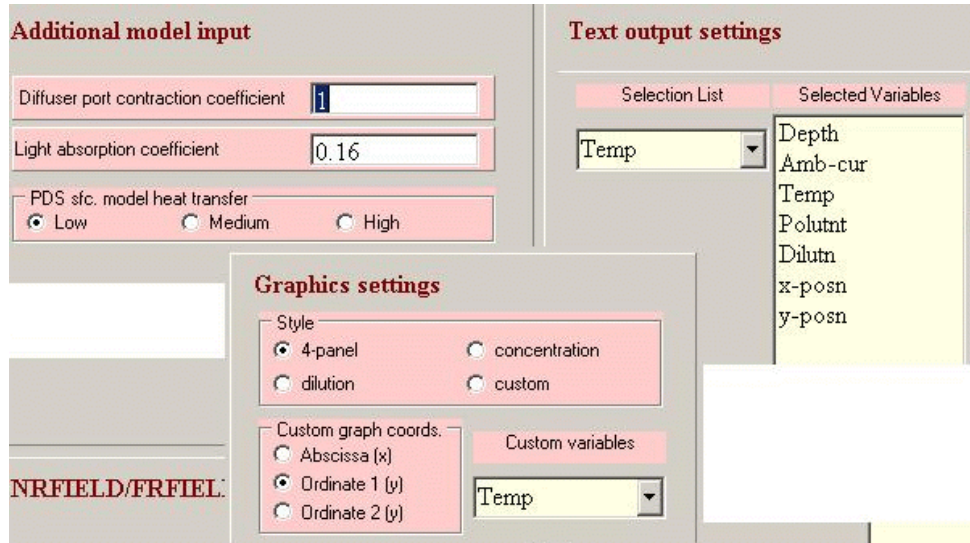**Figure 6.22** A foreshortened diffuser tab showing *A PDS run* project input.

Figure 6.23 A distorted version of the *Special Settings* tab showing relevant PDS project settings.

PDS output

     PDS may be runs from the *Models* menu or by pressing <Ctrl-P>. Depending on system configuration, the DOS window may have to be closed when PDS is finished running. Figure 6.24 shows the *4-panel* graphic display.
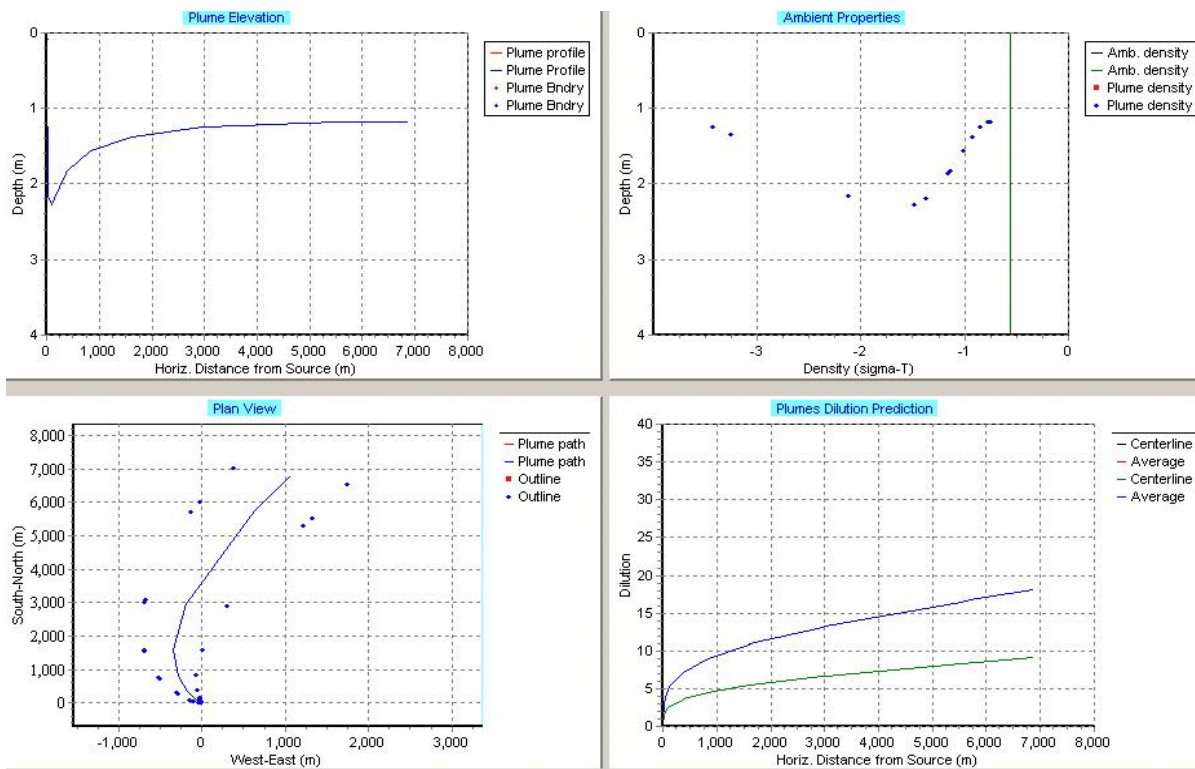
Figure 6.24 A foreshortened diffuser tab showing *A PDS run* project input.

The most relevant project output is the custom graphic shown in Figure 6.25. Based on it, the 2.0 degree excess temperature (15 C in this case) on the plume centerline is about 3400 m from shore. (Surface plumes dilute much slowly than submerged plumes). VP output does not give the areas within isotherms but the original PDS output does. This can be found by using a word processing package or text editor to view the PDS.out file in the VP default directory; it lists a much more detailed output than the VP text output given below. At the bottom of the file, you will see a table giving surface areas within selected isotherms. For this example, you will see the 2.0 degree surface isotherm encloses an area of about 4,000,000 m^2.
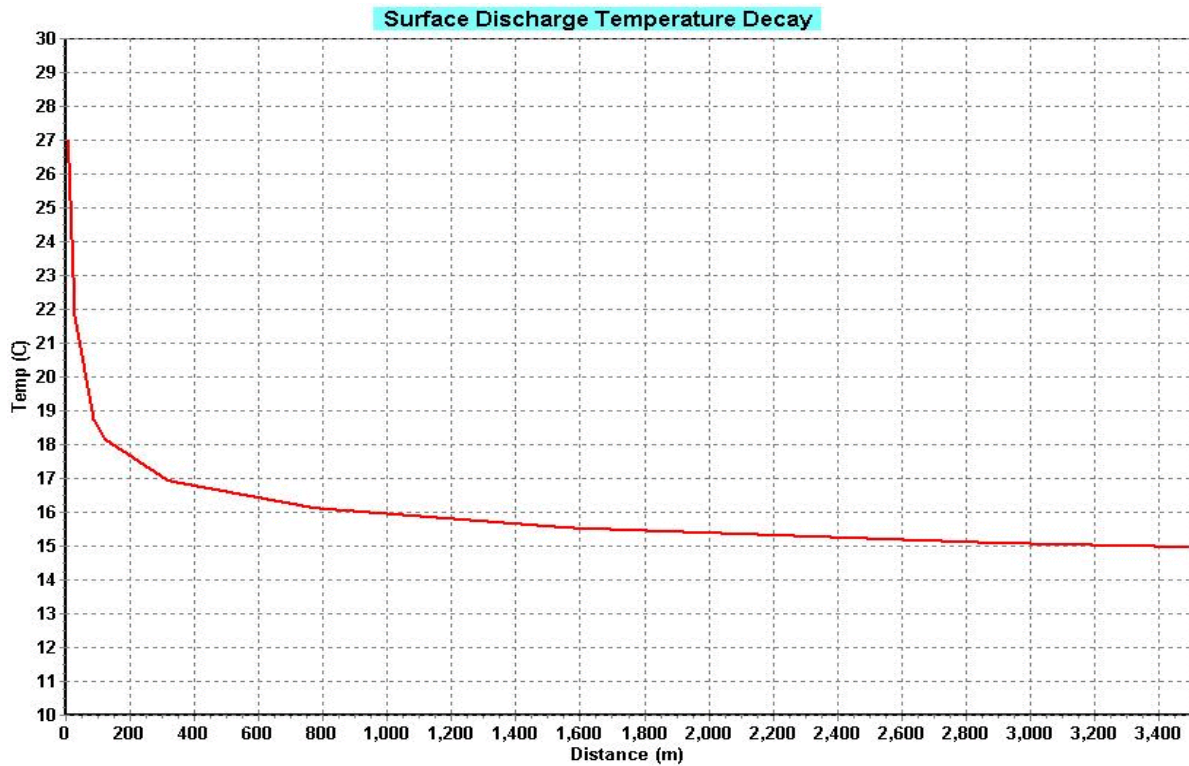
Figure 6.25 Temperature vs. distance for *A PDS run*. For one-time purposes, many of the graphics display features have been modified for greater readability.

```
/ PDS surface discharge model
Case 1; ambient file C:\Plumes\A PDS run.000.db; Diffuser table record 1:
---------------------------------
Cnduit wCnduit d H-angle     ROI Ttl-flo Eff-sal   Temp
   (ft)     (ft)  (deg)      (m)   (MGD)  (psu)     (C)
   30.0      4.0   90.0   8000.0   230.0 1.00E-3    27.0
      Amb-cur    Temp Polutnt  Dilutn    x-posn   y-posn
Step    (m/s)     (C)   (ppb)     ()       (m)      (m)
  1      0.05    27.0 5.000E-8    2.0     -9.44    10.56;
  2      0.05   26.35 4.762E-8    2.1    -10.14    11.34;
 15      0.05   21.79 3.135E-8    3.19   -22.05    25.84;
 28      0.05   18.74 2.049E-8    4.88   -61.63     84.7;
 33      0.05   18.17 1.845E-8    5.42    -82.3    121.0;
 93      0.05   16.97 1.420E-8    7.04   -168.5    315.4;
105      0.05   16.88 1.387E-8    7.21   -179.0    344.9;
225      0.05   16.11 1.114E-8    8.97   -286.7    782.0;
226      0.05    16.1 1.111E-8    9.0    -288.1    790.2;
367      0.05   15.54 9.200E-9   10.87   -340.1   1572.2;
368      0.05   15.54 9.183E-9   10.89   -339.9   1580.5;
536      0.05   15.07 7.599E-9   13.16   -190.6   2952.2;
542      0.05   15.06 7.559E-9   13.23   -181.2   3001.4;
788      0.05    14.6 6.057E-9   16.51    545.4   5499.4;
```

6.29

6.5   Ocean outfall problem

<u>Background</u>

This problem represents an actual technical assistance request, as it developed. Some preliminary discussions are omitted. The email explaining the needs of a mixing zone analysis included suggestions:

"A good analysis will have all the mandatory information on the diffuser and flow: port diameter, number of ports, port spacing, vertical angle, bell or sharp edged port, port depth, effluent flow, effluent salinity and temperature, and effluent pollutant conc..
"For ambient, we will need at specified depths, salinity and temperature, current speed (and direction, if available), background pollutant concentration, decay rate (if not conservative). If all isn't available, we will have to find other sources or use estimates.
"You say the applicant doesn't have much data. If they did, we could talk about time-series files and more sophisticated analysis."

In response, the following information was provided:

"Will you still be able to do the initial dilution calculation for Sadog Tase WWTP? I am also interested in determining what the dilution factor would be 49 feet from the diffuser (i.e., at the boundary of the Zone of Initial Dilution under 301(h) regulations) and 100 meters from the diffuser (per the mixing zone allowed under EPA's Ocean Discharge Criteria, 40 CFR 125.121(c)).
"I've been able to track down the following information for the Sadog Tase WWTP outfall."

<u>Problem description</u>

Data provided by the permit reviewer with comments by the analyst in italics. Note: some parameters and conditions were changed to increase the instructive value of the problem.

1. Port diameter: 6 inches. (From AS-BUILT drawing: Sewer outfall (off-shore) detail trench area and diffuser assembly, DWG. NO. SH-0S-004, SHT. NO. 4/4.)
[*English units should be reminder to change the units before inputting data.*]

2. Number of ports: 6. The last three ports are closed (with a blind flange). The end of the diffuser is also closed with a blind flange. (From AS-BUILT drawing: Sewer outfall (off-shore) detail trench area and diffuser assembly, DWG. NO. SH-0S-004, SHT. NO. 4/4.)
[*Some confusion about whether there are three open ports or six, therefore will do both analyses.*]

3. Port spacing: 19.68 feet. (From AS-BUILT drawing: Sewer outfall (off-shore) detail trench area and diffuser assembly, DWG. NO. SH-0S-004, SHT. NO. 4/4.)

4. Vertical angle: 90 degrees (?). Each port is on a "riser" which rises vertically from the diffuser pipe lying on the sea floor. The upper portion of each riser makes a 90 degree turn, so that when

viewed from the side (i.e., elevation view), diffuser ports open towards the viewer *[therefore, horizontal discharge, not 90 deg]*. The opening of each diffuser port alternates 180 degrees from the adjacent port. (From AS-BUILT drawing: Sewer outfall (off-shore) detail trench area and diffuser assembly, DWG. NO. SH-0S-004, SHT. NO. 4/4.)
[Spacing is effectively 19.68ft if all ports are open, otherwise, twice that distance is assumed. Note, for a diffuser with only three ports, end effects will be important if merging occurs.]

5. Bell or sharp edge port: The opening of the port is not a "square" shape, it is a "circle" shape. Does this mean that it is "bell" shaped? (From AS-BUILT drawing: Sewer outfall (off-shore) detail trench area and diffuser assembly, DWG. NO. SH-0S-004, SHT. NO. 4/4.)
[Note confusion about port shape; the sharp-edged port, assumed (see settings tab), could be tested by changing the contraction coefficient to 0.61.]

6. Port depth: 49 feet below sea level. (From 1995 NPDES application, Standard Form A.) *[Note, depth is the basis of the size of the mixing zone in 301(h) applications.]*

7. Effluent flow: 3.0 MGD annual average (1998); 4.8 MGD (average daily design). (From Mike Lee in fact sheet for 08/12/99 draft permit.)
[High flow is modeled.]

8. Effluent salinity: 1.7 - 19.0. (From 1992 301(h) application, pp. 45 - 46.)
[As this is an ocean outfall, the lower salinities will generally produce higher dilution. However, the very high effluent dilutions appear to be outliers. The reason for modeling 4.5psu no longer known.]

9. Effluent temperature: 20.0 - 30.0 degrees Celsius. (Estimated range from 1992 301(h) application, pp. 44 - 46.)
[Permit reviewer and analyst decided to model the range, i.e., both 20 and 30C.]

10. Effluent pollutant concentration:

  Enterococci: 3,000 #/100 mL [analyzed past holding time of 30 hours]
  Nitrate-nitrogen: 4.6 mg/L
  Total nitrogen: 2.19 mg/L
  Orthophosphate: 0.71 mg/L
  Total phosphorous: non-detect [error?]
  Unionized ammonia: 0.23 mg/L
  Copper: 8.4 ug/L (using RP approach in 1991 TSD, statistically est. as high as 62 ug/L)
  Nickel: 1.2 ug/L (using RP approach in 1991 TSD, statistically est. as high as 81 ug/L)
  Silver: 1.5 (using RP approach in 1991 TSD, statistically estimated as high as 11 ug/L)
  Zinc: 58 ug/L (using RP approach in 1991 TSD, statistically est. as high as 429 ug/L)

[Each pollutant could be the basis for another project (sub-project).]

NOTE: CUC (permittee) staff have indicated that the effluent plume surfaces most of the time. This is based on fecal coliform data collected at the surface above the outfall (i.e., they always find it at the surface above the diffuser). Based on this assumption, do we need salinity and temp. data at different depths, or is this fine for a first attempt modeling run?
[*Information on stratification is not available but would be invaluable. As it stands, all that can be done is to do a sensitivity analysis.*]

11. Ambient salinity: 30 - 36 ppt.

12. Ambient temperature: 25 - 30 degrees Celsius.

13. Current speed: Assume 0.
[*Zero current is not necessarily the most conservative assumption. Both zero and 5cm/sec currents will be modeled. The higher current case will be assumed to have a small density stratification.*]

14. Background concentration: 0.

15. Decay rate: ?
[*The pollutant modeled is Coliform. In tropical water a decay rate corresponding to a T-90 time of one hour is fairly representative. With zero current, the travel time to the mixing zone boundaries will be very long. A more realistic velocity might be 5cm/sec.*]

Comments about currents and stratification

This problem is fairly typical of the kind of information available to the permit writer. As will be seen, the available information often is not as complete as a scientist, or even, permit writer, would like. For example, currents and density stratification are two variables that strongly affect the initial dilution process, for which there frequently is limited data. After all, ambient sampling and data collection is expensive. However, one of the strengths of VP is the ability to determine how sensitive the problem may be to variations in conditions that are only partially specified.

You might try creating a project to analyze this problem on your own. The no-current assumption is quite common, often being considered to represent a worst case analysis. However, the far-field current cannot be zero (the effluent must be able to move to the mixing zone boundary). Concerning the "worst-case" assumption, when the dilutions of interest are to be specified at fixed distances, like mixing zone boundaries, the worst case may not correspond to the no-current assumption, rather, to some higher current speed. It is easy to set up VP to determine likely worst-case current speeds, as is shown subsequently.

The choice of graphing the effective dilution is actually irrelevant, since there is no background pollution.
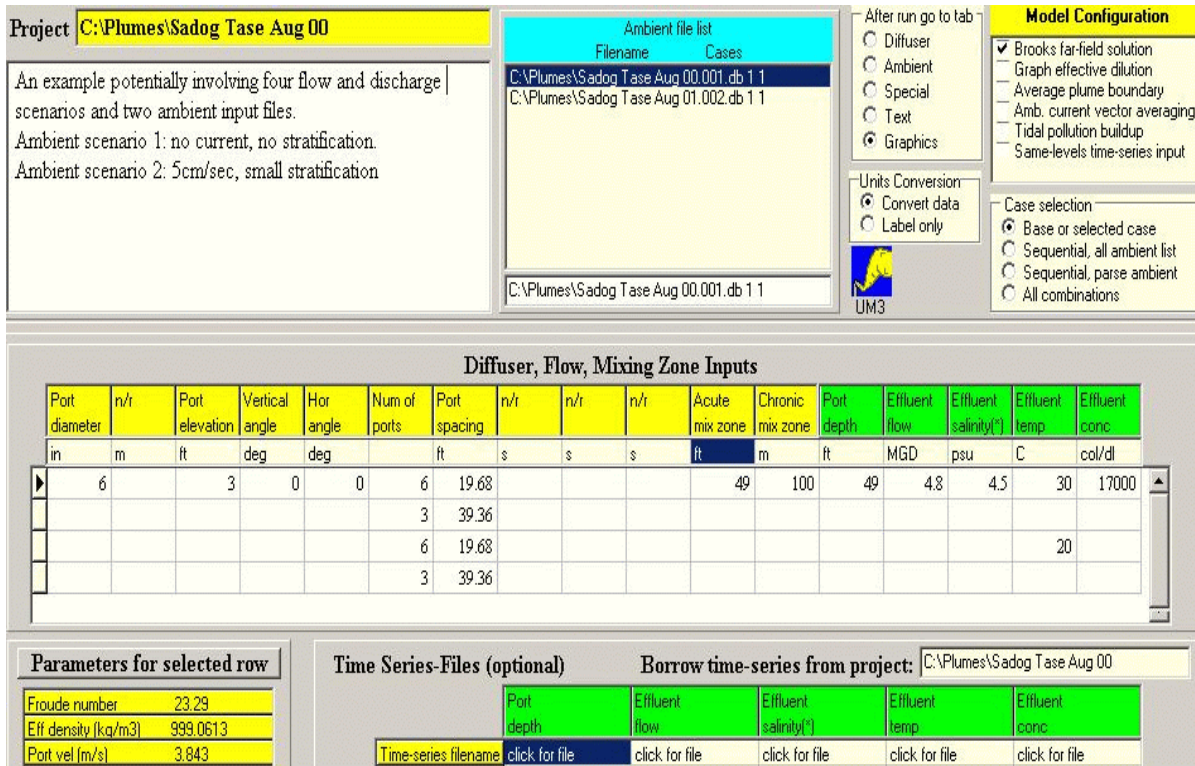
Project C:\Plumes\Sadog Tase Aug 00

An example potentially involving four flow and discharge scenarios and two ambient input files.
Ambient scenario 1: no current, no stratification.
Ambient scenario 2: 5cm/sec, small stratification

Ambient file list
Filename   Cases
C:\Plumes\Sadog Tase Aug 00.db 1 1
C:\Plumes\Sadog Tase Aug 01.002.db 1 1

C:\Plumes\Sadog Tase Aug 00.001.db 1 1

After run go to tab
- Diffuser
- Ambient
- Special
- Text
- ⦿ Graphics

Units Conversion
- ⦿ Convert data
- Label only

UM3

Model Configuration
- ✔ Brooks far-field solution
- Graph effective dilution
- Average plume boundary
- Amb. current vector averaging
- Tidal pollution buildup
- Same-levels time-series input

Case selection
- ⦿ Base or selected case
- Sequential, all ambient list
- Sequential, parse ambient
- All combinations

Diffuser, Flow, Mixing Zone Inputs

Port diameter	n/r	Port elevation	Vertical angle	Hor angle	Num of ports	Port spacing	n/r	n/r	n/r	Acute mix zone	Chronic mix zone	Port depth	Effluent flow	Effluent salinity(°)	Effluent temp	Effluent conc
in	m	ft	deg	deg		ft	s	s	s	ft	m	ft	MGD	psu	C	col/dl
6		3	0	0	6	19.68				49	100	49	4.8	4.5	30	17000
					3	39.36										
					6	19.68									20	
					3	39.36										

Parameters for selected row	
Froude number	23.29
Eff density (kg/m3)	999.0613
Port vel (m/s)	3.843

Time Series-Files (optional)   Borrow time-series from project: C:\Plumes\Sadog Tase Aug 00

	Port depth	Effluent flow	Effluent salinity(°)	Effluent temp	Effluent conc
Time-series filename	click for file	click for file	click for file	click for file	click for file

Figure 6.26 Diffuser tab for the Sadog Tase outfall.

Diffuser tab representation

The diffuser tab is shown in Figure 6.26. Notice the association of several ambient files shown in the Ambient file list. Once these files are cloned from a template file, they can be added to the list by right-clicking the *Ambient file list* window. Clicking on the name will then place the selected file on the diffuser and ambient tabs where it may be modified as appropriate.

Creating new ambient files and editing the *Ambient file list*

To create a new ambient file, from the ambient tab select the *Save ambient file as* command from the *File* menu. VP will attempt to increment the current filename. If that file is selected and already exists, you are asked whether to overwrite the existing file. Generally, it is a good idea to edit the filename before accepting it. However, if a name is overwritten, the filename will appear twice on the *Ambient file list*. The redundant name can be removed on the diffuser tab by selecting the desired file and then right clicking to reveal the pop-up menu in Figure 6.27. Choose

Edit
Add file
Add all xxx
Insert file
Insert all xxx
Remove selected files
Remove selected and following files
Help
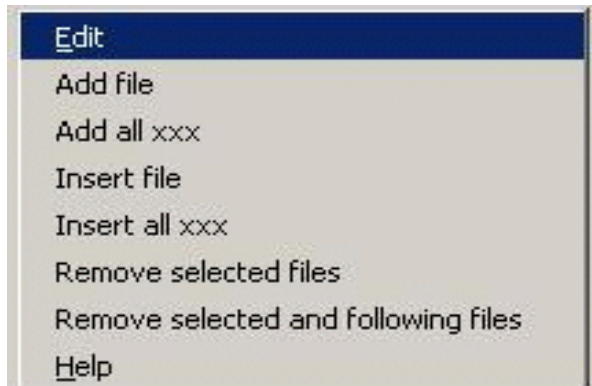
Figure 6.27 *Ambient file list* edit pop-up menu.

the *Remove selected files* command to remove the redundant filename from the list. The file is not affected, i.e., it is not deleted.

The unstratified ambient scenario

The unstratified ambient scenario is shown if Figure 6.28. For greater readability, the right-click menu has been used to increment the font size. The user may wonder why this was done above.

Ambient Inputs

Measurement depth or height	Current speed	Current direction	Ambient salinity	Ambient temperature	Background concentration	Pollutant solar radn(*)	Far-field current speed	Far-field current direct	Far-field diffusion coeff
	depth	depth	depth	depth	depth	depth	depth	depth	depth
	constant	constant	constant	constant	constant	constant	constant	constant	constant
	constant	constant	constant	constant	constant	constant	constant	constant	constant
m	m/s	deg	psu	C	ppb	ly/hr	m/s	deg	m0.67/s2
0	0	0	29	30	0	52.651	0.01	0	0.0003
50									

Figure 6.28 Unstratified ambient scenario showing increased font size using right-click menu.

6.5.1 The Mancini bacteria decay model

The actual input to the *Pollutant decay rate* column was one hour T-90 time (90 percent of the organism will die in one hour). The unit was then changed to ly/hr showing that for the given conditions of salinity, temperature, and absorption (no effect, as this is a surface value), a one hour T-90 time corresponds to a solar radiation intensity of 52.651ly/hr.

	Measurement depth or height	Pollutant decay rate(*)	Pollutant solar radn(*)	
Depth or Height		depth	depth	
Extrapolation (sfc)		constant	constant	
Extrapolation (btm)		constant	constant	
Measurement unit	m	T90hr	ly/hr	
	0	1	52.651	
	15.24	1	140.66	

Figure 6.29 Composite graphic showing a conversion from decay rate to the corresponding solar intensity.

Some points should be emphasized. The Mancini model (1978) can be used at different levels. The UM3 is model is the only model that can dynamically implement the Mancini model at depth, in other words, using absorption coefficient to modify the decay rate at depth. For the other models, VP will convert the solar insolation values to the primary units, per sec. Even VP will give different predictions depending on whether or not the Mancini model is implemented dynamically, which happens when the chosen unit is ly/hr.

6.34

Figure 6.29 shows how a conversion from T-90 time to ly/hr works in the *Pollutant decay/Pollutant solar radn* column. First inputting T-90 time values and then converting to solar intensity units (ly/hr) yields the **surface** solar intensities required to cause the desired T-90 time at depth. This shows the effect of the absorption of radiation in the water column. At the surface it only requires 52.65 ly/hr to produce a decay rate T-90 time of one hour, however, at 15.24m (50ft) it would require 140.7 ly/hr to produce the same decay rate. In reality, the T-90 time will increase with depth; bacteria in submerged effluent have a much better chance for survival.

Known bug. This exercise revealed a bug. The Mancini value is not properly converted when the water column (first column on the ambient tab) depths are in units other than meters. Thus, before using the Mancini model the water column depths should be returned to units of meters. The *Measurement depth or height* column is unique, to change its units one first double clicks in the blue units cell.

6.5.2   Assumptions, the user's domain

Information provided by the analyst includes the assumption about an open-water far-field diffusion coefficient. A value that is in widespread use is 0.000452. Here a more conservative value of 0.0003 is used.

The extrapolation (sfc) cell in the header grid above the ambient table shows that values above and below the indicated depths will be held constant. However, if there were additional entries in any column in the first row, VP will interpolate the indicated values between the given depths, but extrapolate outside of those depths if the *constant* specification were changed to *extrapolate*.

Graphic output with pan and zoom

The graphical output for the unstratified base case is shown if Figure 6.30. To run the base case with the stratified case, one would click on the stratifed ambient file in the *Ambient file list*. For the plume elevation, the pan capability (right-button drag) was used to center the graphic. Other adjustments are apparent. A help button is available to provide instructions on manipulating the graphics.
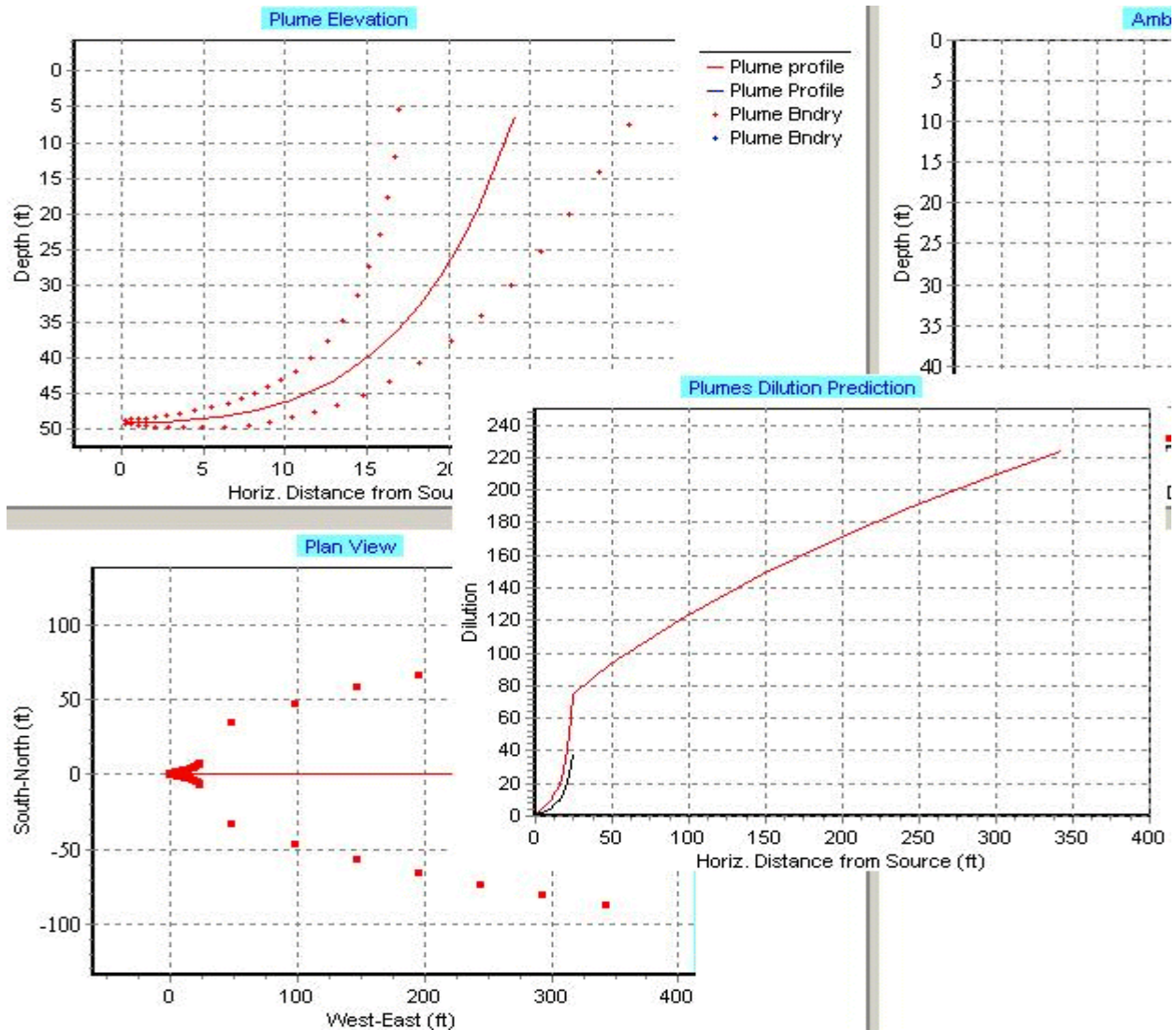
Figure 6.30 Unstratified ambient scenario output (retouched graphics).

The plan view shows the far-field algorithm predictions as the widely spaced points. Note that VP plots a single plume up to the point of merging or until the initial dilution phase is complete. If plumes do not merge before the far-field algorithm takes over, the dispersion will be considerably underestimated.

The dilution panel shows the classical difference between the initial dilution phase (rapid increase in dilution) and the far-field phase (less rapid dilution).
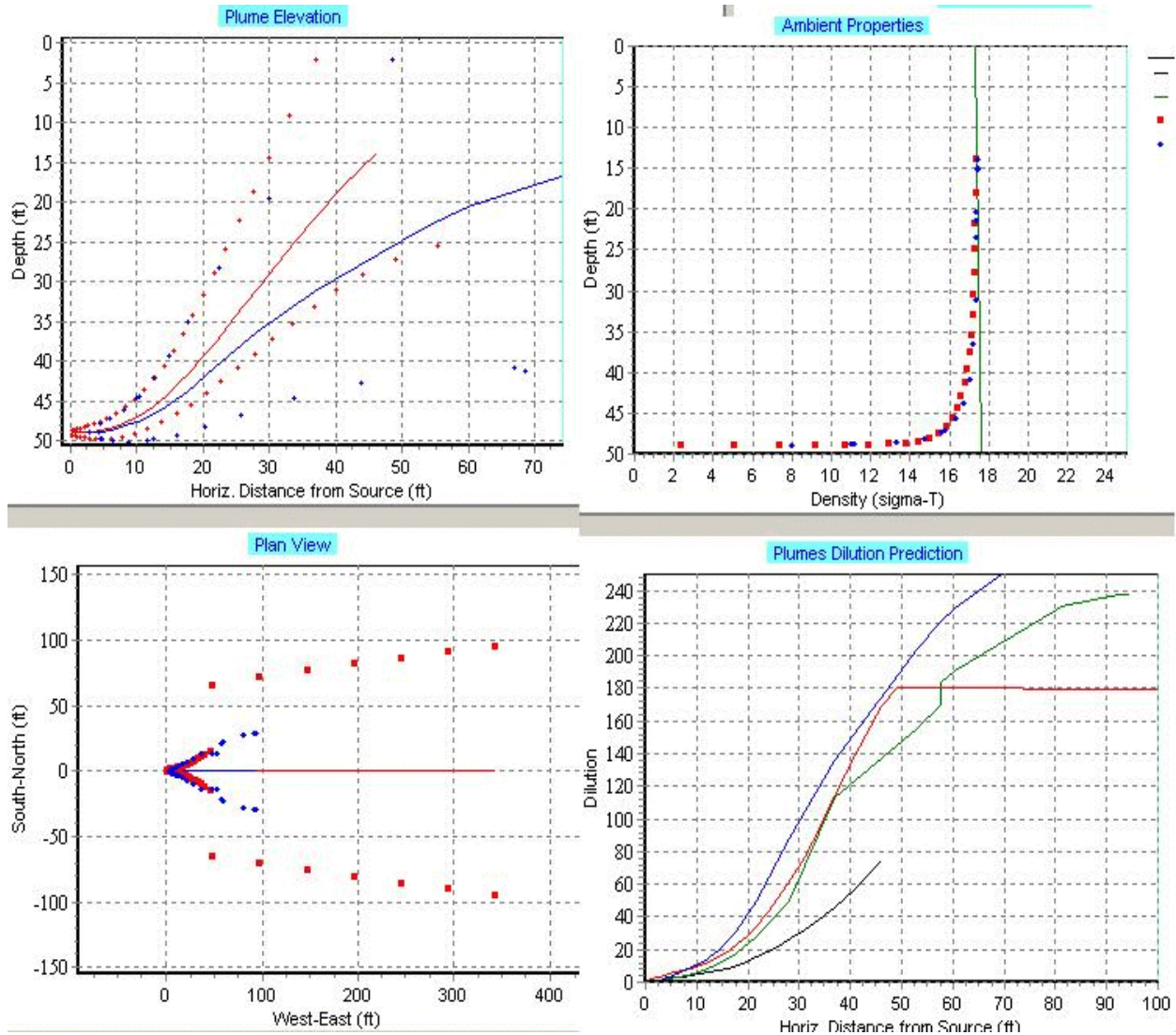
Figure 6.31 Stratified ambient scenario output (retouched graphics), UM3 (red), DKHW (blue).

Stratified case

Figure 6.30 shows UM3 and DKHW runs for the stratified scenario. The difference for the stratified scenario is the entry of 29C at 50ft depth. The stratification is plotted in the upper-right hand panel. The plumes still reach the surface but the effect of stratification is apparent. DKHW predicts considerably less rise, therefore the predicted plume has a much longer initial dilution phase. This effect is exaggerated by the fact that UM3 stops the initial dilution calculation upon any portion of the plume impacting the surface whereas DKHW continues this phase until the centerline hits the surface. The greater DKHW pollution prediction is also apparent in the dilution graphic.

6.37

A bug is apparent in the plan view. DKHW far-field prediction is based on the unmerged waste field width, whereas UM3 uses the width of the waste field (the sum of one plume diameter and the width of the diffuser). This discrepancy may be corrected on any future version of VP.

Running multiple cases

To run all cases, select the *Sequential, all ambient list* option on the *Case selection* radio group. Existing text and graphics might first be cleared with the appropriate buttons on the text and graphics tabs.
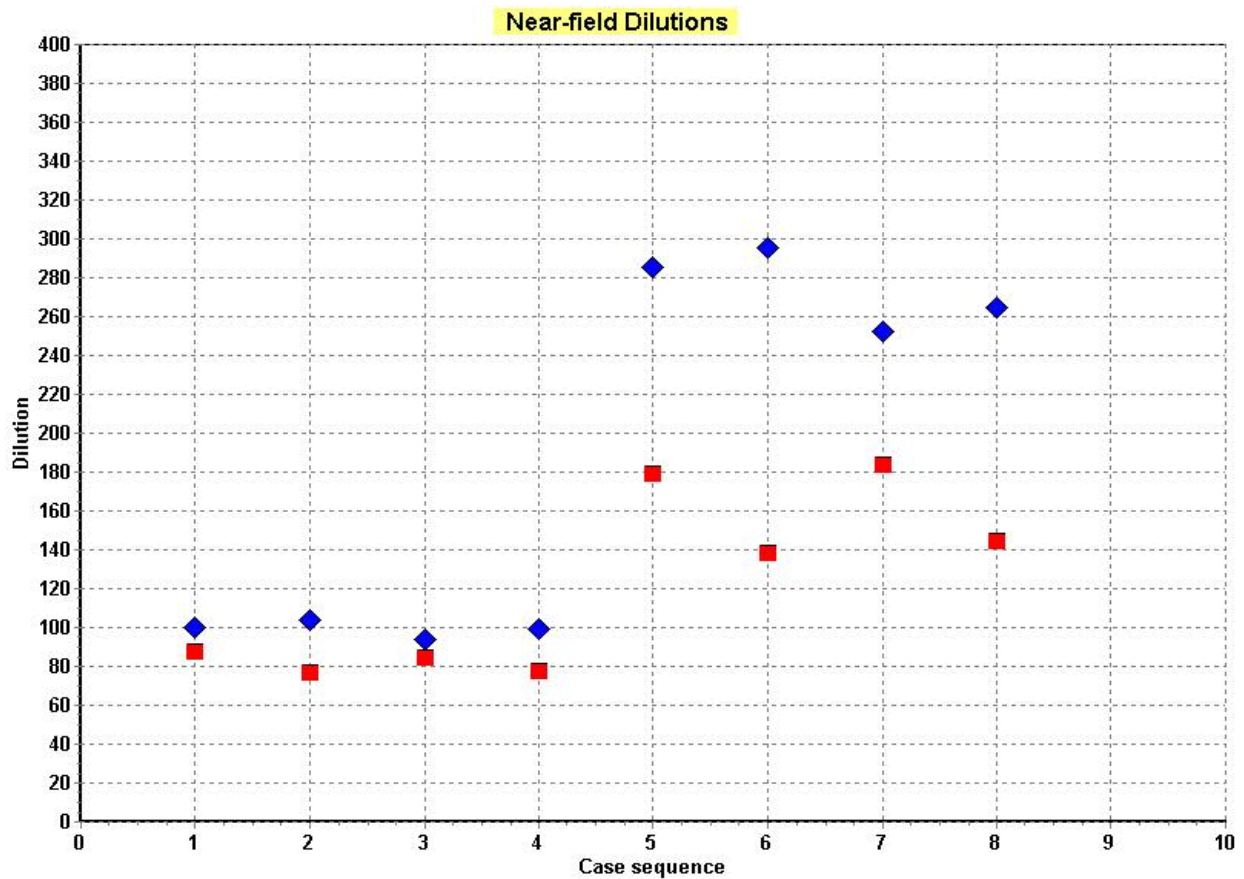
Figure 6.32 Stratified ambient scenario concentrations, UM3 (red square), DKHW (blue). Graphic customized to improve visibility of title and labels.

The dilutions for the unstratified case are lower than for the stratified in part due to the higher current and the surface impaction condition. To show cases where higher currents lead to reduced dilution, more experiments are necessary. Also, it should be borne in mind that the mixing zone predictions (graphed by VP as triangles) have been cleared from Figure 6.31.

<u>Text output</u>

Below is a partial listing of the text output for this problem. The text was edited in VP to remove excessive lines of output (indicated by ....). Also, cases 1,2, and 8 are shown for UM3, followed by a sample for DKHW (Case 1).

```
/ Windows UM3. 11/20/2000 11:42:18 PM
Case   1;   ambient   file   C:\Plumes\Sadog   Tase   Aug   00.001.db;   Diffuser   table   record   1:
---------------------------------
   P-dia  P-elev V-angle H-angle   Ports Spacing AcuteMZ ChrncMZ P-depth Ttl-flo Eff-sal   Temp Polutnt
    (in)   (ft)  (deg)  (deg)     ()     (ft)    (ft)     (m)    (ft)   (MGD)   (psu)    (C)(col/dl)
     6.0    3.0    0.0    0.0      6.0   19.68   49.0   100.0    49.0    4.8     4.5    30.0 17000.0
Froude number:      11.64
      Amb-cur   P-dia Polutnt   Dilutn   x-posn
Step   (m/s)    (in) (col/dl)     ()     (ft)
   0      0.0     6.0 17000.0     1.0      0.0;
 100      0.0    39.75  2427.7   6.876    7.117;
 200      0.0   108.3    482.6   33.81   19.51;
 248      0.0   188.3    181.0   87.45   24.81; surface,
Plumes not merged, Brooks method may be overly conservative.
Const Eddy Diffusivity.  Farfield dispersion based on wastefield width of      15.04 m
    conc  dilutn   width distnce    time
(col/dl)         (m)     (m)    (hrs)   (ppb) (ly/hr)   (m/s)(m0.67/s2)
 106.839  93.38   20.58   14.94   0.205    0.0   51.61    0.01 3.00E-4
....
 0.16261  223.6   53.09  104.5    2.694    0.0   51.61    0.01 3.00E-4
count: 7
/ Windows UM3.
Case   2;   ambient   file   C:\Plumes\Sadog   Tase   Aug   00.001.db;   Diffuser   table   record   2:
---------------------------------
   Ports Spacing
     ()    (ft)
    3.0   39.36
Froude number:      23.29
      Amb-cur   P-dia Polutnt   Dilutn   x-posn
Step   (m/s)    (in) (col/dl)     ()     (ft)
   0      0.0     6.0 17000.0     1.0      0.0;
 100      0.0    42.58  2342.9   7.133    7.62;
 200      0.0   147.8    476.8   34.24   30.64;
 241      0.0   233.2    205.9   76.95   41.79; surface,
Plumes not merged, Brooks method may be overly conservative.
Const Eddy Diffusivity.  Farfield dispersion based on wastefield width of      14.13 m
    conc  dilutn   width distnce    time
(col/dl)         (m)     (m)    (hrs)   (ppb) (ly/hr)   (m/s)(m0.67/s2)
 179.181  77.03   15.93   14.94   0.061    0.0   51.61    0.01 3.00E-4
....
 0.25748  195.6   49.58  104.5    2.55     0.0   51.61    0.01 3.00E-4


....


/ Windows UM3.
Case   8;   ambient   file   C:\Plumes\Sadog   Tase   Aug   01.002.db;   Diffuser   table   record   4:
---------------------------------
   Ports Spacing   Temp
     ()    (ft)    (C)
    3.0   39.36   20.0
Froude number:      24.95
      Amb-cur   P-dia Polutnt   Dilutn   x-posn
Step   (m/s)    (in) (col/dl)     ()     (ft)
   0      0.08    6.0 17000.0     1.0      0.0;
 100      0.08   40.21  2343.2   7.146    7.446;
 200      0.08  192.0    314.7   51.68   41.12;
 218      0.08  245.0    217.9   73.81   49.09; acute zone,
 252      0.08  395.8    107.7   144.7   67.32; trap level, surface,
Plumes not merged, Brooks method may be overly conservative.
Const Eddy Diffusivity.  Farfield dispersion based on wastefield width of      34.05 m
    conc  dilutn   width distnce    time
(col/dl)         (m)     (m)    (hrs)(col/dl) (ly/hr)   (m/s)(m0.67/s2)
 95.5343  144.8   36.16   29.87  0.0519    0.0   52.7     0.05 3.00E-4
 32.9124  162.0   49.94  104.5   0.467     0.0   52.7     0.05 3.00E-4
count: 6
```

/ DKHW

```
Case   1;   ambient   file   C:\Plumes\Sadog   Tase   Aug   00.001.db;   Diffuser   table   record   1:
---------------------------------
    P-dia  P-elev V-angle H-angle   Ports Spacing AcuteMZ ChrncMZ P-depth Ttl-flo Eff-sal   Temp Polutnt
     (in)   (ft)   (deg)   (deg)      ()    (ft)    (ft)     (m)    (ft)   (MGD)   (psu)    (C)(col/dl)
      6.0    3.0     0.0     0.0      6.0   19.68    49.0   100.0    49.0     4.8     4.5    30.0 17000.0
Froude number:       11.64
       Amb-cur   P-dia  Polutnt   Dilutn   x-posn
Step    (m/s)    (in)  (col/dl)     ()     (ft)
   0     0.0      6.0  17000.0      1.0      0.0;
   2     0.08    16.42  8812.9     1.929    2.908;
  26     0.08    24.92  5800.1     2.931    3.825;
  27     0.08    25.67  5632.9     3.018    3.905;
  39     0.08    39.45  3643.4     4.666    5.406;
  48     0.08    60.08  2329.1     7.299    7.714;
  49     0.08    62.68  2220.8     7.655    8.011;
  53     0.08    74.72  1795.7     9.467    9.435;
  54     0.08    79.13  1665.9    10.21     9.969;
  56     0.08    87.4   1449.0    11.73    10.99;
  58     0.08    95.04  1276.3    13.32    11.92;
  60     0.08   102.2   1135.2    14.97    12.77;
  63     0.08   115.6    919.1    18.5     14.26;
  64     0.08   122.0    834.6    20.37    14.92;
  68     0.08   147.5    594.6    28.59    17.08;
  69     0.08   153.8    551.4    30.83    17.52;
  74     0.08   198.3    352.1    48.28    20.01;
  79     0.08   254.7    223.7    76.0     22.39; merging,
  83     0.08   288.8    170.5    99.7     24.06; local maximum rise or fall,
Plumes not merged, Brooks method may be overly conservative.
Const Eddy Diffusivity.  Farfield dispersion based on wastefield width of       31.33 m
     conc  dilutn   width distnce    time
  (col/dl)            (m)    (m)    (hrs)(col/dl) (ly/hr)   (m/s)(m0.67/s2)
  154.586   99.78   33.01   14.94  0.0422    0.0   52.66    0.05 3.00E-4
   42.065  116.5    48.62  104.5   0.54      0.0   52.66    0.05 3.00E-4
```

6.40

7 Model Theory

7.1 Visual Plumes

The text in this section is presently limited. It is planned for expansion.

7.1.1 Bacteria models

The traditional implementation of pathogen decay in VP is a simple first order, or exponential, decay. However, four-stressor (salinity, temperature, solar insolation, and water column absorption) Mancini bacteria model (1978) is implemented in VP and described in limited detail in Section 6.5.1. Further information may be found in Keyes (1999).

7.1.2 Tidal buildup capability

The tidal pollution background buildup capability is described in some detail in Section 6.2. Future additions to this capability to include other main tributaries (or sources) to the tidal channel are planned.

7.1.3 Nascent density

The simplest analogy to the nascent density effect in plumes is pond overturning in winter. It is well known that the equation of state for water is non-linear. Unlike most substances that contract continuously upon cooling, fresh water expands before freezing below about four degrees Celsius. A pond cooling warming slowly in late winter may overturn because, as the surface water warms above 0C, it becomes denser than the bottom water with the result that the pond will suddenly "overturn," with denser, but warmer, surface water replacing the colder, but less dense, bottom water. This mixing will typically be accompanied by increased turbidity.

A similar principle can have profound effects on thermal plumes (Frick and Winiarski, 1978). Consider a fresh water thermal discharge at 60C to a uniformly cooled lake at 0C. The temperature in the plume will cool continuously to zero as it mixes in lake water (see Figure 7.1). The density will increase correspondingly, and the plume will begin to rise, until the average plume temperature approaches approximately 8C. However, below that temperature the plume's density will be greater than the ambient density at 0C and the plume will first decelerate its upward motion, then reverse its upward velocity, and finally sink to the bottom.

For a long time it was believed that the nascent density effect was potentially important only in cold, fresh receiving water. Many discharges, like the sewage effluent in Figure 7.1, behave in the traditional way. However, some DOS PLUMES users found other circumstances in which the
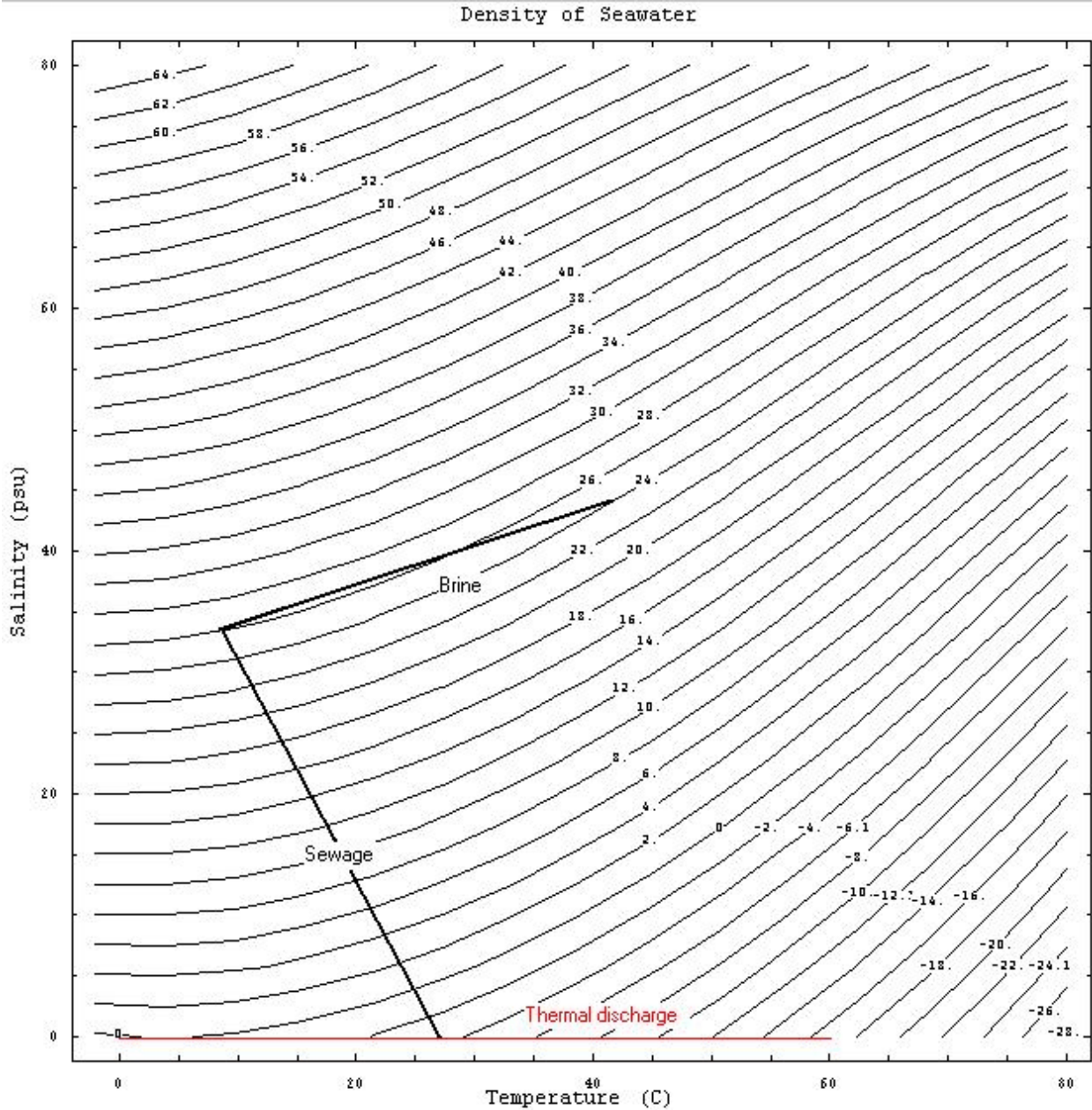
Figure 7.1 Density diagram for fresh and seawater. Three mixing lines are shown for a warm brine, a typical sewage effluent, and a thermal discharge.

nascent density effect is important. Figure 7.1 shows the mixing line for a brine, perhaps derived from a water desalination process, discharged to ocean water. It is discharged at about 42C and 43psu. As it mixes with seawater it first becomes denser, but, at below about 28C it also becomes denser. This characteristic of starting buoyant but becoming denser has led to this process being called the nascent density effect.

The fact that buoyant plumes, which ordinarily are expected to rise to the surface given no ambient density stratification, can end up at the bottom of the water column has profound

implications on regulatory decision making. It would mean that the benthic, not the pelagic, biological community would be affected by the effluent.

File 7.2 shows plume dilutions from simulated plumes all possessing the same buoyancy on discharge (effluent density in all cases is 10 sigma-T units, or 1010kg/m³), discharging to three ambient scenarios (ambient discharge level density about 15.4 sigma-T units), and otherwise identical. The eight effluent cases have temperatures ranging from 0C to 70C in 10C increments, with salinity adjusted to give the constant effluent density. Based on similarity, as expressed by sharing the same densimetric Froude number, all plumes should perform identically. However, dilutions vary widely at maximum rise.

This example illustrates the weakness of some schemes that use linear similarity theory to categorize flow, as these will lead the analyst to conclude that all of these plumes will develop identically in these three flow scenarios. On the other hand, the agreement between UM3 and DKHW is remarkably good in this range.

Additional information on the nascent density effect is available in the DOS PLUMES manual and in the PowerPoint presentation *lagdensity.ppt* file on the Visual Plumes CD (*Winiarski* subdirectory). This is rudimentary guidance planned for completion.
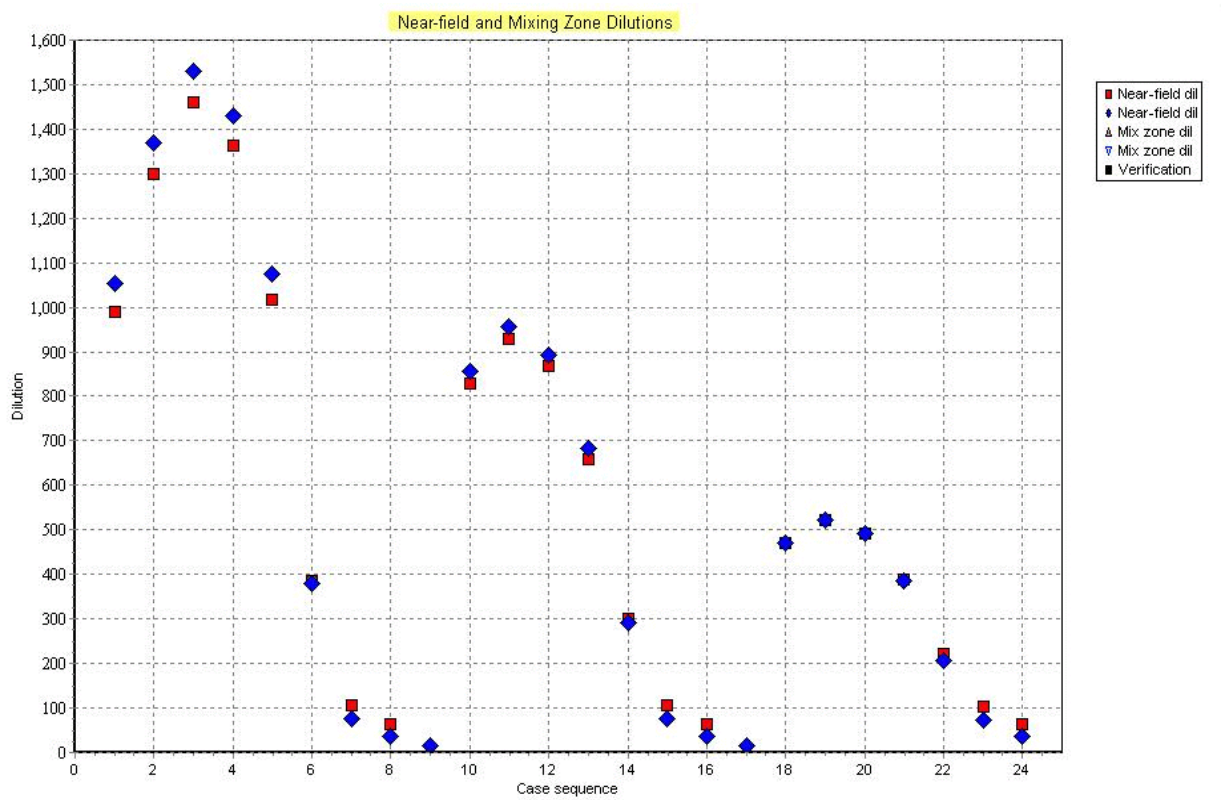
Figure 7.2 Maximum rise dilutions from eight equal density effluent flow scenarios discharged to three ambient receiving waters. Red squares correspond to UM3 output, blue diamonds to DKHW output.

7.3

7.2   UM3 theory

7.2.1   Established theory

UM3 is a Lagrangian initial dilution plume model based on the UM model described in some detail in Baumgartner, Frick, and Roberts (1994), also known as PLUMES (now DOS PLUMES to differentiate it from Visual Plumes. The complete PLUMES manual is available on the Visual Plumes CD in the *3rd Edition, DOS PLUMES* directory (*DOS Manual subdirectory*).

DOS PLUMES manual on the Visual Plumes CD

Two versions of the PLUMES manual are included on the CD, in the original WordPerfect format (*wpd* files) and in *pdf* format. With time printer and graphics technology has changed considerably and users typically have had difficulty printing their own manuals. In the CD version many of the original graphics have been scanned and saved as *jpg* files. Unfortunately, this process results in some degradation in graphic quality which is particularly noticeable in the *pdf* version of the manual.

7.2.2   Three-dimensional generalization

Single Ports: Visual Plumes UM has been modified for three dimensions. This change is possible by using vector math and by generalizing the Projected Area Entrainment (PAE) hypothesis (Winiarski and Frick, 1976; Frick, 1984). The modification adds a term to the PAE hypothesis which represents the entrainment entering the plume element from the side represented by a vector pointing at right angles to the plane formed by the instantaneous direction of motion of the plume element and the gravitational acceleration vector. Thus, for example, a port discharging eastward will form a plume element that will turn northward if there is a component of the current directed northward.

Multiple Ports: The behavior of multi-port plumes is the same as single-port plumes up to the point of merging. After merging the sideway component of entrainment is distributed over all plumes. The problem of parallel currents is solved by assuming that for angles of less that 20 degrees, measured between the plane of the individual plume element motion and to a horizontal line at right angles to the local current, there is no further reduction in the effective spacing between adjacent plumes, which would otherwise reduce to zero when currents are parallel to the orientation of diffuser pipe. This approach is suggested by the findings of Roberts (1979)

.

8   Model Availability and Performance

The text in this section is presently limited. It is planned for expansion.

VP is available on CD. Until it may be obtained through EPA's Center for Exposure Assessment Modeling, Ecosystems Research Division, Athens, GA, USA webpage: ftp://ftp.epa.gov/epa_ceam/wwwhtml/products.htm, Visual Plumes may be obtained by contacting Walter Frick, frick.walter@epa.gov.

8.1   Model verification and comparison

The Visual Plumes models have a long history of verification beginning for UM3 in 1975 (Frick and Winiarski, 1975; Frick, 1984; Baumgartner, Frick, and Roberts, 1994). The DKHW model is similarly represented by a long history (Kannberg and Davis, 1976; Davis,1999). Independent verification exercises are also available, e.g., Fergen, Huang, and Proni (1994).

One of the aims of Visual Plumes is to provide a platform for independent mixing zone models. In this vein, DKHW, NRFIELD, and UM3 are three models that are all designed to predict the same behavior of plumes in the initial dilution region. They may be seen to be competing models. By supporting competing models Visual Plumes is intended to encourage continued model innovation.

One way to further encourage model development is to provide the means to compare model predictions with experimental or field data. The *Verify* button on the graphics tab serves this purpose by enabling the user to link in and display verification files and data. The Fan-Run-16 project based on Fan (1967) illustrates one way this capability may be used (Fig. 8.1). Expanded examples of plume verification are planned for future releases of Visual Plumes.

8.2   Troubleshooting

8.2.1   Problems and solutions

Base case disappears from view

Use the scroll bar on the diffuser table to move it back into view.

File corruption

Files can become corrupted through a crash or other abnormal termination. Normally, at program termination or before opening new projects (i.e., before storing the current project files), VP inserts header information at the top of the database files that helps it interpret the data, e.g., the
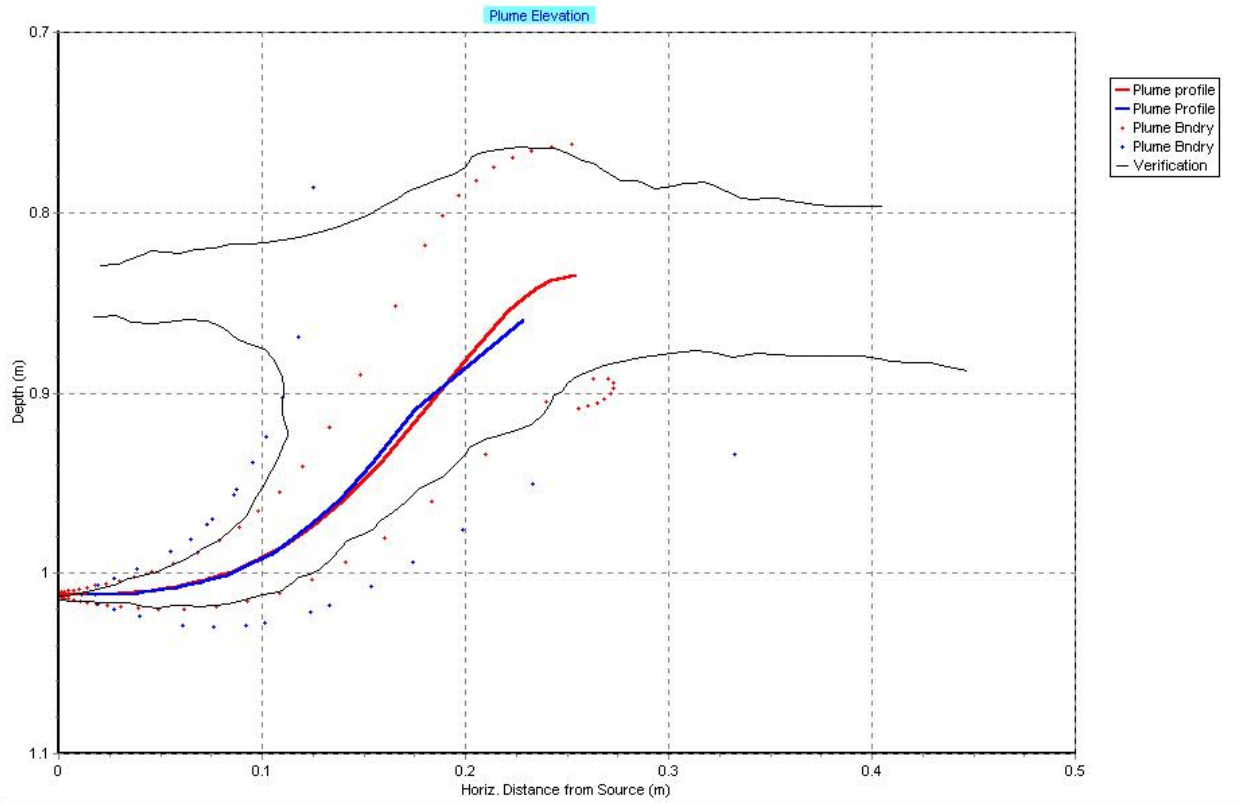
Figure 70 Comparison of plume trajectories and plume diameters: DKHW (blue shorter line) and UM3 (red line) with Fan Run 16 (Fan 1967), jagged black line.

units of each column, the next time the project is run. These header lines are absent from the file when the program terminates abnormally. When VP is next run it examines the input data to check for the presence of the header information. If it does not find it, VP alerts you to the corruption. VP will then attempt to reestablish the integrity of the file by reading the project *List*, if available. Thus, unless the user has incorrectly modified the *List* file, a full recovery will be accomplished (except for customizing details that are not saved in the *List* file). If a *List* file is not found, VP goes through a standard recovery procedure that provides default header information, including primarily MKS units (the units at the top of the list of units in each units popup) and other default data.

When a corrupted file message is issued, it is a good idea to check the units listed for all your variables. If they are not what they originally were, select the *Label only* radio button on the diffuser tab and change the units of each variable to the desired values. This will not change any of the values, just the labels so they agree with the values. Once done, be sure to change back to the default *Convert data* option.

File not found: linking time-series files after save as new name

If an old project that used time-series files is reopened and run, the *File not found* error message can appear if in the interim the time-series files have been moved or renamed. Check to be sure the time-series files exist and are in the correct project or borrow paths.

8.2

Graphics panels are empty

If no graphics appear try using the *Scale* button to rescale the axes. If there is still nothing shown, check the graphics settings on the *Special Settings* tab need to be revised. For example, the *Start case for graphs* number may be higher than the number of cases run.

Parameters for selected row panel does not change

Calculation of densimetric Froude number and effluent velocity, and, conversion to density on the *Parameters for selected row* panel are done only upon pressing the button. The other values are updated when clicking on the cells in the diffuser table or on the tab key being used. Moving up or down in the column using the arrow keys does not update the conversions.
When time-series files are linked in, only the first case is evaluated in this panel.

Using backup files to recover from a crash

Data corruption in VP is now fairly rare. An alternate way of handling corrupted files is to use the backup files VP creates upon opening each project. The backup files all have extensions that begin with an ~ so that backed-up .db files look like **.~db** files and the backed-up *List* files look like **.~ls** files. These files contain the last used values but not your most recent changes. Something like Windows Explorer can be used to manually rename the files.

Substitute backup file commands

CAUTION: The ambient file substitution command on the *Edit* menu only works for the first file in the ambient file list.

Quitting VP to force variables to be reinitialized

VP is a substantial program; it is difficulty to foresee all the problems that users will sooner or later encounter. Sometimes you may get errors you don't understand, some of which may mean that some variable has not been reinitialized. Under these circumstances, try exiting VP and re-starting it. If you think the problem is important to you, please contact one of the authors.

Models do not run when the tables show][in the row pointer

The diffuser and ambient tables must show the row pointer (▸) before the models will run.

Values do not changed when a new unit is selected

Make sure the *Convert data* option is selected on the *Units Conversion* radio group.

Wrong ambient file shown in text output

This can happen when multiple ambient files are listed on the *Ambient file list* and the *Sequential, all ambient list* option is selected on the *Case selection* radio group. Be sure that the row pointer (▸) points to the base case before beginning the run.

8.2.3   Hints

Direct access files

VP's *db* files are direct access files, changes are permanent. On exiting, when the *Pre-exit: File Editing Dialogue Window* appears all files have been closed and may be manipulated as files.

File filters

The *Pre-exit: File Editing Dialogue Window* has filters to help with the disposal of unwanted files in the *Files of type* pull-down edit box. The files displayed in the window can be changed to reflect filtering. Care should be taken when selecting the files for deletions.

Froude number, model parameters

The *Parameters for selected row* must be pushed to update the first three rows of information on this panel. For the action to work the *Base or selected case* option must be selected on the *Case seletion* radio group and the row pointer (▸) must be showing on the selected row.

Reestablish backup file by saving

VP makes backup files upon opening a project. Sometimes it is a good idea to quit VP and restart just before making exploratory changes, this way desirable changes can be reestablished by substituting the back up files if the exploratory changes do not work out.

Tidal buildup capability

Remember to adjust the coast bin when changing the segment length.

Creating VAR files

When running DOS PLUMES from VP, VP can only define the independent variables, it does not provide dependent variables (white in DOS PLUMES), the stop criterion, output table variables, and other DOS PLUMES settings.

8.2.3   Known or suspected bugs

Far-field dilution not graphed to the end of the simulation

This problem appears to be corrected by quitting VP and restarting.

Graphics panel and *Style* button disagree

This is fixed by selecting the desired *Style* button again.

Mancini model usage

Be sure that *Measurement depth or height* unit is returned to *m* before running the models. To change the unit, double click on the blue cell to put it into the change mode, then click again and change the unit. Also, remember that in general VP does a conversion to decay rate unit *s-1* (per second) when the models are run. Only UM3 uses the Mancini model in a dynamic sense. See Section 6.5.1.

9 Extended Bibliography

9.1 Selected references

Davis, L.R., 1999. Fundamentals of environmental discharge modeling. CRC Press, Baton Raton, FL.

Fergen, R.E., H. Huang, and J.R. Proni, 1994. Comparison of SEFLOE II field initial dilution data with two EPA models: UM and CORMIX. WEFTEC '94, Chicago, IL.

Frick, W.E., D.J. Baumgartner, L.R. Davis, W.S. Lung, and P.J.W. Roberts, 2000. Modeling consistency, model quality, and fostering continued improvement. Proceedings of Marine Waste Water Discharges 2000. Genoa, Italy, 28 Nov to 1 Dec, 2000.

Frick, W., A. Sigleo, S. Emerman, and J. Keyes, 1999. An innovative theory of turbulence. A Power-Point presentation forming the basis for a presentation to the Estuarine Research Federation Meeting, New Orleans, LA, September 1999. This slide show gives the user a broad understanding of turbulence and its relationship to entrainment; in file: WF99436b.ppt.

Keyes, J., W. Frick, and A. Dufour, 1999. "A Case Study Using the EPA's Water Quality Modeling System, the Windows Interface for Simulating Plumes (WISP)." Hydrology Days (1999) conference, Colorado State Univ., Ft. Collins, CO.
This work presents a tour of VP capabilities, including an application of the time-series capability and the Mancini bacterial decay model. In file: paper.wpd,

Keyes, J., 1999. A case study using the EPA's water quality modeling system, the Windows Interface for Simulating Plumes. A Power-Point presentation expanding on the materials covered in Keyes et al., 1999 (above); in file: Presentation2.ppt.

Mancini, J.L., 1978. Numerical estimates of coliform mortality rates under various conditions. *Journal of the Water Pollution Control Federation*, Nov 1978, pp. 2477-2484.

Massachusetts Water Resources Authority (MWRA), 2000. What about the new outfall?
http://www.mwra.state.ma.us/harbor/html/outfall_update.htm

Roberts, P.J.W., 2000. Experiments providing information on plume peak-to-mean ratios. Personal communication.

9.2 Extended bibliography

The bibliography found in the DOS PLUMES manual is reproduced here.

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. EPA/600/3-91/073, 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., W. Frick, P. Roberts. 1994. Dilution Models for Effluent Discharges (3rd Ed). EPA/600/R-94/086, U.S. Environmental Protection Agency, Pacific Ecosystems Branch, Newport, Oregon.

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).

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).

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.

Brater, E.F. and H.W. King, 1976. Handbook of hydraulics for the solutions of hydraulic engineering problems, Sixth Edition. McGraw-Hill, NY.

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).

Cheung, V., 1991. Mixing of a round buoyant jet in a current. Ph.D. Thesis, Dept. of Civil and Structural Engineering, University of Hong Kong, Hong Kong.

Davis, L.R., 1999. Fundamentals of Environmental Discharge Modeling. CRC Press, Boca Raton, FL.

Davis, L.R. and E. Hsiao, 1991. An experimental/analytical investigation of buoyant jets in shallow water. Oregon State University, Corvallis OR.

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., D.J. Baumgartner, and C.G. Fox, 1994. Improved prediction of bending plumes. Journal of Hydraulic Research, Vol. 32, No. 6, pp. 935-950.

Frick, W., C. Sproul, and D. Stuart. 1997. Bacterial Impacts of Ocean Outfalls: Legal Challenges. Journal of Environmental Engineering, Vol. 123, No. 2, February, 1997.

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., 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., 1984. Non-empirical closure of the plume equations. Atmospheric Environment, Vol. 18, No. 4, pp. 653-662.

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. 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. and L.D. Winiarski, 1975. Comments on "The rise of moist buoyant plumes". J. of Applied Meteorology, Vol. 14, p. 421.

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.

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.

Jirka, G., 1992. Review comments of "Dilution models for effluent discharges (draft)."

Jirka, G. and S.W. Hinton, 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.

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.

Koestler, A., 1964. The act of creation. Macmillan Company, New York, NY.

Lee, J.H.W., 1992. Private communication. Letter of 24 Nov 1992.

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.

Lee, J.H.W., Y.K. Cheung, and V. Cheung, 1987. Mathematical modelling of a round buoyant jet in a current: an assessment. Proceedings of International Symposium on River Pollution Control and Management, Shanghai, China, Oct 1987.

Ludwig, R. 1988. Environmental Impact Assessment: Siting and Design of Submarine Outfalls. EIA Guidance Document (1988). MARC Report Number 43.

Mancini, J., 1978. Numerical estimates of coliform mortality rates under various conditions. Journal of Water Pollution Control Federation. November 1978.

Martin, J.L. and S.C. McCutcheon, 1999, Hydrodynamic and transport for water quality modeling, Lewis Publ, Boca Raton, FL

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).

National Research Council (NRC), 1984. Ocean disposal systems for sewage sludge and effluent. Washington, DC. National Academy Press, 126pp.

Okubo, A., 1962. A review of theoretical models of turbulent diffusion in the sea. Chesapeake Bay Institute, The Johns Hopkins Univ., Tech Report 30, Reference 62-20.

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., 1993. "Hydraulic Model Study for the Boston Outfall. I: Riser Configuration," To be published in 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 Plan, Sacramento. (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 ù 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 Engineering., ASCE, 110(5), pp 643-656.

Appendix: Notes and important recent changes

Revised: 6 March 2002, pre-CEAM-release revision 24 July 2001, 30 April 2001

Adapted from the Visual Plumes Readme file.

>> Abstract <<

Visual Plumes is Windows software for simulating jets and plumes and for assisting in the preparation of mixing zone analyses, TMDLs, and other water quality applications. This file contains information to help you install and run Visual Plumes, as well as product news, including descriptions of recent additions and changes, hints, caveats, and other relevant comments.

>> Table of Contents <<

>> Introduction <<

The Visual Plumes compact disk (CD) or the U.S. EPA Athens web page contains all the software and documentation necessary to implement the U.S. EPA Visual Plumes model. The Visual Plumes setup software is found in the Visual Plumes (VP) Setup folder. For convenience, the latest executable is also found in a sub-folder called Latest-executable. WordPerfect and PDF versions of the manual are found in the Visual Plumes Guidance folder.

Visual Plumes has passed peer review, i.e., it has been peer reviewed, the reconciliation process has been completed, and permission to disseminate Visual Plumes has been granted.

Events accompanying "small-science" software development make it difficult to synchronize final versions of both software and the manual. For example, an update of the context-sensitive help is not available at this writing. The passage of time has given users time to identify problem areas and to request additions and changes. Consequently, some screen depictions shown in the manual (and other guidance) are no longer identical to current renditions. The authors apologize for the problems associated with this situation. It may help to remember that only the senior author is an EPA employee and most of the other work represents donated time and effort. Users are encouraged to let EPA know the value of mixing zone modeling software and guidance so that more resources may be devoted to it in the future.

The instructional material found in the folder entitled "Mixing zone course units" is based on the California State Water Resources Control Board mixing zone course (February 2000, presented by Walter Frick and Debra Denton) and subsequent similar courses. These materials are intended as classroom visual aids, however, they will hopefully also prove useful in an unassisted context.

>> Installation <<

Visual Plumes must be installed before it will run correctly. The installation software for Visual Plumes (VP) is found in the Visual Plumes (VP) Setup folder. The filename is Setup.exe. When installing it is recommended that you adopt the default folders and settings, creating a file folder called Plumes. When done, the VP executable, Plumes.exe, will be found in that folder. The name Plumes continues the DOS tradition. The executable can be renamed, for example, you might call it VP.exe. For updates normally only Plumes.exe will need to be replaced.

It is recommended that you contact me, the senior author, for the latest compile as the web version of Plumes.exe is updated relatively infrequently. Send email to frick.walter@epa.gov

Test run Plumes.exe. If you have problems it may be necessary to first change the Windows properties of the files. That is, before you access the example files you may have to first deselect their 'read only' property: in Windows Explorer, select all the project files (the project database files with extension .db, and project list files with extension .lst) and then right click: select Properties, and click off the Read-only option.

>> Filename convention <<

Be aware of the new naming convention. For the main project file (which stores the diffuser table), it is the project name followed by .vpp.db, for example, MD-Pathogen.vpp.db (project name: MD-Pathogen, followed by project marker: .vpp, followed by database file extension: .db). There should be no periods in the project name itself. When opening a project from within VP, VP will only show the project names in the dialog box.

The ambient files follow a similar format, a project name followed by .XXX.db, where XXX is a numeric sequence like 001, 002, etc., for example, MD-Pathogen.001.db or Winter-such-and-such.010.db. Note there can be many ambient files and, internally, a project might refer to ambient files from other projects. (Spaces are not recommended in the file name.)

>> Project List file <<

Visual Plumes creates and maintains project "list" files. List files record many (but not all) customized settings. The list filename format is the project name followed by .lst, for example, MD-Pathogen.lst. (For advanced users, the time-series files follow a similar format.)

>> Support <<

After trying VP, you may wish to call me at 706/355-8319 for a short phone tutorial. If you are doing official work you are always welcome to let me know so that I can provide the latest version of VP.
Let me know if you have questions. (Email: frick.walter@epa.gov).

>> Hints <<

In the ambient tab input table: to extrapolate below a pair of ambient data points an additional line reflecting deeper depths should be appended, even if depth is the only value on the line. This makes the appropriate changes to the ambient input table on the Text tab and stratification profile on the Graphics tab. A brief troubleshooting section is found in the manual.

>> Limitations <<

Important: In the usual depth mode, a surface (zero depth, 0) line must be specified in the ambient table. Without this line the ambient array is not set up correctly.
One user reported that VP does not run DKHW (and presumably other models) when installed in the E: partition. The same problem occurs when VP is installed as a subdirectory, as for example, to Program Files.

>> Recent changes <<

6 March 2002

* 20 Feb 2002: fixed a bug that prevented VP from updating the ambient profile when running multiple NRFIELD runs.
* 7 Feb 2002: fixed a bug that prevented proper plotting of density profile on the 4-panel graphics tab. Fixed ambient-table bug affecting interpolation of far-field velocity.
* 23 Jan 2002: fixed bug that resulting in the incorrect depth being passed to NRFIELD.
* 29 Nov 2002: fixed failure to graphic initial DKHW run when running multiple cases.
* 14 Ang 2002: fixed file opening bug when running NRFIELD.
* Date uncertain: fixed far-field algorithm bug resulting in incorrect concentrations when running in the constant-eddy-diffusivity mode.

The special settings tab now offers two equations of state, the new one being a function of pressure as well as salinity and temperature (Text output settings panel). There is also a button to explicitly select the farf-field diffusivity option (Additional model input panel).

Previous changes

* Additional controls on display of the input tables on the Text tab are found on the Special settings tab. The ambient table is now echoed at run time. Display can show the tables filled in or show only those numbers explicitly entered in the tables. The latter option makes the differences between runs more apparent. This selection is not recorded in the List file.

* UM3 includes a new output criterion, the "matched energy radial vel[ocity]" message. For a plume in shallow water in which the radius becomes greater than the depth of the element, UM3 computes the depth and velocity at which the sum of the kinetic and potential energies in the plume element correspond to the total energy of flow out of an annulus (a cylinder with a vertical axis) surrounding the element. An experimental program is being developed to estimate additional dilution attained in the spreading region around plumes discharged vertical into shallow water (as, for example, large thermal discharges). For more information, call (see Support).

* VP now does a better job creating DOS PLUMES VAR files. The biggest problem revolves around translating VP's three-dimensional input into DP's 2-D input. Fidelity between the two inputs is not always easy to establish because UM and UM3 are substantially different models. However, RSB output should still be identical. New users wishing to run DOS PLUMES should consult the DOS PLUMES guidance bundled with VP.

* Corrected is a serious bug in the far-field 4/3-power-law solution.

* VP now supports three graphics series: red, blue, and green. (Also, black for verification).

* An exit-without-saving option has been added to the File menu.

* VP now maintains a list of previously accessed projects on the File menu.

* VP maintains more information in the list file, e.g., the graphics tick increment is now maintained.

>> **Earlier changes** <<

   These changes were made after the WA and OR mixing zone courses, Dec 2000, but before April 2001.

* The farfield increment is now on the Special Settings tab, use it to control the number of intermediate steps between the acute and chronic mixing zone distances.

* Input for the UM3 Taylor (aspiration) entrainment coefficient has been added to the Special Settings tab.

* The second ordinate has been enabled on the Custom-graph-coords radio control box on the Special Settings tab. This allows output to a second y-axis on the Graphics tab (custom graphic). Control over the right axis is different from the other graphics settings. Instead of double-clicking in the margins of the graphics panel, the Right-axis settings button on the custom graphic is used to reveal the control panel for this axis (see next comment).

* The Right-axis-settings button on the custom graph panel of the Graphics tab may be used to create the right axis and control the corresponding settings and values. When struck, the settings panel becomes visible. When satisfied with the settings, click the Show right axis radio button and strike the Apply settings button to show the right axis plots. The variable plotted is specified on the Special Settings tab (see previous comment).

* As part of our effort to provide a common platform for several competing models to help researchers refine their products and to make it easier to compare models both between them and with verification, a "verification" capability has been added to VP. An example is available in the files associated with the Fan-Run-16 project, including the Fan16.txt verification data file. For further information see the note in the project's memo pad when running this example.

Verification in x-y format can now be superimposed on Visual Plumes predictions on the Graphics tab. Clicking the "Verify" button gives access to ASCII input file. An example of file input data excerpted from the example Fan16.txt file is:

side view
  0.0001 1.0145
  0.0068 1.0157
  0.0149 1.0158

  0.0197 1.0161
  0.0264 1.0159
density profile
 17.3 0.0
 25.2 1.0

Blank lines will cause a space between data. The key words (side, profile, path, dilution, effdilution, concentration, and generic) will shift plotting to the corresponding graphic panel, namely, the elevation, density profile, plan view, and dilution on the four-panel graphs, and dilution, concentration, and the generic custom panels. Units should correspond to the ones chosen in Visual Plumes.

* A clear graphics button has been added to the Graphics tab to allow verification data to be cleared from the graphics panels.

* The "Thick" button on the Graphics tab increases the thickness of the solid lines when increased legibility is needed.

>> Troubleshooting <<

Sometimes users encounter problems. When those problems can be clearly linked to a cause the necessary changes are made to prevent the problem in the future. However, some file handling and initialization problems are difficult to identify or may be system specific.

Try to remember to *Exit without saving* when encountering a serious problem. This can protect important work. Then back up important project files before experimenting further.

A remedy that sometimes works is to exit VP and restart. If that still does not work the offending files can sometimes be reconditioned using the reset commands on the Edit menu or the right-click options on the diffuser and ambient data tables. If that does not work the project files should be replaced by backup files and the VPSETUP file might be erased before restarting VP. Please help us improve VP by contacting us about problems (Walter Frick, 706-355-8319 or frick.walter@epa.gov).

>> Extra (optional) files <<

Visual Plumes software includes many peripheral files that are not typically pertinent to the user. Noteworthy in this regard are many graphics files, for example, encapsulated postscript files with the EPS extension. For example, the DOS-PLUMES folder has a Original Files folder containing a file called FBF93G8.eps. This is Figure 8 from Frick, Baumgartner, and Fox, 1994, showing the difference between UM model predictions with and without correction for the negative volume anomaly (relating to the "overlap" message sometimes encountered in running DOS PLUMES). It may viewed in an application like GSview32.exe Version 3.4 (copyright 1993-2000, Ghostgum Software Pty Ltd. Files with the PLT extension are Hewlett-Packard graphical language files that may be accessed by special purpose applications like AutoCAD (copyright). Such programs have usage requirements and idiosyncracies too numerous to list.

Users may peruse these files when wishing to use graphics used in Visual Plumes. In general, the pertinent files are linked to the guidance files and have little other value.

>> Mixing zone course material: Introduction <<

The instructional materials are concentrated in PowerPoint presentations in the "Mixing zone course units" folder and are roughly listed in introductory to advanced categories by the "A1" through "A9" prefixes. For those users who do not have access to PowerPoint, a viewer is found in the "A Powerpoint Viewer" folder.

VP software and documentation is contained in the "Visual Plumes (VP) Setup" and "Visual Plumes Guidance" folders. WordPerfect and PDF versions of the draft manual may be found in the "VP Manual" folder of the "Visual Plumes Guidance" folder.

>> Mixing zone course material: Table of contents <<

MZ1 Introduction:
  mzintro-Dec 2000.ppt: a general introduction to plumes (and mixing zone analysis)
  Other material includes zipped copies of graphics.

MZ2 Fischer-et-al: (graphics reproduced courtesy of Academic Press)
  Fischer-et-al.ppt: site and environmental complexities, model uncertainty, more on siting

MZ3 JennKeyes:
  KeyesHydDays.ppt: Jennifer Keyes' Hydrology Days 1999 VP presentation
    (A good look at VP capability but many changes to VP since then)
  jenngraf.wpd: tide movement for 2 fresh water flows illustrating problem of recirculation
  paper2wf.wpd: Keyes, Frick, and Dufour paper published in Hyd. Days 1999
    proceedings

MZ4 Lagrange:
  Lagmodels-21Sep2001.ppt: Look at a big outfall, general plume behavior, data stuff,
    Lagrangian model
  Lagmodels2.ppt: model complementarity and comparison, verification, plume element,
    gravitational collapse, 3-D model
  PhilBoston outfall turret design work, experimental visualization
  Shallow Water Capability.ppt: DOE course slides illustrating very-shallow-water capability
  Liseth graphics files showing cross-diffuser attraction

MZ5 Winiarski:
  lagdensity.ppt: the very important topic of nascent density
  WinFrick.ppt: Lagrangian Plume Modeling 101. Plume element, dynamics, time stepping

MZ6 HydroQual Wu
  HydroWu.ppt: graphics from a bigger project, a good example

MZ7 Class-Notes
    VP-Tips.ppt

MZ8 Diverse-Issues
    Material from the 2001 Alaska mixing zone course; Phil Roberts verification and papers;
    material on tides

MZ9 Verification
    Verification.ppt
    ModelTestsMemo01.pdf: thermal discharge verification data

MZX Walter-etc
    Chapman.ppt: critical tide factors
    Royw.ppt: Circulation model capabilities courtesy of Roy Walters
        Such models could help determine beach impacts

>> Mixing zone course material: **Optional Folders <<**

DOS-PLUMES
    Contains DOS manual (3rd Edition) and software (background reference for VP)

Optional-Presentations
    HydrologyDays2001Parta.ppt: Part 1 of a poster presented at AGU Hydrology Days 2001,
        Colorado State Univ, 2-5 April 2001. Compressible flow and turbulence theory
        inspired by the concept of entrainment into plumes.
    HydrologyDays2001Partb.ppt: Part 2 of a poster presented at AGU Hydrology Days 2001,
        Colorado State Univ, 2-5 April 2001.
    plumemovie.ppt: excellent LIF movie courtesy of Phil Roberts
        Note: if click does not animate, avi movie files may need to be re-linked
    VisualB&P.ppt: A poster describing Visual Plumes and Visual Beach (under development),
        prepared for EPA, Athens Open House, 27 April 2001.
    Walt.ppt: more LIF courtesy of Phil Roberts

Public-domain Powerpoint Viewer
    For those who don't have it, a public domain PowerPoint viewer

>> Example: Fan-Run-16 (optional exercise) <<

    One way to get a feeling for new software is to run a prepared example. The Fan-Run-16 project
provides an informative simple example of a Visual Plumes project. To run this example, start
Visual Plumes by clicking (or double clicking) on the Plumes.exe program in Windows Explorer
(or similar platform, or the Plumes icon if the program has been put on the desktop). The first time
it is worthwhile to read the credits window notes.

If this is the first time Plumes.exe is run, VP will not yet have established a file called VPSETUP, which contains the name of the last project run in VP. If this file is absent, VP starts with the "How to proceed..." panel that helps to customize the program tabs. Based on the target model selected, VP specifies required columns and attaches the chosen model to the yellow and blue model icon. The headers on the required columns can be seen to change as different models are targeted. (This does not prevent the user from subsequently selecting other models from the Models menu, however, the input requirements can change when that is done.)

For this exercise, select the UM3 model and press the Continue button.

VP establishes a new project called "VP Plume 0" (or other number if a project by that name already exists). There are always at least two data files attached to any given project; the first is called the project file and has a ".vpp.db" extension and the second is an ambient file that will have an extension like ".001.db".

One of the hardest ideas for new users to get used to is that the "db" files are direct access files, which means that the files are updated any time a value is entered or modified. This means that a "Save" option is redundant, the files are constantly being saved. The down side is that sometimes users make changes they end up not wanting. VP does allow the re-establishment of backup files, which it creates when it opens a project. One can also exit without saving. For important files, it is highly recommended that backup files are kept in a separate folder.

This "getting started" exercise is a quick tour of the Fan-Run-16 project. To open it, choose the "Open project" command from the file menu, navigate to the folder where VP has been installed, and select and open the "Fan-Run-16.vpp.db" project file.

Take a look at the tab pages by clicking on the tabs at the top. Read the project memo on the Diffuser tab; to read all of it, click in the memo space and use the arrow keys to navigate. Note one ambient file is listed in the Ambient file list, the model icon should show "UM3" under it. The "base case" is specified for running. Input data are listed on the first line of the diffuser table in the middle of the tab. Since all the required diffuser and ambient data have been specified, pressing the "Parameters for selected row" button will calculate the (densimetric) Froude number for the case. Note that columns headed by "n/r" are not required and have not been filled with data. The optional panel at the bottom contains information for linking time-series files and may be ignored.

Notice that most components have hints on the status bar at the bottom of the screen.

A nice feature about VP is the ability to select units. On the diffuser tab, click on the Effluent Flow column unit (which should show about 0.000267MLD or million liters per day). A pop-up list appears. Clicking on one of the selections makes the conversions. The Effluent density(*) column allows conversions between salinity and density units. The * reminds the user that this is a mixed units column, essentially there is a three-way relationship between density, salinity, and temperature. See the manual for more detail.

Establish the graphics page by clicking on the Graphical Output tab. Click on the Style buttons to get a feel for the available graphics panels; the "cus" option plots variables specified on the Special Settings tab. For now, select the "4-pl" (four panel option), those are the panels for which verification data are available.

Click the Verify button at the bottom left and select the Fan16.txt verification file from the dialog window (some navigation may be necessary). The file contains x-y data points (referenced in the project memo on the diffuser tab) outlining the plume to be simulated on the Plume Elevation panel and the density stratification on the Ambient Properties panel. The plume outline should now be

visible, if not, the axes may have to be adjusted. The Help button at the upper left gives basic instructions on manipulating graphics. The Scale is often useful when scaling graphics for the first time.

It is useful to bear in mind that many settings were previously customized when this project was created. VP keeps settings information in the project "list" file, in this case, "Fan-Run-16.lst". If the settings are changed the more important ones (like axis scales) are updated and maintained in this file.

Fan Run 16 is a buoyant plume discharged to stagnant, stratified ambient. Its entry angle into the trapping level (level of neutral buoyancy) causes some of the plume material to penetrate against the prevailing motion of the plume, which is generally from left to right. Note that depth does not correspond to the actual depth in the original experiment, in fact, often plume experiments are inverted. However, the density stratification is faithfully expressed.

Now run UM3 by selecting it from the Models menu or by finding the model icon and clicking on it or by pressing the <control>u hot key. The predicted plume using the inputted and default settings appears in red. Notice that UM3 predicts variables in addition to those that have corresponding verification data available. Also, notice the "diln" and "con" are not very interesting because the plot only has a single value. These two "summary" graphic panels are intended for showing the end results of multiple runs.

Go to the diffuser tab and check the Average plume boundary option on the Model Configuration checklist and run UM3 again. VP now plots an internal plume boundary at which the concentration matches the average plume element concentration (based on the 3/2 power profile simulation). This helps illustrate the fact that concentration tends to be greatest at the center of the plume and "feathers" out to a value that matches the ambient. At the plume boundary it is difficult to distinguish plume from ambient.

Select the Text Output tab and examine it. Notice that the trapping level is at 89.2cm depth. The average dilution at maximum rise is 57.8. To add or remove output variables, use the "Text output settings" panel on the Special Settings tab. For example, the Time variable might be added by pressing the down-arrow symbol on the Selection List window. The identifier "Time" will then appear on the Selected Variables list. Click on the word "Time" to add it to the list. Variables may be removed from the list by toggling the selection again. This is the time for the plume element to reach the specified point and factors into the decay of some pollutants, like bacteria.

Return to the graphics tab and select Series 2 (blue). VP automatically goes to the diffuser tab, assuming some input changes are coming or that another case is to be selected to be run. Notice, if there were multiple cases (like there are in the Astoria project) the solid right-pointing arrow will indicate the selected case. Simply clicking on a line of data will select the case.

However, right now move to the Special Settings tab and modify the value of the aspiration coefficient to 0.12. UM3 has only one tuned coefficient and this is it. Generally, a value of 0.1 is recommended. Note that the status line indicates this value is not updated in the project list file. On exiting the project the new value will revert back to 0.1.

Run UM3 again. Note the differences in rise and plume position. To view the outer boundary, the Average plume boundary option on the diffuser tab must be turned off. To avoid clutter, the blue inner plume symbols might be removed by pressing the Clear 2b button (note that the status bar explains what is cleared by this button). Now run UM3 again.

This kind of exercise for many runs might be done to optimize the aspiration coefficient. In this case it appears that the higher entrainment coefficient possibly better simulates the plume trajectory. The 0.1 coefficient has the advantage of usually (but not necessarily always) giving a more conservative dilution estimate (see lower right dilution panel).

Much more could be done and deduced from this exercise. For example, not much entrainment occurs beyond the trapping level where the plume element enters the spreading layer. This is quite different from the DOS Plumes UM prediction because UM3 addresses the "overlap" problem described in the DOS Plumes manual. The user could try running the other models (except PDS, which is a surface discharge model, and NRFIELD, which is a multi-port model).

Further experimentation is encouraged. Remember that if a backup folder of the files has not been made it may be difficult to remember the original contents of the project. A backup folder will help avoid re-installing VP to recover the original files.

To terminate this session, press the close button at the upper right corner or select Exit from the File menu. Remember, your diffuser and ambient input data changes have been saved all along.

For polished work the graphics can be substantially customized. Double click in a graphics margin to bring up the customizing dialog window for each graphic. Axis, titles, fonts, and more can all be changed, however, remember only the more important settings are maintained in the project list file.

The To File button can be used to make bitmap copies of the graphics panels (under the same project name). They can then be further processed in programs like Paint. The 4-panel graphics are first enlarged before they are saved and are considerably higher quality than the VP graphics.

The plume manual provides a step-by-step tutorial on developing VP project files.

>> Another optional exercise <<

The "nonlinear-vs-linear" project is interesting in that it illustrates multiple runs and also the importance of the non-linear equation of state on plume behavior. Linear theory holds that all three basic runs contained in the file (one for each ambient profile, or file on the ambient list) would give the same prediction for all eight plume temperatures on the diffuser tab. This is because the cases have been set up to have the same densimetric Froude number. (Test this by establishing a case on the diffuser table by clicking on a line, followed by pressing the Parameters for selected case button in the lower left panel. The Base or selected case on the Case selection radio button box must be pressed to use the Froude number button. When satisfied, select the "Sequential, all ambient list" option again.)

It is recommended that both DKHW and UM3 be run with this data (change the plotting color for the second model). Reveal the dilution graph panel by pressing the "diln" radio button on the Style box. Notice the wide range of maximum rise dilutions for the three sets of eight cases. On the other hand, both models predict similarly. This exercise illustrates the potential dangers inherent in similarity (or length scale) modeling and dimensional analysis when fluids wit non-linear equations of state are being considered.

The three ambient input files may be viewed by clicking on the corresponding file names on the ambient file list on either the diffuser or ambient tabs. These files are manipulated through pop up

menus that appear upon right clicking on the ambient file list on the diffuser tab. More detail is available in the manual.

>> References <<

Frick, WE, DJ Baumgartner, and CG Fox, 1994. Improved prediction of bending plumes. Journal of Hydraulic Research, Vol 32, No 6

Index