COUPLED NEAR AND FAR FIELD THERMAL PLUME ANALYSIS USING
FINITE ELEMENT TECHNIQUES

by

John T. Kaufman and
E. Eric Adams

Energy Laboratory Report No. MIT-EL 81-036
October 1981
COUPLED NEAR AND FAR FIELD THERMAL PLUME ANALYSIS

USING FINITE ELEMENT TECHNIQUES

by

JOHN TAYLOR KAUFMAN

and

E. ERIC ADAMS

Energy Laboratory

and

R.M. Parsons Laboratory
for
Water Resources and Hydrodynamics

Department of Civil Engineering

Massachusetts Institute of Technology

Cambridge, Massachusetts 02139

Sponsored by

Northeast Utilities Service Company

and

New England Power Company

under

M.I.T. Energy Laboratory Electric Utility Program

Energy Laboratory Report No. MIT-EL 81-036

October, 1981
The use of the open cycle cooling process for thermal power plants requires significant effluent discharges into aquatic environments. Both engineering and environmental considerations require accurate prediction of resulting temperature distribution in the receiving waters. Most predictive models have looked at one of two distinct regions of the discharge—the near or the far field—to the neglect of the other.

A methodology is developed in this work to combine the attributes of both near and far field models. A finite element far field code is utilized which calculates both the circulation and heat distribution over a large area of the domain. From the far field coarse grid, a semi-circular area is removed which corresponds to the near field region of the discharge. At the new edge of the domain, which represents the near–far field boundary, mass flux and temperature boundary conditions are specified which simulate both the discharge into and entrainment out of the domain resulting from the surface discharge jet.

Initial verification and testing of the model's characteristics is carried out in a hypothetical idealized domain. A more realistic verification is done at two prototype sites by comparing calculated results to previously acquired field data. The two sites are Millstone Nuclear Power Station (on Long Island Sound near Waterford, Connecticut) operating with two units and Brayton Point Generating Station (in Mt. Hope Bay near Somerset, Massachusetts) operating with three units on open cycle (existing conditions) and with four units on open cycle (proposed future condition). These comparisons suggest that the model can realistically describe the far
field flow patterns associated with near field mixing thus making the model a useful tool in evaluating induced circulations, the source of entrained organisms, etc. These flow patterns are a direct function of the near field entrainment and discharge distributions which are specified as model boundary conditions and are thus easily calibrated and, if necessary, modified. Comparison between measured and predicted temperatures indicates that the predicted lengths and areas of isotherms are similar to measured lengths and areas. Predicted temperatures generally indicate greater dispersion than measured temperatures thus leading to overprediction of intake recirculation. Also, because boundary conditions on the near-far boundary have been assumed constant, the shape of predicted isotherms is not as responsive to changes in ambient current direction (e.g., tidal variations) as the measurements indicate.

Future efforts should emphasize grid and program coding refinement to improve computational efficiency, use of methods to reduce numerical dispersion and incorporation of time-varying near-far field boundary conditions.
ACKNOWLEDGMENTS

The work described in this report was performed in connection with the project "Near and Far Field Interaction of Thermal Discharges" through the MIT Energy Laboratory Electric Utility Program. The authors wish to thank Northeast Utilities Service Company and New England Power Company for their generous support and utility steering committee members Mr. Ron Klattenberg, Dr. William Renfro, Dr. Jerry Holm and Mr. Andrew Aitken for their cooperation.

The research was performed by Mr. John Kaufman, a Research Assistant in the Department of Civil Engineering under the supervision of Dr. E. Eric Adams of the Department of Civil Engineering and the MIT Energy Laboratory. Substantial assistance was also provided by Mr. Peter Ostrowski, a Research Assistant in the Department of Civil Engineering and Dr. Keith Stolzenbach, Associate Professor of Civil Engineering.

Finally, thanks are due to Mrs. Carole Soloman for her assistance in preparing the report.
TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Section Title</th>
<th>Page No.</th>
</tr>
</thead>
<tbody>
<tr>
<td>TITLE PAGE</td>
<td>1</td>
</tr>
<tr>
<td>ABSTRACT</td>
<td>2</td>
</tr>
<tr>
<td>ACKNOWLEDGMENTS</td>
<td>4</td>
</tr>
<tr>
<td>TABLE OF CONTENTS</td>
<td>5</td>
</tr>
<tr>
<td>LIST OF FIGURES</td>
<td>6</td>
</tr>
<tr>
<td>LIST OF TABLES</td>
<td>10</td>
</tr>
<tr>
<td>CHAPTER I INTRODUCTION</td>
<td>11</td>
</tr>
<tr>
<td>CHAPTER II BACKGROUND</td>
<td>13</td>
</tr>
<tr>
<td>2.1 Nature of Problem</td>
<td>13</td>
</tr>
<tr>
<td>2.2 Near Field vs. Far Field Problem</td>
<td>16</td>
</tr>
<tr>
<td>2.3 Review of Models</td>
<td>18</td>
</tr>
<tr>
<td>CHAPTER III THEORETICAL DESCRIPTION</td>
<td>28</td>
</tr>
<tr>
<td>3.1 General Approach</td>
<td>28</td>
</tr>
<tr>
<td>3.2 Far Field Model</td>
<td>29</td>
</tr>
<tr>
<td>3.3 Near Field Model</td>
<td>35</td>
</tr>
<tr>
<td>3.4 General Use of the Model</td>
<td>41</td>
</tr>
<tr>
<td>CHAPTER IV IMPLEMENTATION</td>
<td>48</td>
</tr>
<tr>
<td>4.1 Idealized Domain</td>
<td>48</td>
</tr>
<tr>
<td>4.2 Millstone Plant Site</td>
<td>61</td>
</tr>
<tr>
<td>4.3 Brayton Pt. Site</td>
<td>79</td>
</tr>
<tr>
<td>CHAPTER V SUMMARY AND CONCLUSIONS</td>
<td>117</td>
</tr>
<tr>
<td>5.1 Summary</td>
<td>117</td>
</tr>
<tr>
<td>5.2 Conclusions</td>
<td>117</td>
</tr>
<tr>
<td>5.3 Suggestions for Further Research</td>
<td>119</td>
</tr>
<tr>
<td>REFERENCES</td>
<td>121</td>
</tr>
</tbody>
</table>
## LIST OF FIGURES

<table>
<thead>
<tr>
<th>No.</th>
<th>Title</th>
<th>Page No.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Figure 2-1:</td>
<td>Characteristics of a Discharge from a Surface Channel</td>
<td>14</td>
</tr>
<tr>
<td>Figure 2-2:</td>
<td>Characteristics of a Submerged Multiport Diffuser</td>
<td>15</td>
</tr>
<tr>
<td>Figure 2-3:</td>
<td>Coupling of Near and Far Fields (after Adams et al., 1975)</td>
<td>23</td>
</tr>
<tr>
<td>Figure 2-4:</td>
<td>Schematic of Parameterized Temperature Prediction Model (after Stolzenbach, 1971)</td>
<td>25</td>
</tr>
<tr>
<td>Figure 2-5:</td>
<td>Schematic of Cooling Pond Model Formulated by Watanabe et al., 1975</td>
<td>27</td>
</tr>
<tr>
<td>Figure 3-1:</td>
<td>Description of Surface Jet Parameters Used in Model Schematization</td>
<td>45</td>
</tr>
<tr>
<td>Figure 3-2:</td>
<td>Near Field Surface Jet Properties</td>
<td>46</td>
</tr>
<tr>
<td>Figure 4-1:</td>
<td>Discharge and Entrainment Flow Relationship</td>
<td>54</td>
</tr>
<tr>
<td>Figure 4-2:</td>
<td>Results from CAFE Run at Practice Domain with Constant Cross-Current and No Intake</td>
<td>56</td>
</tr>
<tr>
<td>Figure 4-3:</td>
<td>Results from CAFE Run at Practice Domain with No Cross-Current and an Intake</td>
<td>57</td>
</tr>
<tr>
<td>Figure 4-4:</td>
<td>DISPER Calculation with Practice Domain Corresponding to CAFE Run Shown in Figure 4-3</td>
<td>60</td>
</tr>
<tr>
<td>Figure 4-5:</td>
<td>Site of Millstone Nuclear Power Station</td>
<td>62</td>
</tr>
<tr>
<td>Figure 4-6:</td>
<td>Grid Used for Millstone Simulations</td>
<td>67</td>
</tr>
<tr>
<td>No.</td>
<td>Title</td>
<td>Page No.</td>
</tr>
<tr>
<td>-------</td>
<td>----------------------------------------------------------------------</td>
<td>----------</td>
</tr>
<tr>
<td>Figure 4-7:</td>
<td>CAFE Run Results at Millstone Under Stagnant Conditions</td>
<td>70</td>
</tr>
<tr>
<td>Figure 4-8:</td>
<td>CAFE Run Results at Millstone Under Maximum Flood Tidal Conditions</td>
<td>71</td>
</tr>
<tr>
<td>Figure 4-9:</td>
<td>CAFE Run Results at Millstone Under Maximum Ebb Tidal Conditions</td>
<td>72</td>
</tr>
<tr>
<td>Figure 4-10:</td>
<td>DISPER Results at Millstone Under Stagnant Conditions</td>
<td>73</td>
</tr>
<tr>
<td>Figure 4-11:</td>
<td>DISPER Results at Millstone Under Flood Tidal Conditions</td>
<td>74</td>
</tr>
<tr>
<td>Figure 4-12:</td>
<td>DISPER Results at Millstone Under Ebb Tidal Conditions</td>
<td>75</td>
</tr>
<tr>
<td>Figure 4-13:</td>
<td>Site of Brayton Point Generating Station</td>
<td>80</td>
</tr>
<tr>
<td>Figure 4-14:</td>
<td>Large, Coarse Grid Used at Brayton Point Simulation</td>
<td>83</td>
</tr>
<tr>
<td>Figure 4-15:</td>
<td>Transitional Grid Portion Between Near-Field-Far-Field Interface and Coarse Grid</td>
<td>84</td>
</tr>
<tr>
<td>Figure 4-16:</td>
<td>Large View of Mt. Hope Bay CAFE Results Under Maximum Flood Conditions for Three Unit Operation</td>
<td>90</td>
</tr>
<tr>
<td>Figure 4-17:</td>
<td>View of Region Near Discharge at Brayton Pt. Showing CAFE Results Under Maximum Flooding Conditions For Three Unit Operation</td>
<td>91</td>
</tr>
<tr>
<td>No.</td>
<td>Title</td>
<td>Page No.</td>
</tr>
<tr>
<td>-----------</td>
<td>----------------------------------------------------------------------</td>
<td>----------</td>
</tr>
<tr>
<td>Figure 4-18:</td>
<td>View of Region Near Discharge at Brayton Pt. Showing CAFE Results Under High Slack Conditions for Three Unit Operation</td>
<td>92</td>
</tr>
<tr>
<td>Figure 4-19:</td>
<td>Large View of Mt. Hope Bay CAFE Results Under Maximum Ebb Conditions for Three Unit Operation</td>
<td>93</td>
</tr>
<tr>
<td>Figure 4-20:</td>
<td>View of Region Near Discharge at Brayton Pt. Showing CAFE Results Under Maximum Ebb Conditions for Three Unit Operation</td>
<td>94</td>
</tr>
<tr>
<td>Figure 4-21:</td>
<td>View of Region Near Discharge at Brayton Pt. Showing CAFE Results Under Low Slack Conditions for Three Unit Operation</td>
<td>95</td>
</tr>
<tr>
<td>Figure 4-22:</td>
<td>Large View of Mt. Hope Bay CAFE Results Under Maximum Flood Conditions for Four Unit Operation</td>
<td>96</td>
</tr>
<tr>
<td>Figure 4-23:</td>
<td>View of Region Near Discharge at Brayton Pt. Showing CAFE Results Under Maximum Flood Conditions for Four Unit Operation</td>
<td>97</td>
</tr>
<tr>
<td>Figure 4-24:</td>
<td>View of Region Near Discharge at Brayton Pt. Showing CAFE Results Under High Slack Conditions for Four Unit Operation</td>
<td>98</td>
</tr>
<tr>
<td>Figure 4-25:</td>
<td>Large View of Mt. Hope Bay CAFE Results Under Maximum Ebb Conditions for Four Unit Operation</td>
<td>99</td>
</tr>
<tr>
<td>No.</td>
<td>Title</td>
<td>Page No.</td>
</tr>
<tr>
<td>---------</td>
<td>----------------------------------------------------------------------</td>
<td>----------</td>
</tr>
<tr>
<td>Figure 4-26:</td>
<td>View of Region Near Discharge at Brayton Pt. Showing CAFE Results Under Maximum Ebb Conditions for Four Unit Operation</td>
<td>100</td>
</tr>
<tr>
<td>Figure 4-27:</td>
<td>View of Region Near Discharge at Brayton Pt. Showing CAFE Results Under Low Slack Conditions for Four Unit Operation</td>
<td>101</td>
</tr>
<tr>
<td>Figure 4-28:</td>
<td>DISPER Results at Brayton Pt. for Three Unit Operation Under Maximum Flood Conditions</td>
<td>103</td>
</tr>
<tr>
<td>Figure 4-29:</td>
<td>DISPER Results at Brayton Pt. for Three Unit Operation Under High Slack Conditions</td>
<td>104</td>
</tr>
<tr>
<td>Figure 4-30:</td>
<td>DISPER Results at Brayton Pt. for Three Unit Operation Under Maximum Ebb Conditions</td>
<td>105</td>
</tr>
<tr>
<td>Figure 4-31:</td>
<td>DISPER Results at Brayton Pt. for Three Unit Operation Under Low Slack Conditions</td>
<td>106</td>
</tr>
<tr>
<td>Figure 4-32:</td>
<td>DISPER Results at Brayton Pt. for Four Unit Operation Under Maximum Flood Conditions</td>
<td>110</td>
</tr>
<tr>
<td>Figure 4-33:</td>
<td>DISPER Results at Brayton Pt. for Four Unit Operation Under High Slack Conditions</td>
<td>111</td>
</tr>
<tr>
<td>Figure 4-34:</td>
<td>DISPER Results at Brayton Pt. for Four Unit Operation Under Maximum Ebb Conditions</td>
<td>112</td>
</tr>
<tr>
<td>Figure 4-35:</td>
<td>DISPER Results at Brayton Pt. for Four Unit Operation Under Low Slack Conditions</td>
<td>113</td>
</tr>
</tbody>
</table>
LIST OF TABLES

<table>
<thead>
<tr>
<th>No.</th>
<th>Title</th>
<th>Page No.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Table 2-1:</td>
<td>Integrated Analysis Solutions and Parameter Definition</td>
<td>26</td>
</tr>
<tr>
<td>Table 3-1:</td>
<td>Summary of Near Field Surface Jet Properties</td>
<td>42</td>
</tr>
<tr>
<td>Table 4-1:</td>
<td>Near Field Parameters Computed for Idealized Domain Simulation</td>
<td>51</td>
</tr>
<tr>
<td>Table 4-2:</td>
<td>Near Field Parameters Computed for Millstone Simulation (Two Units)</td>
<td>65</td>
</tr>
<tr>
<td>Table 4-3:</td>
<td>Comparison of Measured Isotherm Lengths (range, ft) and Predicted Isotherm Length (value in parentheses, ft) for Various Tidal Phases at Millstone Station</td>
<td>78</td>
</tr>
<tr>
<td>Table 4-4:</td>
<td>Near Field Parameters Computed for Brayton Pt. Simulation</td>
<td>86</td>
</tr>
</tbody>
</table>
CHAPTER I
INTRODUCTION

For power plants employing once through cooling, it is necessary to accurately predict induced temperatures and flow fields in the receiving water body. Such prediction may be necessary for two general reasons: first, accurate prediction of induced temperatures and velocities is a first step in assessing potential environmental impacts. Possible impacts include entrainment at the intake of small organisms into the condenser cooling system, impingement of larger organisms onto intake screens and exposure of organisms to elevated temperatures within the discharge plume. Federal and state agencies often place specific limitations on induced temperatures and/or induced velocities out of respect for such possible impacts.

The second consideration involves design limitations on intake temperature. Power plant efficiency decreases with increasing intake temperature. In considering the design (or redesign) of intake and discharge structures, a utility must account for possible increases of temperatures at the intake, due to recirculation of the thermal plume, which might lead to a lower plant efficiency or may even require derating. Accurate thermal prediction models may, therefore, aid significantly in optimizing cooling system design.

The wide range of time and space scales characterizing the thermal structure of the water body increases the difficulty in plume modeling. A characteristic of many thermal discharge flow fields is the presence of two distinct regions of flow: the near field and the far field.
Most modeling efforts concentrate on one of these regions to the neglect of the other. This is because of major problems in dealing with the two very different scales of the two regions. The difficulty in coupling the two regions leads to very approximate means of modeling the actual physics of the problem.

This thesis describes a method which combines existing knowledge of near field induced temperature and flow disturbances with available far field numerical models. The resulting tool can be used to address these near and far field interactions. In applying this technique, emphasis will be placed on evaluating the model's ability to simulate near-far field interaction with respect to both environmental considerations and engineering design constraints.
CHAPTER II
BACKGROUND

2.1 Nature of Problem

Steam-electric power plants using once-through condenser cooling systems are often located near large water bodies. Shorelines of coastal areas, large lakes and river banks serve this purpose and are utilized widely. This study concentrates on coastal sites due to the interests of the sponsors. The general approach could easily be extended to a river application, though.

The heated effluent from a power plant is typically discharged into the aquatic environment either through a surface discharge channel or a submerged diffuser. Drawings of the two configurations are shown in Figures 2-1 and 2-2. In the surface discharge case heated water with flow rate $Q_o$ and density $\rho_o$ enters through a discharge channel with a certain geometry into receiving waters with variable depth $H$, ambient density $\rho_a$ and ambient current velocity distribution $u_a$. A submerged diffuser may be characterized by similar discharge variables except that the method of discharge would be through a number of discharge ports near the bottom of the receiving body rather than through a shoreline channel. A detailed discussion of these two discharge types may be found in Adams et al., 1979. The work done in this research has focused on surface discharges; however, the approach could be extended easily to submerged diffusers.
Figure 2-1: Characteristics of a Discharge from a Surface Channel.
Figure 2-2: Characteristics of a Submerged Multiport Diffuser.
2.2 Near Field vs. Far Field Problem

A number of techniques have been developed for predicting the excess temperature distribution induced in the receiving water by the heated discharge from power plants. Yet, new methods are desired to accurately deal with the diverse influences acting on these temperature distributions. Both natural and man-made processes affect these distributions. Any temperature prediction technique must address this problem of combined ambient and man-made thermal influences. Yet it is hard to model all of these processes at a single time in a single analysis, due to the widely varying spatial scales of the near and far fields.

The near field is defined as that region whose characteristics are dominated by the initial discharge conditions. These include the discharge velocity and corresponding momentum, and the discharge temperature rise and corresponding buoyancy. In the near field, the temperature reduction within the plume results primarily from induced mixing of the discharge with the receiving water. Large velocity gradients exist relative to those found at greater distances from the point of discharge. The length of the near field is typically of the order of 1000 ft. and the temperature rise is in the range 1-10°F above ambient after mixing.

The near field circulation induced by the cooling water discharge depends on the type of outfall structure. The discharge may be taken to be one or more mass and momentum jet(s) discharging to the ambient region. These jets can alter the natural flow field by inducing flow from the far field for entrainment along the sides of the jet. Also,
the jet itself has a large influence on the flow field. After dilution, the jet flow is typically of order 10,000 cfs which is significant!

The far field is defined as that region of the plume which is largely independent of the initial characteristics of the discharge. This region is dominated by the natural processes of water movement, dispersion, and surface heat loss. Tidal flushing and its associated dispersive effects dominate the far field temperature distribution at the type of coastal plant sites studied in this work. Surface heat loss is the ultimate sink (from the water's point of view) over the large areas represented by the far field. In rough absolute terms, the far field extends out from the area effected by the discharge jet to the end of the domain being modeled. A range of distance may be estimated as 0.5 to 10 miles.

At intermediate distances from the point of discharge the plume is still being affected by the initial buoyancy and momentum, but it is also being influenced by natural water movement and dispersive processes. Because of the distinctive properties of this transition region it is often referred to as the intermediate field. The scale of the intermediate field ranges from about 1000 ft to the order of 1 mile, depending on the discharge characteristics. The large size of this region is a major reason why a link between near and far field modeling is necessary.

Instead of combining near and far field effects, models usually concentrate primarily on either the near or the far field. Near field models (either physical, integral jet or numerical) typically assume
that the far field characteristics (receiving water temperature, velocity, etc.) are known with the consequence that the effect of the near field on the far field can't be computed. Far field models, on the other hand, tend to over simplify near field effects by considering simply an influx of mass and heat (or in some instances only an influx of heat) at the point of discharge. Near field mixing, if accounted for at all, must be treated by the assignment of artificial values of dispersion coefficient. This procedure discounts many influences of the near field on the far field. Because of the difficulty in near-far field coupling, certain aspects of interest, such as mixing zones, correct entrainment, flow fields and shoreline impacts are difficult to assess.

2.3 Review of Models

Near Field Surface Jet

Most near field surface jet analyses can be divided into three types: integral jet models, numerical models or physical (hydraulic scale) models. Examples of integral jet analyses include: Stolzenbach and Harleman, 1971; Shirazi and Davis, 1974; and Prych, 1972. In the integral approach, the 3-D equations are integrated over the jet cross-section resulting in a numerically one-dimensional formulation. That is, numerical integration is only required in the longitudinal coordinate. Jet properties such as trajectory, width, depth, centerline velocity, centerline temperature, and flow rate are thus determined as a function of longitudinal coordinate. This procedure has usually
described well the near field characteristics of the jet, particularly at sites with simple geometry, deep water and relatively high discharge Froude number. One problem inherent with these models, however, is the failure to consider re-entrainment influences on the jet and near field region from the far field. In general, these integral jet models require explicit specification of boundary conditions which represent the influence of the far field on the near field model. This is a limitation as transient effects within the far field's heat distribution aren't accounted for within the specified boundary conditions. It is not possible to know the real thermal boundary conditions for this problem due to the effects of surface heat exchange, convective mixing, land boundaries and other far field temperature influences. Another problem with these models is that their range of applicability is limited to a comparatively short distance from the discharge point where the near field processes indeed govern the flow. (A more detailed discussion on this distance may be found in Section 3.) Often, however, temperature estimates are required in the intermediate field, requiring that the near field results be extrapolated—perhaps unrealistically.

Other approaches to the near field have been developed using both physical and numerical modeling techniques. Physical modeling presents a similar problem due to the boundaries on the model. Sides (and ends) of containing tanks where the models are built are often too close to the discharging jet to simulate the open receiving waters of most coastal areas. Numerical models of the near field suffer similar pitfalls in the specification of far field influenced boundary conditions.
Far Field Models

Several models have been developed which concentrate on the transport of constituent concentration or heat through the far field (Ahn and Smith, 1972; Leimkuhler, 1974; Wang and Connor, 1975; Eraslan, 1974; Siman-Tov, 1974). These models generally share many of the same characteristics. For instance, due to the generally large and complicated nature of the far field, most models are numerical. Many have the characteristic of dealing only with a 2-D formulation to simplify the calculations. Because of the longer time scales of interest for the far field, most models are transient.

Numerical models of the far field have utilized both finite difference and finite element solution techniques. For example, to compute horizontal circulation (i.e. surface elevation $\eta$ and horizontal velocities $u$ and $v$), a finite difference method would approximate the governing differential equations of continuity and of $x$ and $y$ momentum conservation by corresponding difference equations associated with discrete grid points. The result is a set of simultaneous linear algebraic equations which is solved by matrix techniques. The finite element approach uses the Galerkin method of weighted residuals. In this method a continuous solution for $\eta$, $u$ and $v$ would be assumed by interpolating between trial values of $\eta$, $u$ and $v$ defined at nodal points. The method seeks those trial values for which the weighted residual (error) between the trial and real solution is minimized when integrated over the element.
Characteristic of the two methods is the shape of the elements used in the schematization of the domain. The finite difference method typically utilizes square elements of constant size with the nodal points at the four corners or the center. This is a constraint on modeling the domain as it is often not possible to adequately match land boundaries using square shaped elements of fixed size. Finite elements are more flexible in both size and shape. Their absolute size is restricted only by the consideration of describing the physical gradients in the system. In other words, the elements must be small enough to allow sufficient discretization of areas where large gradients in velocities or depth exist. The shape of the elements should approximate equilateral triangles, though this is not a strict requirement. The degrees of freedom in the elements' size and shape allow the irregular boundaries of domains to be modeled quite closely. It also makes possible the transition from smaller elements in areas of large gradients to larger elements in the less varying far field. More comparisons between finite difference and finite element techniques may be found in Pinder and Gray, 1977.

In general, it is difficult to model accurately the numerous processes taking place in a far field region. Specific problems concern the specification of boundary conditions, especially at open boundaries, and near the point of discharge. Inability in the latter regard means that it is difficult to represent the near field influence in these far field models. To illustrate, if a jet discharging into a receiving water body has an initial flow rate of $Q_o$ and a dilution ratio of 10,
the total amount of flow entering the far field region will be $10Q_o$, and the amount of flow leaving the far field (associated with jet entrainment) would be $9Q_o$. Most standard far field models would typically specify only the net input of $1Q_o$.

Integrated or Complete Field Models

Models which attempt to combine near and far field characteristics seem to hold the most promise in predicting heat distributions about thermal discharges. Koh and Fan, 1970, presented a coupled integral near field model with a two dimensional (longitudinal and vertical) numerical model of the far field. This type of coupling allows the near field effects to be represented as boundary conditions in the far field, but does not allow far field effects to be represented in the near field. For example, it cannot handle reversing currents where far field heat is returned to the near field.

Adams et al., 1975, modeled far field temperature distributions using the concept of relative diffusion. The far field model transported heat in the form of discrete patches which were advected, dispersed and decayed in accordance with physical far field processes. The initial size and temperature of these patches was derived from near field analysis. A major advantage of this type of model is that it can model the influence of transient ambient currents which can advect heat back into the near field (see Figure 2-3).

In the work of Stolzenbach, 1971, an integrated approach was presented which incorporated the major interaction between power plant and receiving water in essentially a three box model including power
Figure 2-3: Coupling of Near and Far Fields (after Adams et al., 1975).
plant intake, near field mixing region and far field zone. See Figure 2-4. Linkage between these boxes was described by the plant operating variables (e.g., flow rate and temperature rise) and by several parameters describing the feedback between regions (e.g., an intake recirculation coefficient, near field dilution and re-entrainment coefficient, and far field flushing and heat loss parameters).

While quite simple, such a model can represent all of the relevant interaction in complex receiving waters such as enclosed tidal embayments. The solution for intake, near field and far field temperature rises, along with parameter definition are provided in Table 2-1.

Watanabe et al., 1975, developed a two layer cooling pond model which describes both major regions, the near and far fields (see Figure 2-5). The entrance mixing zone was modeled by the Stolzenbach-Harleman surface jet model. The far field surface layer was analyzed using a finite element method formulated in terms of temperature, stream function and vorticity. The lower layer was represented by the Ryan-Harleman mathematical model for a stratified reservoir. The primary interest of this model to the present study is the manner in which Watanabe specified the discharge flow from the jet into the computational domain. He accounted for entrained flux into the sides of the jet through specified flows, \( Q_{EH} \) and \( Q_{EV} \). This additional mass was then added to the discharge into the domain at a rate \( Q_M \). These boundary concepts were utilized directly in the present model's formulation.
Figure 2-4: Schematic of Parameterized Temperature Prediction Model (after Stolzenbach, 1971).
Table 2-1: Integrated Analysis Solutions and Parameter Definition

\[ T_F = T_A + \frac{Q_p \Delta T_p}{Q_F + kA_F} \]

\[ T_N = T_F + \frac{\Delta T_p}{S(1-\beta) + \beta - \alpha} \]

\[ T_I = T_F + \frac{\alpha \Delta T_p}{S(1-\beta) + \beta - \alpha} \]

\( T_A \) = Ambient temperature of the far field region
\( T_F \) = Averaged far field temperature
\( A_F \) = Effective surface cooling area of far field
\( Q_F \) = Far field flushing flow
\( Q_p \) = Condenser water flow
\( \Delta T_p \) = Condenser temperature rise
\( T_N \) = Near field temperature
\( T_I \) = Intake temperature
\( S \) = Near field dilution, i.e. ratio of mixed flow leaving near field to discharge flow (\( S=1 \) implies no mixing)
\( \beta \) = Fraction of the entrained flow originating in the near field at temperature \( T_N \) (\( \beta=0 \) implies no re-entrainment)
\( \alpha \) = Fraction of intake water drawn from the near field. The remainder is supplied by the far field (\( \alpha=0 \) implies no recirculation)
Figure 2-5: Schematic of Cooling Pond Model Formulated by Watanabe et al., 1975.
CHAPTER 3
THEORETICAL DESCRIPTION

3.1 General Approach

The formulation of this model begins with a far field schematization of the main portion of the domain under study. Into this domain (which may represent a bay, lake or coastal area), a jet discharge enters near the power plant site. In this model, a semi-circular area is carved out around the discharge jet plume and physically removed from the schematized domain. The portion left represents the intermediate and far fields of the region. It is assumed that the near field area which was removed can be modeled by specifying (inside) boundary conditions on the far field which match those influences by the discharge jet. In these examples, the near field characteristics were determined using analytical and empirical results from other surface jet studies. More detailed discussion of these determinations will be covered in this chapter.

The model used for this work makes use of two distinct, yet interconnected numerical models designed to calculate flow and mass concentration distributions, respectively. The finite element circulation model CAFE was developed by Wang and Connor, 1975. The mass dispersion finite element model DISPER was written by Leimkuhler, 1974, with an original emphasis on sediment transport processes. The two programs were written in conjunction which made their coupled use easier.
3.2 Far Field Model

The CAFE and DISPER Models

CAFE is formulated for vertically integrated variables in the far field. Circulation patterns and surface elevation changes are calculated at individual nodal points to simulate continuous movement throughout the domain. This technique eliminates dependence on the vertical coordinate. This is justified when little variation in the variables takes place over depth.

Most concern over temperature rises in power plant receiving waters takes place in the summer months when temperature levels are at their highest. This period coincides with the greatest stratification due to surface warming, both from the ambient meteorology and the artificial thermal loading, from the plant. This justified use of the 2-D mixed layer approach of CAFE since the density structure would resemble two distinct layers. Because of the density difference between the two layers upon stratification, mixing between them is reduced.

One of the output sets from CAFE contains nodal fluxes and water depths which are used by DISPER to represent the circulation in the domain being modeled for concentration distribution. To this field ambient artificial heat inputs were added. Previous applications of DISPER involved concentration of sediment from a proposed sediment disposal (Pearce and Christodoulou, 1975), the concentration of larval fish near a power plant (Chau, 1977), and various other applications dealing with pollutant dispersion in coastal areas (Pagenkopf et al., 1976).

Leimkuhler wrote DISPER to compute vertically averaged concentra-
tions over the same prescribed depth as used in CAFE. In traditional uses of such models, the user has the option of running the models with the whole depth, fully mixed (as may be necessary in the wintertime when the whole depth is fully mixed), or with a smaller stratified top layer which is fully mixed in its depth. This is an important point as a basic limitation of a 2-D model is its inability to accurately represent the third dimension. This study's model was concerned with induced stratification on the upper surface layer. The far field plume depth was determined from near field jet properties (Eq. 3.3-7) and this value was input as the constant depth throughout the far field.

Ostrowski, 1980, in a related effort used full depths in both of the same numerical models. His work was more concerned with natural temperature distributions as contrasted with the description of the thermal plume in the intermediate field in this study. For another comparison, NUSCO, 1975, modeled circulation near Millstone using the full depth and dispersion using partial depth.

The forms of the governing equations used by CAFE are given as follows. For a complete derivation of these and related expressions, see Wang and Connor, 1975.

\[
\frac{\partial n}{\partial t} + \frac{\partial q_x}{\partial x} + \frac{\partial q_y}{\partial y} = q_I \tag{3.2-1}
\]

\[
\frac{\partial q_x}{\partial t} + \frac{\partial uq_x}{\partial x} + \frac{\partial uq_y}{\partial y} = f q_y - g(h+n) \frac{\partial n}{\partial x} + \frac{\tau_s}{\rho} \tag{3.2-2}
\]

\[
-C_f \frac{(q_x^2 + q_y^2 q) \frac{\partial q}{\partial x}}{(h+n)^2} + \frac{\partial F_{xx}}{\partial x} + \frac{\partial F_{yx}}{\partial y} \tag{3.2-2}
\]

\[
30
\]
\[
\frac{\partial q_y}{\partial t} + \frac{\partial q_x}{\partial x} + \frac{\partial q_y}{\partial y} = -q_x g(h + \eta) \frac{\partial \eta}{\partial y} + \tau^S_y \rho
\]

\[
-C_f \frac{(q^2_x + q^2_y)q_y}{(h + \eta)^2} + \frac{\partial F}{\partial x} + \frac{\partial F}{\partial y}
\]

where,

\[\eta = \eta(x,y,t)\] is the surface elevation

\[q_x = \int_{-h}^{h} u \, dz\]

\[q_y = \int_{-h}^{h} v \, dz\]

\[q_I = \text{a mass input to the system}\]

\[u,v = \text{the flow velocities in the x,y directions}\]

\[f = \text{coriolis parameter, } 2\omega_{\text{earth}} \sin \phi\]

\[\omega_{\text{earth}} = \text{angular frequency of the earth's rotation}\]

\[\phi = \text{latitude (N) of the location}\]

\[g = \text{gravitational acceleration}\]

\[h = \text{the depth of the water body at mean low water}\]

\[\tau^S = C_p \rho_{\text{AIR}} U^2_{30}\]

\[\rho_{\text{AIR}} = \text{air density [\sim 0.075 pcf]}\]

\[U_{30} = \text{wind speed 30 ft above sea surface}\]

\[C_D = (1.1 + 0.0536 U_{30}) 0.10^{-3}\]

\[\rho = \text{density of water}\]

\[C_f = g \frac{n^2}{h^{1/3}} \quad \text{(Manning's n)}\]

\[F_{xx} = E \frac{\partial q_x}{\partial x}\]
$$F_{xy} = E_{xy} \left( \frac{\partial q_x}{\partial y} + \frac{\partial q_y}{\partial x} \right)$$

$$F_{yy} = E_{yy} \frac{\partial q_y}{\partial y}$$

$E_{ij}$ = the eddy viscosity coefficient

As in the solution of all differential equations, it is necessary to specify both initial and boundary conditions. In all cases these specifications are chosen to model as closely as possible the real physical domain. For points in the interior of the domain, initial conditions may be specified as,

$$(q_x, q_y) = (q_{x_0}(x,y), q_{y_0}(x,y)) \text{ for all } (x,y)$$

in the interior domain at $t = 0$

$$H = h + \eta = H_0(x,y) \text{ for all } (x,y) \text{ at } t = 0$$

The discharge boundary conditions are summarized for the normal and tangential directions as,

$$q_n = \alpha_n x q_x + \alpha_y y q_y = q_n^*$$

$$q_s = -\alpha_y x q_x + \alpha_n y q_y = q_s^*$$

where,

$$\alpha_n = \cos(n,x); \alpha_y = \cos(n,y)$$

and the superscript $^*$ signifies a prescribed value. A detailed description of the method in which these nodal boundary conditions are applied in the context of an idealized site may be found in Sec. 4.1.

DISPER solves the vertically integrated form of the conservation of energy equation. The final governing expression is,
\[
\frac{\partial \theta}{\partial t} + \frac{\partial u \theta}{\partial x} + \frac{\partial v \theta}{\partial y} = \frac{\partial}{\partial x} Q_x - \frac{\partial}{\partial y} Q_y + \frac{\phi_s}{\rho c_v}
\]  \hspace{1cm} (3.2-4)

where,

\[
\theta = \int_{-h}^{h} T dz = \overline{T}H \hspace{1cm} (3.2-5)
\]

\(T\) = the water temperature at various depths

\(Q_x = -E_x \frac{\partial \theta}{\partial x} -E_y \frac{\partial \theta}{\partial y}\)

\(Q_y = -E_y \frac{\partial \theta}{\partial x} -E_y \frac{\partial \theta}{\partial y}\)

\[\frac{\phi_s}{\rho c_v} = \int_{-h}^{h} \frac{\phi}{\rho c_v} dz = \Phi_s\]

\(\phi = \) heat flux source \([\text{Joules/m}^3 \text{sec}]\)

\(c_v = \) specific heat of water at constant volume

The model may be formulated in terms of either actual or excess temperature above ambient (T in 3.2-5). The appropriate choice depends on whether the model is used to predict natural temperatures as well as artificially induced temperature variation. Ostrowski, 1980, was concerned with natural warming and computed actual temperatures while this study focuses primarily on excess temperatures. Because the time scales of processes which affect excess temperatures are shorter (order of one day rather than order of a week for natural temperature prediction), this requires less computation time. The disadvantage is that one has to estimate the background temperature if one wants to estimate actual temperatures from excess temperatures.
Program Modification

A number of modifications have been made to CAFE and DISPER in order to make them more flexible and able to perform computations directly with heat (temperature), rather than mass (concentration). Ostrowski, 1980, modified CAFE to allow time-varying fluxes (inflows and outflows of water) to be used as boundary conditions. Both constant and sinusoidal fluxes can be specified at the edges of the domain. This makes it possible to model tidal flows into and out of the domain without depending on the raising and lowering of ocean boundaries to simulate tidal forcing. By specifying the fluxes, more direct control exists over the flow in the domain. Such a modification was also necessary to handle the fluxes at the near field-far field (inner) boundary. An accounting of the verification of this is in Stolzenbach et al., 1980.

DISPER was revised by Ostrowski to solve directly for temperature in the basic convective-diffusion equation rather than constituent concentrations. As stated previously, calculations can either be made for excess temperature or for the actual temperature. Excess temperature calculations are made using the source/sink terms $\phi_s$, assuming first order dependence between heat loss and excess temperature. Thus, $\phi_s = K\theta$

where,

$\theta = $ excess temperature

$K =$ the equilibrium heat transfer coefficient

$K$ is treated as a constant so this approach doesn't take into account the temporal variation of meteorology.
For actual temperature calculations, Ostrowski, 1980, added a subroutine which computes surface heat transfer at each element using net heat flux equations with time-varying meteorology. The model takes the meteorological inputs from which it calculates the heat loss or gain $\dot{q}_s$. This simulation is superior to the constant sink or source option in allowing temporal variability of heat loss calculations. The details of these heat calculations may be found in Ostrowski, 1980.

Murakami developed a scheme to optimally number nodal points in the finite element grid (Stolzenbach et al., 1980). This numbering scheme was necessary to minimize the band width of the matrix used in the numerical routine which solves the matrix equations. The basis for this program was found in references on finite element applications to structural analysis.

3.3 Near Field Model

A separate group of governing equations may be derived for the surface jet models which most near field analyses relevant to this work have used. These expressions have been left out of this work, but can be found in other references (e.g., Stolzenbach and Harleman, 1971).

From these governing equations an important scaling parameter, the local densitmetric Froude number, is derived:

$$\hat{F}_L = \frac{u}{\sqrt{\Delta \rho \cdot \frac{g}{\rho_a}}}$$

(3.3-1)

where,

$u, \Delta \rho$ and $\ell$ represent characteristic values of jet velocity, density deficiency and length at varying positions along the jet.
In a buoyant jet, the value of $I_{FL}$ decreases along the axis. Near the point of discharge of the jet into the receiving waters $I_{FL}$ is typically in the range of 5 to 15. In developing the near field characteristics which were used to determine boundary conditions on the intermediate field portion of the domain, several physical parameters were derived as a function of the densimetric Froude number at the origin.

It should be noted that the discharge densimetric Froude number may be calculated using two different jet length scales. It is normally defined (symbol $I_{F0}$) in terms of the depth, $h_o$ of the discharge channel. However, it may also be defined (symbol $I_{F}'_o$) using the square root of one-half of the channel crosssectional area, $l_o$, as characteristic length. Thus

$$I_{F}'_o = \frac{u_o}{\sqrt{g'_{lo}l_o}} = I_{F}(h_o/b_o)^{1/4}$$

(3.3-2)

where,

$$b_o = \text{the half-width of the discharge channel}$$

$$l_o = (h_o b_o)^{1/2}$$

(3.3-3)

The significance of $I_{F}'_o$ is that it utilizes a more representative characteristic jet length scale and has been found to provide better correlation of results of different length to width ratios.

The near field properties used in this model, are based on Jirka et al., 1981. That paper describes surface jet properties using the Stolzenbach and Harleman surface jet model, along with laboratory and field data. These properties are defined in Figure 3-2 and discussed briefly below.
As mixing takes place the jet spreading rate \( \frac{db}{dx} \) increases and the local Froude number \( F_L \) decreases. The distance at which \( F_L \) becomes of order 1 is referred to as the transition distance because it is at this point that buoyant spreading begins to dominate jet mixing and thus the underlying assumptions of the near field no longer hold. This condition is usually signaled in integral jet models by a condition of rapid jet spreading and/or by a singularity in the matrix of differential equations. For the Stolzenbach-Harleman model, this transition occurs at \( F_L = 1.6 \). The distance, \( x_t \), at which this occurs correlates with,

\[
x_t = 12 \ell \frac{F'}{o} \left( \frac{h}{o} \frac{b}{o} \right)^{-0.2}
\]

or, for moderate values of the aspect ratio \( (1 \leq \frac{h}{b} \leq 2) \), \( x_t \) may be given by,

\[
x_t = 15 \ell \frac{F'}{o}
\]

The maximum depth \( h_{\text{max}} \) to which a plume would spread in deep receiving waters provides an indication of the ultimate plume thickness in the intermediate field. This depth is predicted from the Stolzenbach-Harleman model as

\[
h_{\text{max}} = 0.42 \ell \frac{F'}{o}
\]

Measurements in the laboratory and the field (e.g., Stolzenbach and Harleman, 1971; and Stolzenbach and Adams, 1979, respectively) suggest that the plume thickness in the intermediate field is approximately one half of the maximum jet thickness or

\[
h_{\text{far}} = 0.21 \ell \frac{F'}{o}
\]
Of course other processes acting in the intermediate and far fields will affect the plume thickness, so Eq. 3.3-7 is just an approximation.

The distance to the plume region of maximum penetration is given as,

$$x_{\text{max}} = 5.5 \bar{F}'_{\text{o}}$$  \hspace{1cm} (3.3-8)

Equation 3.3-8 will be used later to analyze the effects of shallow receiving water on jet dilution.

As the discharge jet enters the receiving waters the total amount of flow in the jet body increases as water is entrained from the sides. This dilution is significant in reducing temperatures in the near field and in creating flow which must enter the far field. For our purposes, jet mixing is characterized by the total, or stable, volumetric dilution $S_s$ defined as the ratio of the jet flow $Q$ to the discharge flow $Q_o$, i.e.,

$$S_s = Q/Q_o$$  \hspace{1cm} (3.3-9)

This dilution is referred to as stable because it is the asymptotic value of dilution which is reached beyond the transition distance at which point buoyancy effects have succeeded in damping further turbulent entrainment. For the Stolzenbach-Harleman model under conditions of deep receiving water and $\bar{F}'_o > 3$,

$$S_s = 1.4 \bar{F}'_o$$  \hspace{1cm} (3.3-10)

The volumetric dilution $S_s$ is inversely related to the average temperature rise at the end of the near field, $\overline{\Delta T_s}$. Thus,

$$\frac{\Delta T^o}{\overline{\Delta T_s}} = S_s = 1.4 \bar{F}'_o$$  \hspace{1cm} (3.3-11)
Peak (centerline) temperatures at the end of the near field $\Delta T_{cs}$ are higher than the average temperature by a factor which depends on the lateral profiles of temperature and velocity in the jet. With the Stolzenbach-Harleman model, the predicted stable centerline temperature rise is given by

$$\frac{\Delta T_o}{\Delta T_{cs}} = S_{cs} = 1.0 \frac{F'}{F_o}$$

(3.3-12)

where $S_{cs}$ is the corresponding centerline dilution.

The total jet entrainment, $E_s$, can be expressed as,

$$E_s = S - 1$$

(3.3-13)

This entrainment may be broken up into the components of total entrainment in the vertical and horizontal directions. These factors, $E_v$ and $E_h$, are given for $F' > 1$ as,

$$E_s = E_v + E_h$$

(3.3-14)

$$E_v = 1.2 \left( \frac{F'}{F_o} - 1 \right)$$

(3.3-15)

$$E_h = 0.2 \left( \frac{F'}{F_o} + 1 \right)$$

(3.3-16)

These relationships indicate how flow conditions with large $F'_o$ lead to large vertical entrainment relative to the horizontal entrainment. As $F'_o \to \infty$, $E_v/E_h \to 6$. This implies small far-field lateral recirculation for moderate or large $F'_o$ and vice versa. However, it is noted that in shallow water, vertical entrainment may be wholly or partly inhibited, thus decreasing overall dilution and enhancing the effect of lateral recirculation.
Shallow waters have a significant effect on jet behavior and mixing characteristics, particularly in light of the large bottom entrainment contribution indicated for deep water jets (Eq. 3.3-14-16). For shallow conditions, bottom entrainment flow must approach laterally through a restricted fluid layer under the jet. Induced velocities become higher leading to more frictional dissipation, pressure deviations below hydrostatic and a reduced vertical entrainment flow. Very shallow receiving waters with bottom attachment lead to reduced mixing capacity and distorted jet cross-sectional geometry.

To account for the effects of shallowness on the dilution ratios, a new parameter is defined—\( h_{\text{max}} / H \)—where \( h_{\text{max}} \) is the computed maximum depth of an equivalent deep water jet (3.3-6) and \( H \) is the depth of water at the point of maximum jet penetration \( x_{\text{max}} \) (3.3-8). Generally, for small values of \( h_{\text{max}} / H \), laboratory and field measurements indicate good agreement with deep water model results; for larger values of \( h_{\text{max}} / H \), induced temperatures are increased and the ultimate mixing is decreased. The ratio of observed to calculated values of centerline dilution, \( S_{\text{cs}} \), gives an indication of the degree of shallowness,

\[
r_s = \frac{\hat{S}_{\text{cs}}}{S_{\text{cs}}}
\]

(3.3-17)

where,

\( r_s = \) shallow water dilution reduction factor

\( \hat{S}_{\text{cs}} = \) observed centerline dilution in shallow water

\( S_{\text{cs}} = \) predicted centerline dilution for deep water
As the value $\frac{h_{\text{max}}}{H}$ increases, this dilution factor decreases. From data accumulated on the effects of dilution, it appears that for ratios of $h_{\text{max}} < 0.75$ the deep water dilution prediction $S_{cs}$ (Eqn. 3.3-12) is a reliable value. Therefore, a criterion which is used to determine shallow water conditions is,

$$\frac{h_{\text{max}}}{H} > 0.75$$

(3.3-18)

Further field and laboratory work compiled by Jirka et al., 1981, has shown that a reasonable factor for adjusting dilution values to account for shallowness is,

$$r_s = \left(\frac{0.75}{h_{\text{max}}/H}\right)^{0.75} \text{ for } h_{\text{max}}/H > 0.75.$$  

(3.3-19)

### 3.4 General Use of the Model

The various surface jet properties described in the previous section are summarized in Table 3-1. Figure 3-1 shows how this information is used in the model schematization. The reader is also referred to the description of the idealized domain covered in Section 4.1.

The transition distance, defined by Eqn. 3.3-5, was used as a guideline in establishing the inside boundary of the computational domain between the near and far fields. This boundary consisted of a layer of small elements which surrounded the near field. The elements were small here to give greater resolution of velocity and temperature, since it is at this point that the largest gradients exist.

Moving out from the near-far boundary, the elements increase in size. This corresponds to the lower gradients as the distance from the
TABLE 3-1 Summary of Near Field Surface Jet Properties

SCALING PARAMETERS:

\[ F'_o = \frac{u_o}{\Delta \rho_o} \frac{\rho_o}{(-g \rho_o)^{1/2}} \]

DENSIMETRIC FROUDE NO.

\[ \xi_o = \sqrt{\frac{h_o}{b_o}} \]

DISCHARGE CHANNEL CHARACTERISTIC LENGTH

DEEP WATER PROPERTIES:

\[ x_t = 15 \xi_o F'_o \]
TRANSITION DISTANCE

\[ h_{max} = 0.42 \xi_o F'_o \]
MAXIMUM PLUME DEPTH

\[ h_{far} = 0.21 \xi_o F'_o \]
FAR FIELD PLUME DEPTH

\[ x_{max} = 5.5 \xi_o F'_o \]
DISTANCE FROM DISCHARGE TO MAX PLUME DEPTH

\[ S_s = 1.4 F'_o \]
STABLE VOLUMETRIC DILUTION = \( \frac{Q}{Q_o} = \frac{\Delta T^o}{\Delta T_s} \)

\[ S_{cs} = 1.0 F'_o \]
STABLE CENTERLINE DILUTION = \( \frac{\Delta T^o}{\Delta T_{cs}} \)

\[ E_v = 1.2 (F'_o - 1) \]
VERTICAL JET MASS ENTRAINMENT

\[ E_h = 0.2 (F'_o + 1) \]
HORIZONTAL JET MASS ENTRAINMENT

SHALLOW WATER DILUTION CORRECTION:

\[ r_s = \left[ \frac{0.75}{h_{max}/H(x_{max})} \right]^{0.75} \quad (\text{for } \frac{h_{max}}{H(x_{max})} > 0.75) \]

SHALLOW WATER DILUTION

DEEP (UNAFFECTED) WATER DILUTION
jet source increases. Numerically, the model has a limit on the difference in size of two adjacent elements. Nodes are then spaced to reduce the gradient in element size. As the distance from the near field increases, the larger size of the far field elements becomes roughly constant. A majority of the domain has elements of similar size because of this. The average length of a far field element side is about 15-20 times that of an average element adjacent to the near field.

Another consideration in developing the finite element grid is the optimum element shape for maximizing numerical stability. It has been found from previous finite element models that equilaterally shaped triangular elements serve best. In developing the grids for this study's domains, elements of equal side lengths were used as extensively as possible. However, in some cases departures from the equilateral shape were necessary. This was caused by both the increasing element size from near to far and the real physical constraints of land boundaries. Actual shapes of the elements were kept as close as possible to equilateral, but sometimes the constraints yielded significantly non-uniform elements. A general "limit" which was adopted for element uniformity was that no internal angles should be greater than 90°.

Boundary conditions for the two numerical models used, CAFE and DISPER, are satisfied by designating the type of node and the type of boundary (element side) for each element on the boundary of the domain. Any land or ocean boundary condition, with or without fluxes may be accounted for with this system. In CAFE, a land boundary is one in which boundary fluxes are specified. These boundaries are identified to the program by naming the contiguous nodal points forming each
boundary. All other boundaries in the CAFE model are designated ocean boundaries. On these boundaries, the (tidal) elevation \( \eta \) is specified but there is no explicit constraint on fluxes.

CAFE allows the classification of boundary nodes in several different categories depending on whether the boundary is land or ocean. Land boundary nodes require that normal flux be specified. This flux is specified at each node but represents, both physically and to CAFE, the flux which passes through the land boundary corresponding to each land boundary node. Thus this flux is specified as flow per unit width (e.g., \( \text{ft}^2/\text{sec} \)). This boundary condition was used to represent the discharge from the near field, the entrainment flow to the near field, the plant intake flow, river inflow, and the vertical entrainment flow leaving the domain (see Figure 3-1). In certain cases, flux across a land boundary was also used to represent ambient currents. At all other land boundaries, a condition of no normal flux was specified.

For each land boundary node, the option was available to specify no flux tangential to the boundary at a boundary node (no slip condition). This was necessary in situations where sharp corners existed on the boundary. By specifying the no tangential flux boundary condition, the flow was routed away from these areas preventing accidental loss of mass (and heat in the DISPER calculation) across these boundaries. In CAFE, purely land boundary nodes are designated as either type 1 (allowing tangential flux) or type 4 (no tangential flux allowed).

Ocean boundary nodes in CAFE have their tidal amplitude specified. Using a sinusoidally varying function to describe the ambient raising
Figure 3-1: Description of Surface Jet Parameters Used in Model Schematization.
Figure 3-2: Near Field Surface Jet Properties.
and lowering of the tide, flow was forced into the domain through the boundary sides. There also was available a provision in the model to input phase lags in the tidal elevation for each of the boundary nodes. The expression used in the model which governs the tidal forcing is,

\[ \eta = a[1 - \cos \left( \frac{2\pi}{T} (t - \phi) \right)] \]  

(3.4-1)

where,

\( \eta \) = the height of the water level above the mean low water mark
\( a \) = the amplitude of the tidal fluctuation
\( T \) = the tidal period of the region under study
\( \phi \) = the phase lag

The phase lag can be used to account for spatial and temporal variations of the flow coming into the domain. For instance, a long open ocean boundary may have a tidal flow which comes into one portion of the boundary prior to coming into the other parts.

It is also possible to combine the node type specifications of normal/tangential fluxes and tidal heights. The tidal heights and normal flux specifications may or may not include the constraint on the tangential flux. The specification of normal flows and tidal heights correspond to points where land and ocean boundaries meet.

Since DISPER utilizes the flow field produced by CAFE, the specification of a no flux boundary has no meaning as far as mass flux is concerned. This type of boundary does constrain the heat from crossing the boundary, however. This distinction is important to consider in avoiding problems with artificial heat buildup. Individual boundary node types aren't specified in the running of DISPER.
CHAPTER IV
IMPLEMENTATION

To test the working characteristics and validity of the model, it was necessary to apply it to field sites with the necessary descriptive data with which to compare the results. Isotherms measured at various phases of a tidal cycle at the Millstone and Brayton Pt. sites were used to check against the results of the numerical computations. Prior to this step, though, the basic nature of the model was tested using an idealized domain. This was done to simulate the use of the coupled near-far parameterization and to check sensitivities to certain numerical factors.

4.1 Idealized Domain

Purpose

Up to the beginning of this project the computer models, CAFE and DISPER, were used only for far field analysis. The modification of these models to include near field influences represented a large change in their use which called for certain interim checks. Various program modifications were tested and are reported in a project progress report (Stolzenbach et al., 1980). Application of the program was tested using an idealized practice domain discussed here. By making use of the relatively simple and symmetrical geometry the results of various tests could be examined more quickly than at an actual site.

Grid Design

The domain which was used can be seen in Figs. 4-2 and 4-3. It was rectangular shaped (2740 ft. x 5480 ft.) with a constant depth of
13 ft. This constant depth included the side boundaries, both water and land. In designing the practice domain it was desired to use a simple, symmetrical scheme to provide, as much as possible, a control in which the formulation and numerical properties of the model could be tested. This justified the use of the constant depth specification.

The near field region was carved from the base of the rectangular domain. From this symmetric, semi-circular boundary the triangular finite elements radiated out to a limiting maximum size. The transition was gradual to avoid large differences in member lengths. The minimum element side length on the grid was 82 ft. This is contrasted with the maximum element side length of 1250 ft. The radius of the semi-circular near field area was roughly 360 ft.

To simulate the influence of the near field on the far field model, flux boundary conditions were specified on the near field-far field interface for CAFE. These boundary fluxes were selected to be representative of typical near field mixing from a surface discharge. See Table 4-1 and Figs. 4-2 and 3. Over the central three nodes of the transition circle, the discharge was represented by flux values of 27.3 ft$^2$/sec. Both sets of side entrainment nodes had flux values of 8.4 ft$^2$/sec specified at each nodal point. Sensitivity to alternate lateral entrainment relationships is discussed later in this chapter. To account for the vertical entrainment from the domain into the jet plume, mass was removed from the domain along the top boundary of the grid at a rate of 0.75 ft$^2$/sec. Also, in certain runs, an intake was established by removing flux from the lower right-hand corner of the domain. The flux
specified was $3.3 \text{ ft}^2/\text{sec}$ over a distance of 1400 ft. Several runs were made using specified influx on the left end of the domain and an equal and opposite outflux along the right boundary of the domain. This was to simulate a steady cross current into which the plume was discharged. The value specified was a uniform current velocity of 0.66 ft/sec. The full set of near field boundary conditions on the far field may be seen in Table 4-1.

**Boundary Node Specification**

In specifying boundary conditions for the practice domain, it was desired to use a realistic, yet relatively simple configuration. For the CAFE run, both domain side boundary types and boundary node types were specified. A land boundary was specified around the entire practice domain. All nodes were type 1 (specified normal) except the six corner ones which were type 4 (specified normal and no tangential). The two ends of the rectangular domain had specified fluxes to simulate a steady, constant cross current. This could have been modeled as well by tidal amplitude specifications on the boundary.

The boundaries in DISPER were specified such that only the lengths where no mass flux was removed were left as a land boundary. The open boundaries included the near-far field boundary, the two open ends, and the top boundary where vertical entrainment flow was removed.

**Time Step**

The time step used while running CAFE was found to be dependent on four physical characteristics of the problem: (i) Maximum flow
Table 4-1: Near Field Parameters Computed
for Idealized Domain Simulation

<table>
<thead>
<tr>
<th>Parameter*</th>
<th>Symbol</th>
<th>Simulation Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Discharge Flow Rate</td>
<td>$Q_o (m^3/sec)$</td>
<td>62.9</td>
</tr>
<tr>
<td>Discharge Temperature Rise</td>
<td>$\Delta T_o (^0C)$</td>
<td>13.5</td>
</tr>
<tr>
<td>Transition radius</td>
<td>$r_t (m)$</td>
<td>118.9</td>
</tr>
<tr>
<td>Near Field Volumetric Dilution</td>
<td>$S$</td>
<td>4.5</td>
</tr>
<tr>
<td>Vertical Mass Entrainment</td>
<td>$E_v$</td>
<td>1.0</td>
</tr>
<tr>
<td>Horizontal Mass Entrainment</td>
<td>$E_h$</td>
<td>2.5</td>
</tr>
<tr>
<td>Far Field Plume Depth</td>
<td>$h_{far} (m)$</td>
<td>4.0</td>
</tr>
<tr>
<td>Shallow Water Dilution Correction</td>
<td>$r_s$</td>
<td>1.0</td>
</tr>
</tbody>
</table>

*see Table 3-1
velocity, \( u \); (ii) Minimum nodal spacing, \( \ell \); (iii) Eddy viscosity, \( E_{ij} \); and (iv) Wave speed, \( c = \sqrt{gh} \). The practice domain's minimum spacing was 82 ft., the maximum velocity was 2.7 ft/sec, and the wave speed was 20.5 ft/sec, and for most runs, the eddy viscosity was 165 ft\(^2\)/sec. The most successful timestep was 2.0 secs. Values significantly greater than this caused numerical instability characterized by uncontrolled spatial oscillations in the water elevation at certain nodes, ultimately resulting in negative values of \( \eta \) which exceeded, in absolute value, the actual water depth \( H \). Values of \( \Delta t \) less than 2.0 secs produced similar circulation patterns as observed with 2.0 sec but with increased computation time.

The following general criteria were available to determine the necessary timestep for running DISPER (Leimkuhler et al., 1975):

\[
10 \quad \Delta t < \frac{\ell^2}{D} \quad \text{(4.1-1)}
\]

\[
10 \quad \Delta t < \frac{\ell}{u} \quad \text{(4.1-2)}
\]

where,

\( \Delta t = \) time step

\( D = \) dispersion coeff.

Applying the physical characteristics to 4.1-1 and 4.1-2, a range of \( \Delta t \) from 2 to 10 secs. was determined corresponding to a range in the value of \( D \) from 110 to 22 ft\(^2\)/sec.

In addition to the above criterion on the time step, a condition for stability involving just the three physical quantities has been observed (Leimkuhler et al., 1975):
\[
\frac{2u}{D} < 2
\]  

(4.1-3)

For a given \( k \) and \( u \), this expression gave guidelines on the appropriate range of \( D \) (\( D > 100 \)) while Eqns. 4.1-1 and 4.1-2 then dictated time step. It should be noted that Eqn. 4.1-3 in particular is not a precise constraint. Use of values of \( D \) lower than suggested resulted in spatial oscillations of predicted temperatures. However, even for values of \( D \) significantly greater than suggested (215-275 ft\(^2\)/sec), these oscillations weren't completely eliminated. It is also worth noting that the value of dispersion coefficient required for numerical purposes, given by Eq. 4.1-3, is about an order of magnitude greater than that which would exist physically due to real sub-grid scale dispersion effects. See further discussion under simulation results. The choice of \( D \) thus involves a tradeoff between numerical accuracy (suggesting a large value) and physical accuracy (suggesting a small value). Results presented later in this section were run with a \( D \) value of 110 ft\(^2\)/sec and a time step of \( \Delta t = 13 \) sec. The fact that these integration parameters didn't fall within the limits of the numerical stability criteria point out the approximate nature of this criteria.

**Entrainment Distributions**

Several distributions of discharge and entrainment boundary conditions at the near-far field interface were investigated in the practice domain. One featured a gradual decrease of entrainment from the semi-circular base to the transition from entrainment to discharge (Figure 4-1a). This attempt also had a peak discharge value at the top
a) Gradual Variation between Entrainment and Discharge.

b) Constant Discharge and Entrainment Flow Boundary with Sharp Transition.

c) Discharge and Entrainment Flow Distribution Used in Simulations.

Figure 4-1: Discharge and Entrainment Flow Relationship.
node with smaller discharge values on the two adjacent nodes. Another
distribution (Figure 4-2b) had a constant influx value across the two
entrainment sides with an abrupt shift to a constant outward flux at the
three discharge nodes. Both of these flux distributions led to numerical
instability, however. A final configuration was chosen which had con-
tant entrainment values along the two sides and over the discharge nodes.
The two nodes between the entrainment and discharge were left with no
specified normal flow, though. This avoided a sharp discontinuity in
flux values of adjacent nodes which enhanced stability. This form is
shown in Figure 4-1c.

Simulation Results

Initially, the simplest types of CAFE simulations (stagnant water,
no intake) were made to investigate sensitivity to time step and the
different configurations of the flux boundary conditions on the near-
far boundary. Eventually the optimum flux set was determined to be
that depicted in Figure 4-1c.

Subsequent to this CAFE runs were made with an intake and with a
0.2 m/s cross-current. Steady state circulation patterns corresponding
to a 0.2 m/s cross-current with no intake and stagnant conditions with
an intake are shown in Figures 4-2 and 4-3.

After experimenting with CAFE, the heat dispersion model DISPER
was implemented. This involved specifying excess temperature boundary
conditions of 3.0\(^\circ\) C at the discharge nodes and using the circulation
patterns created by CAFE. The excess temperature of 3\(^\circ\) C corresponds
to a power plant discharge temperature of 13.5\(^\circ\) C and a dilution of
Figure 4-2: Results from CAFE Run at Practice Domain with Constant Cross-Current and No Intake.
Figure 4-3: Results from CAFE Run at Practice Domain with No Cross-Current and an Intake.

Plant Intake
$S = 4.5$. See Table 4-1. The major emphasis in running DISPER involved studying the tradeoff and relationship between the time step and dispersion coefficient.

Selection of the dispersion coefficient was a delicate process. Physically based methods for choosing values of the dispersion coefficients in numerical models has been dealt with in the work of Christodoulou, 1976. One technique suggested employed the mixing length hypothesis,

$$\hat{u} = \frac{Au}{\Delta L} L \quad (4.1-4)$$

where $\hat{u} =$ the r.m.s. turbulent velocity fluctuation

$$\frac{Au}{\Delta L} = \text{the velocity gradient over the distance } L$$

From this basis came an expression for the dispersion coefficient based on velocity gradients,

$$D = L^2 \sqrt{\phi} \quad (4.1-5)$$

where,

$L = 0.12 \Delta S$

$\Delta S = \text{a characteristic side length of an element}$

$$\phi = 2\left[\frac{\partial u}{\partial x}\right]^2 + 2\left[\frac{\partial v}{\partial y}\right]^2 + \left[\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right]^2$$

For the practice domain, this expression yielded a value of about 11 ft$^2$/sec as the physically based sub-grid scale dispersion coefficient. This is significantly lower than the values which were necessary to use for numerical reasons in the model. In general, while lower values of $D$ allowed greater $\Delta t$'s (a desirable effect from a computational cost standpoint), they also led to amplitude oscillations of
temperature calculated in the domain. In general, the value of dispersion coefficient which was chosen was the lowest one which still gave acceptably small temperature oscillations.

It should be mentioned that the amplitude of temperature oscillations (associated, for example, with small values of D) depends on whether DISPER calculations are made with excess temperature or actual temperature. Ostrowski, 1980, used DISPER to compute natural warming. He found that his temperature oscillations were reduced if he subtracted a background temperature from his actual temperature before making his DISPER calculations. The background temperature was then added back in when computing the surface heat transfer and when plotting the results. This can be viewed in the context of the previous examples with the idealized domain by assuming that the initial and background temperatures were 20°C and that the discharge temperature was 33.5°C and thus that the temperature of the diluted flow entering the far field was 23°C. Rather than working with temperatures in the range of 20°C to 23°C, however, 20°C was subtracted from all temperatures and added back when computing surface heat transfer and plotting results. Since the present calculations include excess temperature directly, such a procedure was not necessary.

An example of a DISPER calculation is shown in Figure 4-4. Conditions correspond to the CAFE output shown in Figure 4-3 and the temperature boundary conditions discussed above. They were run with a dispersion coefficient of 45 ft²/sec., a time step of 8 sec. and an elapsed time of 1 hour. The influence of the station intake is clearly seen.
Figure 4-4: DISPER Calculation with Practice Domain Corresponding to CAFE Run Shown in Figure 4-3. Values in (°C).
in deflecting the plume to the right and the effects of model dispersion can be seen in the temperature built-up near the base of the transition zone--especially on the left-hand side. It is expected that this dispersion could be reduced by employing a smaller dispersion coefficient together with a smaller grid size (as required by Eqs. 4.1-3) and a smaller time step (as required by Equations 4.1-1 and 2).

4.2 Millstone Plant Site

Site Description

The Millstone Nuclear Power Station is located in Waterford, Connecticut, on the north shore of Long Island Sound. The area of the site is approximately 500 acres. The main station area of about 80 acres is sited on a point of land which is bounded on the east by Jordan Cove and on the west by Niantic Bay which forms the entrance to the Niantic River estuary (see Figure 4-5).

Tides in the Niantic Bay area are semi-diurnal with mean and spring ranges of 2.7 ft. and 3.2 ft., respectively. Water depths range from several to 100 ft. Field data collected at the site indicate that tidal currents dominate natural water movement in the vicinity of the station. In particular, the flow into and out of Niantic Bay forms a strong current past the station along a line running from the plant site through Twotree Island Channel. Currents in Niantic Bay are also relatively strong as a result of flow into and out of the Niantic River. In contrast, the currents in Jordan Cove, even during the strength of ebb and flood tides, are relatively weak. Thermal and salinity induced strati-
Figure 4-5: Site of Millstone Nuclear Power Station.
fication isn't a significant factor in the vicinity of the plant site although considerable natural temperature variation is observed, particularly in shallower regions of the shoreline area. Mathematical modeling of this natural temperature variability has been the subject of earlier study under this project (Ostrowski, 1980).

Already located at the site are two operational nuclear reactor units. Unit 1 employs a boiling water reactor having a net electrical output of 652 MWe, and Unit 2 employs a pressurized water reactor with a net electrical output of 879 MWe. Water enters the station through intake structures located on the west side of Millstone Pt. After the water passes through the condensers, the heated water is discharged in a southeasterly direction through a 1200 ft. long 100 ft. deep quarry and then into Long Island Sound through a 60 ft. wide channel.

Objectives

Extensive field data has been collected at the Millstone site under two unit operation. A primary project objective was to check the validity of the near-far field model against this data. A second objective was to examine the induced flow associated with the discharge and intake flows. A final project objective was to examine natural (as opposed to plant-induced) warming in shallow areas such as Jordan Cove and Niantic Bay. This issue was addressed in Ostrowski, 1980.
Site Schematization

In applying this model to Millstone it was necessary to choose an appropriate layer depth for the far field based on the well mixed assumption. The depth was determined by looking at the vertical profile of the discharge plume using the near field surface jet analysis. Taking one half of the maximum near field plume depth, the value used was 11.5 ft.

The near field transition distance for two units was computed as a function of tidal stage and is presented in Table 4-2; the average distance was about 720 ft. The grid used a somewhat smaller value of \( r_t \) equal to 475 ft. in order to match the peninsula's physical geometry. At Millstone the water depths surrounding that outfall are deep resulting in negligible bottom effects on the discharge and entrainment. The parameters used for the dilution rate, \( S \), along with the values of the horizontal and vertical entrainment, \( E_h \) and \( E_v \), respectively, can be found in Table 4-2.

The grid was developed in two stages. Initially, the large area coarse element grid was produced. This was the same grid used by Ostrowski to model natural conditions in the area prior to plant discharge effects. The area of the coarse grid near the plant's discharge point was subsequently refined to account for the high velocity induced flows. The near-far field boundary was lined with small triangular elements which in turn were bounded by larger ones at greater radii. This pattern continued for 3-5 layers until the new elements' size matched the original elements' size.
Table 4-2: Near Field Parameters Computed

for Millstone Simulation (Two Units)

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Symbol</th>
<th>Value*</th>
</tr>
</thead>
<tbody>
<tr>
<td>Discharge Flow Rate</td>
<td>$Q_o$ (cfs)</td>
<td>2290</td>
</tr>
<tr>
<td>Discharge Temperature Rise</td>
<td>$\Delta T_o$ ($^\circ$F)</td>
<td>23</td>
</tr>
<tr>
<td>Discharge Velocity</td>
<td>$u_o$ (fps)</td>
<td>3.6–4.5</td>
</tr>
<tr>
<td>Discharge Channel Depth</td>
<td>$h_o$ (ft.)</td>
<td>9.2–11.7</td>
</tr>
<tr>
<td>Discharge Channel Half-Width</td>
<td>$b_o$ (ft.)</td>
<td>27.5</td>
</tr>
<tr>
<td>Discharge Channel Characteristic Length</td>
<td>$\ell_o$ (ft.)</td>
<td>15.9–17.9</td>
</tr>
<tr>
<td>Densimetric Froude No. (based on $h_o$)</td>
<td>$\mathbb{F}_o$</td>
<td>3.0–4.3</td>
</tr>
<tr>
<td>Channel Aspect Ratio</td>
<td>$h_o/b_o$</td>
<td>0.33–0.42</td>
</tr>
<tr>
<td>Densimetric Froude No. (based on $\ell_o$)</td>
<td>$\mathbb{F}'_o$</td>
<td>2.4–3.3</td>
</tr>
<tr>
<td>Transition Radius</td>
<td>$r_t$ (ft.)</td>
<td>655–782 (475)</td>
</tr>
<tr>
<td>Near Field Volumetric Dilution</td>
<td>$S$</td>
<td>3.4–4.6 (4.0)</td>
</tr>
<tr>
<td>Vertical Mass Entrainment</td>
<td>$E_v$</td>
<td>1.7–2.7 (2.2)</td>
</tr>
<tr>
<td>Horizontal Mass Entrainment</td>
<td>$E_h$</td>
<td>0.7–0.9 (0.8)</td>
</tr>
<tr>
<td>Maximum Plume Depth</td>
<td>$h_{\text{max}}$ (ft.)</td>
<td>18.3–21.9</td>
</tr>
<tr>
<td>Far Field Plume Depth</td>
<td>$h_{\text{far}}$ (ft.)</td>
<td>9.2–11.0 (11.5)</td>
</tr>
<tr>
<td>Shallow Water Dilution Correction</td>
<td>$r_s$</td>
<td>1.0</td>
</tr>
</tbody>
</table>

*Ranges given refer to variation over tidal cycle. Values in parentheses were used in simulation.
Actual land boundaries were followed closely given the nature of matching straight element sides to them. Normal fluxes were set to zero along the coarse grid land boundary. The ocean boundary at the southern and eastern sides of the bay was extensive owing to Niantic Bay's opening to Long Island Sound. To account for the spatial variation of tidal flow along this boundary, a phase lag was utilized causing the tide to propagate from east to west. The tidal period was 44640 sec., the amplitude was 0.5 ft. and the phase lag was 12 min. between the top node on the eastern boundary and the far western node.

With the exception of tidal amplitude, these are the same tidal parameters used by Ostrowski, 1980, in his analysis of natural temperatures at the Millstone site. The tidal amplitude has been reduced to about one-third of its observed prototype value to allow for the fact that actual water depths at the site exceed the constant value which was modeled by about a factor of three. On average, therefore, the modeled tidal currents should be of correct magnitude. Recalling the discussion in Chapter 3, the modeled currents will be somewhat excessive in the deeper portions and somewhat reduced in the shallower areas.

At the semi-circle intermediate field boundary it was necessary to specify all normal fluxes consistent with the near field analytical model's induced flow field. The discharge was 32.3 ft$^2$/sec. along the center 280 ft. Side entrainment was 2.5 ft$^2$/sec. over 390 ft on each side. The discharge location on the tip of Millstone Pt. made development of the grid straightforward because of lack of physical interferences. A diagram showing the grid used is in Figure 4-6. The
Figure 4-6: Grid Used for Millstone Simulations.
location of the intake, critical when evaluating results, was placed over three nodes on the west side of Millstone Pt. This coincided closely with the location and geometry of the intake. The intake flux per unit width was specified as $0.12 \, \text{m}^2/\text{sec.}$ along 1700 ft. of the peninsula. The amount of mass entering the jet plume through vertical entrainment left the domain through the ocean boundaries along the southern base of the domain.

As shown in Table 4-2, the incremental rise in discharge temperature over ambient was $23^\circ\text{F}$. After dilution with the entraining fluid in near field mixing, the absolute temperature rise of the discharge is reduced by a factor of 4.5 to $5.1^\circ\text{F}$. This value was used as a constant temperature source at the discharge inflow nodes. Heat was extracted from the domain through the intake and the entrainment boundary fluxes. To model heat loss to the atmosphere a first order decay term—equal to $2.5 \times 10^{-6} \, \text{sec.}^{-1}$—was used. This value is consistent with summertime meteorological conditions and the modeled water depth.

**Time Step**

The time step used for CAFE was determined by considering the following four properties: (i) $u = 2.56 \, \text{ft./sec.}$; (ii) $\lambda = 80 \, \text{ft.}$; (iii) $E_{ij}$ was a constant $810 \, \text{ft.}^2/\text{sec.}$; (iv) $c = 19 \, \text{ft./sec.}$ These values led to a time step of 2.0 secs. for CAFE.

The DISPER time step was dependent on the dispersion coefficient $D$, the minimum length scale $\lambda$, and the maximum velocity $u$. Using stability criteria discussed in Sec. 4.1, a dispersion coefficient of $D = 110 \, \text{ft.}^2/\text{sec.}$ and a time step of $\Delta t = 8 \, \text{secs.}$ were used.
Results

Figures 4-7 through 4-9 show the CAFE results at Millstone for three different conditions which approximate the tidal conditions observed at the site. They are, respectively: (i) stagnant (representing high or low slack) in Figure 4-7, (ii) maximum flooding phase in Figure 4-8 and (iii) the maximum ebbing condition in Figure 4-9. Full tidal cycle simulations weren't run at Millstone. Instead, the stagnant tidal condition was first set up. Then the two conditions of maximum flooding and maximum ebbing were produced. Flooding was simulated by running the circulation model CAFE for a three hour period from two hours prior to the maximum flooding condition until one hour after max flood. DISPER runs were performed by spreading this three hour velocity file out to six hours of input. This was effective in creating max flood conditions over a long period of time. In an exactly analogous manner, the maximum ebbing condition was simulated.

All three conditions seem to show physically consistent behavior. The circulation patterns look reasonable with respect to the entrainment, ocean boundaries, and intake. The circulation in the Niantic River area is small and shows little variation with respect to tidal stage. In general, these results seemed accurately done.

The DISPER computations at Millstone all seemed to be physically reasonable. Figures 4-10, 4-11 and 4-12 show the results for the stagnant, maximum flood and maximum ebbing cases, respectively. The stagnant run case corresponds to the slack tidal conditions, both high and low.

In comparing the three sets of isotherms, one can make out the
Figure 4-7: CAFE Run Results at Millstone Under Stagnant Conditions.
Figure 4-8: CAFE Run Results at Millstone Under Maximum Flood Tidal Conditions.
Figure 4-9: CAFE Run Results at Millstone Under Maximum Ebb Tidal Conditions.
Figure 4-10: DISPER Results at Millstone Under Stagnant Conditions.
Figure 4-11: DISPER Results at Millstone Under Flood Tidal Conditions.
Figure 4-12: DISPER Results at Millstone Under Ebb Tidal Conditions.
distinctions between the various tidal phases and their influence on
the isotherm locations. The flood case shows the isotherm shifted
farthest to the left (or west). The ebb case isotherms, conversely
are shifted farthest to the east. However, it is clear that com-
pared to field data collected at the site for a typical summer day
(e.g., July 29, 1977, as reported in reference 25), the calculated plume
doesn't shift laterally as much as data indicate it should for the ebb
and flood cases. This is because the near field parameterization was
selected independent of the tidal crossflow, thus permitting the plume
to deflect downstream only in the far field (computational) domain.
In reality, the plume should be deflected in the near field as well.
The extent of near field deflection as a function of ambient current
speed can be parameterized well using formulae similar to those used
for the other near field properties discussed in Chapter 3. To more
accurately simulate the plume trajectory, this parameterization could
be used as a basis for prescribing a time varying flux boundary condi-
tion. This would involve allowing the discharge and entrainment
fluxes to increase and decrease around the near-far interface boundary.
By synchronizing this variation with tidal crossflows, the shifting
back and forth of the plume with tidal phase could more accurately be
accomplished. It should be noted that the CAFE model has been modified
to allow time varying flux boundary conditions. However, to adopt the
above approach would require that an additional subroutine be added to
prescribe these boundary conditions as a function of time.

To compare, in a quantitative sense, the size of the calculated
plumes with field data, information on the size of particular isotherms
(reported in reference 25) was viewed relative to that measured from the simulated isotherms. For example, under the maximum flood conditions the 40°F isotherm is found from field data to be from 900 to 2400 ft. in length. This range reflects two days' measurements and two forms of data acquisition (thermal measurements and the use of dye). The centerline distance of the same calculated isotherm was found to be 2000 ft. Under the same maximum flood conditions, the 1.50°F field isotherm was 5000 - 6200 ft. in length. The calculated isotherm length for 1.50°F was around 5700 ft.

The maximum ebbing condition was also compared. For the 40°F field isotherm, the length of the thermal plume was 1500 to 4900 ft. in length. The calculated plume was 900 ft. For 1.50°F, the field data yielded a length of 3200 to 9200 ft. The calculated 1.50°F isotherm length was around 5000 ft.

The stagnant run case could be compared to the slack cases, both high and low. This was reasonable because the computations did not distinguish between high and low tide. For slack cases, measured plume lengths for the 40°F isotherm ranged between 1000 ft. and 3800 ft. The calculated 40°F isotherm under stagnant conditions was 2500 ft. in length. The 1.50°F slack isotherms measured in the field ranged in length from 2700 ft. to 4000 ft. The calculated 1.50°F isotherm was 5800 ft. in length.

The general shape—if not the orientations—of the calculated isotherms seemed to match the field data, considering the limitation of the discharge not being properly influenced by the tidal flow as
already discussed. With two exceptions, the lengths of the calculated isotherms fell into the range of lengths of the field data plumes. A comparison of simulated and observed plume lengths is summarized again in Table 4-3.

Computational Time

The time taken by the CAFE program to perform a simulation is significant. While actual connect time will vary from one system to another, CPU time is comparable. A typical Millstone three-hour simulation required five hours of CPU time on MIT's Multics System (Honeywell HISI 68/DPS). It should be added that these calculations were performed without consideration of cost optimization and could be streamlined by paying greater attention to grid layout or by alteration in the basic CAFE code. Consideration of cost should be a primary objective of any continuation study.

Table 4-3

Comparison of Measured Isotherm Lengths (range, ft) and Predicted Isotherm Length (value in parentheses, ft) for Various Tidal Phases at Millstone Station

<table>
<thead>
<tr>
<th>Tidal Phase</th>
<th>Slack (High and Low)</th>
<th>Max Ebb</th>
<th>Max Flood</th>
</tr>
</thead>
<tbody>
<tr>
<td>4°F</td>
<td>1000 - 3800</td>
<td>1500 - 4900</td>
<td>900 - 2400</td>
</tr>
<tr>
<td></td>
<td>(2500)</td>
<td>(900)</td>
<td>(2000)</td>
</tr>
<tr>
<td>1.5°F</td>
<td>2700 - 4000</td>
<td>3200 - 9200</td>
<td>5000 - 6200</td>
</tr>
<tr>
<td></td>
<td>(5800)</td>
<td>(5000)</td>
<td>(5700)</td>
</tr>
</tbody>
</table>
The time for running DISPER was significantly shorter than that necessary for CAFE. In running Millstone, a typical five hr. simulation time would require 75 min. of CPU time. Again, a refinement of the program coding and grid layout would reduce this simulation time.

4.3 Brayton Pt. Site

Site Description

The Brayton Point Generating Station is located in Somerset, Massachusetts, at the confluence of the Lee and Taunton Rivers at the northern end of Mount Hope Bay. See Figure 4-13. At low tide, Mount Hope Bay has a length (along its north-south axis) of approximately 7 mi., a surface area of 15.6 mi\(^2\), and a volume of 8.3 billion ft\(^3\). The average tidal range is 4.4 ft. which results in a tidal prism volume of approximately 1.2 billion ft\(^3\). Approximately 70% of the Bay area has an average depth of less than 18 ft. at mean low water while the main shipping channels average 30 ft. in depth at mean low water. With the exception of the abrupt increase in depth at the edge of the shipping channels and the rapid shoaling in the area of Spar Island, the bottom contours of much of the Bay are rather even, with a steady increase in depth from the head of the Bay to the two southerly passages.

Circulation in Mount Hope Bay is driven primarily by tides and secondarily by wind, and fresh water inflow from the Taunton River at the north end. Residence time within the bay has been estimated to be within the range of 6 to 12 days (MRI, 1978).

Temperatures within Mount Hope Bay vary with the tidal stage and are quite responsive to meteorological conditions due to the relative
Figure 4-13: Site of Brayton Point Generating Station. (Note location of Generating Units 1, 2 and 3 and Generating Unit 4 on Brayton Point.)
shallowness of the bay. In spring and summer, mild thermal stratification (3° - 5° F) may be found while temperatures are generally vertically well mixed in the fall and winter. In mid-summer, surface and bottom temperatures beyond the influence of Brayton Point Station's thermal plume may reach into the high 70's while water temperatures during winter may occasionally reach the freezing point.

There are four generating units at Brayton Point with a combined capacity of 1600 MWe. A once-through condenser cooling system is used for Units 1, 2, and 3. The design intake flow for the three units is 620,000 gpm, and the average design temperature rise is 14.8°F. The intake for these three units is located on the eastern side of the plant site on the banks of the Taunton River. The discharge back to the bay is via a 3200 ft. channel which terminates at the southern tip of the plant site at a venturi designed to promote mixing. At mean low water, exit velocity is approximately 8 ft/sec.

Objectives

The condenser cooling system for Unit 4 is presently closed-cycle. New England Power Company hopes to convert Unit 4 to open cycle cooling. In that case, the intake flow for the fourth unit would be drawn through a porous dike intake to the west of the plant site, on the banks of the Lee River. The discharge from Unit 4 would be combined with the existing discharges from Units 1, 2 and 3. An analysis of the added thermal loading to Mount Hope Bay and the potential for intake recirculation, both associated with the conversion of Unit 4 to open cycle cooling, provide the motivation for the application of the near-far field modeling.
approach to the site. Specific objectives include the following:
(i) generally improved capability to simulate the transient nature of
the thermal plume as a function of tidal stage; (ii) improved assessment
of the influence of background temperature on the extent of the plume;
(iii) determination of the distribution of heated water in the Lee and
Taunton Rivers which could lead to intake recirculation. Portions of
these objectives can be addressed with the present methodology while
other portions require separate analysis.

Site Schematization

The finite element grid is shown in Figure 4-14, 15. The triangular
elements of the grid allowed the computational domain to model the
boundaries and general shape of the actual site well. In accordance with
the discussion in Section 3.3 the domain had a maximum depth of 11.5 ft.
corresponding to the estimated far field plume thickness listed in
Table 4-4. The depth was adjusted in shallow areas to account for
depths less than 11.5 ft. This was only a concern in the area of the
Lee River and along the shore boundaries (where the shoreline nodal
points had values specified as 6.6 ft.). The near field region was carved
out of the domain in the region surrounding Brayton Point. The discharge
channel at Brayton Pt. runs along side the peninsula. The geometry of
the discharge called for a modification in design from the semi-circular
shape used at Millstone and the idealized domain. At Brayton Pt.
the semi-circle was extended up from its ends to form what would have
been a full circle except for the width of the peninsula. As before,
Figure 4-14: Large, Coarse Grid Used at Brayton Point Simulation.
Figure 4-15: Transitional Grid Portion Between Near Field–Far Field Interface and Coarse Grid.
this near-far field boundary was lined with the smallest elements. The smallest element side length was on the order of 150 ft. The maximum side length, typical of many of the elements in the central portion of the domain, was about 2000 ft.

The near field influences within the plume lead to a transition distance between the near and far field of around 950 ft. for three units and 1270 ft. for four units (a value of about 1000 ft. was used). The value of $r_s$, the shallow depth dilution reduction factor was 0.65 for three units and 0.52 for four units. The dilution $S$, along with the values of the horizontal and vertical entrainment coefficients, $E_h$ and $E_v$, are summarized in Table 4-4.

Two different sets of discharge and entrainment fluxes were specified at the near-far field interface, corresponding to the simulation of three and four unit operation. In both cases, three nodal points at the bottom (southern end) of the interface had specified normal fluxes into the domain. The nodes adjacent to this string of three had zero normal flux and all the remaining nodes had specified entrainment fluxes out of the domain (and into the theoretical near field jet region). For 3 unit simulation, the discharging nodal values were 11.0 ft$^2$/sec. over a length of 980 ft. For 4 unit operation, the discharging nodal values were 16.6 ft$^2$/sec. As before it was decided that the most appropriate distribution of entrainment fluxes about the circular near-far boundary was an equal normal flux value at each set of nodes on the two sides of the discharge. For three unit simulation, the fluxes on the west side were 0.37 ft$^2$/sec. over 2700 ft. and on the east side were 0.56 ft$^2$/sec.
Table 4-4: Near Field Parameters Computed for Brayton Pt. Simulation

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Symbol</th>
<th>3 Unit Values*</th>
<th>4 Unit Values*</th>
</tr>
</thead>
<tbody>
<tr>
<td>Discharge Flow Rate</td>
<td>$Q_o$ (cfs)</td>
<td>1380</td>
<td>1960</td>
</tr>
<tr>
<td>Discharge Temperature Rise</td>
<td>$\Delta T_o$ ($^\circ$F)</td>
<td>14.8</td>
<td>15.7</td>
</tr>
<tr>
<td>Discharge Velocity</td>
<td>$u_o$ (fps)</td>
<td>4.0– 6.4</td>
<td>5.6– 9.1</td>
</tr>
<tr>
<td>Discharge Channel Depth</td>
<td>$h_o$ (ft.)</td>
<td>7.4–11.9</td>
<td>7.4–11.9</td>
</tr>
<tr>
<td>Discharge Channel Half-Width</td>
<td>$b_o$ (ft.)</td>
<td>14.6</td>
<td>14.6</td>
</tr>
<tr>
<td>Discharge Channel Characteristic Length</td>
<td>$l_o$ (ft.)</td>
<td>10.4–13.2</td>
<td>10.4–13.2</td>
</tr>
<tr>
<td>Densimetric Froude No. (based on $h_o$)</td>
<td>$F_o$</td>
<td>4.2– 8.5</td>
<td>5.7–11.7</td>
</tr>
<tr>
<td>Channel Aspect Ratio</td>
<td>$h_o/b_o$</td>
<td>.51–.82</td>
<td>.51–.82</td>
</tr>
<tr>
<td>Densimetric Froude No. (based on $l_o$)</td>
<td>$F'_o$</td>
<td>4.0– 7.2</td>
<td>5.5–10.0</td>
</tr>
<tr>
<td>Transition Radius</td>
<td>$r_t$ (ft.)</td>
<td>790–1120</td>
<td>1080– 1550</td>
</tr>
<tr>
<td>Near Field Volumetric Dilution</td>
<td>$S$ **</td>
<td>5.6–10.1</td>
<td>7.6–13.9</td>
</tr>
<tr>
<td>Vertical Mass Entrainment</td>
<td>$E_v$ **</td>
<td>3.6– 7.4</td>
<td>5.4–10.7</td>
</tr>
<tr>
<td>Horizontal Mass Entrainment</td>
<td>$E_h$</td>
<td>1.0– 1.6</td>
<td>1.3– 2.2</td>
</tr>
<tr>
<td>Maximum Plume Depth</td>
<td>$h_{max}$ (ft.)</td>
<td>22.1–31.4</td>
<td>30.2–43.4</td>
</tr>
<tr>
<td>Far Field Plume Depth</td>
<td>$h_{far}$ (ft.)</td>
<td>11.1–15.7</td>
<td>15.1–21.7</td>
</tr>
<tr>
<td>Shallow Water Dilution Correction</td>
<td>$r_s$</td>
<td>0.65</td>
<td>0.52</td>
</tr>
</tbody>
</table>

* Ranges given refer to variation over tidal cycle. Values in parenthesis were used in simulation.

**These parameters were altered for model use to account for shallowness effects: $S_{\text{actual}} = r_s S$; $E_{v,\text{actual}} = E_v - (S - S_{\text{actual}})$. 

86
over 2100 ft. Under four unit simulation, the west side fluxes were 0.67 ft.²/sec. and the east side fluxes were 1.04 ft.²/sec.

The intake for the first three units was simulated by removing flux from three nodes within the shipping channel inlet as shown in Figure 4-13. The specified flux was 0.42 ft.²/sec. over 4500 ft. In the four unit simulation, the intake for the fourth unit only was added to the west side of Brayton Pt., or on the Lee River boundary. This specified normal flux was 0.43 ft.²/sec. over a length of 2000 ft.

In accounting for entrainment into the bottom of the plume at Brayton Pt., fluxes were specified to leave the domain through boundary nodes in the lower half of the domain; this removal was distributed over long distances to ensure that it would have a small effect on the circulation pattern. With three unit simulation, the fluxes used were 0.18 ft.²/sec. over a distance of 20860 ft. The fluxes used for four units were 0.25 ft²/sec.

To simulate ambient circulation in the domain, the two strings of nodes along the bottom of the bay were designated ocean boundaries. These nodes had specified tidal amplitude of 2.95 ft. (as determined from a tidal gauge at the station), period = 44640 sec., and no phase lag. The resulting tidal motion dominated the ambient circulation pattern in the bay. (It should be noted that this circulation is somewhat exaggerated, especially in the deeper areas of the bay where the actual water depth exceeds the 11.5' which was modeled.)

In addition to the tide, an inflow of 435 cfs from the Taunton River was established as a flux of 0.22 ft.²/sec. over a length of 2000 ft. In
order to provide a general comparison with field data collected at
different times, the simulations were performed with zero wind speed.

To simulate the heat being discharged into the receiving water,
equal temperatures were specified at each of the three discharge nodes.
In the running of the Bryaton Pt. simulation, values were input con-
sistent with the recorded discharge temperature rise at the time being
studied.

The model was run in the excess temperature mode. The surface heat
loss was simulated by making use of actual meteorological measurements
recorded during the time period of interest. Average values of the
various meteorological inputs were obtained from the week preceeding
the day for which the plume field data were given (August 25, 1976).
The average meteorological data were used to compute a surface heat
transfer coefficient of $K = 157 \text{ BTU/ft.}^2\cdot\text{F-day}$. The corresponding
first order decay coefficient $k = 2.5 \times 10^{-6} \text{ sec.}^{-1}$ was based on $K$ and
a water depth of 11.5 ft.

**Time Step**

The four physical properties which the CAFE time step depended on
were (i) $u$ (3 units) = 0.96 ft./sec. and $u$ (4 units) = 1.45 ft./sec.;
(ii) $l = 150 \text{ ft.}$; (iii) $E_{ij}$ which varied from 1100 ft.$^2$/sec. (for small
elements) to 5400 ft.$^2$/sec. (for large elements); (iv) $c = 19 \text{ ft./sec.}$
These values led to a time step of 2.0 secs. for CAFE. In determining
the time step necessary for DISPER, the general physical relationships
discussed earlier were utilized. These criteria along with test runs
using various values of the dispersion coefficient, D, resulted in the final use of $D = 110 \text{ ft}^2/\text{sec.}$ and $\Delta t = 12.0 \text{ secs.}$

**Results**

Figures 4-16 through 4-27 show the circulation pattern in Mt. Hope Bay for three and four unit operation over various phases of the tide: 1 hour before maximum flooding, high slack, one hour before maximum ebbing, and low slack. These figures illustrate the overwhelming influence of the tide.

An inspection of the flow east of the discharge for the three unit case shows an inconsistency in the flow field; the region between the near-far and the shoreline boundaries displays a circulation pattern (eddy) which does not appear to be physically realistic. This anomalous circulation is absent in the four unit case. One explanation for this circulation is that it is the result of numerical problems in handling the relatively large fluxes (from the discharge) being forced against the no-flux boundary (eastern shore of Mt. Hope Bay). Related problems with instability in this area of the domain (in earlier model runs) led us to adopt the spatial variation of eddy viscosity coefficients within the model. By specifying larger coefficients within the elements most effected by this problem, gradients in the flow were reduced, thus lessening the likelihood for instability. A limit on the maximum eddy viscosity was established by the choice of a 2 sec. time step. It is probable that the circulation pattern could have been improved, at the expense of greater computational time, by choosing a larger eddy viscosity coefficient near the near-far boundary, and a shorter time step.
Figure 4-16: Large View of Mt. Hope Bay CAFE Results Under Maximum Flood Conditions for Three Unit Operation.
Figure 4-17: View of Region Near Discharge at Brayton Pt. Showing CAFE Results Under Maximum Flooding Conditions For Three Unit Operation.
Figure 4-18: View of Region Near Discharge at Brayton Pt. Showing CAFE Results Under High Slack Conditions for Three Unit Operation.
Figure 4-19: Large View of Mt. Hope Bay CAFE Results Under Maximum Ebb Conditions for Three Unit Operation.
Figure 4-20: View of Region Near Discharge at Brayton Pt. Showing CAFE Results Under Maximum Ebb Conditions for Three Unit Operation.
Figure 4-21: View of Region Near Discharge at Brayton Pt. Showing CAFE Results Under Low Slack Conditions for Three Unit Operation.
Figure 4-22: Large View of Mt. Hope Bay CAFE Results Under Maximum Flood Conditions for Four Unit Operation.
Figure 4-23: View of Region Near Discharge at Brayton Pt. Showing CAFE Results Under Maximum Flood Conditions for Four Unit Operation.
Figure 4-24: View of Region Near Discharge at Brayton Pt. Showing CAFE Results Under High Slack Conditions for Four Unit Operation.
Figure 4-25: Large View of Mt. Hope Bay CAFE Results Under Maximum Ebb Conditions for Four Unit Operation.
Figure 4-26: View of Region Near Discharge at Brayton Pt. Showing CAFE Results Under Maximum Ebb Conditions for Four Unit Operation.
Figure 4-27: View of Region Near Discharge at Brayton Pt. Showing CAFE Results Under Low Slack Conditions for Four Unit Operation.
A possible reason why the four unit calculations don't display this problem is that the influence of the additional intake on the west side of Brayton Pt. induces the discharge plume over towards this intake. This would lessen the impact of the discharge momentum on the shoreline.

The results from the DISPER computations for three units at Brayton Pt. on August 25, 1976, are shown in Figures 4-28 through 4-31. The isotherm temperature values reflect the discharge temperature as recorded by the station for the particular hour presented. Superimposed on these computed isotherms is the field data describing the actual plume during the time period of interest. This field data has been depth averaged to be comparable to the computational results.

It should be added that additional field data was available for subsequent phases of the tidal cycle (extending into August 26, 1976). However, because only surface data was available, it was not realistic to attempt comparison with the depth-averaged calculations. Comparison of measured excess surface temperatures one tidal cycle apart (e.g., high slack at 0800 on August 25 compared with high slack at 2000 on August 25) indicates significant variability on a scale similar to the variability between measured and computed isotherms shown in Figures 4-28 through 4-31. This indicates the general difficulty of matching the results of the computations with available field data.

In general, the calculated three unit plumes don't extend as far into the domain as the field data indicates. While this isn't the case for all isotherms in all four phases, it seems to be a general pattern. It is probable that the anomolous circulation patterns to the southeast
Figure 4-28: DISPER Results at Brayton Pt. for Three Unit Operation Under Maximum Flood Conditions.
Figure 4-29: DISPER Results at Brayton Pt. for Three Unit Operation Under High Slack Conditions.
Figure 4-30: DISPER Results at Brayton Pt. for Three Unit Operation Under Maximum Ebb Conditions.
Figure 4-31: DISPER Results at Brayton Pt. for Three Unit Operation Under Low Slack Conditions.
of the discharge have reduced the advection of heat into the receiving waters and thereby shortened the plume length.

Table 4-5 shows the areas in \( \text{ft.}^2 \) of the various isotherms (both calculated and field) for three unit operation at Brayton Pt. as well as the calculated isotherm for four unit operation. This information shows that, while the field isotherms are generally a little larger than the calculated ones, their size differential is not as great as a qualitative comparison may indicate. In particular, a comparison of the Max Ebb case shows the field data 1.3\(^\circ\) F isotherm to be 26 \( \times 10^6 \) \( \text{ft.}^2 \) in area and the calculated 1.4\(^\circ\) F isotherm to be 21 \( \times 10^6 \) \( \text{ft.}^2 \). The Max Flood case shows a bigger discrepancy for these low temperature isotherms with a field 1.2\(^\circ\) F area of 49 \( \times 10^6 \) \( \text{ft.}^2 \) and a calculated 1.3\(^\circ\) F of 25 \( \times 10^6 \) \( \text{ft.}^2 \).

A check can also be made of the higher temperature isotherms. For example, at high slack, the field data indicate a 3.3\(^\circ\) F isotherm extending just beyond the transition circle in reasonable correspondence with the calculations. In general the field temperatures in the range of 2–3\(^\circ\) F appear to be similar to the corresponding predictions. Because temperatures in this range reflect the near field the agreement could be improved by adjusting the near field dilution parameters. Recall that the plume is bottom attached in the near field and that the estimated dilution was computed based on discharge conditions and on local water depth, but without specific calibration to the Brayton Pt. field data. If the depth used in calculating the dilution reduction was greater (less) than the actual depth, the computed near field temperatures would be too low (high) compared with field data.
<table>
<thead>
<tr>
<th></th>
<th>3 UNITS</th>
<th></th>
<th>4 UNITS</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>FIELD</td>
<td>CALCULATED</td>
<td>FIELD</td>
<td>CALCULATED</td>
</tr>
<tr>
<td>ΔT(°F)</td>
<td>AREA(10^6 ft.²)</td>
<td>ΔT(°F)</td>
<td>AREA(10^6 ft.²)</td>
<td>ΔT(°F)</td>
</tr>
<tr>
<td>MAX</td>
<td>1.7</td>
<td>17</td>
<td>2.5</td>
<td>6.3</td>
</tr>
<tr>
<td>FLOOD</td>
<td>1.2</td>
<td>49</td>
<td>1.3</td>
<td>25</td>
</tr>
<tr>
<td>HIGH</td>
<td>3.3</td>
<td>1.8</td>
<td>3.3</td>
<td>0.8</td>
</tr>
<tr>
<td>SLACK</td>
<td>2.8</td>
<td>3.5</td>
<td>2.6</td>
<td>5.8</td>
</tr>
<tr>
<td>MAX</td>
<td>1.8</td>
<td>18</td>
<td>2.8</td>
<td>5.4</td>
</tr>
<tr>
<td>EBB</td>
<td>1.3</td>
<td>26</td>
<td>1.4</td>
<td>21</td>
</tr>
<tr>
<td>LOW</td>
<td>3.4</td>
<td>2.2</td>
<td>2.6</td>
<td>1.2</td>
</tr>
<tr>
<td>SLACK</td>
<td>2.2</td>
<td>32</td>
<td>2.0</td>
<td>6.5</td>
</tr>
<tr>
<td></td>
<td>1.0</td>
<td>28</td>
<td>1.2</td>
<td>94</td>
</tr>
</tbody>
</table>
In comparing the shape and location of the calculated plumes with the field data, it appears that the calculated plumes remain somewhat to the left of the field plume. This may largely be explained with the fact that wind effects haven't been included in the simulation of Brayton Pt. On the day being modeled, the wind was generally blowing from west to east. If added to the model (a simple procedure), the simulated plume would have reflected this effect by shifting to the right somewhat, more in keeping with the field data.

The fact that the simulated plume shapes don't exactly match the field data also reflects the average conditions used for input specifications of the model. Meteorology, mass flow rate and distributions, dilution and plume depths are all model properties which were treated as constants.

The four unit DISPER calculations were based on the parameters listed in Table 4-4. Isotherms are drawn in Figures 4-32 through 4-35 and the isotherm areas are summarized in Table 4-5. The shapes of the four unit isotherms appear more realistic (and are more similar to the three unit measured isotherms) than the computed three unit isotherms. This improvement is attributed to the improved circulation patterns observed with the four unit calculations.

It is noted from Table 4-4 that both the discharge temperature rise $\Delta T_o$ and the diluted temperature entering the far field $\Delta T_o/S$ are similar for three and four units. Thus the size of the near field isotherms should be similar for both three and four units. However, because the heat loading for four units is about 1.5 times that for
Figure 4-32: DISPER Results at Brayton Pt. for Four Unit Operation Under Maximum Flood Conditions.
Figure 4-33: DISPER Results at Brayton Pt. for Four Unit Operation Under High Slack Conditions.
Figure 4-34: DISPER Results at Brayton Pt. for Four Unit Operation Under Maximum Ebb Conditions.
Figure 4-35: DISPER Results at Brayton Pt. for Four Unit Operation Under Low Slack Conditions.
three units, the far field isotherms for four units are expected to be significantly larger. Examination of Table 4-5 suggests that the areas of the lower temperature isotherms for four units are about two to three times the equivalent areas for three units.

In both the three and four unit simulation there was a build-up of heat in the area surrounding the Taunton River intake. This didn't match well with the three unit field results which showed the area to be relatively unaffected by the thermal plume. The reasons for the computed rise in temperature are thought to be the implicit assumption that the intake flow was drawn from the surface (upper 11.5 ft.) and the relatively high value of dispersion coefficient which the model requires. In reality depths near the intake range between 20 and 35 feet so a portion of the intake flow could be drawn from the lower (colder) depths and therefore not represent the recirculation of previously discharged water from the surface.

The relationship between modeled and "true" dispersion coefficient was analyzed using formulae in Christodoulou et al., 1976 (Equation 3.9 of this report). Two different locations in the domain were examined—one representing the intermediate field portion having large velocity gradients and the other representing the far field domain with smaller ambient currents. Both locations yielded identical values of the "true" dispersion coefficient, 2.6 ft.\(^2\)/sec. This was approximately an order of magnitude lower than the value utilized in the running of the model. Lower values of the dispersion coefficient had been tried in the running of DISPER; however, this had led to numerical oscillations between
temperatures at adjacent nodes. It was concluded that it would be necessary to reduce the size of the grid to numerically simulate the lower dispersion. An alternative, which could be more economically tested, would be to specify different values of (local) longitudinal and lateral dispersion.

In addition to the general circulation and plume movement in the domain, the sponsors were interested in the specific thermal structure of the Lee River basin to the west of Brayton Pt. Summer field data of this region showed a gradient of surface temperature in the basin between the entrance near the tip of Brayton Pt. and the upper basin area around the proposed intake of approximately 30°F. This was of engineering concern due to the possibility of withdrawing high temperature intake water which would decrease the efficiency of the fourth unit.

It was believed that the addition of the intake on the west side of Brayton Pt. would induce more water from the cooler far field to enter the Lee River. The main question involved with this theory was how much, if any, of the intake flow into the Lee River would be from the near field of the thermal plume. The near field plume would be warmer than the far field and might serve to increase, rather than decrease, temperatures in the Lee River.

The results of both the three and four unit DISPER simulations show approximately the same amount of induced warming in the Lee River.
basin. In neither case does the lowest plotted isotherm reach the proposed fourth unit intake location. This fact suggests that the additional water being pulled in by the fourth unit intake is mainly from the cooler far field.

In a related effort, an analytical model has been formulated to address the heat distribution in the Lee River basin. The fourth unit CAFE results from the present work can be used to estimate the percentage of intake of flow entering the Lee River basin, which originates from the thermal plume. This recirculation coefficient is similar to the value of $\alpha$ defined in Figure 2-4 and is estimated to be about 0.25.

The background temperature of the domain and its influence on the plume were not directly assessed by the present modeling effort. The simulation was carried out without considering boundary condition temperatures on the domain (e.g., at the entrance to Mt. Hope Bay) other than at the points of discharge. This simulation is consistent with the concept of calculating excess temperatures over the ambient conditions. Nor were the computations allowed to continue long enough to reach a thermodynamic equilibrium with the atmosphere. Such calculations would require on the order of 10 days and would be hypothetical in the sense that actual meteorological conditions would change during this period. The calculations which were presented, then, should be viewed as excess temperatures above a background (bay-wide temperature). Based on a bay-wide heat budget analysis, the background temperature rise (averaged over the upper 11.5 ft) is estimated to range between .5 and .75°F for 3 units and between .75 and 1.1°F for 4 units.
CHAPTER V
SUMMARY AND CONCLUSIONS

5.1 Summary

This study has described a methodology for coupling near field and far field models to describe the effects of a surface thermal discharge on an aquatic environment. This was accomplished by (1) making use of analytical expressions (based on theoretical and experimental results) for near field jet dynamics, (2) using this knowledge to develop boundary conditions on the far field region, and (3) applying these boundary conditions in a 2-D, transient numerical analysis using the finite element models CAFE and DISPER. In this way the natural influences of tidal circulation and dispersion could be combined with the influences from the discharge. The model was applied to an idealized domain and to two prototype domains: (1) the Millstone Nuclear Power Station and (2) the Brayton Point Generating Station.

5.2 Conclusions

Physically, the formulation of the model with the near field-far field boundary is very realistic. While the numerical program utilized is a far field type, the near field jet effects on the flow and heat patterns are accounted for directly through the boundary conditions which can be directly related to observed near field mixing. This is an advantage of the model over other models which do not take into account the combined influence of near and far field regions.

The patterns of circulation produced by this model are generally reasonable, showing well the influence of the near field discharge and
entrainment on the ambient circulation. Drawbacks to the present form of the calculations are (1) the use of steady (rather than tidally-varying) boundary conditions between near and far field and the use of an essentially constant layer depth in the far field. The former assumption prevents the plume trajectory from changing in the near field while the latter assumption results in exaggerated ambient (e.g., tidal) velocities in regions where the actual depths are significantly deeper than modeled. Both of these assumptions should be explored in future extensions of this work.

Temperatures are predicted using an excess temperature formulation using steady meteorological conditions (constant heat transfer coefficient). The capability also exists to perform absolute temperature calculations. This option was exercised in the related work by Ostrowski (1980), but was not considered cost effective for the present applications because of the additional computational time involved.

Comparison between measured and predicted isotherms at the two sites indicate generally good agreement with regards to isotherm lengths and area. There seems to be too much dispersion in the model, however, relative to actual physical dispersion at the sites. This could be partially rectified by reducing the minimum element side length \( L \), and the time step \( \Delta t \). This wasn't done in the present study due to the large amount of time required for this sort of sensitivity. Further effort needs to be devoted to the study of numerical dispersion (and ways to reduce it) in the context of the present application.

A major limitation of the numerical scheme is the time required
for running the model. Actual running times were mentioned at the conclusion of the Millstone results section. These long times limited the calculations to the order of one tidal cycle of circulation analysis and prevented extensive sensitivity studies.

5.3 Suggestions for Further Research

A major area requiring more investigation is numerical sensitivity analysis. Much of this should be carried out at a practice domain, similar to the one used here, where more control exists over the physical characteristics of the site.

The relationships between eddy viscosity and dispersion, bottom bathymetry, grid design and numerical time steps should be explored in more detail in relationship to computational cost and accuracy. Further refinement in the specification of boundary conditions, to improve comparisons between field and calculated isotherms, would be useful. This would include the development of the time-varying boundary condition as described in 4.2. Finally, different numerical routines, especially simplified, more cost effective codes appropriate for steady state analysis, should be explored.

It would also be worthwhile to apply the model to other discharging conditions. Submerged diffusers may be of interest, particularly in assessing the effects of high discharge velocity (and attendant rip-type currents) on the ambient circulation pattern. A diffuser plume, because of the high degree of mixing, could be more appropriately analyzed as virtually well-mixed (over the entire water column) than the surface discharge plumes treated in this study. A
river discharge would be another possible application - made simpler because the crossflow could be considered steady.
REFERENCES


