University of Minnesota
St. Anthony Falls Hydraulic Laboratory
Mississippi River at 3rd Avenue S.E.
Minneapolis, Minnesota 55414

Project Report No. 241

BUOYANCY INDUCED PLUNGING FLOW INTO RESERVOIRS
AND COASTAL REGIONS

by

Gerard J. Farrell
and
Heinz G. Stefan

Prepared for:
NATIONAL SCIENCE FOUNDATION
Washington, D. C. 20550
Grant No. CEE 8308471

July, 1986
The University of Minnesota is committed to the policy that all persons shall have equal access to its programs, facilities, and employment without regard to race, creed, color, sex, national origin, or handicap.
TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page No.</th>
</tr>
</thead>
<tbody>
<tr>
<td>List of Figures</td>
<td>iv</td>
</tr>
<tr>
<td>List of Tables</td>
<td>ix</td>
</tr>
<tr>
<td>I. INTRODUCTION</td>
<td>1</td>
</tr>
<tr>
<td>A. Plunging Flow</td>
<td>1</td>
</tr>
<tr>
<td>B. Review of Previous Work</td>
<td>1</td>
</tr>
<tr>
<td>C. Outline of Present Work</td>
<td>7</td>
</tr>
<tr>
<td>II. ASPECTS OF PLUNGING FLOW DYNAMICS</td>
<td>10</td>
</tr>
<tr>
<td>A. General</td>
<td>10</td>
</tr>
<tr>
<td>B. Sloping Reservoir</td>
<td>12</td>
</tr>
<tr>
<td>1. Governing equations</td>
<td>12</td>
</tr>
<tr>
<td>2. Dimensional considerations</td>
<td>17</td>
</tr>
<tr>
<td>3. Basic concepts of plunging flow and underflow</td>
<td>20</td>
</tr>
<tr>
<td>4. Two-layer flow analyses</td>
<td>29</td>
</tr>
<tr>
<td>C. Diverging Reservoir</td>
<td>33</td>
</tr>
<tr>
<td>1. Governing equations</td>
<td>33</td>
</tr>
<tr>
<td>2. Dimensional considerations</td>
<td>35</td>
</tr>
<tr>
<td>3. Two-layer analysis</td>
<td>36</td>
</tr>
<tr>
<td>a. Free surface flow in a diverging channel</td>
<td>37</td>
</tr>
<tr>
<td>b. Two-layer flow in a diverging channel</td>
<td>43</td>
</tr>
<tr>
<td>4. Separated Flow</td>
<td>51</td>
</tr>
<tr>
<td>III. NUMERICAL PROCEDURE</td>
<td>54</td>
</tr>
<tr>
<td>A. Introduction</td>
<td>54</td>
</tr>
<tr>
<td>B. Numerical Scheme</td>
<td>54</td>
</tr>
<tr>
<td>1. General procedure</td>
<td>54</td>
</tr>
<tr>
<td>2. Computation of velocity field</td>
<td>55</td>
</tr>
<tr>
<td>3. Unsteady problems</td>
<td>56</td>
</tr>
<tr>
<td>4. Summary</td>
<td>58</td>
</tr>
<tr>
<td>C. Transient test problem</td>
<td>58</td>
</tr>
<tr>
<td>D. Numerical diffusion</td>
<td>60</td>
</tr>
<tr>
<td>E. Boundary conditions</td>
<td>63</td>
</tr>
</tbody>
</table>
IV. NUMERICAL SOLUTIONS—LAMINAR FLOW

A. Sloping Reservoir
   1. Governing equations ....................... 68
   2. Qualitative model testing ................. 70
      a. Boundary and initial conditions ....... 72
      b. Grid size and time step ............... 72
      c. Results and conclusion ................. 73
   3. Quantitative Model Testing ............... 81
      a. Non-dimensional equations .............. 81
      b. Initial examination ..................... 83
      c. Effect of grid size and time step ....... 85
      d. Physical considerations .................. 89
B. Diverging Reservoir
   1. Governing equations ....................... 94
   2. Qualitative model testing .................. 98
   3. Quantitative model testing ............... 102
C. Closure ....................................... 104

V. NUMERICAL SOLUTIONS—TURBULENT FLOW

A. Sloping Reservoir
   1. Governing equations ....................... 107
      a. Mean flow equations ...................... 107
      b. Turbulence model equations .............. 109
      c. Equation summary ........................ 110
   2. Boundary and initial conditions .......... 110
      a. Boundary conditions ..................... 110
         Velocity
         Temperature
         Turbulent kinetic energy
         Kinetic energy dissipation rate
      b. Initial conditions ....................... 115
      c. Effect of inflow and initial conditions . 119
   3. Model reservoir ............................ 119
   4. Computational parameters ................. 120
      a. NTIMES and LAST .......................... 120
      b. Grid size .................................. 122
      c. Time step .................................. 126
   5. Examination of turbulence model coefficients ... 130
      a. Coefficient $C_H$ ....................... 130
      b. Turbulent Prandtl number $\sigma_t$ ....... 132
      c. Coefficient $C_3$ ....................... 134
      d. Summary .................................. 139
   6. Results ..................................... 139
      a. Introduction .............................. 139
      b. Flow field features ...................... 143
      c. Two-layer flow analysis ................. 160
<table>
<thead>
<tr>
<th>Figure No.</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>I-1</td>
<td>Plunging flow on a sloping bottom.</td>
</tr>
<tr>
<td>I-2</td>
<td>Model reservoir configuration with sloping bottom.</td>
</tr>
<tr>
<td>I-3</td>
<td>Model reservoir configuration with diverging sides.</td>
</tr>
<tr>
<td>II-1</td>
<td>Time scales in a reservoir.</td>
</tr>
<tr>
<td>II-2</td>
<td>Reservoir in cylindrical coordinates.</td>
</tr>
<tr>
<td>II-3</td>
<td>Water temperature/density relation.</td>
</tr>
<tr>
<td>II-4</td>
<td>Plunge region and downstream density current.</td>
</tr>
<tr>
<td>II-5</td>
<td>Velocity and temperature profiles in various flow types.</td>
</tr>
<tr>
<td>II-6</td>
<td>Force balance on an underflow.</td>
</tr>
<tr>
<td>II-7</td>
<td>Illustration of effect of density current normal depth on plunge region position.</td>
</tr>
<tr>
<td>II-8</td>
<td>Entrainment situation downstream of the plunge point.</td>
</tr>
<tr>
<td>II-9</td>
<td>Two-layer flow.</td>
</tr>
<tr>
<td>II-10</td>
<td>Diverging reservoir configuration in an $r, \theta, z$ coordinate system.</td>
</tr>
<tr>
<td>II-11</td>
<td>Free surface flow situation.</td>
</tr>
<tr>
<td>II-12</td>
<td>Possible water surface profiles when $G_o &gt; 1$.</td>
</tr>
<tr>
<td>II-13</td>
<td>Possible water surface profiles when $G_o &lt; 1$.</td>
</tr>
<tr>
<td>II-14(a)</td>
<td>Plot of Eq. II-60 for two $G_o$ values ($D = 0.6$ and $\alpha = 0.5$).</td>
</tr>
<tr>
<td>II-14(b)</td>
<td>Section A-A on Fig. 14(a). Represents plot of function $p$ (Eq. II-59) at $R = 14$.</td>
</tr>
<tr>
<td>II-15</td>
<td>Plunging with large $G_o$.</td>
</tr>
<tr>
<td>II-16</td>
<td>Plunging with small $G_o$.</td>
</tr>
<tr>
<td>II-17</td>
<td>Flow situation with Type I plunging ($G_o$ large).</td>
</tr>
<tr>
<td>Figure No.</td>
<td>Description</td>
</tr>
<tr>
<td>-----------</td>
<td>-------------</td>
</tr>
<tr>
<td>II-18</td>
<td>Flow situation with Type II plunging ($G_o$ small).</td>
</tr>
<tr>
<td>II-19</td>
<td>Plunge point densimetric Froude number behavior.</td>
</tr>
<tr>
<td>III-1</td>
<td>Typical control volume and grid layout.</td>
</tr>
<tr>
<td>III-2</td>
<td>Schematic of computational scheme.</td>
</tr>
<tr>
<td>III-3</td>
<td>Analytical and numerical solutions of a front progression.</td>
</tr>
<tr>
<td>III-4</td>
<td>Typical flow situation giving rise to skewness error.</td>
</tr>
<tr>
<td>III-5</td>
<td>Rigid lid approximation and M1 profile in reservoir.</td>
</tr>
<tr>
<td>III-6</td>
<td>Rigid lid approximation and buoyancy effect.</td>
</tr>
<tr>
<td>IV-1</td>
<td>Rectangular reservoir in Cartesian coordinates.</td>
</tr>
<tr>
<td>IV-2</td>
<td>Reservoir with blocked-out sections to simulate a slope.</td>
</tr>
<tr>
<td>IV-3</td>
<td>Schematic of experimental arrangement of Singh and Shah [1971].</td>
</tr>
<tr>
<td>IV-4</td>
<td>Velocity behavior near plunge point for three time steps.</td>
</tr>
<tr>
<td>IV-5</td>
<td>Reservoir flow development in simulation of experiment of Singh and Shah [1971].</td>
</tr>
<tr>
<td>IV-6</td>
<td>Reservoir flow development in simulation of experiment of Singh and Shah [1971].</td>
</tr>
<tr>
<td>IV-7</td>
<td>Reservoir flow development in simulation of experiment of Singh and Shah [1971].</td>
</tr>
<tr>
<td>IV-8</td>
<td>Velocity and temperature field configuration at the plunge point.</td>
</tr>
<tr>
<td>IV-9</td>
<td>Comparison of predicted plunge points with data of Singh and Shah [1971].</td>
</tr>
<tr>
<td>IV-10</td>
<td>Comparison of experimental and numerical velocity profiles near the plunge point.</td>
</tr>
<tr>
<td>IV-11</td>
<td>Non-dimensional reservoir layout.</td>
</tr>
<tr>
<td>IV-12</td>
<td>Plunge depth as a function of inflow densimetric Froude number.</td>
</tr>
<tr>
<td>IV-13</td>
<td>Effect of control volume size on predicted plunge depth.</td>
</tr>
<tr>
<td>IV-14</td>
<td>Predicted plunge depth with two Reynolds numbers.</td>
</tr>
</tbody>
</table>
Plunge depths for a constant densimetric Froude number (2.1) and control volume size as a function of Reynolds number.

Flow arrangement in an axisymmetric reservoir.

Flow field in simulation number Exp. 2.

Flow field on beach in simulation number Exp. 2.

Developed flow on beach in simulation number Exp. 1.

Definition sketch for velocity at near wall grid point.

Schematic profile of various quantities of the flow inlet plane.

Typical initial velocity field.

Model reservoir for turbulent flow simulation.

Grid size reduction pattern.

Plunge point movement with various grid patterns.

Locally defined grid pattern.

Surface velocity behavior downstream of the plunge point with various grid patterns.

Plunge point behavior with different time steps.

Surface velocity behavior downstream of the plunge point with different time steps.

Surface velocity behavior downstream of the plunge point for two $C_\mu$ values.

Vertical temperature profiles with two $\sigma_t$ values.

Surface velocity behavior downstream of the plunge point for three $C_3$ values.

Plunge point moment with two $C_3$ values.

Change in plunge point position with a $C_3$ value of unity.

Vertical profile of eddy viscosity just downstream of the plunge point.

Reservoir flow patterns.
Temperature contours of partially developed flow field.

Temperature contours of developed flow field.

Temperature contours some time after cold front reached dam.

Plunge point densimetric Froude number behavior for four runs on a slope of .02.

Plunge point densimetric Froude number behavior for three runs on a slope of .01.

Plunge depths given by numerical simulation.

Contours of shear production, \( P \).

Contours of effective viscosity, \( \nu_{\text{eff}} \).

Contours of relative numerical diffusion \( \nu_{\text{num}} / \nu_{\text{eff}} \).

Vertical profiles of temperature and velocity downstream of the plunge point.

Underflow discharge pattern downstream of the plunge point for three elapsed times.

Underflow mean velocity as a function of buoyancy flux.

Reservoir flow pattern showing underflow normal depth.

Plunge depth as a function of downstream normal depth.

Initial mixing rate, \( \gamma \), as a function of downstream normal densimetric Froude number, \( F_N \).

Typical developed velocity field in an axisymmetric situation.

Plunge point densimetric Froude number behavior (axisymmetric reservoir).

Experimental facility.

Schematic of flow arrangement.

Experimental arrangement for sloping reservoir experiments.

Behavior of layer normal depth, \( H_N \), and densimetric Froude number, \( F_N \).

Velocity and temperature profile at two locations in the sloping reservoir.
Figure No.

VI-6  Experimental data on plunge depths.

VI-7  Data from present study compared to data of Singh and Shah [1971].

VI-8  Comparison of various theories with the experimental data.

VI-9  Variation of $\Delta Q/Q_o$ with distance along reservoir.

VI-10  Experimental arrangement of diverging reservoir study.

VI-11  Schematic of flow types.

VI-12  Flow type classification.

VI-13  Flow regimes with an aspect ratio of 0.48.

VI-14  Flow regimes with a rounded transition.

VI-15  Flow situation with a separated flow.

VI-16  Plunge point densimetric Froude number, $F_p$, as a function of sidewall angle, $\delta$.

VI-17  Plunge region configuration in a type A3 flow.

VI-18  $F_p$ variation in attached flows.

VI-19  Initial mixing coefficients, $\gamma$, as a function of sidewall angle, $\delta$. 

viii
<table>
<thead>
<tr>
<th>Table No.</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>IV-1</td>
<td>Source Terms and Diffusion Coefficients for Variables with Laminar Flow in the $r, \theta$ Coordinate System</td>
</tr>
<tr>
<td>IV-2</td>
<td>Variable Ranges in the Experimental Work of Singh and Shah [1971]</td>
</tr>
<tr>
<td>IV-3</td>
<td>Variable Values for a Single Experiment</td>
</tr>
<tr>
<td>IV-4</td>
<td>Details of Numerical Simulation of the Experiments of Singh and Shah [1971].</td>
</tr>
<tr>
<td>IV-5</td>
<td>Source Terms and Diffusion Coefficients in the Non-dimensional, $r, \theta$ Equations</td>
</tr>
<tr>
<td>IV-6</td>
<td>Computer Runs to Determine Plunge Depth as a Function of Inflow Densimetric Froude Number</td>
</tr>
<tr>
<td>IV-7</td>
<td>Compute Runs to Examine Plunge Depth Variation with Various Numerical Parameters</td>
</tr>
<tr>
<td>IV-8</td>
<td>Plunge Depth Variation with Control Volume Size at a Reynolds Number of 1000</td>
</tr>
<tr>
<td>IV-9</td>
<td>Computer Runs to Examine Plunge Point Variation with Reynolds Number</td>
</tr>
<tr>
<td>IV-10</td>
<td>Source Terms and Diffusion Coefficients for Variables with Laminar Flow in the $r, z$ Coordinate System</td>
</tr>
<tr>
<td>IV-11</td>
<td>Details of Axisymmetric Reservoir Simulations</td>
</tr>
<tr>
<td>IV-12</td>
<td>Source Terms and Diffusion Coefficient in the Non-Dimensional $r, z$ Equations</td>
</tr>
<tr>
<td>V-1</td>
<td>Diffusion Coefficients and Source Terms in the Governing Equations</td>
</tr>
<tr>
<td>V-2</td>
<td>Summary of Boundary Conditions for Sloping Reservoir Simulations</td>
</tr>
<tr>
<td>V-3</td>
<td>Runs to Test Effect of Grid Size</td>
</tr>
<tr>
<td>V-4</td>
<td>Runs to Test Effect of Time Step</td>
</tr>
<tr>
<td>V-5</td>
<td>Runs to Test Effect of Turbulent Prandtl Number, $\sigma_t$</td>
</tr>
<tr>
<td>Table No.</td>
<td>Description</td>
</tr>
<tr>
<td>-----------</td>
<td>-------------</td>
</tr>
<tr>
<td>V-6</td>
<td>Runs to Test Effect of Coefficient C₃</td>
</tr>
<tr>
<td>V-7</td>
<td>Coefficient Values used in the Turbulence Model</td>
</tr>
<tr>
<td>V-8</td>
<td>Details of Reservoir Flow Simulation Runs</td>
</tr>
<tr>
<td>V-9</td>
<td>Computed Properties of the Simulated Flow Fields</td>
</tr>
<tr>
<td>V-10</td>
<td>Summary of Diffusion Coefficients and Source Terms in the Governing Equations for the Axisymmetric Situation</td>
</tr>
<tr>
<td>V-11</td>
<td>Summary of Boundary Conditions for the Axisymmetric Reservoir</td>
</tr>
<tr>
<td>V-12</td>
<td>Details of Simulations in Axisymmetric Reservoir</td>
</tr>
<tr>
<td>V-13</td>
<td>Details of Runs to Test Effect of Initial Radius</td>
</tr>
<tr>
<td>VI-1</td>
<td>Details of Experiments on Sloping Bottom Reservoir</td>
</tr>
<tr>
<td>VI-2</td>
<td>Parameters for Experiment of APR 11/86</td>
</tr>
<tr>
<td>VI-3</td>
<td>Parameters of Experiments on Sloping Bottom Reservoir</td>
</tr>
<tr>
<td>VI-4</td>
<td>Description of Flow Types</td>
</tr>
<tr>
<td>VI-5</td>
<td>Details of Experiments Showing Type I Plunging</td>
</tr>
<tr>
<td>VI-6</td>
<td>Details of Initial Mixing Experiments in the Diverging Reservoir</td>
</tr>
<tr>
<td>A1-1</td>
<td>Recommended Constant Values for the k-ε Model</td>
</tr>
</tbody>
</table>
I. INTRODUCTION

A. Plunging Flow

The water in a river flowing into a reservoir, lake or coastal region is rarely of exactly the same density as the ambient water in the waterbody. The density difference may be due to a difference in temperature or in concentration of dissolved or suspended substances. Small density differences can have dramatic effects on the flow patterns that develop in the waterbody. In particular, when the river water is denser than the ambient, the incoming flow dips beneath the ambient water and flows along the reservoir bottom or beach as a density current. Such flows are termed plunging flows.

Figure I-1 shows a plunging flow situation over a sloping bottom with various aspects of the flow illustrated. The position on the water surface where the flow actually plunges is known as the plunge point or plunge line. It will frequently be delineated by a collection of floating debris held by the reverse current generated in the ambient water by the plunged flow. After plunging, the flow becomes a density current underflow. The dynamics of such currents are reasonably well understood [Ellison and Turner, 1959].

The region surrounding the plunge point and encompassing the transition region between the river inflow and the density current is termed the plunge region. This region can be characterized by its location in the reservoir, as expressed in the case illustrated by the depth at the plunge line \( H_p \), and by the amount of mixing that occurs in the region between the inflow and ambient waters. A study of flow in the plunge region with particular reference to these two characteristics forms the subject matter of this report.

An understanding of the plunging phenomenon is important from the point of view of water quality modelling, reservoir sedimentation studies, and effluent mixing analyses. Ford and Johnson [1983] review a number of cases in which plunging flow had large water quality implications. The hydraulics of reservoir sedimentation are reviewed by Graf [1983a, b]. Stefan et al. [1984] describe some effects of plunging on effluent mixing characteristics.

B. Review of Previous Work

The first authoritative documentation of a plunging flow appears to be that of Forel [1892], who reported on the plunging of a dense inflow in Lake Geneva, Switzerland. Hinwood [1970] gives a brief review of some pre-1945 studies on dense inflows. Ford and Johnson [1983] review some more recent work. Systematic documentation and analysis of plunging flows
Fig. I-1. Plunging flow on a sloping bottom.
commenced with the study of density currents in Lake Mead on the Colorado River. These density currents are described by Grover and Howard [1938] and subsequent work is outlined by Howard [1953]. The Conference at which this latter paper was presented yielded a number of important papers on the plunging and underflow phenomenon. The first quantitative work concerning the plunge point position appears to be that of Fan [1960]. He approached the problem theoretically using a two-layer approach and also examined the phenomenon experimentally.

Singh and Shah [1971] examined the plunge flow phenomenon experimentally. Using a tilting flume about 12.0 m in length to represent the reservoir, they reproduced plunging flow by letting a heavy saline solution flow into ambient reservoir water. Water was withdrawn at the downstream end of the apparatus to preserve flow continuity. By introducing dye into the inflow, Singh and Shah were able to observe the progress of the dense flow through the reservoir. They provide a useful description of the development of plunging flow. Depths of the plunge point were measured and related to the parameter \( \left( \frac{q^2}{g'} \right)^{1/3} \) by a regression analysis to give the relation for the depth at the plunge point, \( H_p \), as

\[
H_p = 1.3 \left( \frac{q^2}{g'} \right)^{1/3} \quad (I-1)
\]

In Eq. (I-1), \( q \) is the river discharge per unit width and \( g' = \Delta \rho g / \rho_o \), where \( g \) is the acceleration due to gravity, \( \Delta \rho \) the density difference between ambient water and inflow solution, and \( \rho_o \) the density of the inflow solution. In the experiments, the densimetric Froude number at the plunge point, \( F_p \), where

\[
F_p = \left( \frac{q^2}{g'H_p^3} \right)^{1/2}
\]

varied over the range 0.3 to 0.8. In the experiments of Fan [1960], the densimetric Froude number at the plunge point had an average value of about 0.78.

Wunderlich and Elder [1973] analyzed field data available for the Natahala River inflow into Fontana Reservoir. They found that the densimetric Froude number at the plunge point for that situation was 0.5. This value corresponds to that given by a simplified analysis of Savage and Brimberg [1975]. These authors carried out an analysis analogous to the cavity flow analysis of Benjamin [1968] and the contained oil slick analysis of Wilkinson [1972]. Philpott [1978] adopted a similar approach with entrainment taken into account. Savage and Brimberg [1975] found that the densimetric Froude number at the plunge point should be 0.5. This result requires that supercritical flow occurs downstream of the plunge point. Jain [1980] points out that as such flow cannot be sustained in a horizontal channel, then energy dissipation must necessarily occur at the plunge point, and this renders the simple analysis of Savage and Brimberg [1975] invalid.
Hebbert et al. [1979] also used the momentum conservation approach of Benjamin [1968]. They considered a channel of triangular cross section and gave a relation for the depth at the plunge point in terms of the densimetric Froude number of the downstream underflow at normal depths. In Wellington Reservoir, which they considered specifically, their relation reduces to

\[ H_p = 1.16 \left( \frac{Q^2}{g'} \right)^{1/5} \]

where \( Q \) is the total inflow discharge and \( g' \) is as defined in Eq. (I-1).

Many researchers have examined plunging flow using a two-layer flow analysis. Bata [1957], while not specifically looking at plunging flow, had used this approach to compute the form of the interface profile of a warm water wedge formed by the plunging of cold inflow in a cooling water intake channel. Fan [1960] also examined plunging flow in this way. He suggests that a critical section should occur at the plunge point so that \( F_p \) should be unity. However, this is not necessarily true.

Savage and Brimberg [1975] also used the two-layer approach and integrated along the interface profile numerically to find the plunge point. They gave the relation for the plunge depth in a reservoir with a mild bottom slope as

\[ H_p = \left( \frac{q^2}{g' F_0^*} \right)^{1/3} \]

where \( q \) and \( g' \) are as defined in Eq. (I-1) and

\[ F_0^* = 2.05 \left( \frac{S}{f} \right)^{.478} \]

where \( S \) is the bottom slope, \( f \) the bottom friction factor, and \( \alpha \) the ratio of interface to bottom friction factor.

Jain [1980] also carried out a two-layer flow analysis. He integrated the interface equation in a new non-dimensional form and presented a comprehensive diagram giving the plunge depth on mild and steep slopes in terms of various reservoir parameters. His results are discussed further in Section IIB.4.

Akiyama and Stefan [1984] recognized that mixing in the plunge region was important in modifying the flow physics. They examined the balance of force in the plunge region using the momentum equation with mixing taken into account and presented new relations for the plunge depth. These relations on mild and steep slopes, respectively are
\[
H_p = \frac{1}{2} \left[ \frac{2 + \gamma}{2} + \frac{S}{K} + \sqrt{\left( \frac{2 + \gamma}{2} + \frac{S}{K} \right)^2 - \frac{4}{1 + \gamma} \left( \frac{S}{K} \right)} \right] \left( \frac{q^2}{g \epsilon} \right)^{1/3}
\] (I-2)

and
\[
H_p = \frac{1}{2} \left( \frac{2 + \gamma}{2} + S + \sqrt{\left( \frac{2 + \gamma}{2} + S \right)^2 - \frac{4}{1 + \gamma} S} \right) \left( \frac{1}{S} \right) \left( \frac{q^2}{g \epsilon} \right)^{1/3}
\] (I-3)

where \( K = \frac{f_t}{S_2} = \) constant

\( f_t = \) total friction factor

\( q = \) inflow per unit width

\[
S_2 = \frac{1}{\epsilon_b h_b} \int_0^\infty g \left( \frac{\rho_a - \rho(z)}{\rho_a} \right) \frac{\rho_a - \rho(z)}{\rho_a} \, dz
\] (I-4)

\[
S_1 = \frac{1}{\epsilon_b h_b} \int_0^\infty 2g \left( \frac{\rho_a - \rho(z)}{\rho_a} \right) z \, dz
\] (I-5)

\( \epsilon = (\rho_b - \rho_a)/\rho_a = \Delta \rho_b/\rho_a \)

\( \rho_b = \) underflow density

\( \rho_a = \) ambient water density

\( h_b = \) depth of underflow downstream of plunge point

\( \gamma = \) initial mixing coefficient (defined below)

\( \rho(z) = \) density variation with depth, \( z \)

Akiyama and Stefan [1984] provide a summary table of the various formulae available for plunge depth prediction. Ford and Johnson [1983] also summarise a number of plunging flow studies and give details of a recent series of observations in West Point Reservoir and DeGray Lake, Arkansas. They quote 0.1 to 0.7 as being the range in which typical \( F_p \) values occur. Ford and Johnson [1983] also present a comparison of the performance of various prediction formulae with their field data. On the basis of this comparison they recommend that the equations of Savage and Brimberg [1975], Hebbert et al. [1979], Jain [1980] and Akiyama and Stefan [1984] be used for plunge point predictions.

The formulae given above can be (and frequently are) applied to flow in diverging reservoirs by calculating the unit discharge \( q \) by reference to the width in the reservoir section of interest. This process is of necessity trial and error and ignores various phenomena which may be associated with a diverging situation.
Akiyama and Stefan [1986] are the only researchers to take channel divergence specifically into account in the analysis of plunging flow. Akiyama and Stefan [1986] examined plunging flow in a diverging channel theoretically and experimentally. They found densimetric Froude numbers at the plunge point varying from 0.55 to 1.0 with the Fp value being directly related to divergence angle. A series of observations at a divergence angle of $\alpha = 15^\circ$ showed sidewall flow separation for all experimental conditions.

Philpott [1978] presents the results of an experimental examination of three-dimensional laterally unconfined plunging flow over a sloping beach and gives a description of the plunge region configuration in such a flow. Hauenstein and Dracos [1984] also examined laterally unconfined flow over a sloping beach. They analyzed the flow using an integral model based on global fluxes and also examined the flow experimentally.

Hamblin and Carmack [1978] and Fischer and Smith [1983] provide some useful studies of plunging flow in situations in which the plunge behavior was modified by other hydrodynamical effects. Hamblin and Carmack [1978] explore the totality of river-lake interactions and the dynamics of lake circulation. They highlight in particular the effect of the Coriolis force in deflecting the inflow towards the righthand lake shoreline. Fischer and Smith [1983] describe the results of a field study on a plunging flow in Lake Mead. This study showed that nutrients in the plunged flow were transported to the lake surface waters by an interfacial wave motion due, most likely, to wind effects and by the formation and destruction of a temporary thermocline close to the water surface.

The amount of mixing that occurs between inflow and ambient waters in the plunge region is termed the initial mixing and is usually expressed in terms of the value of the initial mixing coefficient, $\gamma$, defined as

$$\gamma = \frac{Q_{amb}}{Q_0}$$

where $Q_{amb}$ is the amount of ambient reservoir water entrained and $Q_0$ is the inflowing river discharge.

One of the earliest references to initial mixing is that of Ryan and Harleman [1971] who recognized the importance of this parameter in water quality models. Ryan and Harleman [1971] quote mixing values of 0.5 and 2.0 as occurring in a laboratory reservoir, with the lower value referring to warm, i.e. positively buoyant inflow. Ryan and Harleman [1971] mention that river geometry influences these values but do not give details.

The field study of Elder and Wunderlich [1972] in the Tennessee River basin gave mixing of 0.02 to 0.04 for the Chilhowee Reservoir and 0.30 to 0.46 for the Norris Reservoir. Ford et al. [1980] computed mixing values of 0.04 to 0.28 from field data collected in DeGray Lake, Arkansas. A careful field study of Fischer and Smith [1983] in Lake Mead, Nevada, yielded an average initial mixing value of about 0.67. Japanese researchers quoted by Akiyama and Stefan [1986] found $\gamma$ values in the range 0.25 to 1.0 in field studies and 0.1 to 2.0 in laboratory studies.
No basic analytical studies have been done on the initial mixing phenomenon and in fact, as is discussed in Section II.B.3, the definitions of this quantity used in the literature are not uniform. A number of authors [Imgerger and Hamblin, 1982; Ford and Johnson, 1983] have suggested that a formula developed by Jirka and Wanatabe [1980] for mixing in cooling ponds be used to estimate initial plunge region mixing.

C. Outline of Present Work

In this report plunging flow is examined using two separate and distinct approaches. In the first approach, plunging reservoir flow is simulated by extracting numerical solutions of the governing equations of flow. In the second, plunging flow is studied experimentally by reproducing the phenomenon in a laboratory model reservoir.

In real reservoirs and coastal regions, the flow geometry is frequently nonuniform with irregularities in shoreline and bottom being common. As pointed out by Ford and Johnson [1980] such features may lead to incomplete lateral mixing of the inflow. These effects cannot of course be incorporated in a general study. For the purpose of this work, attention is concentrated on two model geometrical configurations which embody two broad waterbody types.

These model configurations were selected on the basis of how they model the velocity behavior in the prototype. The flow velocity in a reservoir with sloping bottom and parallel (or gently diverging) sides is influenced solely (or primarily) by the increasing depth due to the bottom slope. This situation is modelled using the two-dimensional sloping configuration shown in Fig. I-2. If the waterbody sidewalls diverge rapidly, such as occurs in estuaries and some reservoirs, the increasing width can have the primary influence on flow velocity. This situation is modelled using the axisymmetric configuration shown in Fig. I-3.

Apart from flow geometry, various other extraneous forces can exert an influence on the plunging flow dynamics. Wind and waves will exert an influence and wind, in particular, can be expected to move the plunge point on the water surface. Stratification in the reservoir will have also an effect. This effect will be mainly on the density current part of the flow, though if a thermocline exists sufficiently close to the surface, then as shown by Fischer and Smith [1983] some water from the plunge region may enter the reservoir at this level. Finally if the density differences are due to suspended material, the possibility exists that the buoyancy flux may change [Akiyama and Stefan, 1985; Parker et al., 1986] and this may influence the plunging flow dynamics. The possible effects of the above factors are not examined in this study. Attention is focused on laterally homogeneous inflow in which the buoyancy flux is conserved into a reservoir filled with homogeneous light ambient water in the absence of wind and waves.

Various aspects of plunging flow dynamics are outlined in Chapter II. This includes a review of the basic concepts of plunge region position and initial mixing. The numerical work is described in Chapters III, IV, and V. Both laminar and turbulent models are used to simulate the flow. The
Fig. I-2. Model reservoir configuration with sloping bottom.

Fig. I-3. Model reservoir configuration with diverging sides.
laminar model was used initially but was found to have serious limitations for modelling this flow so a turbulence model, the k-ε model, had to be used for the flow simulations. The experimental work is described in Chapter VI. Chapter VII contains the summary and conclusions from the study.
II. ASPECTS OF PLUNGING FLOW DYNAMICS

A. General

Plunging flow is an intrinsically unsteady flow. It does not exist in steady state. The steady solution to the situation of cold inflow into a reservoir full of warm water with balancing outflow is a reservoir full of cold water. Heat fluxes at the water surface are unlikely to be of sufficient magnitude or duration to preserve true steady-state plunging conditions.

Plunging flow can be considered to exist in a quasi-steady state. Figure II-1 illustrates in a simple model the various order of magnitude of time scales/velocity scales encountered in reservoir dynamics. The longest time-scale—that involving changes in hydrologic and meteorological conditions—can be considered as one week. Flash floods and sudden temperature changes can occur, of course, but mean conditions change rather slowly. In particular, the reservoir water surface elevation usually changes very slowly. The other end of the spectrum contains the velocities with a response time of seconds or minutes. The head of the gravity current running along the reservoir bottom is a particularly unsteady phenomenon (with respect to overall reservoir dynamics) and would have to be examined in terms of seconds or minutes.

The plunge region time scale falls somewhere between these two extremes. Field and laboratory observations indicate that the plunge region once formed tends to remain in more or less the same position in the reservoir. This is intuitively acceptable since the balance of momentum and buoyancy forces in the plunge region should not change markedly as the gravity current head moves down the reservoir provided the reservoir is of sufficient size to provide a supply of ambient water for entrainment.

Eventually if conditions persist, all the ambient water will be entrained and withdrawn, the plunge region position will move forward to the dam and a reservoir full of cold water will persist as the steady-state solution. In a reservoir with a reasonably large residence time such a complete washing out, if it could occur, would take an extremely long time.

In a prototype reservoir, the forward movement of the plunge region would have a time scale of at least one day. Thus, conditions in the plunge region in terms of the hydrodynamics may be considered to be quasi-steady. The various theoretical approaches outlined in the previous Chapter all approach plunging flow on this basis. In the following sections, steady state or quasi-steady state conditions are assumed also to prevail and various simple concepts regarding plunging flow and underflows are presented.
Fig. II-1. Time scales in a reservoir.
B. Sloping Reservoir

1. Governing equations

Plunging reservoir flow is governed by the general dynamic equations of motion and on associated density equations. These equations are usually presented in Cartesian coordinates. However, in the present case, if in the classical sloping reservoir shown in Fig. I-2, the water surface is assumed to be a horizontal rigid lid, then a cylindrical $r, \theta, z$ coordinate system can be used. The reservoir shape is easily accommodated in this system as shown in Fig. II-2. The reservoir is assumed to extend to infinity in the lateral direction so that the flow is two-dimensional. The included angle of the wedge to be studied gives the reservoir bottom slope $S$. The initial radius $R_0$ determines the inflow depth.

Since free surface phenomena such as wind or wave effects are not being considered in this study, the rigid lid approximation is reasonable. The assumption is discussed in more detail in Section III.C.

Before the governing equations are presented, three simplifications are introduced. These are the Boussinesq approximation, the introduction of a reduced pressure, and the introduction of a simplified density relation. The simplifications are illustrated below, using, for convenience, the equation for the vertical velocity in a cartesian system.

For a simple cartesian system with the $y$ axis pointing vertically downwards the relevant equation is

$$\rho \frac{\partial v}{\partial t} + \rho u \frac{\partial v}{\partial x} + \rho v \frac{\partial v}{\partial y} = -\frac{\partial p}{\partial y} + \mu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) + \rho g$$  (II-1)

where $v$ is the velocity in the $y$ direction, $u$ the velocity in the $x$ direction, $\mu$ the dynamic viscosity, $\rho$ the local fluid density, and $g$ the acceleration due to gravity.

The density can be written as

$$\rho = \rho_o + \Delta \rho$$

where $\rho_o$ is some reference density (taken in the present case as reservoir water density). With this expression Eq. (II-1) becomes

$$\left( \rho_o + \Delta \rho \right) \frac{\partial v}{\partial t} + \left( \rho_o + \Delta \rho \right) u \frac{\partial v}{\partial x} + \left( \rho_o + \Delta \rho \right) v \frac{\partial v}{\partial y} = -\frac{\partial p}{\partial y}$$

$$+ \mu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) + \rho_o g + \Delta \rho g$$

Introducing the reduced pressure $p^*$ where
Fig. II-2. Reservoir in cylindrical coordinate.
\[ p^* = p - \rho_0 gy \]  

and dividing across by \( \rho_0 \), Eq. (II-1) becomes

\[ \left(1 + \frac{\Delta \rho}{\rho_0}\right) \frac{\partial v}{\partial t} + \left(1 + \frac{\Delta \rho}{\rho_0}\right) u \frac{\partial v}{\partial x} + \left(1 + \frac{\Delta \rho}{\rho_0}\right) v \frac{\partial v}{\partial y} = - \frac{1}{\rho_0} \left( \frac{\delta p^*}{\delta y} \right) \]

\[ + v \left( \frac{\delta^2 v}{\partial x^2} + \frac{\delta^2 v}{\partial y^2} \right) + \frac{\Delta \rho}{\rho_0} g \]

Now neglecting the \( \Delta \rho/\rho_0 \) part of the multiplier \( 1 + \Delta \rho/\rho_0 \) in the
temporal and convective terms (a step known as the Boussinesq approximation) yields the equation of motion as

\[ \frac{\delta v}{\delta t} + u \frac{\delta v}{\delta x} + v \frac{\delta v}{\delta y} = - \frac{1}{\rho_0} \frac{\delta p^*}{\delta y} + v \left( \frac{\delta^2 v}{\partial x^2} + \frac{\delta^2 v}{\partial y^2} \right) + \frac{\Delta \rho}{\rho_0} g \]  

The relationship at (II-2) is meaningful only when used in conjunction
with the rigid lid assumption. If the solution algorithm is such as to
include the computation of a free surface configuration then the expression
at (II-2) is not meaningful.

The reduced pressure \( p^* \) appearing in the \( y \) direction momentum
equation must appear in the momentum equations in all other coordinate
directions also for consistency. Since from (II-2)

\[ \frac{\delta p^*}{\delta x} = \frac{\delta p}{\delta x} \]

this condition is satisfied.

In cylindrical coordinates, as shown in Fig. II-2, the reduced
pressures is defined as

\[ p^* = p - g \rho_0 r \sin \theta \]

This absorbs part of the gravity force in both the \( \theta \) and \( r \) direction
momentum equation and satisfies the condition of similar pressure
definition in both directions.

The buoyancy term in Eq. (II-3), i.e. the term \( (\Delta \rho/\rho_0)g \), can be
treated further. Figure II-3 shows a plot of water density as a function
of water temperature. In a lake with cold inflow the maximum temperature
difference that could be expected to occur would rarely, if ever, exceed
Fig. II-3. Water temperature/density relation (from Table 93 of Dorsey [1940]).
10°C. (Typical fall conditions might have the lake at a temperature of 75°F and the inflow river at a temperature in the 60 - 65°F range.) For this temperature range the density-temperature relation can be linearized, i.e. the density \( \rho \) can be expressed as

\[
\rho = \rho_0 [1 - \beta (T - T_0)] \quad \text{or} \quad \Delta \rho = \rho - \rho_0 = \beta \rho_0 (T_0 - T) \quad (\text{II-4})
\]

where \( T_0 \) and \( \rho_0 \) refer to a reference state, in this case reservoir conditions. \( \beta \) is the coefficient of thermal expansion. If the density difference is due to a difference in dissolved substance, then the density-concentration relation is perfectly linear. Using Eq. (II-4), the buoyancy term in Eq. (II-3) can be written as

\[
\frac{\Delta \rho}{\rho_0} g = \beta g (T_0 - T) \quad (\text{II-5})
\]

Substituting this expression into Eq. (II-3) yields the equation of motion in the form it will be studied here as

\[
\frac{\partial \delta v}{\partial t} + u \frac{\partial \delta v}{\partial x} + v \frac{\partial \delta v}{\partial y} = - \frac{1}{\rho_0} \frac{\partial \delta p}{\partial x} + v \left[ \frac{\partial^2 \delta v}{\partial x^2} + \frac{\partial^2 \delta v}{\partial y^2} \right] + \beta g (T_0 - T) \quad (\text{II-6})
\]

Similarly in an \( r, \theta \) coordinate system, the buoyancy terms become \( \beta g (T_0 - T) \cos \theta \) in the \( u_r \) equation and \( \beta g (T_0 - T) \sin \theta \) in the \( v_r \) equation. When these simplifications are introduced into the \( r, \theta, z \) equations then, referring to the coordinate system in Fig. II-2, and letting \( u = u_\theta \) and \( v = v_r \) for convenience, the equations governing reservoir flow are:

**The momentum equations**

\[
\frac{\delta u}{\partial t} + v \frac{\delta u}{\partial r} + \frac{u}{r} \frac{\delta u}{\partial \theta} + \frac{uv}{r} = - \frac{1}{\rho_0} \frac{1}{r} \frac{\delta p}{\partial \theta} \\
+ v \left[ \frac{1}{r} \frac{\delta}{\partial r} \left( r \frac{\delta u}{\partial r} \right) + \frac{1}{r^2} \frac{\delta^2 u}{\partial \theta^2} \right] + v \left[ - \frac{u}{r^2} + \frac{2}{r^2} \frac{\delta v}{\partial \theta} \right] \\
+ g \beta \cos \theta (T_0 - T) \quad (\text{II-7})
\]
\[
\frac{\delta v}{\delta t} + v \frac{\delta v}{\delta r} + \frac{u}{r} \frac{\delta v}{\delta \theta} - \frac{u^2}{r} = -\frac{1}{\rho_o} \frac{\delta p^*}{\delta r}
\]

\[
+ v \left[ \frac{1}{r} \frac{\delta}{\delta r} \left( r \frac{\delta v}{\delta r} \right) + \frac{1}{r^2} \frac{\delta^2 v}{\delta \theta^2} \right] - v \left[ \frac{v}{r^2} + \frac{2}{r^2} \frac{\delta u}{\delta \theta} \right]
\]

\[
+ g \delta \sin \theta (T_o - T)
\]

The continuity equation

\[
\frac{1}{r} \frac{\delta}{\delta r} (rv) + \frac{1}{r} \frac{\delta u}{\delta \theta} = 0
\]

The energy equation

\[
\frac{\partial T}{\partial t} + v \frac{\partial T}{\partial r} + \frac{u}{r} \frac{\partial T}{\partial \theta} = \alpha' \left[ \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial T}{\partial r} \right) + \frac{1}{r^2} \frac{\partial^2 T}{\partial \theta^2} \right]
\]

where \( \alpha' ( = k / \rho c_p ) \) is the thermal diffusivity (k is the conductivity and \( c_p \) is the specific heat). This quantity can also be expressed as \( \alpha' = \nu / \text{Pr} \) where Pr is the Prandtl number and \( \nu = \mu / \rho \). The energy input from viscous dissipation is neglected. All other symbols are as previously defined.

2. Dimensional considerations

The important parameters governing plunging reservoir flow can be identified by carrying out a standard dimensional analysis. The parameters of interest can be more readily identified, however, if the governing equations are put in non-dimensional form.

The equations given in Section B.1 are put in non-dimensional form in the usual manner using a characteristic velocity \( V \) (taken as inlet river velocity), characteristic length \( d \) (taken as the inlet river depth), dimensionless pressure \( P \), where

\[
P = \frac{P^*}{\rho_o V^2}
\]

and a dimensionless temperature \( T' \) defined as

\[
T' = \frac{T_o - T}{T_o - T_{in}}
\]

where \( T_o \) is the reservoir water temperature and \( T_{in} \) the inflowing river water temperature. The resulting dimensionless equations are given below. Primes have been dropped for convenience.
The momentum equations

\[
\frac{\delta u}{\delta t} + v \frac{\delta u}{\delta r} + \frac{u}{r} \frac{\delta u}{\delta \theta} + uv = -\frac{1}{r} \frac{\delta p}{\delta \theta}
\]

\[+
\frac{1}{Re} \left[ \frac{1}{r} \frac{\delta}{\delta r} \left( r \frac{\delta u}{\delta r} \right) + \frac{1}{r^2} \frac{\delta^2 u}{\delta \theta^2} - \frac{u}{r^2} + \frac{2}{r^2} \frac{\delta v}{\delta \theta} \right] + T \frac{\cos \theta}{F_o^2} \tag{II-11}
\]

\[
\frac{\delta v}{\delta t} + v \frac{\delta v}{\delta r} + \frac{u}{r} \frac{\delta v}{\delta \theta} - \frac{u^2}{r} = -\frac{\delta p}{\delta r}
\]

\[+
\frac{1}{Re} \left[ \frac{1}{r} \frac{\delta}{\delta r} \left( r \frac{\delta v}{\delta r} \right) + \frac{1}{r^2} \frac{\delta^2 v}{\delta \theta^2} - \frac{v}{r^2} - \frac{2}{r^2} \frac{\delta u}{\delta \theta} \right] + T \frac{\sin \theta}{F_o^2} \tag{II-12}
\]

The continuity equation

\[\frac{1}{r} \frac{\partial}{\partial r} (rv) + \frac{1}{r} \frac{\delta u}{\delta \theta} = 0 \tag{II-13}\]

The energy equation

\[
\frac{\partial T}{\partial t} + v \frac{\partial T}{\partial r} + \frac{u}{r} \frac{\partial T}{\partial \theta} = \frac{1}{Re Pr} \left[ \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial T}{\partial r} \right) + \frac{1}{r^2} \frac{\partial^2 T}{\partial \theta^2} \right] \tag{II-14}
\]

All velocities and lengths are now dimensionless. The various dimensionless numbers appearing are

- \(Re = \frac{Vd}{v}\) the inflow Reynolds Number
- \(F_o = \frac{v}{\sqrt{g'd}}\) the inflow Densimetric Froude number (\(g' = \frac{\Delta \rho}{\rho_o} g\)).
- \(Pr = v/a'\) the Prandtl number.

The above equations are notable in one regard. They do not contain the Froude number. The absence of the Froude number is a direct consequence of the rigid lid assumption. This assumption allowed the absorption of the hydrostatic gravity term and effectively converts the reservoir flow into an internal-flow problem in which the through flow is specified independent of gravity. Buoyancy forces are gravity controlled and hence the densimetric Froude number appears in the equations.

The reservoir geometry and bed condition can be characterised by the inflow depth, \(d\), the bottom slope \(S\) and a bottom roughness height \(h\). Considering that the Prandtl number for water may be taken as a constant,
the above consideration shows that plunging reservoir flow is governed by the parameter groupings.

\[(Re, F_o, S, h_r/d)\]

The analysis may be taken further by looking specifically at the quantities of primary interest here, viz, the plunge depth \(H_p\), and the initial mixing coefficient, \(\gamma\).

The plunge depth \(H_p\) must appear in a relation of the form

\[H_p/d = f(Re, F_o, S, h_r/d)\] (II-15)

Now, while \(d\) is a necessary geometrical parameter for specifying the reservoir geometry, it is unlikely that \(H_p\) depends on this quantity. The plunge region will be positioned at a location where buoyancy and momentum forces are in balance. It should not depend on a arbitrarily selected inflow depth. Thus, relation (II-15) must be such that \(d\) and \(F_o\) do not appear specifically. The relation satisfying this constraint is

\[H_p = d F_o^{2/3} f(Re, S, h_r/d)\]

This, of course, reduces to the form

\[H_p = f(Re, S, h_r/d) \left( \frac{d}{g r} \right)^{2/3} \] (II-16)

which is similar in form to many of the relations outlined in Section I.B.

Since by definition

\[H_p = \frac{1}{F_p^{2/3}} \left( \frac{d}{g r} \right)^{1/3} \] (II-17)

where \(F_p\) is the densimetric Froude number at the plunge point, then relations (II-16) and (II-17) suggest that

\[F_p = f(Re, S, h_r/d)\] (II-18)

The initial mixing coefficient, \(\gamma\), must appear in a relation of the form
\[ \gamma = f(Re, F_o, S, h_r/d) \]

Since the \( F_o \) value is arbitrary and merely depends on the location selected for the inflow it is unlikely to have any effect on mixing dynamics which occur downstream. Thus \( \gamma \) can be written as

\[ \gamma = f(Re, S, h_r/d) \]  

(II-19)

3. Basic concepts of plunging flow and underflow

Various aspects of the plunging flow situation are presented in this section. The plunge region and underflow situation are shown in Fig. II-4, where various symbols are defined. The plunge region influences and is influenced by the density current. As Akiyama and Stefan [1984] point out the underflow layer forms the downstream boundary condition for the plunge region. An outline of the properties of the underflow thus occupy a large part of the considerations in this section.

Figure II-5 shows three types of velocity and temperature profiles that can occur downstream of the plunge point in a plunging flow field. The temperatures \( T_o \) and \( T_{in} \) are as defined above and \( T_{mix} \) is the temperature resulting from the mixing of inflow and ambient waters. Figure II-5(a) is an inverted wedge type flow with no mixing between light ambient and dense inflow. The interface is sharp and well defined. This is the type of situation implicitly assumed in the two-layer flow analyses. Figure II-5(b) shows a well mixed layer situation. The upstream flowing ambient is completely entrained by the underflow, and the entrained water is thoroughly mixed through the underflow layer. The interface between ambient and mixed water is well defined and coincides with the zero velocity point.

Figure II-5 shows a situation more representative of what would occur in a turbulent flow in a real situation. The ambient and cold water are separated by a mixed layer and an interface is difficult to define. Various methods can be used to define the underflow layer depth. Ellison and Turner [1959] use a layer depth based on momentum considerations, i.e. a momentum layer. In the present case the simple approach is adopted of defining the underflow as being that portion of the flow below the zero velocity point. This effectively converts the partial mixing case into the complete mixing situation shown in Fig. II-5(b).

Entraining density currents were first studied by Ellison and Turner [1959]. They presented the dynamic equation for such a current and showed that a density current quickly adopts an equilibrium situation with a constant flow densimetric Froude number, \( F_N \). This densimetric Froude number is defined as
where \( u \) is the layer velocity, \( h \) its depth, and \( g' \) is the reduced gravitational acceleration as defined above. As in free surface open channel flow, the flow state is termed subcritical if \( F_N < 1 \), critical if \( F_N = 1 \) and supercritical if \( F_N > 1 \). Unlike free surface open channel flow \( h \) is not a constant but increases in the downstream direction due to entrainment. Using \( u_e \) as the velocity of inflow into the lower layer, Ellison and Turner [1959] define an entrainment ratio \( E \) by the relation

\[
u_e = E u \]

and \( E \) is a function of the layer densimetric Froude number. Experimental measurements of entrainment [Ellison and Turner, 1959; Ashida and Egashira [1977] show that \( E \) is well represented by the relation

\[
E = 0.0015 F_N^2
\]

(II-21)

When \( F_N \) is constant, \( E \) is constant and the layer depth may be written as

\[
h = h_0 + E x
\]

where \( x \) is the distance from the location at which equilibrium conditions were reached and \( h_0 \) is the depth at that location.

An estimate of the constant discharge underflow normal densimetric Froude number may be obtained from a simple force balance on the underflow layer. Referring to Fig. II-6 shows that at equilibrium

\[
\rho g' h S = \tau_b + \tau_i
\]

where \( g' \) is as defined above, \( S \) is the bottom slope and the other symbols are as shown on Fig. II-6.

Letting \( \tau_b = \frac{f}{8} \rho u^2 \)

and \( \tau_i = \frac{f_i}{8} \rho u^2 \)

where \( f_i, f \) are interface and bed friction coefficients and \( u \) is the layer velocity, then
Fig. II-4. Plunge region and downstream density current.
Fig. II-5. Velocity and temperature profiles in various flow types.
Fig. II-6. Force balance on an underflow.

Fig. II-7. Illustration of effect of density current normal depth on plunge region position.
Letting $f_1 = \alpha f$, then

$$F_N^2 = \frac{8S}{f + f_1}$$  \hspace{1cm} (II-22)

Hence, since the friction factors will generally not vary very much, $F_N$ is constant for a given reservoir. Since $E$ is a function of $F_N$ only, it will also be constant for a given reservoir. A relation for $F_N$ can also be obtained from the underflow equation of Ellison and Turner [1959] by imposing the condition $\delta F/\delta x = 0$, where $F$ is the densimetric Froude number of the underflow layer. The ensuing relation is

$$F_N = \left( \frac{S_2 S - \frac{1}{2} S_1 E}{f + E} \right)^{1/2}$$  \hspace{1cm} (II-24)

where $E$ is as defined above, $f$ is the channel bottom friction factor and $S_1, S_2$ are factors to account for the nonuniformity of the velocity and temperature profiles and are as defined in Eqs. (I-4) and (I-5).

When the layer depth is based on the zero velocity point, similar consideration yields the $F_N$ value for the layer as

$$F_N = \left[ \frac{S_2 S - \frac{1}{2} S_1 E}{f + E} \right]^{1/2}$$  \hspace{1cm} (II-25)

where $S_3$ is a momentum adjustment coefficient and, apart from the factor $2h/B_0$, the other quantities are as defined above. The quantity $2h/B_0$, where $B_0$ is the flow width, accounts for side effects if the flow is not pure two-dimensional. This factor goes to zero in two-dimensional flow.

At small slopes $E \sim 0$ and Eq. (II-25) reduces to the simpler form of Eq. (II-23). If the slope is large ($S \sim 0.1$) then, as shown by Britter and Linden [1980], the bottom stress becomes irrelevant as most of the retarding force on the underflow comes from layer entrainment. Real reservoirs or beaches rarely have bottom slopes large enough to put them in this latter category. However, distorted models can have such large slopes and there is a danger that underflow phenomena will not be correctly reproduced in these installations.

For a given set of flow conditions, the buoyancy flux $B$ is a constant where
\[ B = uhg' \] (II-26)

with the symbols as defined above. Combining Eqs. (II-20) and (II-26) yields

\[ u = B^{1/3} F_2^{2/3} \] (II-27)

so the underflow layer velocity is also a constant. However \( u \) is not a constant for a given reservoir, and does depend on the flow conditions. If the layer discharge, \( q \), is known at any section in the density current, the normal depth, \( H_N \), can be computed from the relation

\[ H_N = \left( \frac{q^2}{g'} \right)^{1/3} \frac{1}{F_N^{2/3}} \] (II-28)

with \( g' \) given by Eq. (II-26). At the downstream end of the plunge region

\[ q = q_o (1+\gamma) \]

and

\[ g' = \frac{g'_o}{1+\gamma} \]

where \( q_o \) and \( g'_o \) refer to inflow conditions. Hence,

\[ H_N = \left( \frac{q_o^2}{g'_o} \right)^{1/3} \frac{(1+\gamma)}{F_N^2} \] (II-29)

Thus, the downstream normal depth increases in direct proportion to the amount of initial mixing.

The impact of the downstream layer normal depth on the plunge region position can be appreciated from the sketch in Fig. II-7. This shows plunging situations with two different normal depths. It is clear from this that the downstream normal depth must appear as an important parameter in any formula for the plunge depth. In fact, the formulae reviewed in Section I.B can all be expressed in terms of the downstream normal depth.

If the flow is supercritical, the plunge region position can be expected to be controlled by the singular point at the critical depth. Subcritical conditions exist in the great majority of practical situations. The friction factors \( f_f \), \( f_1 \) appearing in the expression for the normal depth are functions of the layer Reynolds number (= \( q/v \)) and the bottom
relative roughness, $h_r$. Hence the appearance of $f$, $f_1$, and $S$ in the expression for $H_N$, and by the above physical argument in the expression for $H_p$, is in line with the expectations from the dimensional analysis and suggests that $F_p$ is a function of $F_N$.

The entrainment situation downstream of the plunge point will be as shown in Fig. II-8. The quantity $\Delta Q$ is the amount of ambient water entrained into the underflow and $Q_o$ is the inflowing river discharge. Only vertical entrainment (by three-dimensional motion if necessary) can occur in this flow since two-dimensional average motion is being considered. Lateral entrainment is discussed in Section II.C.4. Also only developed plunging flow is under consideration here. Mixing that occurs in that portion of water which first enters the reservoir and establishes the quasi-steady plunged flow is not considered. The ratio $\Delta Q/Q_o$ is zero at the plunge line. Its initial rate of increase depends on the amount of initial mixing. The end of the plunge region is defined as being that section where developed two-layer flow starts. This section can be recognized on the $\Delta Q/Q_o$ versus $x$ plot as the position where $\Delta Q/Q_o$ starts to increase linearly as outlined above and is shown as section a-a on Fig. II-8. The value of $\Delta Q/Q_o$ at this position is by definition the initial mixing coefficient, $\gamma$. Viewed in this way the initial mixing is the amount of mixing that is required to bring the flow from conditions at the plunge line, characterized by $F_p$, to conditions of equilibrium layer flow characterized by $F_N$. These considerations indicate that $\gamma$ will be related to the difference between $F_p$ and $F_N$. Since as seen above $F_p$ is expected to be a function $F_N$, then $\gamma$ can be written as a function of $F_N$ only, i.e.

$$\gamma \propto F_N$$

It has been suggested [Imberger and Hamblin, 1982; Ford and Johnson, 1983] that the initial mixing in the plunge region may be estimated from research on cooling ponds by Jirka and Watanabe [1980]. Jirka and Watanabe [1980] recommend the relation

$$\gamma = 1.2(F_o - 1)$$

(II-30)

for entrance mixing in cooling ponds (from the vertical direction), where $F_o$ is the inflow densimetric Froude number. This relation is for jet type entrance mixing and is not appropriate for the situation considered here.

The quantity $\Delta Q/Q_o$ thus increases along the reservoir length. Measurements of $\Delta Q/Q_o$ at Section b-b, say, will show a large underflow compared to the original river inflow. This measured quantity is not the initial mixing coefficient but contains a large contribution from layer entrainment.

Much of the quoted figures regarding initial mixing of plunging flow may suffer from this discrepancy. The data of Elder and Wunderlich [1972] were taken 1.0 to 2.0 miles downstream of the plunge point. The mixing values in the Chilhowee reservoir can in fact all be attributed to layer entrainment so that the initial mixing rates become zero. Ford et al. [1980] calculated their mixing values by reference to the depth at which
Fig. II-8. Entrainment situation downstream of the plunge point.
the underflow entered a stratified reservoir. Some layer entrainment is thus also included in this case. Ford and Johnson [1983] were aware of these discrepancies and used the term entrance mixing to describe all mixing down to the measuring point. Ryan and Harleman [1971] do not give details of how their mixing values were computed. However, their use of the term 'entrance mixing' and their quotation of some work on buoyant surface jets suggests that their interest may be with jet type entrance mixing. Fischer and Smith [1983] measured concentrations of tracer just above and in the bottom water just below the plunge point so their results refer to real plunge region mixing.

The effect of flow separation and subsequent jet type mixing with lateral entrainment is discussed in Section II.C.4 in the context of flow in a diverging reservoir.

4. Two-layer flow analyses

As outlined in section I.B, many investigators have examined plunging flow using a two-layer flow analysis. In this section these approaches are summarised and various properties of the governing equation are examined.

All two layer analyses start from the basic equations of Schijf and Schönfeld [1953]. Considering the two-layer flow shown in Fig. II-9, those equations are:

\[
\frac{\partial h_1}{\partial x} + \frac{\partial h_2}{\partial x} + \frac{\partial u_1}{\partial x} = \frac{S_0 - S_{1E}}{g}
\]  

(II-31)

and

\[
(1 - \varepsilon) \frac{\partial h_1}{\partial x} + \frac{\partial h_2}{\partial x} + \frac{\partial u_2}{\partial x} = \frac{S_0 - S_{2E}}{g} \]

(II-32)

where \( \varepsilon = \Delta \rho / \rho_2 \), \( S_0 \) = channel bed slope, \( S_{1E} \), \( S_{2E} \) = layer energy gradients and \( \Delta \rho = \rho_2 - \rho_1 \). The quantities \( x, h_1, h_2, u_1 \) and \( u_2 \) are as shown on Fig. II-9 and \( \rho_1, \rho_2 \) are the layer densities.

The layer continuity equations are

\[
\frac{\partial}{\partial x} (u_1 h_1) = 0
\]

(II-33)

and

\[
\frac{\partial}{\partial x} (u_2 h_2) = 0
\]

(II-34)

It is the simple nature of this approach that the layers are assumed to remain distinct with no entrainment occurring. Equations (II-31), (II-32), (II-33), and (II-34) may be combined to yield a general equation for the interface as
Fig. II-9. Two-layer flow.
\[
\frac{\partial h_2}{\partial x} = \frac{S_0(1 - \theta) - S_{2E} + \theta S_{1E}}{1 - F_2^2 - \theta}
\] (II-35)

where \( \theta = \frac{1 - \varepsilon}{1 - F_2^2} \), and \( F_1, F_2 \) are the layer Froude numbers, i.e.

\[
F_1 = \frac{u_1}{\sqrt{gh_1}} \quad \text{and} \quad F_2 = \frac{u_2}{\sqrt{gh_2}}
\]

Taking the critical section as being where \( \partial h_2/\partial x = \pm \infty \), yields the condition at the critical section as

\[
F_1'^2 + F_2'^2 - 1 = 0
\] (II-36)

where \( F_1', F_2' \) are the layer densimetric Froude number, i.e.,

\[
F_1' = \frac{u_1}{\sqrt{g' h_1}} \quad ; \quad F_2' = \frac{u_2}{\sqrt{g' h_2}}
\]

and \( g' = \varepsilon g \) as before.

This criterion was first given by Stommel and Farmer [1953]. Since the ordinary Froude number is usually small, the above criterion may be written as

\[
F_1'^2 + F_2'^2 - 1 = 0
\] (II-37)

Under various assumptions Eq. (II-35) reduces to more familiar forms. For example, with zero net flow in the upper layer so that \( F_1' = 0 \), and taking \( \varepsilon \ll 1 \) the equation reduces to the interface equation of Fan [1960], of Savage and Brimberg [1975], and Jain [1980]. On imposition of the condition that \( h_1 + h_2 = \text{a constant depth} \), Eq. (II-35) reduces to the equation of Bata [1957] and of Harleman [1961]. In these latter equations, \( S' \) has disappeared, and with \( F_1' = 0 \) the equations may therefore be integrated directly to yield the interface profile.

When \( F_1' = 0 \), the condition at the critical section reduces to the simplest form

\[
F_2'^2 - 1 = 0
\] (II-38)
Energy gradients can be expressed in terms of the boundary stresses as

\[
S_{1E} = \frac{\tau_i}{g \rho_1 h_1}
\]

\[
S_{2E} = \frac{\tau_b - \tau_i}{g \rho_2 h_2}
\]

where \( \tau_i \), \( \tau_b \) are the interface and bed stresses, respectively. As above, using friction coefficients, the stresses may be linked to the mean velocities as

\[
\tau_b = \frac{f}{8 \rho_2 u_2 |u_2|}
\]

\[
\tau_i = \frac{f_i}{8 \rho_1 (u_1 - u_2) |u_1 - u_2|}
\]

where \( f \) and \( f_i \) are the bed and interface friction coefficients. Making the necessary assumptions, the interface equation of Savage and Brimberg [1975] and Jain [1980] is

\[
\delta h_2 \frac{\partial}{\partial x} (S_0 - \frac{\rho_2 r^2 f/8(1+\alpha+\frac{h_2}{h_1} \alpha)}{1 - \frac{\rho_2 r^2}{\delta h_2}}) = 0
\]

This equation is analogous to the dynamic equation of gradually varied flow in open channels. Chow [1959] discusses the open channel flow equation and shows how the water surface profiles that can occur may be arranged into several categories. Stefan [1973], Edinger et al. [1974], and Savage and Brimberg [1975] give a similar analysis of the kinds of flow profiles that can occur in two-layer flow.

A brief examination of Eq.(II-39) shows that, since at plunging it is necessary that

\[
\delta h_2 \frac{\partial}{\partial x} < 0
\]

then plunging can only occur when the densimetric Froude number is less than, or equal to, unity.

in particular exhaustively analyzes Eq. (II-39). He presents his results in graphical form. In terms of Eq. (II-17) for the plunge depth, his results may be generally summarized as stating that on steep slope reservoirs

\[ F_p = 1 \]

and on mild slope reservoirs

\[ F_p = k \frac{F}{p N} \]

where \( k \) is a function of \( \alpha \) and averages about 0.7 for \( \alpha = 0.5 \).

C. Diverging Reservoir

The axisymmetric reservoir is considered in this section. The approach is similar to that used in the two-dimensional case so details are not repeated. In particular many of the basic concepts of the two-dimensional case apply to the axisymmetric flow so that the material in Section II.B.3 is not repeated. The two-layer flow in the axisymmetric situation has properties different from the two-layer case. The two-layer axisymmetric analysis is thus presented in detail.

1. Governing equations

A diverging reservoir configuration can be modelled using the \( r, \theta, z \) coordinate system shown in Fig. II-10. The flow is symmetric in the \( \theta \) coordinate in this case. Using the rigid lid approximation and making similar assumptions as were made for the two-dimensional case in section II.B.1, the governing equations are

The momentum equations

\[
\frac{\delta v}{\delta t} + v \frac{\delta v}{\delta r} + u \frac{\delta v}{\delta z} = \frac{1}{\rho_o} \frac{\delta p}{\delta r} + v \left[ \frac{1}{r} \frac{\delta}{\delta r} \left(r \frac{\delta v}{\delta r}\right) \right] + \frac{\delta^2 v}{\delta z^2} \frac{\nu v}{r^2}
\]

\[
\frac{\delta u}{\delta t} + v \frac{\delta u}{\delta r} + u \frac{\delta u}{\delta z} = \frac{1}{\rho_o} \frac{\delta p}{\delta z} + v \left[ \frac{1}{r} \frac{\delta}{\delta r} \left(r \frac{\delta u}{\delta r}\right) \right] + \frac{\delta^2 u}{\delta z^2} \frac{\nu u}{r^2} + \beta g(T_0 - T)
\]
Fig. II-10. Diverging reservoir configuration in an r, θ, z coordinate system.
The continuity equation

\[ \frac{1}{r} \frac{\partial}{\partial r} (rv) + \frac{1}{r} \frac{\partial u}{\partial z} = 0 \]  

(II-42)

The energy equation

\[ \frac{\partial T}{\partial t} + v \frac{\partial T}{\partial r} + u \frac{\partial T}{\partial z} = \alpha'[ \frac{1}{r} \frac{\partial}{\partial r} (r \frac{\partial T}{\partial r}) + \frac{\partial^2 T}{\partial z^2} ] \]  

(II-43)

All symbols are as defined in Section II.B.1.

2. Dimensional considerations

Again as in the two-dimensional case the governing dimensionless parameters are most easily identified by rendering the governing equations into dimensionless form, using the inflow velocity, inflow depth and dimensionless pressure and temperature as in Section II.B.2. The resulting dimensionless equations are (dropping primes for convenience):

The momentum equations

\[ \frac{\partial u}{\partial t} + v \frac{\partial u}{\partial r} + u \frac{\partial u}{\partial z} = \frac{\delta P}{\delta z} + \frac{1}{Re} \left[ \frac{1}{r} \frac{\partial}{\partial r} (r \frac{\partial u}{\partial r}) + \frac{\delta^2 u}{\delta z^2} \right] + \frac{T}{Fo^2} \]  

(II-44)

\[ \frac{\partial v}{\partial t} + v \frac{\partial v}{\partial r} + u \frac{\partial v}{\partial z} = \frac{\delta P}{\delta z} + \frac{1}{Re} \left[ \frac{1}{r} \frac{\partial}{\partial r} (r \frac{\partial v}{\partial r}) + \frac{\delta^2 v}{\delta z^2} \right] - \frac{v}{r^2} \frac{1}{Re} \]  

(II-45)

The continuity equation

\[ \frac{1}{r} \frac{\partial}{\partial r} (rv) + \frac{\partial u}{\partial z} = 0 \]  

(II-46)

The energy equation

\[ \frac{\partial T}{\partial t} + v \frac{\partial T}{\partial r} + u \frac{\partial T}{\partial z} = \frac{1}{Re Pr} \left[ \frac{1}{r} \frac{\partial}{\partial r} (r \frac{\partial T}{\partial r}) + \frac{\partial^2 T}{\partial z^2} \right] \]  

(II-47)

The dimensionless numbers Re, Fo , and Pr are as defined in section II.B.2.

The reservoir geometry and bed condition for this case can be characterized by the inflow depth \( d \), the inlet radius \( R_0 \) and a bottom roughness height \( h_r \). Since uniform flow cannot occur in this case,
it may be anticipated that the reservoir length, L, must also enter as a parameter.

Hence, assuming that Pr is constant, plunging reservoir flow in an axisymmetric situation is governed by the parameter groupings.

\[(Re, F_o, R_o/d, h_r/d, L/R_o)\]

As in the sloping situation it can be argued that \( F_o \) and \( R_o \) are arbitrary parameters and thus must disappear in the final groupings. Hence, it can be anticipated that the plunge region radius, \( R_p \), must appear in a relation of the form

\[\frac{R_p}{R_o} = F_o \cdot f(Re, h_r/d, \frac{R_o}{d}, \frac{L}{R_o})\]  \hspace{1cm} (II-48)

The densimetric Froude number at the plunge point must be governed by the parameter groupings

\[(Re, h_r/d, F_o \cdot \frac{R_o}{L}, \frac{F_o}{d} R_o)\]  \hspace{1cm} (II-49)

3. Two-layer analysis

The two-layer analysis of flow in the sloping reservoir situation has yielded useful insights into the plunging phenomenon. In this section a two-layer flow analysis is applied to flow in an axisymmetric situation.

Flow in a diverging channel with straight walls may be analyzed as axisymmetric (radial) flow from a central source over a plane. The source flow rate, \( q_{axi} \), can be linked to the actual inflow channel flow rate, \( Q \), using

\[q_{axi} = \frac{Q \pi}{\delta}\]  \hspace{1cm} (II-50)

where \( \delta \) is the channel half-angle of divergence.

If the channel aspect ratio is very small, side wall friction will play a role and a simple axisymmetric treatment will not be strictly applicable. If separation occurs, the axisymmetric treatment is also not valid.

Energy considerations show that the Schijf and Schönfeld equations apply to axisymmetric flow if differentiation with respect to \( x \) is replaced by differentiation with respect to \( r \).
The continuity equations become

\[ \frac{\partial}{\partial r} (u_1 h_1 r) = 0 \]  \hspace{1cm} (II-51)

\[ \frac{\partial}{\partial r} (u_2 h_2 r) = 0 \]  \hspace{1cm} (II-52)

Combining Eqs. (II-31), (II-32), (II-51), and (II-52) and assuming a constant overall flow depth, \( d \), yields the equation for the interface in two-layer axisymmetric flow as

\[ \frac{\partial h_2}{\partial r} = \frac{(S_{2E} - S_{1E})/\epsilon + \left[ F_1 \frac{h_1}{r} - F_2 \frac{h_2}{r} \right]}{F_1^2 + F_2^2 - 1} \] \hspace{1cm} (II-53)

Assuming that there is no net discharge in the upper layer so that \( F_1' = 0 \), and noting that \( \epsilon \ll 1 \), Eq. (II-53) reduces to

\[ \frac{\partial h_2}{\partial r} = \frac{\frac{F_2'}{2} \left[ \frac{f}{8} (1 + \alpha + \alpha \frac{h_2}{h_1}) - \frac{h_2}{r} \right]}{F_2' - 1} \] \hspace{1cm} (II-54)

The notation is the same as for the two-layer analysis in section II.B.4. The energy gradients have been related to the boundary stress and the mean velocities as in section II.B.4. As \( r \to \infty \), the flow becomes effectively parallel and Eq. (II-54) reduces to the equation of Savage and Brimberg [1975] with zero bed slope.

It has not proven possible to directly integrate this equation. However, some useful information can be extracted from features of the equation. Before going on to discuss these features, free surface flow in a diverging channel is examined. This examination will give some indications as to the kinds of flow to be expected in the two layer situation.

**a. Free surface flow in a diverging channel**

The equation for free surface flow in a diverging channel is obtained from Eq. (II-54) by setting \( \alpha = 0 \) and \( \epsilon = 1 \). The equation is

\[ \frac{dd}{dr} = \frac{f/8 - d/r}{1 - 1/F^2} \] \hspace{1cm} (II-55)

where \( d \) is the depth of flow and \( F \) is the Froude number. This equation has been given by Vallentine [1967] who also discussed the flow profiles that can occur in a pure axisymmetric situation.
Fig. II-11. Free surface flow situation.
The flow situation considered herein is shown in Fig. II-11. The diverging channel flow in this case is not purely axisymmetric (apart from the presence of the walls) but is joined on its upstream side to a parallel wall channel. This upstream boundary condition introduces an additional degree of freedom from purely axisymmetric flow. The parallel sided channel is fed from a reservoir with discharge set at a constant value. The channel bottom is horizontal. The downstream boundary condition is assumed to be a free overfall in the diverging channel section.

The flow in the parallel sided channel will always be a drawdown flow from a reservoir with an H2 water surface profile [Chow, 1959]. The flow will be governed by conditions at the end of the parallel sided channel, i.e., at the diverging channel entrance. The depth there will be equal to or greater than (depending on downstream conditions) the critical depth.

The flow profiles in the diverging portion are most readily examined if Eq. (II-55) is rendered into non-dimensional form using the non-dimensional depth and distance \( R = \frac{d}{h_0} \) and \( \frac{r}{r_0} \), respectively, where \( r_0 \) is the radius at the diverging channel entrance and \( h_0 \) is the critical depth there.

The non-dimensional equation is

\[
\frac{dH}{dR} = \frac{G_0 - H/R}{1 - \frac{H^4 R^2}{p^2}}
\]

where \( G_0 = \frac{f r_0}{8h_0} \)

and \( F_c^2 = \frac{Q^2}{4g^2 h_0^3 r_0^2} \)

\( Q \) is the channel discharge.

Thus the flow profiles are a function of the two non-dimensional parameters \( F_c \) and \( G_0 \). Since by definition \( F_c = 1 \), the flow profiles depend on \( G_0 \) only. In particular the form of the flow profiles depends on the magnitude of \( G_0 \) relative to unity.

If \( G_0 > 1 \) only subcritical flow can occur in the diverging channel and the flow will be controlled from downstream. This flow will be referred to as Type I. Two subtypes occur in this category. If \( H_{r=r_0} < G_0 \), then a Type Ia will occur. In this flow \( dH/dR \) is always negative; the water surface will slope continuously downwards. This corresponds to flow in a channel with a very small divergence angle (\( r_0 \gg 1 \) so that \( G_0 \gg 1 \), and the water surface profile is very similar to the ordinary H2 profile.
If $H_o = r_o > G_o$, then a Type Ib will occur. The water surface in this profile will first slope upwards i.e. $dH/DR > 0$, until it intersects the line $H = RG$ where $dH/DR$ must equal zero. Then the surface slopes downwards continuously.

Equation (II-56) was integrated numerically using a Runge-Kutta fourth order scheme with various $G$ values. Two resulting profiles illustrating type Ia and type Ib behavior are shown in Fig. II-12.

If $G_o < 1$, Type II flow can occur. This may be either subcritical or supercritical depending on the downstream water level. If the downstream water level is sufficiently low, then a critical section will form at the parallel diverging channel junction and the initial flow in the diverging channel will be supercritical. This is termed a Type IIa flow. The water surface will slope downwards until it intersects the line $H = G_oR$ at which point $dH/DR$ must equal zero. The surface will then slope upward to intersect the critical depth line vertically. If the beach is sufficiently short, of course, this flow will shoot off the beach end in a supercritical state.

The result of a numerical integration on this profile type is shown in Fig. II-13. A hydraulic jump will form at the singular point. The approximate position of the jump can be found in the normal manner by utilizing the usual hydraulic jump equation as illustrated by Vallentine [1967].

The significance of the regions demarcated by the curves $G_o - H/R = 0$, i.e. $H = G_oR$, and $1 - R^2H^2/F_c^2 = 0$, i.e. $H = R^{-2/3}$, can be appreciated in Figs. II-12 and II-13. The sign the water surface slope must have in the appropriate region is shown in Fig. II-13.

The supercritical portion of the profile in Fig. II-13 is controlled from upstream and the subcritical part from downstream. The singular point is thus not a control point. A controlling singular point where the flow changes from subcritical to supercritical state can never occur in a straight walled diverging channel since such a profile requires a downward sloping subcritical flow into the point and a downward sloping supercritical flow away from it. An examination of Fig. II-13 shows that this profile cannot occur. Such a hydraulic drop can occur at the upstream and downstream ends of a diverging channel but not in the channel itself.

If the water level downstream is sufficiently high, supercritical flow will be suppressed and a profile of the type shown in Fig. II-13, termed Type IIb, will form.

It is of interest that in the case shown in Fig. II-13 lengthening the beach had very little effect on raising the water level downstream of the jump. This is quite unlike flow in a parallel sided channel and is, of course, due to the fact that the addition in length is associated with increasing width. In the particular case shown, it did not appear to be possible to ever drown out the jump by lengthening the beach.

Free surface flow profiles in a diverging channel appear to have some interesting features. These were not pursued as the purpose here was merely to identify controlling parameters and broad profile types.
Fig. II-12. Possible water surface profiles when \( G_0 > 1 \).
Fig. II-13. Possible water surface profiles when $G_0 < 1.$
b. Two-layer flow in a diverging channel

As a first step in examining interfacial profiles, Eq. (II-54) is put in non-dimensional form. Again non-dimensionalization is carried out using conditions at the parallel channel/diverging channel junction. The radius \( r_0 \) is the same as before. The critical depth \( h_0 \) now refers to the reduced gravitational field \( g \varepsilon \). Hence with \( R = r/r_0 \), \( H = h_2/h_0 \) and \( D = d/h_0 \), Eq. (II-54) becomes

\[
\frac{dH}{dR} = \frac{G_0 (1 + \alpha + \frac{\alpha H}{D-H}) - H}{R} \left( 1 - \frac{H^3 R^2}{F_c r_0^2} \right)
\]

where \( G_0 = \frac{f r_0}{8 h_o} \)

and

\[
F_c' = \frac{Q^2}{4 \varepsilon r_o^2 h_o^3 r_0^2}
\]

Thus, the profiles that occur in two-layer axisymmetric flow are characterized by the two nondimensional parameters \( G_0 \) and \( D \). If the above equation had been non-dimensionalized by the overall depth \( d \) instead of \( h_0 \), then the profiles would be characterized by \( G_{in} \) and the inflow densimetric Froude number \( F_0 \) where

\[
G_{in} = \frac{fr_0}{8d}
\]

and

\[
F_0 = \frac{Q^2}{4 \varepsilon d^2 h_0^3 r_0^2}
\]

These are equivalent parameter groupings to the two above. \( F_0 \) and \( D \) are linked by the relation

\[
D = F_0^{-2/3}
\]

Thus, as the actual inflow densimetric Froude number will usually be greater than unity, then \( D \), the dimensionless total depth, will be less than unity.
Figure II-13 shows that the form of the profiles in the free surface case are well defined by the regions demarcated by the curves shown on the diagram. Adopting a similar approach in the two-layer case yields the functions

\[ 1 - \frac{H^3}{R^2} = 0 \]  \hspace{1cm} (II-58)

and

\[ G_0 \left(1 + \alpha + \frac{H}{D-H} \right) - \frac{H}{R} = p \]  \hspace{1cm} (II-59)

The value of the quantity \( p \) is discussed below. Equation (II-58) defines the critical depth curve (in the reduced gravitational field) as in the free surface case. Equation (II-59) defines a surface in \((H, R)\) space. The curves on which this surface intersects the \((H, R)\) plane are given by

\[ H = \frac{1}{2} \left( G_0 R + D \right) \pm \frac{1}{2} \sqrt{(G_0 R + D)^2 - 4 G_0 RD(1+\alpha)} \]  \hspace{1cm} (II-60)

This defines two curves about the straight line \( H = (D+G_0 R)/2 \) and crossing that line at the points

\[ R = \frac{D}{G_0} \left[ (1+2\alpha) \pm 2 \sqrt{\alpha + \alpha^2} \right] \]  \hspace{1cm} (II-61)

Only the curve at low \( R \) values is considered further. This curve springs from the points \( H = 0 \) and \( H = D \) on the central axis. A plot of the curve for a small and large \( G_0 \) value is shown in Fig. II-14. If \( G_0 \) is large, the curve retreats behind the \( R = 1 \) line. Only the part of the curve forward of this line can have any relevance to the flows considered here.

The curve in Fig. II-14(a) shows, as stated above, the intersection of a surface with the \((H, R)\) plane. A section through this surface at the location indicated on Fig. II-14(a) is given in Fig. II-14(b). This shows a curve with the following salient features:

\[ p = G_0 (1 + \alpha) \text{ at } H = 0 \]

\[ p \to +\infty \text{ at } H = D \]

\[ p_{\text{min}} (<0) \text{ at } H = D - \sqrt{DG_0 \alpha R} \]

Thus in Fig. II-14(a), the function \( p \) is negative inside the curve given by Eq. (II-60) and is positive outside.
Fig. II-14(a). Plot of Eq. II-60 for two $G_0$ values ($D = 0.6$ and $\alpha = 0.5$).

Fig. II-14(b). Section A-A on Fig. 14(a). Represents plot of function $p$ (Eq. II-59) at $R = 14$. 
A plot of the curves given by Eq. (II-58) and Eq. (II-60) with large \( G_0 \) is shown in Fig. II-15. The sign the interface slope must have in the regions demarcated by the curves is indicated and is emphasized by the inclined arrows.

It is clear from this diagram that plunging cannot occur until the densimetric Froude number of the flow drops below unity since at plunging the interface must of necessity slope downwards. This conclusion is in line with experimental and prototype data from two-dimensional situations which show plunging in a range of densimetric Froude numbers of 0.3 and 0.7. Flow at small divergence angles should be similar to two-dimensional flow. At small angles of divergence \( G_0 \) is large, and this effectively removes any influence of Eq. (II-60) on the flow.

When \( G_0 \) is small, the picture is less definite. Figure II-16 outlines a situation which may arise. The interface slope signs are again shown on this diagram. At the free overfall at the beach end, the interface must pass through a critical section as shown. The theory indicates that plunging can only occur in subcritical state. However, to get from the plunge point to the dropoff, the interface must traverse a region where its slope must be upward. It appears that an interface such as that marked “profile (a)” in Fig. II-16 cannot exist because it requires violation of the theory along much of the interface. Subcritical plunging in this case would thus appear to be impossible. However, if the flow plunges in a supercritical state, thus violating the theory at the plunge location, it has a ready-made path to the dropoff by adopting an interface such as that marked profile (b) in Fig. II-16. It is speculated that this profile is the one most likely to occur. It can be argued that the strict theory does not hold in the immediate vicinity of the plunge line as the curvature of the streamlines there render the simple one-dimensional approach invalid.

In summary then it is expected that two basic types of interface profiles can occur in two-layer diverging channel flow. That termed “Type I” occurs for large \( G_0 \) values and is the counterpart of plunging flow in two-dimensional situations. An illustration of this type of profile is shown in Fig. II-17. That termed “Type II” may occur at small \( G_0 \) values and consists of plunging in a supercritical state. An illustration of the interface to be expected is shown in Fig. II-18. Whether or not the hydraulic jump forms depends on the beach length.

When Type I plunging occurs the flows are controlled from downstream. Using a downstream boundary condition of a free overfall, Eq. (II-57) was integrated numerically (using a fourth order Runge-Kutta scheme) to yield densimetric Froude number values at the plunge point. As \( r_0 \) and \( F_0 \) are arbitrary parameters, similarity considerations indicate that \( F \) is a function of the parameter groupings.

\[
F \propto (R_{end}^{1.5}, \ G_0^{2.5})
\]

or more conveniently, of
Fig. II-15. Plunging with large $G_0$. 

$D = 0.6$

$\frac{dH}{dR} < 0$

$\frac{dH}{dR} > 0$

EQ II-60

$G_0 = 0.15$

EQ II-58
Fig. II-16. Plunging with small $G_o$. 
Fig. II-17. Flow situation with Type I plunging ($G_0$ large).
Fig. II-18. Flow situation with Type II plunging ($G_o$ small).
(R_{end}^{F_{o}^{-1}}, G_{in}^{F_{o}})

The results are summarized in graphical form on Fig. II-19 in terms of these latter parameters.

The dashed line in the upper part of this figure indicates the boundary between Type I and Type II plunging. Type I or subcritical plunging cannot occur above this line. The demarcation line occurs as $G_{o}$ is reduced (say, by reducing $f$) when the upstream encroaching interface just meets the downstream advancing curve given by Eq. (II-60). Figure II-19 shows that there is no single value of $F_{p}$ associated with the plunge point. Depending on the values of the governing parameters, $F_{p}$ can have any value in the range unity to close to zero.

4. Separated flow

The experimental results reported by Akiyama and Stefan [1986] showed that flow separation occurred in their experiment with a channel sidewall divergence angle of 15°. The considerations in the previous sections for both parallel and diverging flow assume that the flow is attached at all times to both sidewalls. If the flow separates, the flow physics change fundamentally. Flow separation, the possibility of its occurrence, and the consequences if it does occur, are examined in this section.

Nikuradse [1929] experimentally examined flow in two-dimensional diverging channels with no buoyancy effects i.e., with fluid in the channel of the same density as the inflow. He found that for sidewall divergence angles, $\alpha$, less than 3°, the flow velocity profiles were symmetrical and similar to flow in parallel wall channels. When $\alpha$ exceeded 5°, the flow separated with attachment on one wall and backflow along the other. For values in the range 3° to 5°, the flow remained attached, but the velocity profile was skewed to one side. Reneau et al. [1967] have carried out a comprehensive series of experimental observations of flow in two-dimensional diffusers. These researchers have categorized the various flow types that can occur and report that one-wall separation (or stall) occurs at sidewall angles greater than about 10° with the separation angle decreasing as the diffuser length increased.

The above results refer to two-dimensional flow in non-buoyant situations. The addition of bottom effects and buoyancy will modify these results. The consequences of bottom effects are difficult to predict but Townsend [1961] states that the addition of end walls will induce secondary flows, and this will cause separation at smaller divergence angles.

The addition of buoyancy forces will tend to inhibit separation if the flow plunges at the entrance to the diverging channel section so that the buoyancy forces can manifest themselves. If the flow is such that in a non-separated state, plunging starts somewhere down the diverging section, then the flow upstream of the (conceptual) plungeline is effectively non-buoyant, and normal separation will occur if the sidewall divergence angle is sufficiently large. Once separation occurs, warm ambient water will be swept upstream and a complex flow will ensue.
Fig. II-19. Plunge point densimetric Froude number behavior.
Exact flow configurations for buoyant flow in a diverging channel with bottom effects can only be determined experimentally. This experimental work is described in Chapter VI.

Flow separation has large consequences for the attempts to predict plunge region position and initial mixing. It may be anticipated that with separation, a plunge region may be difficult to identify and the association of a single $F_p$ value with the plunge region will be impossible. Mixing can be expected to increase dramatically as lateral entrainment is possible. Initial mixing will be similar to the mixing of a jet entering a waterbody and will depend, as in the jet flow, on the inflow parameters and on entrance geometry.

As with the flow configuration, the actual mixing in the flow separated case can only be determined experimentally. The results of an experimental study are given in Chapter VI.
III. NUMERICAL PROCEDURES

A. Introduction

As has been seen in the previous chapters, plunging flow has been studied in the past using simple models such as a two-layer approach or by bulking parts of the flow together so that a momentum equation approach can be used. These approaches concentrate attention on the plunge region only and effectively isolate the plunge region from the rest of the reservoir.

The plunge region is of course influenced by the general reservoir flow and conversely the reservoir flow pattern is influenced by the plunge region. The entire flow configuration from the downstream dam boundary condition to the upstream river inflow is governed by the general dynamic equations of motion. However, the motion is too complex to allow a complete analytical solution of these equations. This is of course the reason why simplifications have to be made.

At the present time many numerical schemes have been developed which can extract solutions of the Navier Stokes equations. The availability of large and fast computers make feasible the application of these schemes to complex flow situations such as plunging flow.

In the numerical approach the plunge region need not be isolated and the entire reservoir flow can be simulated. The plunge region will then appear in the emerging flow field as part of the overall solution. (In fact, to isolate the plunge region for a numerical approach would introduce complex new problems on the specification of the downstream boundary conditions.)

Numerical schemes have been applied to many different flow problems and appear to be capable of producing realistic and useful results. In the following two chapters plunging flow is examined numerically in the two model reservoir types identified in Chapter I, i.e. a two-dimensional sloping bed reservoir and a horizontal bed diverging reservoir. In the remainder of this chapter the numerical scheme to be used is described and some common features used in the numerical treatment are outlined.

B. Numerical Scheme

1. General procedure

The numerical procedures used for the solution of the relevant equations are those described by Patankar [1980] and incorporated in the computer code of Patankar [1982]. This code was modified in some respects, particularly in extending it so that unsteady flow could be computed but the essential procedures remained unaltered.

The computation technique employs the control volume concept. In this method the area of study is discretized by dividing it into a series of
contiguous control volumes. The constraint that diffusion and convective fluxes, plus rates of change and source term effects, sum to zero over each control volume, yields a series of simultaneous linear equations for solution.

The above constraint is expressed in mathematical terms by direct appeal to the basic physics or alternatively by integration of the governing differential equation over the control volume. (This equation is of course derived in the first place by appealing to the basic physics over a finite control volume and letting ΔVol ≠ 0.)

Figure III-1 shows a typical control volume layout for the computation of the concentration c of some inert substance. Main grid lines are shown passing through the center of each control volume. (These do not have to be central in the control volume, but in this work central grid lines are used in all cases, even when an uneven control volume size is used.) Each control volume contains one intersection of grid lines, and the concentration value is computed at this intersection.

In the computation of source and rate of change terms, the variable value is assumed constant over each control volume at the intersection value. In the computation of fluxes through the control volume faces, profile assumptions linking the grid point values through the faces have to be made. These phenomena are computed using the power law scheme (effectively the exponential scheme) described by Patankar [1980]. The diffusion coefficients are specified at each main grid line intersection. The value at the control volume face is interpolated using a harmonic type average.

In the velocity field calculations, use of a very high viscosity in a control volume adjacent to a boundary will result in the boundary velocity (usually zero) being transferred through the control volume. In this way control volumes can be "blocked out" and irregular boundaries can be simulated.

The simultaneous linear equations resulting from the above numerical scheme may be solved by any of the standard methods of numerical analysis. The method used here is the line by line method. This is a combination of the well-known Tridiagonal-matrix-algorithm (or Thomas algorithm) for one-dimensional situations and the iterative Gauss-Seidel method. Before this procedure is applied, a coarse one-dimensional solution is imposed on each coordinate direction to get boundary condition information into the interior quickly. This coarse solution is then effectively refined by the line by line method.

2. **Computation of velocity field**

The above scheme works well when the concentration of some inert substance is to be computed in a given velocity field. When the velocity field itself is to be computed, two problems arise. The first arises in the nonlinearity of the convective term in the governing equation of motion since the velocity field required to be computed is now being convected by itself. Nonlinearity is handled by iteration. A velocity field is assumed, the coefficients arising from the convective terms are calculated, and a solution of the linear equations is extracted. Using this new velo-
city field, new coefficients are computed, and the procedure is repeated until convergence occurs.

In dealing with nonlinear situations like this the iterative solution method for the linear equations has a distinct advantage over a direct method (gauss elimination, for example). In the iterative method the approximate solution emerging after a small number of iterations is accepted and used to construct new coefficients. This procedure can be repeated a number of times with increasingly better coefficients in the same time that a direct method can extract a complete solution of the linear equations with the old coefficients. The iterative method is a very efficient approach for nonlinear problems.

The second problem in computing the velocity field arises because of the velocity field-pressure field interdependence. If the pressure field is known, then the pressure term becomes a source term in each control volume, and the usual procedure can be used to solve for the velocity field. However, the pressure field is rarely known so some additional procedure is required to handle the velocity field-pressure field interaction. The procedure used here is the SIMPLE procedure of Patankar and Spalding [1972]. This is a method which iteratively adjusts an assumed pressure field using pressure corrections calculated from the mass imbalances in each control volume given by the velocity field arising from the momentum equation with the assumed pressure field. A staggered grid [Harlow and Welch, 1965] is used with the SIMPLE procedure and the velocities are computed at the position of the main grid control volume faces. Hence, in Fig. III-1 the velocities are shown at the actual locations at which they are computed. The pressures are computed at the main grid points so that the velocity across a control volume face is driven by the pressure gradient between the grid points in the adjacent control volumes.

In a typical problem involving, say, the computation of some inert substance, the velocity field is usually solved for first. Then using the known velocity field in the convective terms, the species concentration equation is solved for. In the present problem the energy equation needs to be solved to give the temperature configuration in the reservoir. Now, however, the velocity field cannot be solved for first as temperature appears in the momentum equation via the buoyancy term. This problem is overcome by including the energy equation in the same iteration cycle as the velocities. Then for each approximate velocity field a new temperature field is solved for and this yields better buoyancy terms for the next iteration cycle on the velocity field. Thus, the velocity field proceeds to convergence with a continuously updated and improved buoyancy term.

3. Unsteady problems

All of the above effectively refers to steady-state problems. These problems are usually solved by iteratively extracting a converged solution from the initial guess. Plunging flow is an unsteady phenomenon so an unsteady solution method must be used. Unsteady problems are solved much like a sequence of steady-state problems with the initial guess for each time step being the converged solution at the previous time step. The time history must now of course enter the problem.
Fig. III-1. Typical control volume and grid layout.
Since the governing differential equation is parabolic in the time domain, any of the standard finite difference methods for solution of transient problems may be used. Of these methods three in particular are most commonly used: the explicit, Crank-Nicholson and fully implicit methods. Each has its advantages and disadvantages. The explicit method is the easiest to implement but the time step $\Delta t$ must usually be kept small to satisfy stability requirements. The Crank-Nicholson scheme, although in principle unconditionally stable, can give rise to oscillation and greatly inaccurate solutions if $\Delta t$ is too large. The fully implicit scheme is the only one that can give stable and realistic solutions at large time steps.

In the study of plunging flow where reservoir through-flow is being considered, it is desired to keep the time step large so that a maximum amount of elapsed time may be achieved for a given computational effort. For this reason the fully implicit scheme is used. Thus, at any space point at time $t$, information comes from the surrounding values at time $t$ and from the same space point at time $t = t - \Delta t$.

For a steady state problem many iterations are generally required before a sufficiently converged solution is finally achieved from an initial guess. In the unsteady case, however, the results of each previous time step are usually an excellent guess as to the configuration after a further time step so only a small number of iterations are required before convergence is achieved. The exception to this in plunging flow is that time step over which back flow first appears. The initial and final state (with respect to the time step) is then quite different and the potential exists for inaccuracies to occur. Particular attention is thus paid to this region when selecting the time step to be used in the calculation.

4. **Summary**

The complete computation routine for unsteady problems is shown schematically in Fig. III-2. There are two iteration loops. If the problem is linear, the outer (I) loop is not required since the coefficients in the linear equations are constants. In this case the parameter LAST is set equal to unity and NTIMES is set sufficiently large to extract a full solution of the linear equations. In a nonlinear problem, LAST is large, and NTIMES small. Listings of the computer codes used in the numerical simulations and the details of their implementation are given by Farrell [1986]. All computations were carried out on the Cray 1 computer at the Supercomputer Institute of the University of Minnesota.

C. **Transient test problem**

The computer code was applied to a simple transient problem to see how well it could simulate a developing situation with steep gradients. The problem chosen was that of a front propagation governed by the one-dimensional equation
START with initial or guessed fields

INCREMENT TIME by At

DO I = 1, LAST

LOOP on each of the variables u, v, pressure correction and T.

Form coefficients for the linear equations using (non-linear case) previously computed variable values.

DO I = 1, NTIMES (NTIMES - number of iteration cycles desired)

Extract solution of simultaneous linear equations

STOP if TIME > TEND

Fig. III-2 Schematic of computational scheme.
resulting from a sudden step-up in concentration at an upstream location. In the above equation

\[
\frac{\partial c}{\partial t} + V \frac{\partial c}{\partial x} = D \frac{\partial^2 c}{\partial x^2}
\]  

(III-1)

c = tracer concentration
V = flow velocity (a constant)
D = diffusion coefficient
t = time
x = distance from concentration step-up

This problem is extensively discussed by Dhamotharan, Gulliver and Stefan [1981].

The computed numerical solution with a uniform grid size (\(\Delta x\)) for a single grid Peclet number (\(Pe = V\Delta t/D\)) and dimensionless time step \((\Delta \tau^* = \Delta t V/\Delta x)\) is shown in Fig. III-3. The concentration is shown in dimensionless form as \(conc^* = c/\Delta c\) where \(\Delta c\) is the initial step-up in concentration. The analytical solution of this problem is also shown. It can be seen that the numerical procedures with the implicit time scheme can simulate this situation reasonably well.

The above is a relatively simple one-dimensional problem in which concentration only needs to be computed. However, the gradients in the above problem are exceptionally steep. The fact that the numerical scheme can deal effectively with these and produce a realistic solution indicates that it should be able to deal effectively with similar developing situations in a plunging flow field.

D. Numerical diffusion

Numerical diffusion arises because of the inability of a discrete grid in space and time to perfectly simulate continuous processes. The subject is extensively discussed by Pantankar [1980], Raithby [1976], Roache [1972], and Leonard [1979]. The kind of numerical diffusion of most interest in this study is that part of the numerical diffusion known as "skewness error."

Skewness error arises when flow takes place at an angle to the numerical scheme grid lines. In a flow where recirculation occurs, as in plunging flow, flow at an angle to the grid lines is inevitable. A typical exaggerated situation is shown in Fig. III-4. The line a-a is an interface with warm water above and cold below. The heavy arrows show the real flow field velocity directions. The light arrows show the representation of the velocity field in the numerical scheme.

The temperature at point A in the real velocity field is strongly influenced by the temperature at the point A' by convection, with some influence from its surroundings by diffusion. In the numerical scheme,
Fig. III-3. Analytical and numerical solutions of a front progression.
Fig. III-4. Typical flow situation giving rise to skewness error.
however, point A receives information by convection from point B and point C. The information from point C causes the temperature at point A to be higher (in this case) then it should be and thus causes "smearing" or false or numerical diffusion. It is intuitively clear that the extent of the numerical diffusion depends on the skewness angle to the grid lines. If physical diffusion is very strong and convection weak, then numerical diffusion will usually not be important. In the converse situation, numerical diffusion may be strong enough compared to the physical diffusion to obscure basic details of the flow field.

Numerical diffusion is a serious problem and has been a source of concern to computational fluid dynamicists for many years. The problem is of particular concern if sophisticated turbulence models are being compared as then the numerically induced errors can obscure subtle differences in the models. Recently schemes such as Raithby's [1976] skew differencing scheme and Leonard's [1979] quartic upstream weighted (QUICK) scheme have been introduced specifically to counter the "skewness error" source of diffusion.

The above two schemes have been applied to flow problems with impressive results [Demuren, 1983]. However, it is not clear that various problems in their application have been sufficiently overcome so they are not used here in this applied study. Furthermore the present study is more concerned with the resulting bulk parameters of a flow rather than the fine scale structure so the effects of numerical diffusion, while certainly important, are less disastrous then if sophisticated turbulence models are being compared.

A number of relationships, basically all equivalent, have been given for estimating the magnitude of the numerical diffusion due to skewness. The best known relationship is that of DeVahl Davis and Mallinson [1976] which gives the numerical diffusion, $v_n$, as

$$v_n = \frac{|V| \Delta x, \Delta y \sin 2 \theta}{4(\Delta x \sin^2 \theta + \Delta y \cos^2 \theta)}$$

where $|V|$ is the resultant velocity vector magnitude, $\Delta x$ and $\Delta y$ the grid sizes, and $\theta$ the angle the velocity vector takes with the $y$ direction.

The above relationship is used in this study to determine the ratio of numerical to physical diffusion so the magnitude and location of possible errors can be identified. Numerical diffusion can be expected to be greatest in the region where the cold inflow plunges under the ambient warm water, as here the velocity vectors will be most skewed off the grid lines.

E. **Boundary Conditions**

There are a number of features in the boundary conditions that are common to the various flow types considered in the following chapters. These are set out here for convenience. Various details are provided at the location where each specific problem is considered.
In the temperature calculations in all cases the reservoir bottom is assumed to be adiabatic. The dam face is maintained at the same temperature as the ambient fluid. (This is an arbitrary decision; conditions at the dam face do not influence flow in the plunge region.) Some heat flux will occur at the water surface. The amount and direction of this flux depends on the specific conditions prevailing. In the interest of generality and also to concentrate attention on the pure hydrodynamics, the surface heat flux was taken as zero. Inflow temperatures are specified along the inflow boundary. The specification of the boundary velocities depends on whether the flow is laminar or turbulent, but effectively the no-slip condition is applied to velocities at the reservoir bottom and on the dam face. A river flow with an appropriate velocity profile constitutes the boundary condition at the upstream end. Water is withdrawn at the dam base with the same discharge as the inflow to preserve continuity.

The free water surface is treated as a surface of symmetry for all variables. This is the so-called "rigid lid" assumption where the free surface may be considered replaced by a frictionless lid pressing down on the water surface. Differences in water surface elevation cannot now occur. A non-horizontal water surface manifests itself in terms of the pressure field generated on the underside of the rigid lid by the evolved velocity field. Modeling of free surface phenomena such as waves, seiches or wind set-up are, of course, ruled out with the rigid lid assumption.

Though the rigid lid assumption is usually used without question, it is not error free. Errors arise because even though the rigid lid can "model" a non-horizontal surface by developing varying pressures, the loss (or gain) in waterway leads to velocities that are too high (or low). Errors may arise from two sources.

First, a river invariably enters a reservoir with a classic M1 water surface profile [Chow, 1959]. The extent of the encroachment of this profile into the reservoir increases with increasing river discharge and decreasing reservoir slope and side angle divergence. Figure III-5 illustrates the situation. It is clear that a great extent of pure river flow cannot be modeled with the rigid lid assumption. Provided the modeling starts some distance into the reservoir where the M1 curve has flattened out, errors arising from this source should be minimized. The requirement that a large extent of inlet channel cannot be modeled is not a disadvantage as the developed channel flow velocity profile is merely put as an upstream boundary condition on the reservoir model.

The only situation where problems could occur from this source are if the inflow is modeled too far upstream—at Section A, say, on Fig. III-5. Then the profile of the flow characteristics (velocity, turbulence parameters) may be quite different by the time they reach Section B, the "correct" model inflow point, from what they would be in an actual river inflow.

The second source of error arises because of the density effects. An estimate of this error can be made by considering the simple situation shown in Fig. III-6. This shows warm water (density \( \rho_0 \)) being displaced from left to right by the cold water flow with density \( \rho_0 + \Delta \rho \). In this idealized situation no mixing is occurring at the interface. The super-elevation in the lighter fluid is given by
Fig. III-5. Rigid lid approximation and M1 profile in reservoir.
Fig. III-6. Rigid lid approximation and buoyancy effect.
\[ \Delta H = \frac{\Delta \rho \ H}{\rho_o} \]

and the error (\(\Delta v\)) in the velocity due to the imposition of a rigid lid can readily be shown to be

\[ \frac{\Delta v}{V} = \frac{\Delta \rho}{\rho} \]

Hence, the error in the computed velocities with the rigid lid assumption is of the order of \(10^{-3}\) and is not material.

The hydrodynamics are thus computed with a fixed free surface and instantaneous inflow matching outflow. This is an adequate situation for the study of a particular flow phenomenon occurring in the reservoir. If reservoir modeling over a long time period is required with hydrologic effects taken into account, the basic numerical scheme can be extended with various levels of sophistication to accommodate changes in water surface elevation. This aspect is not considered here.
IV. NUMERICAL SOLUTIONS--LAMINAR FLOW

In this chapter the numerical scheme described in the previous chapter is applied to flow in the two model reservoir shapes: a classical two-dimensional sloping reservoir and a horizontal bed reservoir with diverging sides. Laminar flow is studied.

A. Two-Dimensional Sloping Reservoir

The sloping reservoir shape can be simulated as being rectangular, as shown in Fig. IV-1 to easily accommodate Cartesian axes based calculations, and this has been done in some cases [Oberkampf and Crow, 1976; Kao, Park, and Pao, 1978]. In the present case where the degree of flow confinement is important, this approach is not satisfactory.

Another approach commonly used in reservoir modeling is to use Cartesian coordinates but to numerically block-out some of the lower grid points so that the sloped reservoir bottom is simulated by a staircase effect as illustrated in Fig. IV-2. This will probably simulate correctly the gross flow processes associated with a deepening reservoir. However, the unquantifiable error associated with such a representation is unacceptable if the detailed hydrodynamics are being studied as is the case here.

The above problems can be bypassed if the flow situation is recast in cylindrical \( r, \theta, z \) coordinates. As discussed in Section II.B.1, the reservoir shape is neatly accommodated in this system as shown in Fig. II-2.

1. Governing equations

The governing equations are as given in section II.B.1. The equation forms at Eqs. (II-7) through (II-10) are used directly in the numerical scheme.

The control volume scheme described in the previous chapter encompasses the temporal, convection and diffusion terms in the basic governing equation. All other terms—the buoyancy term and those terms not appearing in the cartesian form of the equations—must be coded separately and added into the calculation as source terms in each control volume. That is, each of the governing equations is considered in the standard form of

\[ \frac{\partial \phi}{\partial t} + \text{convection terms} = \text{pressure term} - \text{diffusion terms} + S_c \]  

(IV-1)

where \( \phi \) is a general variable and \( S_c \) is an all encompassing source term made up of all the terms not specifically listed. The pressure term disappears when \( \phi \) is not a velocity. The source term and diffusion coefficient, \( \Gamma \), for each variable are given in Table IV-1.
Fig. IV-1. Rectangular reservoir in Cartesian coordinates.

Fig. IV-2. Reservoir with blocked-out sections to simulate a slope.
TABLE IV-1. Source Terms and Diffusion Coefficients for Variables with Laminar Flow in the $r, \theta$ coordinate system

<table>
<thead>
<tr>
<th>Variable Name</th>
<th>Diffusion Coefficient</th>
<th>Source Term, $S_c$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$u_\theta$</td>
<td>$v - \frac{uv}{r} - \frac{\nu u}{r^2} + \frac{2v}{r^2} \frac{\delta v}{\partial \theta} + g\theta \cos \theta (T_o - T)$</td>
<td></td>
</tr>
<tr>
<td>$v_r$</td>
<td>$\frac{u^2}{r} - \frac{vV}{r^2} - \frac{2v}{r^2} \frac{\delta u}{\partial \theta} + g\theta \sin \theta (T_o - T)$</td>
<td></td>
</tr>
<tr>
<td>$T$</td>
<td>$v/Pr$</td>
<td>None</td>
</tr>
</tbody>
</table>

2. **Qualitative model testing**

The various procedures and schemes outlined in the previous chapter all appear reasonable and have worked well in many problems of various types. The ultimate test in any application, of course, is that the complete numerical package reproduce results that conform to physical reality as observed in the laboratory or in the field.

To examine how the model behaves under conditions where plunging could occur, solutions were extracted for flows corresponding to the experimental runs of Singh and Shah [1971]. Their experimental reservoir configuration was scaled from their paper and is shown schematically in Fig. IV-3.

The variable ranges in Singh on Shah's work are given in Table IV-2.


<table>
<thead>
<tr>
<th>Variable</th>
<th>Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reynolds Number</td>
<td>600 - 11,000</td>
</tr>
<tr>
<td>Reservoir Slope</td>
<td>.0056 - .0215</td>
</tr>
<tr>
<td>Stream Discharge $(q)$</td>
<td>.5 - 135 cc/cm/sec</td>
</tr>
<tr>
<td>$(q^2/g')^{1/3}$</td>
<td>1.2 - 16 cm</td>
</tr>
</tbody>
</table>
Fig. IV-3. Schematic of experimental arrangement of Singh and Shah [1971].
The quantity \( q^2/\rho R \)^{1/3} was the parameter used by Singh and Shah [1971] to correlate their experimental data. The parameter is related to the inflow densimetric Froude number \( F_o \) as

\[
\left( \frac{q^2}{\rho R} \right)^{1/3} = d F_o^{2/3}
\]

where \( d \) is the inlet channel depth.

Singh and Shah [1971] give precise details of only one of their runs. This is given in Fig. 8 in their paper and listed in Table IV-3 below. (The inflow discharge shown was computed from their velocity profile rather than their given value which does not match their data.)

**TABLE IV-3. Variable Values for a Single Experiment (from Fig. 8, Singh and Shah [1971])**

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stream discharge (cc/cm/sec)</td>
<td>32</td>
</tr>
<tr>
<td>Reservoir Slope</td>
<td>.0104</td>
</tr>
<tr>
<td>( \Delta \rho/\rho )</td>
<td>.0017</td>
</tr>
<tr>
<td>Plunge depth (cm)</td>
<td>12.7</td>
</tr>
</tbody>
</table>

**a. Boundary and initial conditions**

Boundary conditions were as outlined in Section III.E above. A one-seventh power law velocity profile was used at the flow inlet.

The reservoir water was initially at rest with temperature at all points being equal to \( T_0 \). At time \( t = 0 + \) colder water, at temperature \( T_{in} \), was input at the upstream end of the dam and an identical amount was withdrawn through one control volume at the dam base.

**b. Grid size and time step**

A configuration of 42 longitudinal and 22 vertical evenly spaced grid points was used to model the reservoir. As the objective here was to study the overall model behavior, grid dependence was not extensively tested. However, one of the runs (No. 3) was repeated with a 22x12 grid. This gave a plunge depth of less than 7% difference from the original run.

Time steps of 5, 10, and 20 seconds were examined. The potentially most unstable region in the flow field is that behind the plunge point.
Here velocities must decrease as the cold water approaches, then must change direction when it plunges and finally settles down to the new backflow.

The behavior of the velocity at a point on the water surface about four grid points behind the plunge point was examined for the above time steps for a single set of flow conditions (that of Run No. 3 in Table IV-4). The development of the velocity with elapsed time is shown in Fig. IV-4. It can be observed that a time step of 20 seconds is too coarse to model this area. Time steps of 5 or 10 seconds appear to be adequate. A time step of 10 seconds was adopted. This was used for all runs as none of the run parameters varied drastically from those of the test run.

c. Results and conclusions

A number of flows were simulated within the variable ranges given by Singh and Shah [1971]. The details of the runs are given in Table IV-4. A β value (Table IV-2) of .000171 and a Prandtl number value of 5.0 were used in all runs. Run No. 3 uses the variable values specifically given by Singh and Shah [1971] and listed previously in Table IV-3. (An initial depth had to be assumed for this run as this was not given in the paper.) Two flows were simulated at a value of \((q^{2}/g'1/3)\) above the experimental range.

In qualitative terms, the simulations developed in a similar manner to that described by Singh and Shah [1971]. The warm water was initially displaced forward and velocities were forward at all points. Eventually when the cold water had penetrated sufficiently far into the reservoir, a small region of backflow appeared at the water surface. This backflow region grew larger as time elapsed. Eventually the cold water front reached the dam base and the entire warm water zone was a zone of recirculation. The time development of the flow for Run No. 2 is shown in Figs. IV-5, IV-6 and IV-7.

The plunge point was well defined in both the velocity and temperature fields. A typical configuration of the plunge region is shown in Fig. IV-8. No attempt was made to interpolate between the opposing velocities to fix the plunge point position more precisely than given by the numerical grid.

Once formed, the plunge point usually drifted downstream and never really settled into a quasi-steady state. Taking the plunge point as the position where plunging first appears, depths at the plunge point were extracted from each run and are shown in Table IV-4. The values are shown plotted against \((q^{2}/g'1/3)\) in Fig. IV-9.

The experimental data is also shown on that plot, as is the least squares regression line fitted to the data by Singh and Shah [1971]. It can be seen that in terms of plunge point location the agreement between numerical and experimental data is quite good.

The flows with the large plunge depth (Runs 4, 5 in Table IV-4), were run when it appeared that runs 6 and 7 were drifting above the experimental data. However, it can be seen that Runs 4 and 5 confirm the general trend
### TABLE IV-4. Details of Numerical Simulations of the Experiments of Singh and Shah [1971]

<table>
<thead>
<tr>
<th>Run No.</th>
<th>Bottom Inflow Slope</th>
<th>Inflow Depth (cm)</th>
<th>Stream Inflow (q) (cc/cm/sec)</th>
<th>$T_o$ ($^\circ$C)</th>
<th>$T_{in}$ flow ($^\circ$C)</th>
<th>$(q^2/g')^{1/3}$ (cm)</th>
<th>$F_o$</th>
<th>Re</th>
<th>@ Plunge Point Grid Point Depth</th>
<th>Plunge Depth (cm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>SSHAH2</td>
<td>.01</td>
<td>3.5</td>
<td>11.3</td>
<td>25</td>
<td>15</td>
<td>4.24</td>
<td>1.33</td>
<td>1132</td>
<td>15</td>
<td>7.7</td>
</tr>
<tr>
<td>SSHAH3</td>
<td>.0104</td>
<td>8.3</td>
<td>30.8</td>
<td>25</td>
<td>15</td>
<td>8.27</td>
<td>0.99</td>
<td>3080</td>
<td>14</td>
<td>12.4</td>
</tr>
<tr>
<td>SSHAH4</td>
<td>.02</td>
<td>16.0</td>
<td>95.3</td>
<td>18</td>
<td>15</td>
<td>26.2</td>
<td>2.10</td>
<td>9533</td>
<td>32</td>
<td>35.1</td>
</tr>
<tr>
<td>SSHAH5</td>
<td>.02</td>
<td>16.0</td>
<td>59.2</td>
<td>17</td>
<td>15</td>
<td>21.8</td>
<td>1.60</td>
<td>5923</td>
<td>23</td>
<td>29.4</td>
</tr>
<tr>
<td>SSHAH6</td>
<td>.02</td>
<td>8.0</td>
<td>42.1</td>
<td>18</td>
<td>15</td>
<td>15.2</td>
<td>2.62</td>
<td>4206</td>
<td>25</td>
<td>22.7</td>
</tr>
<tr>
<td>SSHAH7</td>
<td>.02</td>
<td>8.0</td>
<td>29.6</td>
<td>18</td>
<td>15</td>
<td>12.0</td>
<td>1.84</td>
<td>2962</td>
<td>18</td>
<td>18.3</td>
</tr>
<tr>
<td>SSHAH10</td>
<td>.02</td>
<td>16.0</td>
<td>6.2</td>
<td>25</td>
<td>10</td>
<td>2.45</td>
<td>.06</td>
<td>615</td>
<td>@ inflow</td>
<td>16.0</td>
</tr>
<tr>
<td>SSHAH11</td>
<td>.02</td>
<td>16.0</td>
<td>95.3</td>
<td>25</td>
<td>10</td>
<td>15.4</td>
<td>.94</td>
<td>9525</td>
<td>11</td>
<td>21.9</td>
</tr>
<tr>
<td>SSHAH1L</td>
<td>.01</td>
<td>3.5</td>
<td>11.3</td>
<td>25</td>
<td>15</td>
<td>4.24</td>
<td>1.33</td>
<td>1132</td>
<td>16</td>
<td>8.0</td>
</tr>
</tbody>
</table>
RUN NO. 3
VELOCITY AT
GRID POINT NO. 21

Fig. IV-4. Velocity behavior near plunge point for three time steps.
Fig. IV-5. Reservoir flow development in simulation of experiment of Singh and Shah [1971].

Elapsed time secs = 200.00

Run NO

0.1 m/sec

Distance into reservoir meters
ELAPSED TIME SECS = 280.00

DISTANCE INTO RESERVOIR METERS

RUN NO

0.1M/SEC

Fig. IV-8. Velocity and temperature field configuration at the plunge point.
Fig. IV-9. Comparison of predicted plunge points with data of Singh and Shah [1971].
of the data. The limit bounds shown about the numerical points are the extent to which the plunge depth would move if it were one grid point upstream or downstream.

One flow with an extremely high buoyancy force (No. 11 with $F = .06$) was simulated. This flow plunged at the reservoir entrance. So, even in the extreme case the model behaved realistically. One simulation (No. 2) was rerun with a laminar velocity profile at the inflow (Run No. L1). This modification yielded some changes in the hydrodynamics but the plunge point position remained basically unchanged.

The experimental velocity profile at the plunge point is given by Singh and Shah [1971] in Fig. 8 of their paper for the experimental run whose parameters they identified. In Fig. IV-10 this velocity profile is compared to the velocity profile just downstream of the plunge point in the equivalent numerical simulation. It can be seen that the profiles are qualitatively different. The numerical profile is of course laminar as demanded by the governing equations. The experimental profile is turbulent ($Re \sim 3000$) with high velocities down close to the reservoir bottom.

The above results show that under the initial and buoyancy conditions prescribed, the numerical model can reproduce plunging and can respond to changed buoyancy forces in a way that parallels the experimental data. However, these results are best interpreted as a general demonstration that the numerical scheme does possess the essential ingredients for modeling plunging flow rather than a quantitative illustration of its powers.

Some features of the procedure are quantitatively examined in detail in the following section.

3. **Quantitative model testing**

The results of the previous section show that the numerical model can reproduce the gross features of a plunging flow. In this section the quantitative prediction capabilities of the model are examined in detail.

This examination is not linked to flow in a specific experimental apparatus. Instead, a non-dimensional situation is investigated. Using this approach the effect of the governing parameters identified in Section II.B.2 can be specifically looked at in the numerical model.

In a later stage of research, if the numerical model proves to be quantitatively sound, the non-dimensional formulation may be used to generate relations between flow field characteristics and the governing parameters.

a. **Non-dimensional equations**

The non-dimensional equations given in Section II.B.2 are used directly in the numerical scheme. The governing dimensionless parameters in these equations are the inflow Reynolds number, $Re$, the inflow densimetric Froude number, $F_o$, and the Prandtl number, $Pr$, as defined in Section II.B.2. Since the Prandtl number for water may be taken as a constant ($\approx 5.0$ was used in the present series of runs) plunging flow is thus governed by the two parameters $Re$ and $F_o$. 
Fig. IV-10. Comparison of experimental and numerical velocity profiles near the plunge point. (Experimental data from Singh and Shah [1971]. Numerical from Run 3, elapsed time 320 secs, grid point 17.)
The non-dimensional equations may be expressed in the standard equation form of Equation (IV-1). Expressed in this way the source terms, $S_c$, and diffusion coefficients, $r$, in the non-dimensional equations are as given in Table IV-5. The quantities $u$ and $v$ are the non-dimensional angular and radial velocities, respectively, and $T$ is the dimensionless temperature.

TABLE IV-5. Source Terms and Diffusion Coefficients in the Non-dimensional $r, \theta$ Equations

<table>
<thead>
<tr>
<th>Variable Name</th>
<th>Diffusion Coefficient</th>
<th>Source Term, $S_c$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$u$</td>
<td>$\frac{1}{Re}$</td>
<td>$-\frac{uv}{r} - \frac{u}{Re r^2} + \frac{2}{Re r^2} \frac{\delta v}{\delta \theta} + \frac{T \cos \theta}{F_0}$</td>
</tr>
<tr>
<td>$v$</td>
<td>$\frac{1}{Re}$</td>
<td>$\frac{u^2}{r} - \frac{v}{Re r^2} - \frac{2}{Re r^2} \frac{\delta u}{\delta \theta} + \frac{T \sin \theta}{F_0}$</td>
</tr>
<tr>
<td>$T$</td>
<td>$\frac{1}{(Re Pr)}$</td>
<td>None</td>
</tr>
</tbody>
</table>

b. Initial examination

An initial examination was carried out with a very large reservoir. The reservoir layout and grid configuration are shown in Fig. IV-11. The reservoir is 3000 units long with a bed slope of 0.01 and an inflow depth of 1.0 units. The upstream one-third of the reservoir (i.e. 1000 units long) was modeled with 20 control volumes evenly sized at 50.0 units each. The remaining 2000 units was modeled using 10 control volumes gradually expanding in size from 100 units adjacent to the uniform grid to 280 units at the dam. In the vertical, 10 uniformly spaced control volumes were used.

This grid layout was used to concentrate the computations in that area where the plunging was expected to occur. The long reservoir and expanding grid was used merely to move the downstream boundary condition far away from the plunge region.

A time step of 25.0 time units was selected. With an average inflow velocity of 1.0 time units, it would thus take two time steps for information to travel from node to node at the inflow.

The boundary conditions were as outlined in Section III.E. Initially the reservoir water was at rest with dimensionless temperature...
Fig. IV-11. Non-dimensional reservoir layout.
everywhere being 0.0. At time \( t = 0^+ \), an inflow with average velocity of 1.0 units and temperature and 1.0 was started and an identical discharge was taken out at the dam base to preserve continuity. The stream inflow was given a one-seventh power law profile.

Four runs were made with this configuration for different densimetric Froude number values, all other parameters being kept constant. The run details and results are given in Table IV-6. The plunge depths shown in that table are the depths at which plunging first occurs. The plunge location is defined in the same way as in the previous section.

The plunge depths are shown plotted against densimetric Froude number in Fig. IV-12. The curve \( H_p/d = 2.4 F_o^{2/3} \) fits through those points very well. This suggests a densimetric Froude number at the plunge point of about 0.27. The curve \( H_p/d = 1.3 F_o^{2/3} \) is also shown in Fig. IV-12. If, based on Singh and Shah's data, the latter curve is taken as being the correct curve, then the numerically generated curve is in error by about 85 percent at all points.

This error is systematically examined in the following sections in an attempt to identify its major sources.

c. **Effect of grid size and time step**

The effect of grid size and time step, the standard features that are checked in all numerical work, are examined in this section. The details of the runs made for this examination are given in Table IV-7.

Since the computer simulations of Singh and Shah's [1971] experiments gave results close to the data, one of these simulations (SSAH4 in Table IV-4) was rerun in non-dimensional terms. The results were identical to the dimensional case. The details of this run (labelled PROD 12) are shown in Table IV-7. Using this run as a basis for comparison, the control volume size was varied over the range 1 unit to 50 units (Runs PROD 13, 14, 17, 27, 25). The depths at which plunging first occurred were extracted and are shown in Table IV-7. These depths are plotted (as full circles) against control volume size in Fig. IV-13. It can be seen that as control volume size increases, plunging moves increasing distances downstream from its "correct" location. The open circles indicate grid points between the numerical plunge point and the location where the data of Singh and Shah [1971] suggest plunging should occur. It is interesting to note that as the grid size decreases, the value \( H_p/d = 1.3(2.1)^{2/3} = 2.13 \) corresponding to Singh and Shah's data is approached.

The runs with control volume size of 30 and 50 units used a bottom slope of .01 against .02 for the other runs. This was done to keep the increase in depth from grid point to grid point within a reasonable range. Thus the latter two points may have a built-in slope effect. Nevertheless the general trend in all the points shows an increasing error as control volume size increases. A control volume size of about \( Dx/d = 4.0 \) appears to be necessary to keep the numerically generated plunge location within 10 percent of the experimental value.

All the above data refer to reservoirs of different lengths. To check that absolute reservoir size is not influencing the plunge point position,
<table>
<thead>
<tr>
<th>Run No.</th>
<th>Reservoir Length</th>
<th>GRID YxY</th>
<th>ΔY</th>
<th>Reynolds Number</th>
<th>F₀</th>
<th>Time Step</th>
<th>Bottom Slope</th>
<th>Inlet Radius</th>
<th>Hₚ/d</th>
</tr>
</thead>
<tbody>
<tr>
<td>Prod 1</td>
<td>3000</td>
<td>32x12</td>
<td>50</td>
<td>10³</td>
<td>2</td>
<td>25</td>
<td>.01</td>
<td>100</td>
<td>3.75</td>
</tr>
<tr>
<td>3</td>
<td>3000</td>
<td>32x12</td>
<td>50</td>
<td>10³</td>
<td>4</td>
<td>25</td>
<td>.01</td>
<td>100</td>
<td>5.75</td>
</tr>
<tr>
<td>4</td>
<td>3000</td>
<td>32x12</td>
<td>50</td>
<td>10³</td>
<td>6</td>
<td>25</td>
<td>.01</td>
<td>100</td>
<td>7.75</td>
</tr>
<tr>
<td>5</td>
<td>3000</td>
<td>32x12</td>
<td>50</td>
<td>10³</td>
<td>8</td>
<td>25</td>
<td>.01</td>
<td>100</td>
<td>9.25</td>
</tr>
</tbody>
</table>
Fig. IV-12. Plunge depth as a function of inflow densimetric Froude number.
TABLE IV-7. Compute Runs to Examine Plunge Depth Variation with Various Numerical Parameters

<table>
<thead>
<tr>
<th>Run No.</th>
<th>Reservoir Length, YL</th>
<th>Grid YXX</th>
<th>Control Volume Size, Δy</th>
<th>Reynolds Number Re</th>
<th>Inlet Densimetric Froude Nr. ( F_o )</th>
<th>Time Step DT</th>
<th>Bottom Slope S</th>
<th>Inlet Radius R(1)</th>
<th>Dimensionless Plunge Depth ( H_p/d )</th>
</tr>
</thead>
<tbody>
<tr>
<td>PROD 12</td>
<td>80</td>
<td>42x22</td>
<td>2</td>
<td>9533</td>
<td>2.10</td>
<td>4</td>
<td>.02</td>
<td>50</td>
<td>2.18</td>
</tr>
<tr>
<td>PROD 13</td>
<td>160</td>
<td>42x22</td>
<td>4</td>
<td>9533</td>
<td>2.10</td>
<td>4</td>
<td>.02</td>
<td>50</td>
<td>2.32</td>
</tr>
<tr>
<td>PROD 14</td>
<td>640</td>
<td>42x22</td>
<td>16</td>
<td>9533</td>
<td>2.10</td>
<td>4</td>
<td>.02</td>
<td>50</td>
<td>2.76</td>
</tr>
<tr>
<td>PROD 17</td>
<td>40</td>
<td>42x22</td>
<td>1</td>
<td>9533</td>
<td>2.10</td>
<td>4</td>
<td>.02</td>
<td>90</td>
<td>2.11</td>
</tr>
<tr>
<td>PROD 21</td>
<td>640</td>
<td>42x12</td>
<td>16</td>
<td>9533</td>
<td>2.10</td>
<td>4</td>
<td>.02</td>
<td>50</td>
<td>2.76</td>
</tr>
<tr>
<td>PROD 25</td>
<td>1000</td>
<td>22x12</td>
<td>50</td>
<td>9533</td>
<td>2.10</td>
<td>25</td>
<td>.01</td>
<td>100</td>
<td>3.25</td>
</tr>
<tr>
<td>PROD 26</td>
<td>80</td>
<td>22x22</td>
<td>4</td>
<td>9533</td>
<td>2.10</td>
<td>4</td>
<td>.02</td>
<td>50</td>
<td>2.32</td>
</tr>
<tr>
<td>PROD 27</td>
<td>600</td>
<td>22x22</td>
<td>30</td>
<td>9533</td>
<td>2.10</td>
<td>15</td>
<td>.01</td>
<td>100</td>
<td>2.65</td>
</tr>
<tr>
<td>PROD 28</td>
<td>160</td>
<td>42x22</td>
<td>4</td>
<td>9533</td>
<td>2.10</td>
<td>1</td>
<td>.02</td>
<td>50</td>
<td>2.24</td>
</tr>
<tr>
<td>PROD 29</td>
<td>160</td>
<td>42x22</td>
<td>5</td>
<td>9533</td>
<td>2.10</td>
<td>16</td>
<td>.02</td>
<td>50</td>
<td>2.56</td>
</tr>
</tbody>
</table>
run number 13 was rerun in a reservoir one-half the original length. This run is numbered 26 and gives identical results to the original. This confirms that the dam position, i.e. the reservoir length, has no influence on the plunge point location (provided of course that the reservoir is of sufficient length for plunging to occur upstream of the dam).

The effect of the time step was examined by rerunning Run No. 13 with time steps of 1 to 16 units (compared to the original 4 units). These runs, numbered 28 and 29, show that the time step has some influence on plunge point location. However, the fact that the plunge point position moves in time means that with the larger time step, plunging may first appear further out in the reservoir than with a smaller time step, even though at that elapsed time the plunge point may be in the same position for both time steps. This consideration will remove most of the discrepancy with the large time step.

d. **Physical consideration**

The above considerations are purely numerical ones. In this section the basic underlying physics are discussed. In the simulation of the Singh and Shah experiments and in the above runs, flow separation invariably occurred on the reservoir bottom. A recirculation zone varying in length from two or three control volume lengths to about one-half the reservoir length was formed in which flow on the reservoir bottom was actually going back uphill against the slope. This is clearly unrealistic behavior and would not occur in a prototype of similar slope where turbulent velocity profiles with high momentum near the bed would prevent such occurrences. The separation problem arises because of the fact that a laminar model is being used.

This flow situation can be analyzed by considering that the steady state flow in a diverging two-dimensional channel (or half symmetrical channel such as the reservoir) obeys the laminar Navier-Stokes equations. If the flow is assumed to be similar at all radial points, then the radial velocity \( V_r \) may be written as

\[
V_r = \frac{v F(\theta)}{r}
\]

where \( v \) is the kinematic viscosity and \( F(\theta) \) is a dimensionless velocity profile. With this assumption the governing partial differential equations reduce to the nonlinear ordinary differential equation

\[
F^2 + 4F + F'' + K = 0 \quad (IV-2)
\]

with the boundary condition \( F(\pm \delta) = 0 \), where \( \delta \) is the channel half angle. The prime indicates differentiation with respect to \( \theta \) and \( K \) is a constant incorporating the radial pressure gradient.

This flow was originally studied by Jeffrey [1915] and Hamel [1916]. Further analysis was carried out by Rosenhead [1940], and Millsaps and Polhausen [1953]. Integration of Eq. (IV-2) yields an expression for \( F \) in terms of an elliptic integral. The integral may be evaluated using either
Jacobian elliptic functions or Weierstrassian functions. The velocity profiles in the channel are a function of both the divergence angle $\delta$ and the Reynolds number $Re$, where $Re$ is as defined in section II.B.2. Imposition of the condition $\delta F/\delta \theta \bigg|_{\theta=\delta} = 0$ yields the relationship

$$\delta Re = 4.71$$

(IV-3)

for an incipient separation velocity profile type, where $\delta$ is in radians.

The above theoretical results refer to steady state radial flow from a central source with a similar velocity profile at all radial locations. Apart from the unsteady aspect, the reservoir flow situation differs from the above flow in a number of ways.

First, the upstream imposed velocity profile means that pure radial similarity cannot exist. If the flow physics demand that separation occur, then it will have to occur in the reservoir at a downstream distance that will depend on the form of the inflow profile and on the initial radius as this determines the magnitude of the adverse pressure gradient. (Strictly speaking, separation never occurs in Jeffrey-Hammel flow as whatever profiles exist, persist from $r = 0$ to $r = \infty$. However, the condition given by Eq. (IV-3) is usually interpreted as a separation criterion.) A flatter inflow profile such as a one-seventh power law can be expected to move separation further downstream than a parabolic (laminar) one. Second, radial similarity conditions are violated also by the outflow condition which is at the dam base rather than by a similar profile over the reservoir depth. Third, the presence of the buoyancy force (if such could exist in steady-state flow) will act to suppress separation.

When the flow is unsteady, as it is here, then the problems of boundary layer growth, the development of the separation flow, and the development of the plunging flow must be added on top of the above complications. All of those factors combine to produce a particularly complex flow field even in the absence of buoyancy forces.

The theoretical separation result given by Eq. (VI-3) will give some idea of the strength of the tendency to separate. In the reservoir flows considered above, bottom slope (i.e. $\delta$) was either .01 or .02. Hence the critical Reynolds numbers for separation are 471 or 236, respectively. The actual Reynolds numbers were well above those values in all cases so it is not surprising that separation occurred.

The computer run results shown plotted in Fig. IV-13 were obtained with a Reynolds number of 9533. Some of those were rerun with a Reynolds number of 1000. The details of these runs are shown in Table IV-8. The resulting plunge depths are plotted in Fig. IV-14. It can be seen that in each case the predicted plunge depth is deeper than the results at a Reynolds number of 9533. It is suspected that this difference is due to the separated region. In the lower Reynolds number flow the separated region occurs further downstream and possibly has displaced the plunge region outwards.

As a further check on Reynolds numbers effects, Run No. PROD 12 (this corresponds to simulation No. 4 of the Singh and Shah [1971] experiments)
Fig. IV-13. Effect of control volume size on predicted plunge depth (F_o = 2.1).
<table>
<thead>
<tr>
<th>Run No.</th>
<th>Reservoir Length, YL</th>
<th>Grid Size, Δy</th>
<th>Reynolds Number, Re</th>
<th>Inflow Densimetric Froude Nr. $F_o$</th>
<th>Time Step, DT</th>
<th>Bottom Slope, S</th>
<th>Inlet Radius, (R(1))</th>
<th>Dimensionless Plunge Depth $H_p/d$</th>
</tr>
</thead>
<tbody>
<tr>
<td>PROD 1</td>
<td>3000</td>
<td>32x12</td>
<td>50</td>
<td>2.0</td>
<td>25</td>
<td>.01</td>
<td>100</td>
<td>3.75</td>
</tr>
<tr>
<td>PROD 20</td>
<td>500</td>
<td>52x22</td>
<td>10</td>
<td>2.10</td>
<td>5</td>
<td>.01</td>
<td>100</td>
<td>2.95</td>
</tr>
<tr>
<td>PROD 23</td>
<td>80</td>
<td>42x22</td>
<td>2</td>
<td>2.10</td>
<td>4</td>
<td>.02</td>
<td>50</td>
<td>2.50</td>
</tr>
</tbody>
</table>
Fig. IV-14. Predicted plunge depth with two Reynolds numbers.
was run with four Reynolds numbers. The details are given in Table IV-9 and the plunge depths are plotted against Reynolds number in Fig. IV-15. The two high Reynolds number flows are much the same. Separation and plunging occur at about the same point in each. At smaller Reynolds numbers, changes occur. At Re = 1000 the plunge point moves downstream as discussed above. At an Re of 100 (this is less than the theoretical separation value of 236) separation did not occur. However, a typical laminar profile developed with high velocities at the water surface and plunging flow never developed. The simulation was run long enough to allow the cold water to advance to the dam where the dimensionless depth was 3.4. To check that this latter run was not influenced by changed diffusion in the energy equation, the simulation was rerun with a Prandtl number of 50.0 to give the same Peclet number as with the Reynolds number of 1000. The results remained unchanged.

The use of the one-seventh power law velocity profile will tend to suppress or delay separation since it puts more momentum close to the reservoir bottom than a laminar profile. This effect was examined by rerunning Run No. PROD 13 with a laminar inflow profile. (This run is labelled PROD 32 in Table IV-9.) This change resulted in a much larger separation region with the dimensionless plunge depth being pushed out to a depth of about 2.64 compared to the original 2.32.

A comparison of runs numbered PROD 13 and PROD 26 (Table IV-7) showed identical separation regions. These were for reservoirs of different lengths, and suggests that outflow conditions do not strongly influence the separation zone.

The discussion in this section shows that there are strong physical forces in the laminar flow model which render the basic model results suspect for application to prototype situations. It is possible that these physical forces also interact with the numerical grid and time steps. Further discussion of the limitation of laminar flow models and the conclusion to be drawn is delayed until after the diverging reservoir case is presented. The diverging situation is now examined briefly.

B. Diverging Reservoir

As outlined in Section I.C, if the decrease in flow velocity along a reservoir is due mainly to the increasing width of the reservoir rather than its increasing depth, then an axisymmetric system can be used to model this situation. This flow arrangement is shown in Fig. IV-16. The flow is symmetrical in the θ coordinate and the z axis points vertically downwards. The area shown shaded can be blocked out to simulate flow over a beach with a dropoff at its end.

1. Governing equations

The governing equations are as given in Section II.C.1. The equation forms at Equations (II-40) through (II-43) are used directly in the numerical scheme. Again each of the governing equations may be written in the standard form of Eq. (IV-1) with all non-Cartesian terms are included in the source term $S_c$. The source term and diffusion coefficients for each
### TABLE IV-9. Computer Runs to Examine Plunge Point Variation with Reynolds Number

<table>
<thead>
<tr>
<th>Run No.</th>
<th>Reservoir Length</th>
<th>GRID YxX</th>
<th>( \Delta Y )</th>
<th>Reynolds Number</th>
<th>( F_0 )</th>
<th>Time Step</th>
<th>Bottom Slope</th>
<th>Inlet Radius</th>
<th>Prandtl Number</th>
<th>( H/d )</th>
</tr>
</thead>
<tbody>
<tr>
<td>PROD 23</td>
<td>80</td>
<td>42x22</td>
<td>2</td>
<td>1000</td>
<td>2.1</td>
<td>4.0</td>
<td>.02</td>
<td>50</td>
<td>5.0</td>
<td>2.5</td>
</tr>
<tr>
<td>PROD 12</td>
<td>80</td>
<td>42x22</td>
<td>2</td>
<td>9533</td>
<td>2.1</td>
<td>4.0</td>
<td>.02</td>
<td>50</td>
<td>5.0</td>
<td>2.18</td>
</tr>
<tr>
<td>PROD 33</td>
<td>120</td>
<td>62x22</td>
<td>2</td>
<td>50,000</td>
<td>2.1</td>
<td>4.0</td>
<td>.02</td>
<td>50</td>
<td>5.0</td>
<td>2.14</td>
</tr>
<tr>
<td>PROD 34</td>
<td>120</td>
<td>62x22</td>
<td>2</td>
<td>100</td>
<td>2.1</td>
<td>4.0</td>
<td>.02</td>
<td>50</td>
<td>5.0</td>
<td>No Plunging</td>
</tr>
<tr>
<td>PROD 35</td>
<td>120</td>
<td>62x22</td>
<td>2</td>
<td>100</td>
<td>2.1</td>
<td>4.0</td>
<td>.02</td>
<td>50</td>
<td>50.0</td>
<td>No Plunging</td>
</tr>
<tr>
<td>PROD 32</td>
<td>160</td>
<td>42x22</td>
<td>3</td>
<td>9533</td>
<td>2.1</td>
<td>4.0</td>
<td>.02</td>
<td>50</td>
<td>5.0</td>
<td>2.64 (lam. profile)</td>
</tr>
</tbody>
</table>
Fig. IV-15. Plunge depths for a constant densimetric Froude number (2.1) and control volume size as a function of Reynolds number.
Fig. IV-16. Flow arrangement in an axisymmetric reservoir.
variable are given in Table IV-10. When \( \phi \) is not a velocity, the pressure term vanishes.

<table>
<thead>
<tr>
<th>Variable Name</th>
<th>Diffusion Coefficient, ( \Gamma )</th>
<th>Source Term ( S^c )</th>
</tr>
</thead>
<tbody>
<tr>
<td>( u_z )</td>
<td>( \nu )</td>
<td>( \rho \beta (T_o - T) )</td>
</tr>
<tr>
<td>( v )</td>
<td>( \nu )</td>
<td>( -\nu v/r^2 )</td>
</tr>
<tr>
<td>( T )</td>
<td>( \nu/Pr )</td>
<td>None</td>
</tr>
</tbody>
</table>

2. Qualitative Model Testing

The axisymmetric model was used to simulate two flows in a small model reservoir with a flat beach and diverging sides. A drop-off at the end of the beach was simulated by "blocking out" the area shown shaded at Fig. IV-16. In each case, a grid of 24 horizontal control volumes was used on the beach area with 10 control volumes in the remainder of the reservoir. A grid of 20 vertical control volumes was used on the flow area over the beach and 10 control volumes over the remaining reservoir depth. The effect of varying the above parameters was not examined as this is merely a qualitative test and problems similar to those encountered in the two-dimensional case were anticipated. The run details are given in Table IV-11.

The initial and boundary conditions were as outlined in Section III.E. The flow was started from rest as in the sloping situation. The inflow velocity was given a one-seventh power law profile.

A plot of the numerically generated flow field for run Exp. 2 at an elapsed time of 90 seconds is shown in Fig. IV-17. It can be seen that the flow field is qualitatively as expected with the warm water wedge protruding into the diverging channel and the cold water plunging and falling off the beach end. The beach area is shown to a larger scale in Fig. IV-18. A region of separation can be seen on the reservoir bottom near the inflow.

Simulation number Exp. 1 was basically similar to number Exp. 2 but was allowed to run onto an elapsed time of 400 seconds. Over that time the full development of the flow field was observed. After the initial plunging, a warm water recirculating wedge developed. This is the flow
TABLE IV-11. Details of Axisymmetric Reservoir Simulations

<table>
<thead>
<tr>
<th>Run No.</th>
<th>Grid on Beach</th>
<th>Total Grid</th>
<th>Depth (m)</th>
<th>Beach Length (m)</th>
<th>Inlet Radius (R(l))</th>
<th>Time Step (secs)</th>
<th>Reynolds No.</th>
<th>Densimetric Froude No.</th>
</tr>
</thead>
<tbody>
<tr>
<td>EXP 1</td>
<td>24x20</td>
<td>34x30</td>
<td>.067</td>
<td>4.70</td>
<td>1.24</td>
<td>1.25</td>
<td>5574</td>
<td>1.63</td>
</tr>
<tr>
<td>EXP 2</td>
<td>24x20</td>
<td>34x30</td>
<td>.084</td>
<td>2.34</td>
<td>0.40</td>
<td>1.25</td>
<td>5574</td>
<td>1.21</td>
</tr>
</tbody>
</table>
Fig. IV-17. Flow field in simulation number Exp. 2.
RUN NO. EXP2
ELAPSED TIME secs = 40.

Fig. IV-18. Flow field on beach in simulation number Exp. 2.
pattern expected of the quasi-steady state solution. A small separation region was evident on the reservoir bottom between the inflow and the plunge region. However, a quasi-steady state never really developed. Instead the plunge region continued to migrate downstream and the separated region continued to grow. Eventually the warm water wedge was completely displaced off the beach and the lower half of the flow field on the beach consisted of a separated recirculating region. This developed flow field on the beach area is shown in Fig. IV-19.

This is clearly unrealistic prototype behavior. Some migration of the plunge point can perhaps be expected as the reservoir pool of warm water is entrained and discharged, particularly in the sloping case. In the axisymmetric situation, however, there is a very large pool of warm water compared to the inflow, and entrainment cannot really be used to explain the large plunge point migration.

3. **Quantitative model testing**

Quantitative model testing is best carried out, as in the two-dimensional case, using the non-dimensional equations in a non-dimensional reservoir. An examination was not proceeded with as it was felt that this would merely duplicate in another form the sloping reservoir examination. The governing equations are given below in the standard form for record purposes.

The non-dimensional equations are as given in Section II.C.2. The equations contain the three dimensionless number Re, \( F_o \) and Pr as defined in Section II.B.2.

The non-dimensional equations may be expressed in the standard equation form of Eq. (IV-1). Expressed in this way the source terms, \( S_c \), and diffusion coefficients, \( \Gamma \), in the non-dimensional equations are as given in Table IV-12. The quantities \( u \) and \( v \) are the non-dimensional vertical and radial velocities, respectively, and \( T \) is the dimensionless temperature.

<table>
<thead>
<tr>
<th>Variable Name</th>
<th>Diffusion Coefficient, ( \Gamma )</th>
<th>Source Term, ( S_c )</th>
</tr>
</thead>
<tbody>
<tr>
<td>( u )</td>
<td>( 1/Re )</td>
<td>( T/F_o^2 )</td>
</tr>
<tr>
<td>( v )</td>
<td>( 1/Re )</td>
<td>( - v/(Re r^2) )</td>
</tr>
<tr>
<td>( T' )</td>
<td>( 1/Re ) Pr</td>
<td>None</td>
</tr>
</tbody>
</table>

TABLE IV-12: Source Terms and Diffusion Coefficient in the Non-Dimensional \( r,z \) Equations
C. **Closure**

The results of the previous sections show that in qualitative terms these numerical schemes can reproduce plunging flow. However, some serious drawbacks associated with the use of a laminar model have emerged. In particular the occurrence of flow separation complicates the flow field and prevents quantitative interpretation of the events occurring.

In all the cases where separation occurred, the separation zone ended just upstream of where the plunge region started. As the plunge point moved downstream, the separated region grew accordingly, always ending just upstream of the plunging flow. This can be interpreted in two ways: either the flow is plunging as it would without separation and the separation merely occurs as the plunging moves downstream, or that the separation grows in strength and pushes the plunge region downstream. This latter situation is the most suspect in the axisymmetric situation.

Some of the results show that if buoyancy forces are exceptionally strong (run SSHAH10 for example) the separation can be completely suppressed. The actual flow field in any situation emerges as a balance between the competing buoyancy and separation forces. Even when separation does not appear explicitly, the separation forces are still inherent in the physics and can, unseen, modify the plunging flow field.

If laminar flow were of primary interest, then a matrix of computer results could easily be produced to determine precisely the influence of each separate factor, both numerical and physical. This is not done here as laminar flow is really of little interest in the study of prototype reservoir flows.

To be sure various devices, such as using different inflow profiles or anchoring a high velocity near the reservoir bed, could be used to suppress the laminar physics. Such approaches have their uses in bulk reservoir flow modeling, and in fact the model developed above can probably be profitably used with such devices to study various aspects of reservoir flow. When the actual hydrodynamics are of interest, as in this study, then such artificial devices cannot, of course, be used.

It should be stated that the problems encountered here are not a fault in the numerical scheme or in the model used. These simulate the basic physics only too well. The fault lies in the attempt to apply the model to a situation for which its basic physics are inappropriate in that flow in real reservoirs is turbulent, not laminar. The turbulent flow situation is considered in the next chapter.
V. NUMERICAL SOLUTIONS - TURBULENT FLOW

A. Introduction

Most of the flows encountered by Civil Engineers in practical situations are turbulent flows. In turbulent flow the shear stresses experienced by the flow are many times greater than in an equivalent laminar flow. Mixing and energy dissipation are accordingly increased. In mathematical terms if in the Navier-Stokes equation

\[
\frac{\partial \delta u_i}{\partial t} + \frac{\partial \delta u_i}{\partial x_j} = - \frac{1}{\rho} \frac{\partial \delta p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_i \partial x_j} \tag{V-1}
\]

the instantaneous velocity is broken down into a mean, \( \bar{u} \), and a fluctuating part \( u' \) as

\[
u_i = \bar{u}_i + u'_i \tag{V-2}
\]

then on multiplying and taking a mean, the resulting equation for the mean velocity is

\[
\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = - \frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \nu \left( \frac{\partial^2 \bar{u}_i}{\partial x_i \partial x_j} - \frac{\partial \bar{u}_i}{\partial x_j} \frac{\partial \bar{u}_j}{\partial x_i} \right) \tag{V-3}
\]

where the quantities \( \bar{u}u' \) are the new additional stresses and \( \bar{p} \) is the mean pressure. These stresses are called Reynolds stresses after O. Reynolds [1901] who first examined the Navier-Stokes equation in this way.

The basic practical problem of turbulence reduces to the problem of expressing the unknown Reynolds stresses in terms of mean flow variables. A common technique uses the eddy viscosity concept of Boussinesq [1877]. Using this concept the stresses are related to mean flow gradients, in analogy to the laminar situation, as

\[
\bar{u}u' = - \nu_t \left( \frac{\delta \bar{u}_i}{\delta x_j} + \frac{\delta \bar{u}_j}{\delta x_i} \right)
\]

where \( \nu_t \) is a new turbulent or eddy viscosity much larger than its laminar counterpart. The turbulence problem now concerns the value of \( \nu_t \).

One practice is to consider \( \nu_t \) as a constant. Provided its value is carefully selected, the constant eddy viscosity can give useful results if
the flow field is reasonably uncomplicated. Various empirical formulae have been introduced to impose a variation on $v_t$. Prandtl's mixing length model falls into this category. The relation of Munk and Anderson [1948]

$$v_t = v_{to} (1 + 20 R_i)^{-0.5} \quad (V-4)$$

is another such formula. This is used to impose a variation on the vertical eddy viscosity in the presence of stable density gradients. In the above, the gradient Richardson number $R_i$ is defined as

$$R_i = - \frac{\frac{1}{\rho} \frac{\partial \rho}{\partial z}}{g} \left( \frac{\partial v}{\partial z} \right)^2$$

where the vertical coordinate is $z$ and the horizontal component of velocity is $v$. The viscosity $v_{to}$ is the viscosity value under neutral conditions.

The above formulae can give useful results in various specialized situations. However, when phenomena such as plunging or flow recirculation occur, the simple empirical formulae are usually inadequate and more sophisticated models are required.

In recent years various new schemes have been devised to simulate the Reynolds stresses [Lauder and Spalding, 1972]. One model in particular, the $k-\epsilon$ model, has performed exceptionally well in a wide variety of problems. This model is extensively discussed by Rodi [1980] who also presents examples of its application on a number of hydraulic problems.

The $k-\epsilon$ model is briefly described in Appendix 1. In the model the eddy viscosity is calculated from the value of the turbulent kinetic energy per unit mass, $k$, and the turbulent energy dissipation rate per unit mass, $\epsilon$, at each point in the flow field. The quantities $k$ and $\epsilon$ are obtained from differential equations which incorporate the effects of convection, diffusion and shear and buoyancy effects in the flow field. Thus $v_t$ at a point is computed from local values of $k$ and $\epsilon$ which in turn depend on general mean flow effects. Hence $v_t$ can vary throughout the flow field in response to local conditions.

The $k-\epsilon$ model incorporates what would be considered the minimum number of influences required for a sufficient representation of a real flow field. It is the simplest representation of turbulence that promises reasonable simulation of complex flow situations. Accordingly the model is adopted for use here in the study of plunging reservoir flow.

The analysis is provided in separate sections for the two-dimensional sloping reservoir and the diverging reservoir. The sloping reservoir is considered first.
B. Sloping Reservoir

1. Governing equations

a. Mean flow equations

The equations of the mean motion in the cylindrical case are obtained in exactly the same way as for the cartesian situation. The details of the derivation are outlined in Appendix 2. Introduction of mean and fluctuating velocities

\[ \bar{v}_r = \bar{v}_r + v'_r \]

\[ \bar{u}_\theta = \bar{u}_\theta + u'_\theta \]

and pressure

\[ \bar{p} = \bar{p} + p' \]

results in the appearance of Reynolds stresses in the mean velocity equations. On replacement of these stresses with a model of the form

\[ - \bar{u}'_i u'_j = \nu_t \left( \frac{\delta u_i}{\delta x_j} + \frac{\delta u_j}{\delta x_i} \right) - \frac{2}{3} k \delta_{ij} \quad (V-5) \]

the following momentum equations result (letting \( u = \bar{u}_\theta \) and \( v = \bar{v}_r \))

\[ \frac{\delta u}{\delta t} + \nu \frac{\delta u}{\delta r} + \frac{1}{r} \frac{\delta u}{\delta \theta} + \frac{uv}{r} = - \frac{1}{r} \frac{\delta p^*}{\delta \theta} + \frac{1}{r^2} \frac{\delta}{\delta \theta} \left( \nu_{eff} \frac{\delta u}{\delta r} \right) + \frac{1}{r^2} \frac{\delta}{\delta \theta} \]

\[ \left( \nu_{eff} \frac{\delta u}{\delta \theta} \right) - \frac{1}{r} \frac{\delta}{\delta r} \left( \nu_{eff} u \right) + \frac{1}{r} \frac{\delta}{\delta r} \left( \nu_{eff} v \right) + \frac{1}{r^2} \frac{\delta}{\delta \theta} \left( \nu_{eff} \frac{\delta u}{\delta \theta} \right) \]

\[ + \frac{2}{r^2} \frac{\delta}{\delta \theta} \left( \nu_{eff} \nu \right) + \frac{\nu_{eff} \delta u}{r} - \frac{\nu_{eff} u}{r^2} \]

\[ + \frac{\nu_{eff} \delta v}{r^2} + g \beta \cos \theta \left( T_o - T \right) \quad (V-6) \]

and
\[
\frac{\delta v}{\delta t} + v \frac{\delta v}{\delta r} + u \frac{\delta v}{\delta \theta} - \frac{u^2}{r} = -\frac{\delta p^*}{\delta r} + \frac{1}{r} \frac{\delta}{\delta r} (\nu \frac{\delta v}{\delta r}) + \frac{1}{r^2} \frac{\delta}{\delta \theta} (\nu \frac{\delta v}{\delta \theta})
\]
\[
+ \frac{1}{r} \frac{\delta}{\delta r} (\nu \frac{\delta v}{\delta r}) + \frac{1}{r} \frac{\delta}{\delta \theta} (\nu \frac{\delta u}{\delta r}) - \frac{1}{r^2} \frac{\delta}{\delta \theta} (\nu \frac{\delta u}{\delta \theta})
\]
\[
- \frac{2 \nu \frac{\delta u}{\delta \theta}}{r^2} - \frac{2 \nu \frac{\delta v}{\delta \theta}}{r^2} + g \beta \sin \theta (T_0 - T)
\]

where

\[\nu = \nu + \nu_{\text{eff}}\]

These are the equations for the mean velocity. The overbar symbol has been dropped for convenience. The kinetic energy portion of the stress representation has been absorbed into the pressure term. The Boussinesq approximation has been used and the gravity term has been treated as outlined in section II.B. If desired, the laminar portion of these terms in the above equations which do not appear in the laminar equation can be discarded since \(\nu\) is constant. These terms then appear above multiplied by \(\nu_t\) only instead of \(\nu_{\text{eff}}\). The continuity equation for the mean velocities is

\[
\frac{\delta v}{\delta r} + \frac{v}{r} + \frac{1}{r} \frac{\delta u}{\delta \theta} = 0
\]

The energy equation is treated in the same way as the velocity equations. Decomposition of the temperature into a mean and fluctuating part as

\[T = \bar{T} + T'\]

leads to the appearance of correlations of the form \(\bar{u}_i T'\) in the equation for the mean temperature. On replacement of these correlations with expressions of the form

\[\bar{u}_i T' = -\frac{\nu_t}{\sigma_t} \frac{\delta T}{\delta x_i}\]

where \(\sigma_t\) is a turbulent Prandtl number, the following equation for mean temperature (\(T=\bar{T}\)) results
b. Turbulence model equations

The standard $k-\varepsilon$ model is used with the simplest possible extension to comprehend buoyancy effects. As outlined in Appendix 1, the eddy viscosity $v_t$ is computed from the equation

$$v_t = C_k \frac{k^2}{\mu \varepsilon} \quad (V-10)$$

where $k (=1/2(u'^2 + v'^2 + w'^2))$ is the turbulent kinetic energy and $\varepsilon$ its dissipation rate per unit mass. The quantities $k$ and $\varepsilon$ are obtained from solutions of the following pair of equations:

The $k$ equation:

$$\frac{\delta k}{\delta t} + \frac{v}{r} \frac{\delta k}{\delta r} + \frac{u}{\theta} \frac{\delta k}{\delta \theta} = \frac{1}{r} \frac{\delta}{\delta r} \left( r \nu_{eff} \frac{\delta k}{\delta r} \right)$$

$$+ \frac{1}{r^2} \frac{\delta}{\delta \theta} \left( \nu_{eff} \frac{\delta k}{\delta \theta} \right) + P + G - \varepsilon \quad (V-11)$$

where $\nu_{eff} = \nu + \frac{v_t}{\sigma_k}$ and $\sigma_k$ is an empirical coefficient. The quantities $P$ and $G$ are defined below:

The $\varepsilon$ equation

$$\frac{\delta \varepsilon}{\delta t} + \frac{v}{r} \frac{\delta \varepsilon}{\delta r} + \frac{u}{\theta} \frac{\delta \varepsilon}{\delta \theta} = \frac{1}{r} \frac{\delta}{\delta r} \left( r \nu_{eff} \frac{\delta \varepsilon}{\delta r} \right) + \frac{1}{r^2} \frac{\delta}{\delta \theta} \left( \nu_{eff} \frac{\delta \varepsilon}{\delta \theta} \right)$$

$$+ C_{1\varepsilon} \frac{\varepsilon}{k} (P + C_3 G) - C_{2\varepsilon} \frac{\varepsilon^2}{k^2} \quad (V-12)$$

In these equations $P$ denotes the production of turbulent kinetic energy from the mean flow. This function is discussed in detail in Appendix 2. It is given by
\[
\rho = \nu_t \left[ 2\left( \frac{\delta v_r}{\delta r} \right)^2 + 2\left( \frac{1}{r} \frac{\delta u_\theta}{\delta \theta} + \frac{v_r}{r} \right)^2 + \left( \frac{\delta u_\theta}{\delta r} - \frac{u_\theta}{r} + \frac{1}{r} \frac{\delta v_r}{\delta \theta} \right)^2 \right]
\]  
(V-13)

where the velocities are mean velocities. The quantity \( G \) denotes the production or destruction of turbulent energy by buoyancy forces. This function is also discussed in Appendix 2. It is given by

\[
G = \beta g \nu_t \frac{v_r}{\sigma_t} \left( \frac{1}{r} \frac{\delta T}{\delta \theta} \cos \theta + \frac{\delta T}{\delta r} \sin \theta \right)
\]  
(V-14)

The coefficients \( C_{1e}, C_{2e}, \sigma_k, \sigma_e \) appearing in these equations have been given the recommended standard values as outlined in Appendix 1. The coefficients \( C_1, C_2, \text{ and } C_3 \) are discussed later. It is the strength of the \( k-\varepsilon \) model that the coefficient values were set initially in the development of the model and remain at these values in all applications. If the coefficients had to be tuned for each specific problem, the model would be weaker and much less useful than it is.

c. Equation summary

The equations in the previous sections can be all written in the standard form.

\[
\frac{\delta \phi}{\delta t} + v_r \frac{\delta \phi}{\delta r} + \frac{u_\theta}{r} \frac{\delta \phi}{\delta \theta} = \text{pressure term}
\]

\[
+ \frac{1}{r} \frac{\delta}{\delta r} \left( \Gamma_r \frac{\delta \phi}{\delta r} \right) + \frac{1}{r^2} \frac{\delta}{\delta \theta} \left( \Gamma \frac{\delta \phi}{\delta \theta} \right) + S_c
\]

where \( \Gamma \) is a diffusion coefficient and \( S_c \) is a source term. When \( \phi \) is not a velocity, the pressure term vanishes. Table V-1 summarizes the \( \Gamma \) and \( S_c \) expressions that occur in the standard equation expressions for \( v_r, u_\theta, T, k \) and \( \varepsilon \).

2. Boundary and initial conditions

a. Boundary conditions

To make the flow field determinate, boundary conditions must be specified on the inlet plane and outlet plane, at the free-surface and at the walls. The boundary conditions used were similar to those used for the laminar flow situation. The boundary conditions are considered individually for each separate variable.
### TABLE V-1. Diffusion Coefficients and Source Terms in the Governing Equations

<table>
<thead>
<tr>
<th>Variable</th>
<th>Symbol</th>
<th>Diffusion Coefficient, ( T )</th>
<th>Source Term ( S_c )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Angular velocity</td>
<td>( u )</td>
<td>( v_{\text{eff}} (\omega + v_t) )</td>
<td>(- \frac{1}{r} \frac{\partial}{\partial r} (v_{\text{eff}} u) + \frac{1}{r} \frac{\partial}{\partial \theta} (v_{\text{eff}} \frac{\partial v}{\partial \theta}) + \frac{1}{r^2} \frac{\partial}{\partial \theta} (v_{\text{eff}} \frac{\partial u}{\partial \theta}) )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( + \frac{2}{r} \frac{\partial}{\partial \theta} (v_{\text{eff}} v) ) ( + \frac{v_{\text{eff}}}{r} \frac{\partial u}{\partial r} - \frac{v_{\text{eff}}}{r^2} \frac{\partial v}{\partial \theta} ) ( - uv )</td>
<td>( + g \delta \cos \theta (T_o - T) )</td>
</tr>
<tr>
<td>Radial velocity</td>
<td>( v )</td>
<td>( v_{\text{eff}} (\omega + v_t) )</td>
<td>(- \frac{1}{r} \frac{\partial}{\partial r} (v_{\text{eff}} v) + \frac{1}{r} \frac{\partial}{\partial \theta} (v_{\text{eff}} \frac{\partial v}{\partial \theta}) - \frac{1}{r^2} \frac{\partial}{\partial \theta} (v_{\text{eff}} u) )</td>
</tr>
<tr>
<td>Temperature</td>
<td>( T )</td>
<td>( \frac{v}{T} + \frac{v_h}{\sigma_T} )</td>
<td>NONE</td>
</tr>
<tr>
<td>Turbulent kinetic energy</td>
<td>( k )</td>
<td>( v + \frac{v_k}{\sigma_k} )</td>
<td>( P + G - \epsilon )</td>
</tr>
<tr>
<td>Turbulent energy dissipation rate</td>
<td>( \epsilon )</td>
<td>( v + \frac{v_c}{\sigma_c} )</td>
<td>( c_1 \frac{\epsilon}{k} (p + c_3 \epsilon) - c_2 \frac{\epsilon^2}{k} )</td>
</tr>
</tbody>
</table>
Velocity

The velocity was given a symmetry condition at the free surface. At the reservoir bottom and on the dam face, the no-slip condition was effectively imposed. This was accomplished using the wall functions of Patankar and Spalding [1972]. These use known behavior near a wall to compute the velocity at the near wall grid point. This eliminates the requirement of having to laboriously compute out from the wall through the steep gradients of the viscous sublayer.

The simple boundary layer profile shown in Fig. V-1 is used to compute the near wall velocity. Using the boundary layer profile, the tangential velocity component $V$ (taken as $v_r$ for the reservoir bottom and $u_0$ for the dam face) is set using

$$V = \frac{u_*}{K} \ln \left( \frac{u_* y_p}{v} E_b \right)$$

where the Van Karman constant was taken as $K=0.4$, and $E_b=9.0$. The shear velocity, $u_*$, is as defined below and $y_p$ is as shown in Fig. V-1. It was attempted to keep the near wall grid point in the log law range but in the eventuality that $y^+$, where

$$y^+ = \frac{y_p u_*}{v}$$

fell below 11.5, the velocity was set to its viscous subrange value of

$$V = \frac{u_*^2 y_p}{v}$$

In the constant stress layer, the stress at the point P (Fig. II-1) is approximately equal to that at the wall so the definition of the shear velocity, $u_*$, was generalized [Patankar and Spalding, 1972] to

$$u_* = C_{u' v}^{1/4} k^{1/2}$$  \hspace{1cm} (V-15)

Since the tangential velocity component is close to zero near a wall, then continuity considerations indicate that

$$\frac{\delta u}{\delta y} = 0$$

where $u$ is the normal velocity component and $y$ the direction normal to the wall. This was used as a boundary condition on the normal velocity.
BOUNDARY LAYER PROFILES ASSUMED TO OCCUR IN THIS REGION

NEAR-WALL GRID POINT (P)

V

Y_P

WALL

Fig. V-1. Definition sketch for velocity at near wall grid point.
component (taken as $u_0$ in the case of the reservoir bottom and $v_r$ in the case of the dam face).

At the inflow boundary the radial velocity component, $v_r$, was given a one-seventh power law velocity profile. The angular component, $u_0$, was set to zero.

Outflow occurs through a single control volume close to the dam base. The radial velocity component here was allocated a value to exactly balance inflow. A zero gradient condition was imposed on $v_r$ and $u_0$ at the outflow point.

**Temperature**

As in the laminar case, the reservoir bottom and surface were taken as adiabatic. The dam face was given a temperature equal to the initial temperature of the warm reservoir water. This latter is an arbitrary decision and is not of importance. The inflow temperature was set at a constant value with no variation over river depth. A zero gradient condition was imposed at the outflow point.

**Turbulent kinetic energy**

A zero gradient condition for $k$ was imposed at the free surface, at the reservoir bottom, and on the dam face. The turbulent kinetic energy in the inflow river was given a linear profile according to the equation

$$k = u_*^2 \left(0.3 + 3 \frac{y}{H}\right)$$

where $y$ is the distance from the water surface and $H$ is the river depth. This equation is based on the data of Laufer [1951]. The river shear velocity, $u_*$, was computed from the expression

$$u_* = \frac{V \sqrt{g} \; n}{H^{1/6}}$$  \hspace{1cm} (V-16)

where $V$ is the mean river velocity. This is based on Manning's equation. The friction parameter $n$ was given the value $n = .03$.

At the outflow point, the turbulent kinetic energy was given a zero gradient condition.

**Kinetic energy dissipation rate**

The kinetic energy dissipation rate was given a symmetry condition at the free surface. At the reservoir bottom and on the dam face the dissipation at the near wall grid point was computed from the condition that the length scale is a linear function of distance from the wall. Thus $\epsilon$ was computed using
where \( y_p \) and \( K \) are as defined above and \( u_* \) is as given by Eq. V-15.

In the inflowing river the energy dissipation profile is computed from the condition that the eddy viscosity resulting from the equation

\[
v_t = C \frac{k^2}{\mu \varepsilon}
\]

will have the generally accepted parabolic form

\[
v_t = u_* KH \left[ \frac{y}{H} \frac{y}{H} \left( 1 - \frac{y}{H} \right) \right]
\]

where \( K \) is von Karman's constant (\( = 0.4 \)), \( H \) is the river depth, \( y \) the distance from the river bottom and \( u_* \) is computed from Eq. (V-16). Thus the \( \varepsilon \) profile is in fact

\[
\varepsilon = \frac{C}{\mu} \frac{k^2}{u_* KH \left[ \frac{y}{H} \frac{y}{H} \left( 1 - \frac{y}{H} \right) \right]}
\]

The zero gradient condition was imposed on \( \varepsilon \) at the outflow point.

The boundary conditions on the variables \( v_r, u_\theta, T, k \) and \( \varepsilon \) are summarized in Table V-2. The profile in the inflowing river of \( v_r, u_\theta, T, k \) and \( \varepsilon \) and the \( v_t \) profile resulting from the \( k \) and \( \varepsilon \) profiles are shown in Fig. V-2.

b. Initial Conditions

The initial velocity field consisted of a forward radial velocity, \( v_r \), and a zero angular velocity, \( u_\theta \), at all points except close to the dam. Radial velocity was computed using the expression

\[
v_r = \frac{r_o}{r} v_{ro}
\]

(V-17)

where \( r \) is the radial location, \( r_o \) the inlet radius, \( v_r \) the velocity at radius \( r \) and \( v_{ro} \) the inflow velocity at the equivalent angular position. This formula gives a radially similar velocity field throughout the reservoir. Close to the dam \( u_\theta \) velocities were introduced to give a velocity field converging on the outflow point. A typical initial velocity field is shown in Fig. V-3.
### TABLE V-2. Summary of Boundary Conditions for Sloping Reservoir Simulations

<table>
<thead>
<tr>
<th>Variable Name</th>
<th>Symbol</th>
<th>Inlet Plane</th>
<th>Reservoir Bottom</th>
<th>Free Surface</th>
<th>Dam Face</th>
<th>Outflow Point</th>
</tr>
</thead>
<tbody>
<tr>
<td>Angular Velocity</td>
<td>u</td>
<td>0</td>
<td>( \frac{\delta u}{\delta \theta} = 0 )</td>
<td>( u = 0 )</td>
<td>( u = \frac{U_<em>}{K} \ln \left( \frac{U_</em> Y_p}{v E_b} \right) )</td>
<td>( \frac{\delta u}{\delta r} = 0 )</td>
</tr>
<tr>
<td>Radial Velocity</td>
<td>v ( \nu = V_o \left( \frac{Y}{H} \right)^{1/7} )</td>
<td>( \nu = \frac{U_<em>}{K} \ln \left( \frac{U_</em> Y_p}{v E_b} \right) )</td>
<td>( \frac{\delta v}{\delta \theta} = 0 )</td>
<td>( \frac{\delta v}{\delta r} = 0 )</td>
<td>( \frac{\delta v}{\delta r} = 0 )</td>
<td></td>
</tr>
<tr>
<td>Temperature</td>
<td>T ( T = T_{IN} ) (constant)</td>
<td>( \frac{\delta T}{\delta \theta} = 0 )</td>
<td>( \frac{\delta T}{\delta \theta} = 0 )</td>
<td>( T = T_0 )</td>
<td>( \frac{\delta T}{\delta r} = 0 )</td>
<td></td>
</tr>
<tr>
<td>Turbulent Kinetic</td>
<td>k ( k = U_*^2 (0.3 + 3 y/H) )</td>
<td>( \frac{\delta k}{\delta \theta} = 0 )</td>
<td>( \frac{\delta k}{\delta \theta} = 0 )</td>
<td>( \frac{\delta k}{\delta r} = 0 )</td>
<td>( \frac{\delta k}{\delta r} = 0 )</td>
<td></td>
</tr>
<tr>
<td>Turbulent Energy</td>
<td>( \varepsilon )</td>
<td>( \varepsilon = C_u \frac{k^2}{U_* KH \left( \frac{Y}{H} \left( 1 - \frac{Y}{H} \right) \right)} )</td>
<td>( \varepsilon = \frac{U_*^3}{K Y_p} )</td>
<td>( \frac{\delta \varepsilon}{\delta \theta} = 0 )</td>
<td>( \varepsilon = \frac{U_*^3}{K Y_p} )</td>
<td>( \frac{\delta \varepsilon}{\delta r} = 0 )</td>
</tr>
<tr>
<td>Dissipation Rate</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Fig. V-2. Schematic profile of various quantities of the flow inlet plane.
Fig. V-3. Typical initial velocity field.
The initial temperature field consisted of a constant temperature ($T_0$) throughout the reservoir.

The initial $k$ and $\varepsilon$ fields were computed using similar profiles in the reservoir as in the inflow river. That is, the values at all radial locations were computed from an equation of the same form as Eq. (V-17).

c. **Effect of initial and inflow conditions**

Information on initial conditions is scrambled quickly in this unsteady problem. The plunging flow field that develops is a function of momentum and buoyancy forces and is independent of initial conditions. Initial conditions would only have an effect if they were so unreasonable as to cause divergence of the solution algorithms.

The effect of the inflow values of kinetic energy and dissipation rate was examined by rerunning a simulation with upstream and initial values of $k = k_0/2$ and $\varepsilon = 2\varepsilon_0$, where $k_0, \varepsilon_0$ refer to the values in the standard simulation. The combination of the new $k$ and $\varepsilon$ values give an inflow eddy viscosity one-eighth of the original.

In the new simulation (run TRB 151) energy production increased so that $V_t$ values developed which were of a similar order of magnitude to the original run. The flow field was similar to the original but the plunge point appeared one grid point downstream of the original prediction and stayed one grid point downstream as the flow developed.

Thus, while the effect of different inflow parameter values can be mitigated by altered flow dynamics, there is a residual effect on the predicted flow fields. The inflow values and profiles used here, as described above, were selected as representative of real flow conditions.

3. **Model reservoir**

The results from the laminar flow studies indicate that the phenomenon under study and in particular the plunge point location is sensitive to grid size. Hence, for computational reasons the reservoir in which the flow is to be studied should be as small as possible. However, there are a number of physical considerations which together provide a constraint on the minimum size of reservoir that can be used.

As a first consideration the Reynolds number must be high enough for turbulent energy production to be significant. Otherwise the eddy viscosity will be negligible compared to the laminar viscosity, and the advantage of a variable eddy viscosity model will not be realized. High Reynolds number effectively means a high flow velocity at the river entrance.

Dimensional considerations show that the densimetric Froude number is of importance in this phenomenon. If $F_{RD}$ is the densimetric Froude number at any location in the reservoir, where the depth is $H$, then geometrical considerations show that
where \( F_0 \) is the inflow densimetric Froude number and \( d \) the depth at the river inflow point.

Existing data show that the plunge region is located typically at densimetric Froude numbers of about 0.5. Hence with an inflow densimetric Froude number of about 2.0, the depth at the plunge region is about 2.5 times the inflow depth. Locating the plunge region at the mid-point of the reservoir, half-way between the inflow river and the dam, gives a depth at the dam of about 5\( d \).

Locating the plunge region around the mid-point of the reservoir is intended to try to minimize entrance and end effects. If the plunge region is too close to the river inflow, the entrance conditions may become important and this is undesirable. On the other hand in real situations the dam is always far downstream and is unlikely to influence the plunge region. These real situations would require a very long reservoir for simulation and this is undesirable computationally. A reservoir with a plunge region in its middle region was selected as a compromise on these competing demands.

The length of reservoir required to give a depth of 5\( d \) at the dam depends on the bottom slope. This slope must be realistic. If the slope is too steep, the separation phenomena will become involved, and this is undesirable as such phenomena are unlikely to make a contribution in prototype situations. A slope of 0.02 was selected for reservoir sizing. This slope is steep but does occur in practical situations. Simulations were carried out with slopes greater and less than this value.

An entrance depth of 0.5 m gives a depth at the dam of 2.5 m. With a bottom slope of 0.02 this requires a reservoir 100 m long. A length of 96 m was in fact used. Using a typical entrance velocity of 0.2 m/sec gives a Reynolds number \( Re \) and an ordinary Froude number \( F \) at the entrance of

\[
Re = 10^5 \\
F = 0.1
\]

These are typical prototype values and are acceptable.

The values of the densimetric Froude numbers depend on the selected values of reservoir and inflow temperatures. The final model reservoir as selected is shown in Fig. V-4.

4. **Computational parameters**

   a. **NTIMES and LAST**

   The effect of the values of the numerical scheme parameters NTIMES and LAST (Section III.B) was examined by running simulations with various values of these parameters.
Fig. V-4. Model reservoir for turbulent flow simulation.
The parameter NTIMES was given the values 4 and 1. There was no discernible differences in the two predicted flow fields. This suggests, and this is the general pattern, that the bulk of the information in the linear equations is extracted in the first sweep through of the iteration method. Hence, NTIMES = 1 was used for all calculations.

Selection of a value for LAST was more difficult. This is the number of iterations for each time step. The criterion for selection for LAST was that the maximum mass imbalance (i.e. the residual in the continuity equation) in any control volume should be less than, or equal to, order $10^{-6}$ m$^3$/sec. In the program arrangement NTIMES was set at the beginning of a run and remained constant. It was generally found that any convergence problems that did occur invariably occured at the initial development stage of the flow field. Once this stage was past, the maximum residual error continued to decrease with time. Values of LAST of 20, 15, and 12 and 10 were looked at. Eventually a value of LAST=12 was selected as offering sufficiently good convergence. The LAST values of 20 and 15 gave better converged flow fields initially but this advantage tapered off as time elapsed.

b. Grid size

As discussed above, the reservoir size is fixed by physical considerations. The problem now concerns the selection of a grid to simulate the reservoir flow adequately.

Grid size considerations are of prime importance in the simulation of selected flows, but not for the usual reasons. It is unlikely that fine enough grids can be used to give completely grid independent solutions. What is required is that the selected flow model behaves reasonably even on a coarse grid so that the bulk flow details are adequately modelled.

The laminar simulations of plunging flow showed that the predicted plunge point is extremely sensitive to the grid size. Accordingly, in the study of grid size effect with the k-ε model most attention is devoted to an examination of the plunge point position. The behavior of surface velocities at a point downstream of the plunge point is also examined.

In all grid patterns used, the grid size was progressively reduced approaching the reservoir bottom, the dam face and the free surface. The reduction pattern was the same at all three locations and is shown in Fig. V-5. The increment $\Delta x$ shown refers to the general interior mesh size in that coordinate direction.

Simulations were run with a series of grid arrangements. The simulation details are given in Table V-3. The plunge point position as defined in Fig. IV-8, or a series of positions if the plunge point moved, was extracted from each run. With the finer grids the plunge point was found to move forward for a period after it had formed. One of the runs, number TRB 138 was allowed to run on to a large elapsed time. In this run it was found that the initial movement eventually ceased and the plunge point then remained in a fixed position.

The plunge point movement with time is summarized for all the grid arrangement in Fig. V-6. It can be seen that with the coarser grids the
Fig. V-5. Grid size reduction pattern.
### TABLE V-3. Runs to Test Effect of Grid Size

<table>
<thead>
<tr>
<th>Run Number</th>
<th>Grid Configuration (length x depth)</th>
</tr>
</thead>
<tbody>
<tr>
<td>TRB 120</td>
<td>22 x 14</td>
</tr>
<tr>
<td>TRB 127</td>
<td>42 x 14</td>
</tr>
<tr>
<td>TRB 128</td>
<td>22 x 22</td>
</tr>
<tr>
<td>TRB 132</td>
<td>62 x 14</td>
</tr>
<tr>
<td>TRB 135</td>
<td>28 x 14</td>
</tr>
<tr>
<td>TRB 138</td>
<td>62 x 22</td>
</tr>
<tr>
<td>TRB 301</td>
<td>98 x 14</td>
</tr>
<tr>
<td>TRB 121</td>
<td>33 x 14*</td>
</tr>
</tbody>
</table>

*Refined Locally

### TABLE V-4. Runs to Test Effect of Time Step

<table>
<thead>
<tr>
<th>Run Number</th>
<th>Time Step (Seconds)</th>
</tr>
</thead>
<tbody>
<tr>
<td>TRB 139</td>
<td>5</td>
</tr>
<tr>
<td>TRB 140</td>
<td>10</td>
</tr>
<tr>
<td>TRB 141</td>
<td>2.5</td>
</tr>
</tbody>
</table>
Fig. V-6. Plunge point movement with various grid patterns.
plunge point once formed remained fixed at its initial location. The finer grids show a forward drift of the plunge point.

The horizontal grid appears, not surprisingly, to have the dominant influence. Runs TRB 128 (22x22) and TRB 120 (22x14) show identical results though one has 50 percent more vertical grid than the other. Runs TRB 138 (62x22) and TRB 132 (62x14) while not identical show a very similar plunge point movement pattern.

In an attempt to pick up the initial plunge point motion, a coarse grid (22x14) was refined locally at the expected plunge point location (as indicated by the coarse grid predictions). The grid pattern at the expected plunge region is shown in Fig. V-7. This gave a total grid pattern of 33x14. The results of this run, number TRB 121, are shown on Fig. V-6. Plunging occurred in the interior of the fine grid area but no local movement occurred once the plunge point had formed. In fact the predicted plunge point position is very similar to that given by run TRB 135 which has almost the same number of grid points (28x14) but distributed evenly. This suggests that the plunge point motion is not determined by local conditions but is a function of the overall flow pattern.

Figure V-6 shows that the major differences in the predicted behavior with the different grids is largely confined to the developing stages of the plunging flow. The initial plunge point movement feature is lost with the coarse grids. However, as time elapses, the difference in the predicted plunge point position reduces to about 5 percent over all the grids.

The velocity behavior with time at a point 72 m into the reservoir is shown in Fig. V-8 for each of the grid configurations. It can be seen that all grids produce a similar basic pattern. The finer grids show a sharper changeover from forward to recirculating flow.

A 62x22 grid was adopted as offering sufficiently good definition of the flow. However, the above results show that even with a coarse grid the reservoir hydrodynamics are well simulated. This is a good finding as it indicates the robustness of the numerical schemes and model used. Procedures that require an extremely fine grid before they start to perform well are of little use for the simulation of flow in real waterbodies.

c. **Time step**

The effect of varying the time step was examined in the same manner as the grid size. That is, attention was concentrated on the behavior of the plunge point and the velocity at a point on the water surface downstream of the plunge point.

Time steps of 10, 5, and 2.5 seconds were examined. The run details are given in Table V-4. A 62x22 grid layout was used with all time steps.

Figure V-9 summarizes the plunge point behavior with time for each of the three time steps. It can be seen that the two smaller steps show a similar pattern. The 10 sec time step shows an unusual initial backward movement of the plunge point.
PLUNGE EXPECTED TO OCCUR IN THIS AREA

Fig. V-7. Locally defined grid pattern.
Fig. V-8. Surface velocity behavior downstream of the plunge point with various grid patterns.
Fig. V-9. Plunge point behavior with different time steps.
The behavior of the velocity at surface grid point 44 (72 m into reservoir) is shown in Fig. V-10. The 10 sec time step gives a maximum backflow velocity about 22 percent less than that predicted by the 2.5 sec time step. The 5 sec time step gives a maximum backflow velocity less than 6% different from that of the 2.5 sec time step.

All three time steps gave similar representations of the general flow patterns. However, only the 5 sec and 2.5 sec steps gave good definition of the initial plunge point motion and of the surface backflow. The 5 sec time step was adopted for use.

5. Examination of turbulence model coefficients

The standard k-ε model contains five empirical coefficients. These coefficients have been assigned values by calibrating the model with reference to simple flow conditions. In applications of the k-ε model to situations where buoyancy, or other body forces such as centrifugal force, do not occur, these coefficients may be regarded as constants.

When buoyancy forces are present the coefficient \( C_\mu \) and the turbulent Prandtl number \( \sigma_t \) can no longer be regarded as constants but are functions of flow field parameters. Furthermore, a new coefficient \( C_3 \) makes its appearance.

These three coefficients \( C_\mu, C_3 \) and \( \sigma_t \), their origins and influences are examined separately in the following sections. A philosophy underlying this examination is that the various values will not be altered from their standard values unless there is a very strong reason for doing so.

a. Coefficient \( C_\mu \)

The value of the \( C_\mu \) coefficient is found in the standard k-ε model by noting that the model must hold in the near wall equilibrium layer. In that region by definition

\[ P = \epsilon \]

and it can be readily shown that

\[ C_\mu = \left( \frac{u'v'}{k} \right)^2 \]

where \( u' \) and \( v' \) are the lateral and longitudinal fluctuating velocities. Measurements show that in the equilibrium layer

\[ \left( \frac{u'v'}{k} \right) \approx 0.3 \]

Hence,
Fig. V-10. Surface velocity behavior downstream of the plunge point with different time steps.
This value is used throughout the flow field in the standard k-ε model.

The differential equation for the Reynolds stresses can be simplified by modelling its complex terms to yield algebraic expressions for the stresses [Rodi, 1980]. If there are buoyancy terms in the governing equations then buoyancy terms appear in the expressions for the stresses. On rewriting the stress expression in an eddy viscosity form the C coefficient will appear as a function of flow field parameters including buoyancy effects [Hossain and Rodi, 1980].

Hence, Cμ is not a true constant in a buoyancy affected flow field, and it may be necessary to include a functional form for Cμ if buoyancy affected flow fields are to be correctly modelled. Before attempting to set up this function a sensitivity analysis was carried out to examine the influence of Cμ in an actual simulation of a plunging flow field. In this case this consisted of running two simulations: one with the standard Cμ value of 0.09, the other with a Cμ value of 0.20.

On doing this it was found that the velocity fields in each case were almost identical. The behavior of the surface velocity at grid point 14 (72 m into reservoir) is shown in Fig. V-11 for the two Cμ values. This was taken as indicating that Cμ does not have a strong influence in this problem. Accordingly, it was not attempted to incorporate a functional dependence for Cμ. Instead Cμ was used at its standard value of 0.09.

b. Turbulent Prandtl number σt

The turbulent Prandtl number σt is the ratio of the eddy viscosity to the eddy diffusivity of heat in a given flow. Most calculations give a Prandtl number close to unity. Data for wall flows give an average value for σt of about 0.9.

It is well known from experimental and prototype data that a waterbody's ability to diffuse heat by turbulent motion is more affected than its ability to diffuse momentum under conditions of a stable density gradient. This leads to an increase in the value of the turbulent Prandtl number under such conditions. The well known relationships of Munk and Anderson [1948] for the variation of eddy viscosity and diffusivity under stable stratification incorporate this variation, in that different formulae are given for the two quantities.

A theoretical expression for the variation of σt in the presence of buoyancy forces may be derived from the differential equation governing the behavior of the temperature-velocity fluctuation correlation. Gross simplification of this equation leads to an algebraic expression for the correlation. Then on writing this correlation in an eddy diffusivity form the quantity σt may be extracted. In the resulting expression, the turbulent Prandtl number is seen to depend on a buoyancy parameter incorporating the local temperature gradient. When the gradient dissappears, σt reverts to its value under homogeneous conditions. Gibson and Launder [1976] and Hossain and Rodi [1980] outline this procedure.
Fig. V-11. Surface velocity behavior downstream of the plunge point for two $C_\mu$ values.
A sensitivity analysis was carried out to examine the effect of $\sigma_t$ variation on the predicted flow field. The run details are given in Table V-5. Two $\sigma_t$ values were used; the standard value of 0.9, and a value of 3.0.

Eddy viscosity values in the simulation with the higher $\sigma_t$ value were generally greater than the values in the standard run. Also the plunge point appeared further downstream with the higher $\sigma_t$ value and migrated towards the dam more quickly than in the standard run.

What is actually happening can be understood by examining temperature profiles in the reservoir. Vertical temperature profiles in the reservoir at r-grid point 40 and at elapsed time of 840 secs are shown in Fig. V-12 for the two $\sigma_t$ values. It can be seen that there is some variation in the profiles but not as much as would be expected. The high $\sigma_t$ value should yield a shorter, steeper gradient in temperature. However, numerical diffusion is smearing the interface and preventing the formation of a sharp gradient with the high $\sigma_t$ value. Hence, the $\sigma_t$ values in the buoyancy term in the $\kappa$ equation are operating on similar temperature gradients. For the high $\sigma_t$ value this term is thus artificially low so that proper draining of the turbulent energy is not occurring. This mechanism appears to be the dominant influence of $\sigma_t$ variation and is of course a completely incorrect mechanism.

An ability to vary the value of $\sigma_t$ is undoubtedly important for the very accurate modelling of buoyancy affected flows. In the present case, numerical diffusion is inhibiting the formation of steep gradients and is thus preventing the precise influence of $\sigma_t$ on the flow. This drawback nullifies the advantage of having a variable $\sigma_t$. For this reason it was decided not to incorporate a functional form for $\sigma_t$. Instead a constant value for the turbulent Prandtl number of the standard wall flow value of 0.9 was adopted for use.

c. Coefficient $C_3$

The coefficient $C_3$ appearing in the $\epsilon$ equation is not part of the standard $\kappa-\epsilon$ model discussed by Launder and Spalding [1974]. The coefficient arises in the modelling of the buoyancy term in the $\epsilon$ equation. Before any assumptions are made this term is

$$\frac{\delta T' \delta u'_i}{\delta x_j \delta x_j} - \beta g \frac{\delta}{\delta x_j}$$

where $T'$ denotes the temperature fluctuation and $u'_i$ the velocity fluctuation. This term is modelled by relating it simply to the buoyancy term in the $\kappa$ equation as

$$\frac{\delta T' \delta u'_i}{\delta x_j \delta x_j} = C_1 \epsilon \frac{\epsilon}{k} C_3 u'_i T'$$

134
TABLE V-5. Runs to Test Effect of Turbulent Prandtl Number, $\sigma_t$

<table>
<thead>
<tr>
<th>Run Number</th>
<th>Value of Turbulent Prandtl Number, $\sigma_t$</th>
</tr>
</thead>
<tbody>
<tr>
<td>TRB 138</td>
<td>0.9</td>
</tr>
<tr>
<td>TRB 151*</td>
<td>3.0</td>
</tr>
</tbody>
</table>

*Refined grid at plunge point.

TABLE V-6. Runs to Test Effect of Coefficient $C_3$

<table>
<thead>
<tr>
<th>Run Number</th>
<th>Value of Coefficient $C_3$</th>
</tr>
</thead>
<tbody>
<tr>
<td>TRB 113</td>
<td>0.2</td>
</tr>
<tr>
<td>TRB 114</td>
<td>0.0</td>
</tr>
<tr>
<td>TRB 115</td>
<td>0.6</td>
</tr>
<tr>
<td>TRB 121*</td>
<td>0.0</td>
</tr>
<tr>
<td>TRB 122*</td>
<td>1.0</td>
</tr>
<tr>
<td>TRB 123*</td>
<td>0.6</td>
</tr>
</tbody>
</table>

*Refined grid at plunge point.
Fig. V-12. Vertical temperature profiles with two $\sigma_t$ values.
and since

\[ \frac{u^i}{T} = -\frac{\nu_t}{\sigma_t} \frac{\delta T}{\delta x_i} \]

the buoyancy term becomes

\[ \beta g \frac{C_{ke}}{k} C_3 \frac{\nu_t}{\sigma_t} \frac{\delta T}{\delta x_i} = C_{ke} \frac{C_3}{k} G \]

where G is the buoyancy production in the k equation. Hence the total production term in the \( \varepsilon \) equation becomes

\[ \text{Total} = C_{ke} \frac{\varepsilon}{k} P + C_{ke} \frac{\varepsilon}{k} C_3 G \]

\[ = C_{ke} \frac{\varepsilon}{k} (P + C_3 G) \quad (V-18) \]

\[ = C_{ke} \frac{\varepsilon}{k} P (1 - C_3 R_f) \]

where \( R_f \) is a flux Richardson number defined as minus the ratio of buoyancy production of turbulent kinetic energy to shear production.

Unfortunately, there is no generally accepted value for the coefficient \( C_3 \). Rodi [1980] references various researchers who have found that \( C_3 \) should be close to zero in horizontal shear layers i.e. no buoyancy effects in the \( \varepsilon \) equation, and close to unity for vertical shear layers. Rodi [1980] also discusses an extension of the term which allows some of the variability to be absorbed in a new definition of \( R_f \) and thus enables \( C_3 \) to be constant. The simple model as given by Equation (V-18) is used here. The question now arises as to what numerical value should be given to \( C_3 \). Since reservoir flow is basically a horizontal flow with gravity acting in a normal direction to the mean flow direction, a \( C_3 \) value close to zero would appear to be most appropriate. A sensitivity analysis was carried out to determine the influence of \( C_3 \) in an actual calculation of a plunging flow field. The run details are given in Table V-6.

The first series of simulations were carried out on a coarse grid (22x14). Three \( C_3 \) values were examined: 0.0, 0.2, 0.6. It was found that the plunge point formed at the same location (Grid point 8), and remained fixed there for the duration of the simulation with all three \( C_3 \) values. The behavior of the surface velocity at grid point 14 (72.0 m into the reservoir) is shown in Fig. V-13 for the three \( C_3 \) values. It can be observed that there is little difference between the three predicted patterns.
Fig. V-13. Surface velocity behavior downstream of the plunge point for three $C_3$ values.
Further simulations were run with a refined grid (of the pattern shown in Fig. V-7) in place at the expected plunge point position. Values of $C_3$ of 0.0 and 0.6 were used. The predicted plunge point movements are shown in Fig. V-14. Again it can be seen that the predictions are similar. The behavior of the velocity of the water surface downstream of the plunge point were also very similar for the two $C_3$ values.

As a matter of interest, one simulation was run using a $C_3$ value of 1.0 with the refined grid. This change in $C_3$ produces a marked change in the flow field. The predicted plunge point behavior is shown plotted in Fig. V-15, where it can be compared with that predicted using the $C_3$ values of 0.0 and 0.6.

Thus, it appears that the flow field is unresponsive to changes in $C_3$ until $C_3$ becomes close to unity. This is fortunate because as pointed out above, a $C_3$ value close to zero appears most appropriate for this flow. A $C_3$ value of zero was adopted for use. Although this $C_3$ value appears an almost natural choice for this flow, the fact that a single $C_3$ value cannot be fixed for all problems is a serious defect in the $k$-$\varepsilon$ model in buoyant situations.

d. **Summary**

The coefficient values used in the turbulence model are summarized for convenience in Table V-7:

<table>
<thead>
<tr>
<th>Coefficient</th>
<th>$C_\mu$</th>
<th>$C_{1\varepsilon}$</th>
<th>$C_{2\varepsilon}$</th>
<th>$\sigma_k$</th>
<th>$\sigma_\varepsilon$</th>
<th>$\sigma_t$</th>
<th>$C_3$</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Value</strong></td>
<td>0.09</td>
<td>1.44</td>
<td>1.92</td>
<td>1.0</td>
<td>1.3</td>
<td>0.9</td>
<td>0.0</td>
</tr>
</tbody>
</table>

**6. Results**

a. **Introduction**

The model described in the previous sections was applied to negatively buoyant flow in the standard sloping reservoir for a range of conditions. The run details are given in Table V-8.

All of these flows were simulated using an inflow velocity of about 0.2 m/sec and an inlet depth of 0.5 m. This gives a reservoir Reynolds number of $0.875 \times 10^5$ for all flows. One run (number TRB 161) was carried out with an inflow velocity of 0.4 m/sec to give a Reynolds number twice as large as that above. The results were essentially unchanged from the standard run with the same densimetric Froude number.

In the following section various features of the simulated plunging flow fields are described and assessed. Plunge depths are then extracted and examined and the flow fields are analyzed as two layer flows.
Fig. V-14. Plunge point moment with two $C_3$ values.
Fig. V-15. Change in plunge point position with a $C_3$ value of unity.
TABLE V-8. Details of Reservoir Flow Simulation Runs

<table>
<thead>
<tr>
<th>Run Number</th>
<th>Bottom Slope S</th>
<th>Temperature Difference $T_o - T_{in}$ (°C)</th>
<th>Inflow Densimetric Froude Number $F_o$</th>
</tr>
</thead>
<tbody>
<tr>
<td>TRB 138</td>
<td>.02</td>
<td>10</td>
<td>1.77</td>
</tr>
<tr>
<td>TRB 139</td>
<td>.02</td>
<td>20</td>
<td>1.25</td>
</tr>
<tr>
<td>TRB 142</td>
<td>.02</td>
<td>5</td>
<td>2.5</td>
</tr>
<tr>
<td>TRB 143</td>
<td>.02</td>
<td>15</td>
<td>1.44</td>
</tr>
<tr>
<td>TRB 145C</td>
<td>.01</td>
<td>10</td>
<td>1.77</td>
</tr>
<tr>
<td>TRB 146</td>
<td>.01</td>
<td>15</td>
<td>1.44</td>
</tr>
<tr>
<td>TRB 147</td>
<td>.01</td>
<td>20</td>
<td>1.25</td>
</tr>
<tr>
<td>TRB 156</td>
<td>.005</td>
<td>25</td>
<td>1.12</td>
</tr>
<tr>
<td>TRB 157</td>
<td>.03</td>
<td>10</td>
<td>1.77</td>
</tr>
<tr>
<td>TRB 158</td>
<td>.0075</td>
<td>25</td>
<td>1.12</td>
</tr>
<tr>
<td>TRB 159</td>
<td>.015</td>
<td>15</td>
<td>1.44</td>
</tr>
<tr>
<td>TRB 160</td>
<td>.006</td>
<td>25</td>
<td>1.12</td>
</tr>
</tbody>
</table>
b. **Flow field features**

The flow developed in a similar manner in all the runs. Initially velocities were forward at all points. An eddy viscosity of the order of $10^{-4} \text{ m}^2/\text{sec}$ developed throughout the reservoir. As the cold front advanced into the reservoir, the denser cold water pushed slightly under the warm water. The resulting density gradients gave rise to a local lowering of $\nu_t$ values through the action of the energy draining term $G$ in the $k$ equation. Plunging flow developed in this area of low viscosity. A vertical profile of $\nu_t$ is shown in Fig. V-16 for a location downstream of a developed plunge line. As the flow develops further, the $G$ term in the $k$ equation grows larger and $\nu_t$ effectively disappears over much of the recirculating flow zone.

The lowering of the eddy viscosity appears to be an essential part of the plunging flow physics. One run (TRB153) was made with $G$ set to zero but buoyancy effects retained in the mean flow equations. The flow field was similar in the early stages to the standard runs. However, the eddy viscosity remained high at all points and plunging flow never developed. A mechanism for explaining this behavior is that the tendency to plunge and recirculate creates high velocity gradients which when associated with the high eddy viscosity may create high local stresses which the buoyancy forces cannot overcome.

Velocity fields illustrating the various stages of flow field development for run TRB138 are shown in Figs. V-17(a), (b), (c), (d), and (e). These show a velocity field before plunging, the situation just at plunging with the appearance of a small region of backflow and subsequent development as the recirculating region grows. The plunge point on the water surface where downstream and upstream velocities meet can be seen to be well defined in these velocity fields.

Figure V-17(e) shows the flow pattern shortly after the recirculating region has reached the dam. A small eddy recirculating in the opposite direction to the general flow pattern can be seen at the water surface near the dam. This feature appeared in all the runs. As time elapsed, this eddy grew by extending back upstream at the water surface. This phenomenon is undoubtedly associated with the presence of the dam. It may not occur in large reservoirs where plunging frequently occurs in a side arm so that the downstream boundary condition is different from above. Also, flow conditions may not persist long enough in a straight through large reservoir for the recirculating zone to reach the dam.

Figures V-18(a), (b) and (c) show contours of the temperature field for run TRB138 at the elapsed times indicated. The arrows on these figures are velocity vectors and show the limits of the recirculating region. The flow is still developing in Fig. V-18(a). The position at the end of the recirculating region on the water surface can be seen to give a good indication of the cold front position on the reservoir bottom as would be expected. The cold front and recirculation region have just reached the dam in Fig. V-18(b). The configuration of underflow cold layer and wedge of warmer ambient water can be clearly seen. Again as in the velocity fields the plunge point is well defined.
Fig. V-16. Vertical profile of eddy viscosity just downstream of the plunge point (Run TRB 138, elapsed time 520 seconds, grid point 21).
Fig. V-17(b). Reservoir flow pattern.
Fig. V-17(d). Reservoir flow pattern.
The situation some time after the recirculation zone has reached the dam is shown in Fig. V-18(c). The second eddy has appeared on the water surface as indicated by the velocity vectors. The temperature contours show the upwelling of mixed water at the dam. Because of this the last zone of ambient unmixed water occurs not at the dam but at some distance upstream.

The plunge point in the velocity and temperature fields shown above appears stable, but close inspection shows that a small amount of downstream drift occurs. This movement has been documented in Fig. V-6 in terms of the flow depth at the plunge point. Figures V-19 and V-20 show the behavior with time of the densimetric Froude number at the plunge point, \( F_p \), for the runs on a .02 slope and a .01 slope. This shows \( F_p \) decreasing with time as the plunge point moves downstream. Considering that each step on these plots shows plunge point movement of one grid length, the results indicate that the behavior of the densimetric Froude number of the plunge point is basically independent of inflow buoyancy flux. Defining the plunge point as the most downstream point reached, \( F_p \) values were extracted for all runs and are shown in Table V-9.

The initial forward movement of the plunge point was noted by Singh and Shah [1971]. It is likely associated with the changing balance of forces as the density current moves down the reservoir bottom. The motion in the laminar simulations appeared to be strongly influenced by separation. The plunge point never stabilized but continued to move towards the dam. The pattern emerging in the turbulent simulations is more representative of the results observed by Singh and Shah [1971] and of the observations in large reservoirs which generally show a stable plunge point. Depths at the stabilized plunge point were extracted for each run and are given in Table V-9. These depths are plotted against the quantity \( (q'/g')^{1/3} \) in Fig. V-21 and may be seen to be reasonably well correlated with this parameter. The line shown was fitted to the points by eye. Thus \( H_p \) is given by the equation

\[
H_p = 1.6 \left( \frac{q'}{g'} \right)^{1/3} \tag{V-19}
\]

This result is discussed again in the section on a two-layer flow analysis which follows this section.

As indicated in Section V.A, the important feature of the \( k-\epsilon \) model is that the eddy viscosity, \( \nu_t \), can vary throughout the flow field in response to local conditions. In particular the production, \( P \), of kinetic energy by shear has a large influence on the value of \( \nu_t \). Contours of \( P \) for run TRB138 at elapsed time of 840 seconds are shown in Fig. V-22. These show the large production associated with the steep velocity gradients near the bottom and in the interfacial region. In the absence of the buoyancy term in the \( k \) equation high values of \( \nu_t \) could be expected in these areas.

Contours of \( \nu_{eff} (= \nu + \nu_t) \) in the same flow field are shown in Fig. V-23. Values of \( \nu_{eff} \) of less than \( 1 \times 10^{-6} \text{ m}^2/\text{sec} \) occur but are not specifically shown as when \( \nu_t \) drops below this value the laminar viscosity part of \( \nu_{eff} \) controls the flow. It can be seen that this
Fig. V-18(a). Temperature contours (°C) of partially developed flow field (Run TRB 138, Elapsed time 640 seconds).
Fig. V-18(b). Temperature contours (°C) of developed flow field. (Run TRB 138, Elapsed time 840 seconds).
Fig. V-18(c). Temperature contours (°C) some time after cold front reached dam. (Run TRB 138, Elapsed time 1120 seconds).
Fig. V-19. Plunge point densimetric Froude number behavior for four runs on a slope of .02.
Fig. V-20. Plunge point densimetric Froude number behavior for three runs on a slope of 0.01.
<table>
<thead>
<tr>
<th>Run Number</th>
<th>Plunge Point Densimetric Froude Number</th>
<th>Underflow Mean Velocity V (m/sec)</th>
<th>Plunge Depth H_p (m)</th>
<th>Underflow Normal Densimetric Froude Number</th>
<th>Underflow Normal Depth H_N (m)</th>
<th>Entrainment Constant E</th>
<th>Entrainment Constant E</th>
<th>Friction Factor f</th>
<th>Initial Mixing Coeff. γ</th>
</tr>
</thead>
<tbody>
<tr>
<td>TRB 138</td>
<td>.475</td>
<td>.115</td>
<td>1.102</td>
<td></td>
<td>.828</td>
<td>.0061</td>
<td>.0051</td>
<td>.0022</td>
<td>.064</td>
</tr>
<tr>
<td>TRB 139</td>
<td>.469</td>
<td>.156</td>
<td>.962</td>
<td></td>
<td>.668</td>
<td>.0048</td>
<td>.0013</td>
<td>.0018</td>
<td>.063</td>
</tr>
<tr>
<td>TRB 142</td>
<td>.493</td>
<td>.089</td>
<td>1.478</td>
<td>1.01</td>
<td>1.066</td>
<td>.0051</td>
<td>.0022</td>
<td>.064</td>
<td>.165</td>
</tr>
<tr>
<td>TRB 143</td>
<td>.462</td>
<td>.134</td>
<td>1.066</td>
<td></td>
<td>.723</td>
<td>.0055</td>
<td>.0013</td>
<td>.063</td>
<td>.14</td>
</tr>
<tr>
<td>TRB 145C</td>
<td>.511</td>
<td>.109</td>
<td>1.143</td>
<td></td>
<td>.874</td>
<td>.0011</td>
<td>.0013</td>
<td>.063</td>
<td>.095</td>
</tr>
<tr>
<td>TRB 146</td>
<td>.518</td>
<td>.128</td>
<td>.990</td>
<td>.877</td>
<td>.770</td>
<td>.0019</td>
<td>.0018</td>
<td>.063</td>
<td>.105</td>
</tr>
<tr>
<td>TRB 147</td>
<td>.501</td>
<td>.141</td>
<td>.920</td>
<td></td>
<td>.709</td>
<td>.0023</td>
<td>.0013</td>
<td>.055</td>
<td>.028</td>
</tr>
<tr>
<td>TRB 156</td>
<td>.532</td>
<td>.137</td>
<td>.822</td>
<td>.772</td>
<td>.658</td>
<td>.0001</td>
<td>.0013</td>
<td>.055</td>
<td>.028</td>
</tr>
<tr>
<td>TRB 157</td>
<td>.449</td>
<td>.124</td>
<td>1.250</td>
<td>1.054</td>
<td>.840</td>
<td>.0074</td>
<td>.0015</td>
<td>.09</td>
<td>.19</td>
</tr>
<tr>
<td>TRB 158</td>
<td>.502</td>
<td>.145</td>
<td>.853</td>
<td>.845</td>
<td>.660</td>
<td>.0012</td>
<td>.0016</td>
<td>.054</td>
<td>.095</td>
</tr>
<tr>
<td>TRB 159</td>
<td>.654</td>
<td>.135</td>
<td>1.029</td>
<td>.971</td>
<td>.723</td>
<td>.0047</td>
<td>.0021</td>
<td>.047</td>
<td>.11</td>
</tr>
<tr>
<td>TRB 160</td>
<td>.539</td>
<td>.139</td>
<td>.814</td>
<td>.79</td>
<td>.665</td>
<td>.0005</td>
<td>.0013</td>
<td>.057</td>
<td>.055</td>
</tr>
</tbody>
</table>
Fig. V-21. Plunge depths given by numerical simulation.
Fig. V-22. Contours of shear production, P (Run TRB 138, Elapsed time 840 seconds).
Fig. V-23. Contours of effective viscosity, $\nu_{\text{eff}}$ (Run TRB 138, Elapsed time 840 seconds).
occurs in a large part of the recirculating area. This indicates the influence of buoyancy forces in this flow. The very low values of $v_t$ arise as after plunging the backflow reinforces the density gradients and the buoyancy energy draining term in the $k$ equation becomes large.

The $k-\varepsilon$ model is a high Reynolds number model and strictly should only be used where turbulence is intense and energy production by shear is strong. Use of the model in situations such as the plunging flow situation where buoyant damping is high is questionable. However, the flow fields predicted using the model appear reasonable. It is unlikely that the predictions would change in any major way if a modified model were used.

Even though the $v_{eff}$ values are low in a large part of the recirculating area, the temperature contours shown in Fig. V-18 do not display a sharp interface. The reason for this is that the numerical diffusion, $v_{num}$, discussed in Section III.D is high and is smearing the sharp gradients. Contours of $v_{num}/v_{eff}$ computed using the formula of DeVahl Davis and Mallinson [1978] are shown in Fig. V-24. These illustrate the high numerical diffusion in the recirculating region where the velocities are skewed off the grid lines. If $v_t$ values remained high, as would be the case in a nonbuoyant situation, the numerical diffusion would be insignificant. However, as $v_t$ decreases at the density gradients and the velocity skewness is large there, the quantity $v_{num}/v_{eff}$ is magnified.

As outlined in Section III.D, numerical diffusion is a universal problem in computational fluid mechanics. The various schemes referenced in that section appear to be able to reduce numerical diffusion significantly. It will be worthwhile incorporating such a scheme in this model. The removal of numerical diffusion will constitute a major improvement in itself. Furthermore, it will allow a better assessment of various effects, such as the $v_t$ reduction effect and the $\sigma_t$ variation effect, which at present may be obscured by numerical diffusion.

c. **Two layer flow analysis**

In this section the plunging flow fields are analyzed as two layer flows. The analysis for the 0.02 slope reservoir is presented in detail. Similar results for reservoirs with other slopes are presented in summary form.

Figures V-17(d) and V-18(b) show velocity and temperature fields for run TRB138 just as the cold front reaches the dam. Vertical profiles of velocity and temperature in these fields at a location downstream of the plunge point are shown in Fig. V-25. These profiles show a continuous variation over depth. Reduction of the profiles to a two-layer representation is somewhat arbitrary. As outlined in Section II.B, the zero velocity point is taken as the interface.

The discharge in the lower layer for run TRB 138 was computed at a number of positions along the reservoir for three elapsed times. The data are plotted as $\Delta Q/Q$ in Fig. V-26, where $Q_0$ is the river inflow and $\Delta Q$ is the increase in discharge of the underflow. These show the situation before, as, and after the cold front reaches the dam. The area near the
Fig. V-24. Contours of relative numerical diffusion, $\nu_{\text{num}}/\nu_{\text{eff}}$ (Run TRB 138, Elapsed time 840 seconds).
Fig. V-25. Vertical profiles of temperature and velocity downstream of the plunge point (Run TRB 138, Elapsed time 840 seconds, Grid point 30).
Fig. V-26. Underflow discharge pattern downstream of the plunge point for three elapsed times (Run TRB 138).
dam varies with time and the dam may be exerting an influence here even after the cold front has stabilized. Downstream of the plunge point, but away from the dam, the discharge pattern remains constant so that a true quasi-steady situation has developed. This area also shows a linear increase in discharge as would be expected in a developed two-layer flow.

Plots of $\Delta Q/Q_0$ for the other simulations on slope 0.02 just as the cold front reached the dam were also prepared. All displayed the characteristic area of linear increase in underflow discharge downstream of the plunge point. In each case this linear area was taken as representing the developed part of the underflow.

Mean velocities in the lower layer were computed at all longitudinal grid points in the portion of the underflow with the linearly increasing discharge. These velocities were found to vary little along the reservoir for each run. The values were averaged and the resulting mean velocities are shown in Table V-9. As outlined in Section II.B, in a two layer situation on a fixed slope the underflow velocity should vary as the buoyancy flux raised to a power of $1/3$. The velocities on slope $= .02$ are plotted against $(T_0 - T_{in})^{1/3}$ in Fig. V-27 and can be seen to follow such a relationship. The slope of this relationship indicates using Equation (II-27) that in this case the normal densimetric Froude number of the underflow is of order unity.

The velocities on slope $= .01$ similarly all fell on a line when plotted against $(T_0 - T_{in})^{1/3}$. In this case the normal densimetric Froude number of the underflow was calculated to be .877. The data from the other slopes were similarly used to define a relationship between average underflow velocity and buoyancy flux and hence yield an underflow normal densimetric Froude number, $F_N$. The $F_N$ values calculated in this way are shown in Table V-9.

These densimetric Froude number values are calculated on the basis of a velocity averaged on the profile from the zero velocity point to the reservoir bottom. Any other layer definition would give a depth smaller than the one used. Mean velocities could be correspondingly higher and $F_N$ values would be greater than the values calculated above.

The normal underflow depths calculated on the basis of the above $F_N$ values and the river inflow discharge and density difference are shown in column 6 of Table V-9. The normal depths for two cases are shown superimposed on the appropriate velocity field in Figs. V-28(a) and (b). Since the normal depths were computed from these velocity fields, it is not surprising that the underflows are well scaled, as can be observed, by the normal depths. Visual estimation of the position where the underflow emerges from the plunge region yields locations corresponding to the positions on the $\Delta Q/Q_0$ versus distance plots where the linear increase in underflow discharge commences.

The values of the entrainment constant, $E$, as defined in Section II-B were calculated from the slope of the linear portion of the $\Delta Q/Q_0$ plots and are shown in column 7 of Table V-9. Column 8 shows for comparison purposes the $E$ values given by Equation (II-21), adjusted to account for the different layer definitions. The actual values show little relation to
SLOPE = 0.02

\[ \nu_{AV} \propto (T_0 - T_{IN})^{1/3} \]

Fig. V-27. Underflow mean velocity as a function of buoyancy flux.
Figure A-28(b) - Reservoir flow pattern showing underflow normal depth.

W = 1.066 M

H

0.0

50.0

0.0

50.0

100.0

200.0

Distance Into Reservoir = m
the computed values. The actual entrainment is too high at the higher $F_N$ values and decreases much too quickly as $F_N$ decreases (i.e. as buoyancy increases). The reasons for this behavior are not clear but are probably a combination of inadequacies in the turbulence model and numerical diffusion effects.

Profile shape factors $S_1$, $S_2$, and $S_3$ were computed from the data and had the values 0.71, 0.84, and 1.23, respectively. These values were used with Eq. (II-25) (with $a$ and $h/B_0$ set to zero) to compute the friction factors shown in column 9 of Table V-9. These friction factors appear very high.

As discussed in Section II.B.3, basic physical considerations and also the results of the formal two-layer flow analysis using the Schijf and Schonfeld equations indicate that the plunge depth should be well correlated with normal underflow depth. Accordingly the plunge depths are plotted against normal depths on Fig. V-29. It can be seen that there is some correlation, but this is in fact due to the underlying critical depth as can be seen from Fig. V-21. Hence, introduction of specific reservoir information (Viz., bottom slope $S$ and friction factor, $f$) introduces scatter into the plunge depth relations. This is not surprising of course as the data in Table V-9 show that plunging generally occurs at an $F_p$ value of about 0.5 independent of slope. The relation fitted to the data in Fig. V-21 does in fact correspond to an $F_p$ value of 0.5 when cast in the form of Equation (II-17).

As defined in Section II.B.3, the initial mixing coefficient, $\gamma$, is the value of $\Delta Q/Q_o$ where the underflow starts. These values were read from the plots of $\Delta Q/Q_o$ versus distance into the reservoir. On some plots the position for defining $\gamma$ was clear cut, on others some interpretation was required. The $\gamma$ values as read off the graphs are shown in the final column of Table V-9. Given the uncertainty surrounding the value of $\gamma$, no attempt was made to apply a correction for plunge region length.

On the two slopes for which more than one $\gamma$ value was available, there is no obvious trend with changing buoyancy flux, but scatter is evident in the data. This scatter arises it is felt mainly because of the problems in pin-pointing a $\gamma$ value. An inadequate flow model coupled with numerical diffusion effects may also contribute to the scatter. Although the sample sizes are small, a standard deviation was computed and yielded a value of 0.01 in each case.

The $\gamma$ values do show a trend with the normal densimetric Froude number, $F_N$, of the underflow. This is in line with the expectations outlined in Section II.B. The data are shown plotted against $F_N$ in Fig. V-30. To highlight the fact that the $\gamma$ values are not precisely determined points, all the data points are shown with a ± .01 error band. In the case of the $\gamma$ values for the .01 and .02 slopes the error band is the actual standard deviation. These points are distinguished by arrows on the band limits. In the case of the values on the other slopes, the band is merely an artifact to break the visual impact of a precisely plotted data point. The shaded band encompasses all the data points and the definite trend with $F_N$ can be discerned. The simple straight line fitted to the points as indicated follows the equation...
Fig. V-29. Plunge depth as a function of downstream normal depth.
Fig. V-30. Initial mixing rate, \( \gamma \), as a function of downstream normal densimetric Froude number, \( F_N \).
\[ \gamma = 0.5 \left( F_N - 0.7 \right) \]

This becomes zero at \( F_N = 0.7 \). For \( F_N \) values less than 0.7, \( \gamma \) may be taken as zero. A straight cutoff at \( F_N = 0.7 \) is abrupt as \( \gamma \) probably approaches zero asymptotically as \( F_N \) approaches zero. Thus, the relation for initial mixing becomes

\[ \gamma = 0.5 \left( F_N - 0.7 \right) \text{ for } F_N > 0.7 \]
\[ = 0.0 \text{ for } F_N < 0 \]  \hspace{1cm} (V-20)

Since all upstream moving fluid is assumed to be mixed in the plunge region, the above \( \gamma \) values may be too high. Nevertheless the values given by Eq. (V-20) are much less than the values generally quoted in the literature.

C. **Diverging Reservoir**

1. **Introduction**

The diverging reservoir is considered in this section. As outlined in Section II.C.4, experimental results indicate that flow separation can be expected in the diverging situation when the sidewall angle exceeds about 5°. Separated flow cannot, of course, be modelled in the two-dimensional axisymmetric model used here. For very small angles of divergence (\( \delta < 5° \)) the flow will remain attached. At those angles, however, bottom slope will play an important role in real reservoirs, and the flow can be approximated as a sloping reservoir situation with parallel sides.

For the above reasons the axisymmetric situation is not extensively examined. The model was brought to a stage where it could reproduce plunging reservoir flow but was not applied over a range of conditions. The model equations and computational details are presented in the following section. The results for the application of the model to flow in a single geometrical configuration of about the same size as that used in the sloping reservoir simulations are presented in summary form.

Using the model an attempt was made to reproduce the theoretically predicted behavior described in Section II.C.3. These efforts met with no success and are described briefly in Section V.C.6.

2. **Governing equations**

a. **Mean flow equations**

The details of the derivation of the governing equations are given in Appendix 2. The Boussinesq approximation has been used and the gravity term has been treated as outlined in Section II.B. Letting \( u = u_z \) and \( v = \bar{V}_r \) for convenience, the momentum equations are
\[
\frac{\partial u}{\partial t} + v \frac{\partial u}{\partial r} + u \frac{\partial u}{\partial z} = - \frac{\partial p}{\partial z} + \frac{1}{r} \frac{\partial}{\partial r} \left( \nu \frac{\partial u}{\partial r} \right) + \frac{\partial}{\partial z} \left( \nu \frac{\partial u}{\partial z} \right)
\]

and

\[
\frac{\partial v}{\partial t} + v \frac{\partial v}{\partial r} + u \frac{\partial v}{\partial z} = - \frac{\partial p}{\partial r} + \frac{1}{r} \frac{\partial}{\partial r} \left( \nu \frac{\partial v}{\partial r} \right) + \frac{\partial}{\partial z} \left( \nu \frac{\partial v}{\partial z} \right)
\]

where \( \nu_{\text{eff}} = \nu_t + \nu \) and the other quantities are as previously defined.

The laminar portion of these terms in the above equations which do not appear in the laminar equations can be discarded if desired since \( \nu \) is constant. The terms then appear above multiplied by \( \nu_t \) instead of \( \nu_{\text{eff}} \).

The continuity equation for the mean velocities is

\[
\frac{1}{r} \frac{\partial}{\partial r} (rv) + \frac{\partial u}{\partial z} = 0
\]  

The energy equation is

\[
\frac{\partial T}{\partial t} + v \frac{\partial T}{\partial r} + u \frac{\partial T}{\partial z} = \frac{1}{r} \frac{\partial}{\partial r} \left( \alpha \frac{\partial T}{\partial r} \right) + \frac{\partial}{\partial z} \left( \alpha \frac{\partial T}{\partial z} \right)
\]

where \( \alpha_{\text{eff}} = \frac{\nu_t}{\nu_t} + \frac{\nu}{Pr} \)

As in the sloping reservoir study, the turbulent Prandtl number was set at the value 0.9.

b. **Turbulence model equations**

The turbulence model in axisymmetric coordinates has the same basic structure as the \((r,\theta)\) form. The eddy viscosity, \( \nu_t \), is computed from the equation
where \( k \) and \( \varepsilon \) are, as before, the turbulent kinetic energy and its dissipation rate per unit mass, respectively. The quantities \( k \) and \( \varepsilon \) are determined from the differential equations

\[
\frac{\partial k}{\partial t} + \nu \frac{\partial k}{\partial r} + u \frac{\partial k}{\partial z} = \frac{1}{r} \frac{\partial}{\partial r} \left( \nu_{\text{eff}} \frac{\partial r}{\partial r} \right) + \frac{\partial}{\partial z} \left( \nu_{\text{eff}} \frac{\partial k}{\partial z} \right) + P + G - \varepsilon \tag{V-25}
\]

where

\[
\nu_{\text{eff}} = \nu + \frac{\nu_t}{\sigma_k}
\]

and

\[
\frac{\partial \varepsilon}{\partial t} + \nu \frac{\partial \varepsilon}{\partial r} + u \frac{\partial \varepsilon}{\partial z} = \frac{1}{r} \frac{\partial}{\partial r} \left( \nu_{\text{eff}} \frac{\partial \varepsilon}{\partial r} \right) + \frac{\partial}{\partial z} \left( \nu_{\text{eff}} \frac{\partial \varepsilon}{\partial z} \right) + C_{1\varepsilon} \frac{\varepsilon}{k} (P + C_3 G) - C_2 \varepsilon \frac{\varepsilon^2}{k} \tag{V-26}
\]

where \( \nu_{\text{eff}} = \nu + \nu_t/\sigma_\varepsilon \) and \( u \) and \( \nu \) are as defined in the previous section. The coefficients in the \( k \) and \( \varepsilon \) equations were set at the same values used for the sloping reservoir study. In the above equations the quantity \( P \) gives the rate of production of turbulent energy from the mean flow. The function is discussed in detail in Appendix 2. It is given by the expression

\[
P = \nu_t \left[ 2 \left( \frac{\delta v}{\delta r} \right)^2 + 2 \left( \frac{\nu}{r} \right)^2 + 2 \left( \frac{\delta u}{\delta z} \right)^2 + \left( \frac{\delta v}{\delta z} + \frac{\delta u}{\delta r} \right)^2 \right] \tag{V-27}
\]

The quantity \( G \) denotes the production or destruction of turbulent energy by buoyancy forces. This quantity is discussed in Appendix 2. It is given by the expression

\[
G = \beta g \frac{\nu_t}{\sigma_t} \frac{\delta T}{\delta z} \tag{V-28}
\]

where \( T \) is the mean temperature and the other quantities are as previously defined.
c. Equation summary

All of the equations presented above can be written in the standard form

\[ \frac{\delta \phi}{\delta t} + v \frac{\delta \phi}{\delta r} + u \frac{\delta \phi}{\delta z} = \text{pressure term} \]

\[ + \frac{1}{r} \frac{\delta}{\delta r} \left( \Gamma r \frac{\delta \phi}{\delta r} \right) + \frac{\delta}{\delta z} \left( \Gamma \frac{\delta \phi}{\delta z} \right) + S_c \]

where \( \phi \) is a general variable, \( \Gamma \) a diffusion coefficient and \( S_c \) a source term. When \( \phi \) is not a velocity, the pressure term vanishes.

Table V-10 summarizes the \( \Gamma \) and \( S_c \) expressions that occur in the standard equation forms for \( u_z \), \( v_r \), \( T \), \( k \) and \( \epsilon \).

3. Boundary and initial conditions

The boundary conditions used in the axisymmetric reservoir were identical to those used in the sloping reservoir case. The description and discussion of boundary conditions given in Section V.B.2 can thus be interpreted as referring also to the axisymmetric case. This description is not duplicated here. The boundary conditions used for the variables \( u_z \), \( v_r \), \( T \), \( k \) and \( \epsilon \) are presented in summary form in Table V-11.

The initial fields of velocity and \( k \) and \( \epsilon \) were computed in the same manner as described for the sloping reservoir in Section V.B.2. The initial temperature field consisted of a constant temperature throughout the reservoir.

4. Model reservoir and computational details

A reservoir in the same size range as that used for the sloping case was selected for use. Actual dimensions were 96.0 m long and 1.0 m deep. An initial radius of 15.0 m and an inflow velocity of 0.175 m/sec were used.

As in the sloping situation, the flow area was modelled using 62 longitudinal and 22 vertical grid points. A time step of 5 seconds was used.

The computational parameters NTIMES and LAST were given the same values as for the sloping reservoir calculations, i.e. NTIMES was set to unity and LAST to 12.

5. Results

Three runs were carried out with the geometrical configuration described above. The run details are given in Table V-12. Those flows evolved in the same manner as the sloping reservoir flows with local viscosity lowering appearing as the cold front advanced.

A typical developed velocity field is shown in Fig. V-31. The behavior of the densimetric Froude number at the plunge point for the three
TABLE V-10. Summary of Diffusion Coefficients and Source terms in the Governing Equations for the Axisymmetric Situation

<table>
<thead>
<tr>
<th>Variable</th>
<th>Symbol</th>
<th>Diffusion Coefficient, $\Gamma$</th>
<th>Source term $S_c$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vertical Velocity</td>
<td>$u$</td>
<td>$v_{\text{eff}} = v + v_t$</td>
<td>None</td>
</tr>
<tr>
<td></td>
<td></td>
<td>$\frac{1}{r} \frac{\partial}{\partial r} \left( r v_{\text{eff}} \frac{\partial v}{\partial z} \right) + \frac{\partial}{\partial z} \left( v_{\text{eff}} \frac{\partial u}{\partial z} \right)$</td>
<td></td>
</tr>
<tr>
<td>Radial Velocity</td>
<td>$v$</td>
<td>$v_{\text{eff}} = v + v_t$</td>
<td>$P + G - \varepsilon$</td>
</tr>
<tr>
<td></td>
<td></td>
<td>$\frac{1}{r} \frac{\partial}{\partial r} \left( r v_{\text{eff}} \frac{\partial v}{\partial r} \right) + \frac{\partial}{\partial r} \left( v_{\text{eff}} \frac{\partial v}{\partial r} \right) - \frac{2v_{\text{eff}} v}{r^2}$</td>
<td></td>
</tr>
<tr>
<td>Temperature</td>
<td>$T$</td>
<td>$\frac{v}{Pr} + \frac{v_t}{\sigma_t}$</td>
<td>None</td>
</tr>
<tr>
<td>Turbulent Kinetic Energy</td>
<td>$k$</td>
<td>$v + \frac{v_t}{\sigma_k}$</td>
<td>$P + G - \varepsilon$</td>
</tr>
<tr>
<td>Turbulent Energy Dissipation Rate</td>
<td>$\varepsilon$</td>
<td>$v + \frac{v_t}{\sigma_\varepsilon}$</td>
<td>$C_1 \frac{\varepsilon}{k} (P + C_3 G) - C_2 \frac{\varepsilon^2}{k}$</td>
</tr>
</tbody>
</table>
### TABLE V-11. Summary of Boundary Conditions for the Axisymmetric Reservoir

<table>
<thead>
<tr>
<th>Variable Name</th>
<th>Symbol</th>
<th>Inlet Plane</th>
<th>Reservoir Bottom</th>
<th>Free Surface</th>
<th>Dam Face</th>
<th>Outflow Point</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vertical Velocity</td>
<td>u</td>
<td>u = 0</td>
<td>δu / δz = 0</td>
<td>u = 0</td>
<td>u = u* / k ln (U* Y / v) E_b</td>
<td>δu / δr = 0</td>
</tr>
<tr>
<td>Radial Velocity</td>
<td>v</td>
<td>v = V_0 Y / H (1/7)</td>
<td>v = U* / k ln (U* Y / v)</td>
<td>δv / δz = 0</td>
<td>δv / δr = 0</td>
<td>{ δv / δr = 0</td>
</tr>
<tr>
<td>Temperature</td>
<td>T</td>
<td>T = T_{IN} (constant)</td>
<td>δT / δz = 0</td>
<td>δT / δr = 0</td>
<td>T = T_0</td>
<td>δT / δr = 0</td>
</tr>
<tr>
<td>Turbulent Kinetic Energy</td>
<td>k</td>
<td>k = U_*^2 (0.3 + 3 Y / H)</td>
<td>δk / δz = 0</td>
<td>δk / δr = 0</td>
<td>δk / δr = 0</td>
<td>δk / δr = 0</td>
</tr>
<tr>
<td>Turbulent Energy Dissipation Rate</td>
<td>ε</td>
<td>ε = C_μ U_<em>^3 / U_</em> K H (1 - Y / H)</td>
<td>ε = U_*^3 / K Y_p</td>
<td>δε / δz = 0</td>
<td>ε = U_*^3 / K Y_p</td>
<td>δε / δr = 0</td>
</tr>
</tbody>
</table>
### TABLE V-12. Details of Simulations in Axisymmetric Reservoir

<table>
<thead>
<tr>
<th>Run Number</th>
<th>Temperature Difference (^\circ\text{C})</th>
<th>Inflow Densimetric Froude Number (F_p)</th>
<th>Inflow Reynolds Number (\text{Re})</th>
</tr>
</thead>
<tbody>
<tr>
<td>ATRB 106</td>
<td>10</td>
<td>1.25</td>
<td>1.75x10^5</td>
</tr>
<tr>
<td>ATRB 107</td>
<td>5</td>
<td>1.77</td>
<td>1.75x10^5</td>
</tr>
<tr>
<td>ATRB 108</td>
<td>15</td>
<td>1.02</td>
<td>1.75x10^5</td>
</tr>
</tbody>
</table>

### TABLE V-13. Details of Runs to Test Effect of Initial Radius

<table>
<thead>
<tr>
<th>Run Number</th>
<th>Inflow Radius (\text{m})</th>
<th>Reservoir Length (\text{m})</th>
<th>Time Step (\text{sec})</th>
<th>Temperature Difference (^\circ\text{C})</th>
<th>Inflow Densimetric Froude Number (F_p)</th>
<th>Inflow Reynolds Number (\text{Re})</th>
<th>Behavior of Densimetric Froude Number at Plunge Point (F_p)</th>
</tr>
</thead>
<tbody>
<tr>
<td>ATRB 107</td>
<td>25.0</td>
<td>96.0</td>
<td>5</td>
<td>5</td>
<td>1.77</td>
<td>1.75x10^5</td>
<td>.65 - .56</td>
</tr>
<tr>
<td>ATRB 109</td>
<td>10.0</td>
<td>96.0</td>
<td>5</td>
<td>5</td>
<td>1.77</td>
<td>1.75x10^5</td>
<td>.51 - .46</td>
</tr>
<tr>
<td>ATRB 110</td>
<td>5.0</td>
<td>20.0</td>
<td>2</td>
<td>5</td>
<td>1.77</td>
<td>1.75x10^5</td>
<td>.57 - .51</td>
</tr>
<tr>
<td>ATRB 111</td>
<td>1.0</td>
<td>3.0</td>
<td>0.5</td>
<td>10</td>
<td>1.25</td>
<td>1.75x10^5</td>
<td>.51 -</td>
</tr>
</tbody>
</table>
Fig. V-31. Typical developed velocity field in an axisymmetric situation.
runs is summarized in Fig. V-32. It can be seen that the pattern displayed there is similar to that found in the sloping configuration.

6. **Free overfall and small reservoir simulations**

The theoretical analysis in Section II.C.3 applies to flow which is purely axisymmetric. Thus, the various flow situations described in that section should have some relevance to the numerical model results.

The theoretical work relates to flow controlled by a free overfall at the downstream beach end. Extensive efforts were made to reproduce this boundary condition in the numerical model by blocking out a beach section as was done in the laminar simulation. It was not possible to get the computations to converge with problems arising with the specification of the initial conditions. This is not surprising as the numerical solution of the problem of a steady-state non-buoyant turbulent flow over an expansion in an axisymmetric situation is itself a major exercise. Efforts to impose a beach drop-off were thus discontinued.

Next attention was concentrated on examining the effect of the inflow radius, $R_o$, in a simple axisymmetric situation with the beach drop-off simulated by withdrawal at the dam base. The inflow radius appears in the parameter $G$ defined in Section II.C.3. The objective was to see how the plunge point densimetric Froude number behaved as $R_o$ decreased. In particular, it was of interest to see if type II plunging ever developed. The run details are given in Table V-13 with the $F_p$ range also shown. A constant depth of 1.0 m with a 62x22 grid and an inflow velocity of 0.175 m/sec was used with all runs. It can be seen that all runs display a similar basic pattern with an $F_p$ value of order 0.5. Type II plunging was not seen.

The above may not be completely a fair test of the numerical model as the free overfall boundary condition is simulated by the withdrawal at the dam base and this may not be a good reproduction of that type of boundary condition. Furthermore the theoretical results refer to steady (or quasi-steady) state conditions when downstream conditions are well established. The numerical results give the unsteady-state development of the flow. It may be that as the condition at a free overfall is established, the large pool of ambient water downstream allows the plunge point to migrate upstream and assume a quasi-steady Type II configuration.

D. **Closure**

In this chapter attention has been concentrated on the sloping bottom reservoir as flow separation would render the axisymmetric results inapplicable in many cases. The axisymmetric model details were presented for record purposes as the model may find some use in the future for modelling some specific flow situations.

As has been seen, the flow model used can simulate plunging reservoir flow well. The generated flow fields appear reasonable when examined in terms of bulk flow parameters. The problems associated with the use of a
Fig. V-32. Plunge point densimetric Froude number behavior (axisymmetric reservoir).
liminar model, in particular the problem of bottom flow separation, have been eliminated. However, as illustrated by the poor predictions of the entrainment constant, $E$, and the friction factor, $f$, a number of problems still remain. The turbulence model used is far from perfect for this flow situation and the high buoyant damping is a cause for concern. The turbulence model can be improved by introducing functional terms for $C_h$ and $\sigma_C$ from an algebraic stress model. However, such efforts will not yield any improvement until the numerical diffusion problem is eradicated. The numerical diffusion problem is thus the main outstanding problem with this model and should be the first problem addressed in any attempted upgrading of the model.

The model has been applied only to the specific situation of plunging flow but the model as developed is a general reservoir flow model. It can be used to simulate the whole spectrum of reservoir flow phenomena; overflows and interflows, surface heating and cooling effects, wind effects or dam withdrawal effects. As the model stands at present, even without the improvements listed above, it has the potential for yielding useful insights into these phenomena as it has done in the plunging flow case.

The above comments refer to reservoir hydrodynamics. In fact in principle this model can be used directly to generate velocity and concentration fields in a water quality model. The biological part of those models would then be driven by continuously updated concentration fields. However, such a sophisticated model would be too consuming of computer time to be of much use in practice.

Various approaches could be adopted for incorporating the hydrodynamic model in a water quality model in a simplified manner. One method would be to solve for the velocity field, not continuously, but at intervals throughout the period of interest. This field could be considered as steady while the biological activity is modelled. Boundary conditions could then be changed and a new velocity field derived. Then the biological modelling starts again and so on.
VI. LABORATORY STUDY

The laboratory study is described in this chapter. This aspect of the work has been described in various technical papers and St. Anthony Falls Hydraulic Laboratory reports and memoranda. Where such publications exist reference is made to them rather than repeating the details herein. This chapter is thus a summary of the main results of an extensive experimental program. In this chapter, as in the previous ones, the sloping reservoir and diverging reservoir are considered separately.

A. **Experimental Facility**

1. **Basic equipment**

   All of the experiments were carried out in a large tank, 40 ft long, 16.5 ft wide, and 1.3 ft deep. Sloping reservoir and diverging reservoir shapes were constructed in the tank as required. A moveable platform (4 ft wide) mounted on rails spanned the tank. A small instrument carriage running on a rail on the platform carried temperature and velocity sensing probes. The position of the platform along the tank and of the carriage along the platform were visually read from pointers running along metal measuring tapes fixed to the tank and the platform. Using these, the probes could be positioned with an accuracy of ± 0.05 inch. The necessary recording instrumentation was carried on the platform. A drawing of the experimental facility is shown in Fig. VI-1.

   The density differences were provided by using water of different temperatures for the ambient tank water and inflow water. The cold (heavier) water entered the tank through a 6 ft long rectangular channel from a stilling basin. Thus, at the point of entry, the velocity profile of the cold water approximated fully developed channel flow. The water depth in the channel could be varied between 2 in. and 4 in. and channel width varied between 6 in. and 30 in. to give different aspect ratios. The cold water came directly from the city water supply and for small flows (< 0.5 ft³/sec) was metered through a constant head tank. For larger flows (ranging up to 0.2 ft³/sec), the water entered the stilling basin through a 2 in. pipe fitted with an orifice flow meter.

   The incoming cold water flowed into the appropriate geometrical configuration and plunged as expected. After plunging, the cold water flowed through the model reservoir to the tank bottom where it then formed a layer of mixed water. Withdrawal of this mixed water was accomplished by suspending a full tank width skimmer wall (3/4 in. painted plywood) to within 2-1/2 in. of the tank bottom as shown in Fig. VI-1. This retained the warm water but allowed the mixed water to escape underneath. The water level in the pool downstream of the skimmer wall was controlled by means of an adjustable weir.

   The flow situation described above corresponds to the situation described in the previous chapters in that it is unsteady. Preliminary
Fig. VI-1. Experimental facility.
calculations on this flow arrangement for the flow rates envisaged indicated that the bulk of the warm water would be removed too quickly for sufficient velocity and temperature measurements to be taken in the quasi-steady state. Hence, it was decided to continuously replenish the warm water and thus model an infinitely long reservoir with effectively steady-state conditions in the plunging flow field.

To provide the warm water, some of the water in the sump downstream of the weir was extracted by pump, passed through a heat exchanger, and returned to the tank. The outlet water temperature was thermostatically controlled to within ±1°F. For warm water flows of less than 0.2 ft³/sec a 3 HP pump was used for recirculation. For quantities in the range 0.2 ft³/sec up to 1.0 ft³/sec a 25 HP centrifugal pump was used. The warm water flows were monitored using orifice meters connected to U-tube manometers.

The warm water was introduced at the tank centerline just upstream of the skimmer wall and allocated across the full tank width by means of distribution boxes containing adjustable weirs. Dye visualization was used to insure that the flow was evenly allocated. A painted plywood shelf extending 2.0 ft horizontally out from the skimmer wall and 7 in. below the water surface, formed a bottom to the distribution boxes. This shelf, together with a vertical weir on its front end reaching to within 3 inches of the water surface, acted to prevent interference with the general flow pattern in the area. A series of 10 guide vanes extending 2.0 ft out from the upstream edge of the boxes directed the flow upstream. Two 6" x 1-1/2" timber planks were tethered at the water surface, one 5 ft upstream of the warm water inflow, the other at the end of the guide vanes. These acted as a floating breakwater to damp small wave effects. A schematic drawing of the flow arrangement is shown in Fig. VI-2.

2. Data acquisition system

Temperatures in the flow field were measured using a rake consisting of 16 YSI thermisters, Type 427 (Type 403, up to 6/19/84) at 0.25 in. vertical spacing. If the flow depth was greater than the rake size, a number of positionings of the rake, overlapping if necessary, could be taken to establish a complete vertical temperature profile.

The signals from the 16 rake mounted thermisters were fed into a specially constructed signal processor containing 16 separate amplifiers and thence to an Apple III microcomputer fitted with a 16 channel A/D converter card. The amplifiers were tuned to sense temperature in a 25°C band (10°C to 35°C in summer, 0 to 25°C in winter). The signal from each probe was converted to a temperature using individual calibration data stored in the Apple computer. The calibration curves were slightly non-linear and were very well fitted using a quadratic expression. The system was sensitive to within 0.1°C. The Apple scanned each probe in the rake 30 times over a 50 second period and computed average temperatures and standard deviations.

The temperature of the incoming cold water, of the hot water at the point where it entered the distribution boxes, and at two fixed positions in the tank were manually recorded every 30 minutes during an experiment.
Fig. VI-2. Schematic of flow arrangement.
The temperatures were measured using type 402 YSI thermistors connected to a YSI direct readout telethermometer. The probes connected to the telethermometer were also calibrated to within 0.1°C. The accuracy of these probes was of lesser importance as their primary function was to detect trends in temperature at a fixed point.

The velocity measuring device was mounted on a probe which was traversed vertically over the flow area. Instrument readings were taken at a number of positions on each vertical profile.

In the sloping reservoir experiments the velocities were reasonably robust (order 0.35 ft/sec) and were measured using a Delft miniature propeller flow meter (W.V.M. type). This has a propeller diameter of 0.59 inch (15 mm). The frequency output from this device was converted to voltage, suitably amplified and fed into the Apple microcomputer via an A/D converter card. This signal was converted to velocity using calibration data stored in the Apple computer. In the diverging reservoir experiments, velocities were generally low (on the order of 0.1 ft/sec) and were measured using the instrument described by Ellis and Stefan [1986]. This device uses a fibre optic sensor to determine travel time of a hydrogen bubble released from a fixed distance upstream. The travel time was related to flow velocity by calibration in a test flume. The readings from this instrument were recorded manually and converted into velocities later.

3. **Experimental procedure**

Each experiment lasted a complete day. The cold and hot water flow rates and the hot water thermostat were set as required. General data describing the experiment were entered into the Apple microcomputer. Then using the rake, the temperature profile at a fixed point in the flow field (usually on the tank centerline 12 ft from the channel outlet) was measured every 30 minutes to monitor the establishment of steady-state conditions. Because of the large size of the facility, it usually took about 3 hours for steady-state conditions to be satisfactorily established. When steady-state conditions had been established, temperature and velocity profiles were measured and visualization experiments conducted as desired.

In the case of temperature measurements, the location (x,y) coordinates were entered via the Apple keyboard and sampling began. After sampling, the temperature profile and standard deviations were displayed in numerical and graphical terms. The graphical display especially facilitated the "real time" recognition of flow behavior and of any unexpected appearances in the profiles. The data were immediately printed on an Epson printer and recorded to a diskette. The rake was then removed from the water and moved to a new (x,y) location. Collection of velocity data proceeded in a similar manner.

The flow patterns were made visible by introducing different colored dyes into the cold and warm waters. The dye pattern was recorded manually by sketching and photographically by using a still camera and a color video. Printed cards in the field of view identified the particular experiment. This visual data was later subjected to detailed examination and interpretation. A scale marked on the reservoir bottom or beach aided in the interpretation of this data.
At the end of each experiment a duplicate disk was made. The Apple was fitted with a communications interface card and the data were sent to the University of Minnesota Cyber Computer. Contour and 3-D plots were generated using various plotting routines available on that machine. Vertical profile plots were generated on a Textronic plotter linked to the Apple.

B. Sloping Reservoir Study

The two-dimensional sloping reservoir was simulated experimentally by constructing sidewalls in the large tank which were essentially an extension of the inflow channel walls. The sidewalls were 16 ft long and a constant 1.17 ft (14 inches) apart. The reservoir end was left open to allow warm water to enter. A sketch of the experimental facility is shown in Fig. VI-3.

1. General

Details of the experiments carried out on the sloping bottom reservoir are given in Table VI-1. All these experiments were carried out with a bottom slope of \( S = 0.047 \) (2.7°). The slope in the experimental facility could be varied. However, the data from the above slope indicated that little additional information, either on plunge depth or on initial mixing, would be gained by looking at plunging on the other slopes achievable in the facility. (The minimum slope realistically achievable was \( S = .03 \).)

The plunging flow fields in the experiments were as expected. The cold inflow plunged and flowed along the sloping bottom as a density current. Lateral mixing motions were observed in the plunge region so that the plunge line was rarely a straight line across the channel. Also small amounts of warm water were generally evident, intruding upstream from the plunge region on the surface at the reservoir sides. These intrusions are most likely due to the retarded inflow velocities at the sidewalls and were considered to be hydrodynamically insignificant. Notwithstanding these effects, the plunge region was well defined and a specific position could be assigned to the (mean) plunge line without difficulty. This position remained fixed throughout each experiment.

Secondary motion was observed in the inflow in some experiments. This consisted of an eddy on either side of the channel centerline aligned along the flow and transporting material away from the centerline on the water surface and towards it at the channel bottom. This influenced the velocity profiles and caused some problems with the lateral integrations of the local mass fluxes to find the total discharge.

Detailed data for the experiment of APR 11/86 are given in Table VI-2. Column 2 shows the inflow discharge, \( Q_o \), as measured by an orifice meter. The changes in value are due to pressure changes in the supply system. Column 3 and 4 show the underflow discharge, \( Q_m \), and the ambient upper layer discharge, \( Q_a \), as computed from the velocity profile data. These values were computed from at least three vertical profiles in a cross section. An error evaluation computed as
Fig. VI-3. Experimental arrangement for sloping reservoir experiments.
<table>
<thead>
<tr>
<th>Experiment Date</th>
<th>Inflow Discharge, $Q_o$ ($\text{ft}^3/\text{sec}$)</th>
<th>Inflow Temperature ($^\circ\text{C}$)</th>
<th>Ambient Reservoir Temperature ($^\circ\text{C}$)</th>
<th>Inflow Channel Depth (.320)</th>
<th>Inflow Densimetric Froude Nr, $F_o$</th>
<th>Flow Reynolds Number, Re</th>
</tr>
</thead>
<tbody>
<tr>
<td>MAR 7/86</td>
<td>.145</td>
<td>15.4</td>
<td>29.1</td>
<td>.320</td>
<td>2.16</td>
<td>10,200</td>
</tr>
<tr>
<td>MAR 11/86</td>
<td>.145</td>
<td>15.5</td>
<td>29.2</td>
<td>.320</td>
<td>2.16</td>
<td>10,200</td>
</tr>
<tr>
<td>MAR 14/86</td>
<td>.143</td>
<td>15.4</td>
<td>29.2</td>
<td>.320</td>
<td>2.14</td>
<td>10,100</td>
</tr>
<tr>
<td>MAR 24/86</td>
<td>.123</td>
<td>21.1</td>
<td>37.5</td>
<td>.320</td>
<td>1.53</td>
<td>10,000</td>
</tr>
<tr>
<td>MAR 27/86</td>
<td>.164</td>
<td>9.7</td>
<td>25.5</td>
<td>.320</td>
<td>2.59</td>
<td>9,900</td>
</tr>
<tr>
<td>APR 4/86</td>
<td>.165</td>
<td>10.4</td>
<td>25.2</td>
<td>.319</td>
<td>2.67</td>
<td>10,200</td>
</tr>
<tr>
<td>APR 11/86</td>
<td>.066</td>
<td>15.6</td>
<td>36.6</td>
<td>.161</td>
<td>2.10</td>
<td>4,500</td>
</tr>
</tbody>
</table>
### TABLE VI-2. Parameters for Experiment of APR 11/86

<table>
<thead>
<tr>
<th>Distance from Outlet, X (ft)</th>
<th>Inflow Discharge, $Q_o$ (ft³/sec)</th>
<th>Underflow Discharge, $Q_m$ (ft³/sec)</th>
<th>Upper Layer Discharge, $Q_a$ (ft³/sec)</th>
<th>Error in Continuity of Mass Flux (%)</th>
<th>Error in Continuity of Heat Flux (%)</th>
<th>Underflow Layer Depth, $H_N$ (ft)</th>
<th>Under Layer Densimetric Froude No., $F_N$</th>
<th>$Q_m - Q_o / Q_o$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.5</td>
<td>.066</td>
<td>.066</td>
<td>0.0</td>
<td>0.0</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>6</td>
<td>.064</td>
<td>.0748</td>
<td>.008</td>
<td>+ 3.7</td>
<td>+ 2.3</td>
<td>.299</td>
<td>1.04</td>
<td>1.17</td>
</tr>
<tr>
<td>8</td>
<td>.064</td>
<td>.0801</td>
<td>.0222</td>
<td>- 7.6</td>
<td>- 21.3</td>
<td>.289</td>
<td>1.18</td>
<td>1.25</td>
</tr>
<tr>
<td>10</td>
<td>.065</td>
<td>.0847</td>
<td>.0237</td>
<td>- 4.7</td>
<td>- 13.7</td>
<td>.303</td>
<td>1.19</td>
<td>1.30</td>
</tr>
<tr>
<td>12</td>
<td>.068</td>
<td>.0892</td>
<td>.0291</td>
<td>- 8.8</td>
<td>- 13.6</td>
<td>.317</td>
<td>1.18</td>
<td>1.31</td>
</tr>
<tr>
<td>14</td>
<td>.066</td>
<td>.0952</td>
<td>.0333</td>
<td>- 4.3</td>
<td>- 8.6</td>
<td>.361</td>
<td>1.05</td>
<td>1.44</td>
</tr>
<tr>
<td>15</td>
<td>.066</td>
<td>.0960</td>
<td>.0444</td>
<td>- 15.0</td>
<td>- 22.8</td>
<td>--</td>
<td>--</td>
<td>1.46</td>
</tr>
</tbody>
</table>
The data in columns 7 and 8 of Table VI-2 show the underflow depth, $H_N$, increasing in a downstream direction and the layer densimetric Froude number, $F_N$, quickly attaining a constant value. The $H_N$ and $F_N$ data are shown plotted against distance from the outlet, $X$, in Fig. VI-4. These plots indicate that the underflow layer appears to have stabilized in this case.

Dimensionless velocity and temperature deficit profiles at two locations on the centerline for the experiment of APR 11/86 are shown in Fig. VI-5. Each of these profile pairs is similar in form to the profile types shown in Fig. II-5(c) and to the numerically generated data of Fig. V-25 in that the top of the mixed layer corresponds closely to the zero velocity points.

The plots in Fig. VI-5 illustrate a trend found in all the experimental data of a slightly more peaked velocity profile having more momentum close to the bed developing with downstream distance. This suggests that even though bulk parameters such as $F_N$ indicate an established flow, some internal adjustments are still taking place. This adjustment is more evident in the temperature deficit plots which show the continuing erosion of the cold underlayer. The profile changes are reflected in the computed values of the profile shape factors. While $S_3$ remains generally constant at a value of about 1.18, $S_1$ changes from 0.78 at $X = 6$ ft to 0.42 at $X = 14$ ft with a mean value of 0.61. The factor $S_2$ decreases from 0.88 at 6 ft to 0.70 at 14 ft with a mean value of 0.81.

The channel bottom friction factor was established at $f = 0.038$ by running an open channel flow test. This positioned the channel bottom roughness on the universal friction factor diagram. The diagram indicated that an $f$ value of 0.04 could be expected for the underflow situation of APR 11/86. Use of Eq. (II-25) with the computed $S_1$, $S_2$, $S_3$, $F_N$ and $E$ (see Section VI.B.3) values yielded an $\alpha$ requirement of 1.8 to match the computed to the expected $f$ value. Alternatively, use of Eq. (II-25) with the expected $f$ value and the generally used $\alpha$ value of 0.5 yielded an $F_N$ value of 1.38.

2. **Plunge region position**

The plunge line position, $X_p$, as measured from the outflow point is given for each experiment in column 2 of Table VI-3. Depths, $H_p$ and densimetric Froude numbers, $F_p$, at the plunge line are given in columns 3 and 4, respectively. The $F_p$ values show little variation and have an average value of 0.69. The parameter $(q^2/g')^{1/3}$ is given in column 5. The $H_p$
Fig. VI-4. Behavior of layer normal depth, $H_N$, and densimetric Froude number, $F_N$. 
Fig. VI-5. Velocity and temperature profile at two locations in the sloping reservoir.
### TABLE VI-3. Parameters of Experiments on Sloping Bottom Reservoir

<table>
<thead>
<tr>
<th>Experiment Date</th>
<th>Distance from Outlet to Plunge Line, $X_p$ (ft)</th>
<th>Plunge Depth $H_p$ (ft)</th>
<th>Densimetric Froude No. at Plunge Point, $F_p$</th>
<th>Underflow Layer Densimetric Froude No. $F_N$</th>
</tr>
</thead>
<tbody>
<tr>
<td>MAR 7/86</td>
<td>7.75</td>
<td>.68</td>
<td>.69</td>
<td>.53</td>
</tr>
<tr>
<td>MAR 11/86</td>
<td>7.75</td>
<td>.68</td>
<td>.69</td>
<td>.53</td>
</tr>
<tr>
<td>MAR 14/86</td>
<td>7.75</td>
<td>.68</td>
<td>.68</td>
<td>.53</td>
</tr>
<tr>
<td>MAR 24/86</td>
<td>4.5</td>
<td>.53</td>
<td>.70</td>
<td>.43</td>
</tr>
<tr>
<td>MAR 27/86</td>
<td>10.0</td>
<td>.79</td>
<td>.67</td>
<td>.60</td>
</tr>
<tr>
<td>APR 4/86</td>
<td>10.5</td>
<td>.81</td>
<td>.66</td>
<td>.61</td>
</tr>
<tr>
<td>APR 11/86</td>
<td>3.75</td>
<td>.34</td>
<td>.69</td>
<td>.26</td>
</tr>
</tbody>
</table>
values are plotted against this parameter in Fig. VI-6 and can be seen to fall on or close to the line

\[ H_p = 1.3 \left( \frac{q^2}{g'} \right)^{1/3} \]  

(VI-1)

The data are shown plotted together with the data of Singh and Shah [1971] in Fig. VI-7. It can be seen that the new data points generally confirm the relationship established by Singh and Shah [1971] with all the data pointing to an average \( F_p \) value of 0.68 or 0.69.

Average densimetric Froude numbers, \( F_N \), in the underflow layers downstream of the plunge region are shown in column 6 of Table VI-3. These values were computed using Eq. (II-27) with the mean underflow layer velocity. The low \( F_N \) values may be due to low discharge computations. The values generally show that the underflow on this slope is internally critical or slightly supercritical.

The theoretical results of Jain [1980] and Akiyama and Stefan [1984] may be applied to this case. The relationship predicted by Jain [1980] (as read from Fig. 8 of his paper with \( \alpha = .5 \)) is shown plotted against the data in Fig. VI-8. The predicted line gives a smaller plunge depth than the actual, with predicted \( F_p \) values of about 0.97. The relation of Akiyama and Stefan [1984] for a steep slope i.e. Eq. (I-3) with the measured values of the profile constant \( S_1(=.72) \) and initial mixing (= 0.1, see next section) becomes

\[ H_p = 1.39 \left( \frac{q^2}{g'} \right)^{1/3} \]

This relation is also plotted on Fig. VI-8 and can be seen to predict plunge depths closer to the measured depths. In the absence of specific data on profile shapes and initial mixing Akiyama and Stefan [1984] use the profile constant value measured by Ellison and Turner [1959] (i.e. \( S_1 = 0.2 - 0.3 \)) with \( \gamma = 0 \) to yield the expression

\[ H_p = (1.49 - 1.71) \left( \frac{q^2}{g'} \right)^{1/3} \]

The lines for both coefficient values are shown in Fig. VI-8 and can be seen to predict deeper plunge depths than actual. The numerically generated relationship of Eq. (V-19) is also shown plotted in Fig. VI-8. This gives plunge depths deeper than actual.

The two-layer theory of Jain [1980] assumes a well defined layer continuing back to the plunge line. In reality it is possible that when the upper layer becomes thin, it breaks down in the eddying motion and the plunge region thus appears further downstream than predicted. This effect could be accounted for in the theoretical development by increasing the interfacial friction factor as the plunge region is approached. This would
Fig. VI-6. Experimental data on plunge depths.

\[ H_p = 1.3 \left( \frac{q^2}{g'} \right)^{\frac{1}{3}} \]
Fig. VI-7. Data from present study compared to data of Singh and Shah [1971].
Fig. VI-8. Comparison of various theories with the experimental data.
make the interface steeper, and it would intersect the water surface further downstream as required. However, the $f_1$ increase pattern must be known for this approach to work.

The $S_1$ values measured by Ellison and Turner [1959] are much lower than the values computed in this study. Hence use of the lower $S_1$ values in the theory of Akiyama and Stefan [1984] understandably give plunge depths greater than actual. When used with the measured $S$, and initial mixing coefficient value, the theoretical predictions of Akiyama and Stefan [1984] agree well with the experimental points.

The slope examined above is steep (critical or supercritical flow). The smallest slope that could be achieved in the experimental facility while keeping a reasonable depth change over the reservoir length was about 0.03. Use of the measured values of shape factors, and of $f_1$, $\alpha$ and $E$ in Eq. II-25 indicated that $F_N$ on such a slope would be about 0.93. This was considered to be not sufficiently far into the mild slope range to warrant a series of experiments at the new slope.

True mild slopes of order $S = .001$ such as are found in real reservoirs are difficult to achieve in the laboratory. Recourse to field studies may be necessary to study plunging flow under mild slope conditions.

3. Initial mixing

The underflow layer discharges in Table VI-1 were used to compute the value of the quantities $\Delta Q/Q_0$ shown in column 9 of that table. These data show $\Delta Q/Q_0$ increasing along the reservoir as expected. The $\Delta Q/Q_0$ values are shown plotted against distance along the reservoir in Fig. VI-9. The data can be seen to fall around a straight line as in the computer simulation data. The well defined trend in this data with little scatter around the trend line suggests that the accuracy in the discharge measurement of the underflow layer is much better than that indicated by the error computation in columns 5 and 6 of Table VI-2.

Backprojection to the plunge line of the straight line fitted to the data points gives an initial mixing coefficient for this experiment of $\gamma = 0.14$. Figure VI-9 also shows $\Delta Q/Q_0$ data for the experiments of MAR 24/86 and MAR 14/86. These data indicate initial mixing of about 0.1. The data from the other experiments was not precise enough to enable $\gamma$ values to be obtained, but the general mixing values were of the same order of magnitude as the values for the above experiments.

The experimental data thus yields an initial mixing coefficient, $\gamma$, value of about 0.1 for a normal layer densimetric Froude number, $F_N$, value of about 1.16. The numerically generated relation of Eq. (V-20) suggests a $\gamma$ value of about 0.23. The experimental value is less than this but the value confirms the order of magnitude of the initial mixing involved.

This single data point coupled with the trends established in the numerical data answers many questions about initial mixing of plunging flows. Since it gives little initial mixing at a quite high $F_N$ (critical or supercritical flow), it suggests that in most reservoirs with parallel
Fig. VI-9. Variation of $\Delta Q/Q_0$ with distance along reservoir.
or gently diverging sides initial mixing is practically zero, as real reservoirs generally have small slopes and low $F_N$ values. The high initial mixing reported in some real reservoirs may arise because of flow separation effects. This aspect is considered in the next section.

C. Diverging Reservoir Study

1. General

For the diverging reservoir experiments, a horizontal beach (16 ft x 12 ft) was installed in the experimental tank. The beach consisted of asbestos-cement sheets (each 8 ft x 4 ft) bolted to a metal frame. The frame, which was supported on concrete blocks placed on the tank bottom, was levelled by means of metal shims. The beach surface was set at the same elevation as the channel bottom so that channel bottom and beach formed a continuous horizontal surface. Sidewalls consisting of either L shaped metal or of a perspex upright bolted to a metal base rested on the beach and confined the inflow. Foam rubber on the wall bottom formed a seal between wall and beach. In some experiments where the wall angle was not varied, the wall was sealed to the beach using a bead of silicon. This experimental arrangement is shown schematically in Fig. VI-10. As can be seen on that figure the downstream boundary condition for the cold flow consists of a free overfall at the beach end.

As outlined in Section II.C.4, depending on sidewall angle, flow separation can be expected in this situation. Flow separation was in fact the dominant phenomenon in the flows examined in this experimental series. The separation phenomenon and added buoyancy effects gave rise to a large variety of flows. Order was imposed on this variety by classifying the flows into broad flow types. This aspect of the work is described in the section following. The values of the densimetric Froude number at the plunge point and the initial mixing rates are then examined.

The material in this section is a summary of that presented by Johnson et al. [1986]. Detailed descriptions and discussion of the flow fields, including contour and isometric plots of the temperature fields can be found in that publication.

2. Flow regimes

a. Flow classification scheme

The flow classification scheme used is an extension of that developed by Reneau et al. [1967] for two-dimensional diffusers. In this particular case the cold (bottom) flow and warm (surface) flow can have different configurations and the classifications must be able to reflect this.

The classification scheme adopted for use is based on flow types. This scheme was selected after extensive observations of flow in the experimental facility. The various categories of flow types are described in Table VI-4 and summarized pictorially in Fig. VI-11. Figure VI-11(a) shows the plunge line configuration as delineated by dye in the warm water. Figure VI-11(b) shows the velocity patterns on the channel bottom as made visible by dye in the cold inflow.
Fig. VI-10. Experimental arrangement of diverging reservoir study.
TABLE VI-4. Description of Flow Types

Flow Type

A   Type A regime exhibits no stalled region.

A1  Type A1 exhibits a symmetrical plunge line in the diffuser downstream from the inlet channel. The inflow stream moves downstream with a symmetric velocity profile over the entire width of the cross section. The velocities of the ambient water moving upstream are also symmetrical.

A2  Type A2 exhibits an asymmetrical plunge line. The velocity profile of the ambient water is also asymmetrical. The underflow is radial after plunging. In the plunge region, however, the inflow stream is skewed to one side.

A3  Type A3 occurs when the inflow stream begins to plunge at the transition from the inlet channel to the diffuser/forming a V shaped plunge line. The upstream flow of the ambient water appears to be symmetrical. The underflow is radial and symmetrical after plunging occurs.

B   The Type B regime exhibits a stalled region. The inflow stream moves into the diverging channel radially, then separates from one side and moves downstream attached to the other wall. This attached flow eventually disappears under the ambient water downstream. The stalled region consists of a large eddy which carries both ambient and entrained inflow water upstream into the diffuser.

B   The Type C regime exhibits a wall jet. This wall jet separates from one wall at the mouth of the inlet channel. The resulting stalled region extends upstream to the inlet channel mouth. The attached wall jet may plunge under the ambient water some distance downstream from the point of separation.

E   The final regime (Type E) enters the diffuser as a free jet flow. Stalled regions of recirculation form on both sides of the jet between the jet and the diffuser walls. The recirculation extends upstream along the diffuser walls to the inlet channel mouth. The jet may plunge under the ambient water some distance downstream from the channel outlet.
Fig. VI-11. Schematic of flow types.
It can be seen that the A type flows (A1, A2 and A3) are all attached flows with no separation occurring. The other flow types all exhibit various degrees of separation or stall.

b. **Experimental results**

The effect of a number of independent variables on the flow types was examined. The variables considered were, the inflow densimetric Froude number, \( F_o = \frac{U_o}{(g' H_o)^{1/2}} \); inflow Reynolds number, \( Re = \frac{U_o H_o}{\nu} \), channel divergence angle, \( \delta \); inflow channel aspect ratio, \( AR = \frac{B_o}{H_o} \) and transition type from parallel to diverging channel. In the above expressions \( H_o \) is the inflow channel depth, \( B_o \) its width, \( U_o \) is the mean inflow velocity and \( g' = g(\rho_o - \rho_a)/\rho_o \) is the reduced acceleration of gravity.

It was found that \( F_o \) and \( \delta \) were the dominant variables in determining the flow type. Figure VI-12 shows a typical plot of the experimental data with \( F_o \) and \( \delta \) as the independent variables. The various symbols indicate the flow type as shown. The lines indicate the boundaries between these flow types.

A number of salient features can be observed on this plot. At high Froude numbers (> 5.0), which is basically a non-buoyant situation with momentum dominant, the flow advances through the categories A, B, C, and E as the side wall divergence angle increases. The flow separates at divergence angles greater than about 4°-7°. At low Froude numbers (< 2.0) flow separation is suppressed by the buoyancy driven lateral expansion and the flow advances through the types A1, A2 and A3 as the side wall angle increases. This is a buoyancy dominated flow.

Different channel aspect ratios gave slightly different locations for the flow type boundaries. Figure VI-13 shows a similar plot to Fig. VI-13 for a small aspect ratio of 0.48. The main difference between the two figures is that with the smaller aspect ratio the lower boundary for the separated flow zone moves to higher \( F_o \) values and larger \( \delta \)'s.

The effect of putting in a rounded transition between outlet channel and diverging channel is shown in Fig. VI-14. The main effect of this change is to move to bigger angles the boundary between flow categories C and E. Physically this arose because the flows attached to one wall (types B and C) tended to follow the rounded transition and remain attached as the divergence angle increased. With the sharp transition, this flow tended to spring off and become a free jet at smaller divergence angles.

Figure VI-12 may be taken as generally indicating the flow type regions in terms of the dominant variables \( F_o \) and \( \delta \). Variation of other factors such as the aspect ratio will move the boundaries of these regions slightly but the overall pattern will remain essentially unchanged. More details on the experiments and on the effect of other parameters are given in Johnson et al. [1985].

The particular kind of flow regime occurring in a diverging channel is of prime importance in determining the plunge region location and the degree of mixing involved. If the flow remains attached (basically any of the A type flows), then the analysis presented in Section II.C.3 should
Fig. VI-12. Flow type classification.
Fig. VI-13. Flow regimes with an aspect ratio of 0.48.

Fig. VI-14. Flow regimes with a rounded transition.
have some relevance to the events occurring. If the flow separates, then jet type phenomena become important and mixing rates can be expected to increase significantly.

3. **Plunge region position**

   As in the sloping reservoir situation, the plunge region position can be given by the value of the densimetric Froude number at the plunge point, $F_p$. The values of $F_p$ found experimentally are examined in this section.

   a. **Separated flow**

   When a separated flow regime exists, a plunge region is difficult, if not impossible to define. Figure VI-15 illustrates the situation when a type B or C flow is occurring. Figure VI-15(a) shows the velocity pattern in the cold flow with the plunge line configuration indicated, i.e. the line made visible by dye in the ambient water. Figure VI-15(b) shows the velocity pattern in the warm ambient water.

   A plunge region by definition is a region where bulk momentum and buoyancy terms are in balance. It is clear from an inspection of Fig. VI-15 that such a region is difficult to define in this case. The incoming cold flow distributes its momentum and mass gradually. The elongated line on the water surface merely indicates a local equilibrium where a warm wedge is held by the laterally moving portion of the cold flow. The most downstream portion of this line represents the point where the inflow momentum is sufficiently dissipated that it can no longer displace the warm water.

   In this flow situation the plunge region position can be defined in many ways. Extensive efforts were made to define a plunge region and correlate the resulting $F_p$ data. Johnson et al. [1986] describe an approach using the most downstream distance of the plunge line to define the plunge region. $F_p$ is then computed in terms of the full channel width, $B_p$, at that location. Figure VI-16 shows the $F_p$ values obtained using this approach plotted against channel divergence angle. This plot illustrates that the $F_p$ values span the range 1.0 to almost zero and is of little use for predicting $F_p$ values. Most of the trend in the data of lower $F_p$ values at higher angles of divergence is introduced by the fact that the physical situations were often of similar extent with different angles but the higher angles gave a larger $B_p$ and thus a smaller $F_p$. The separated flows bear many similarities to jet flows (either wall jets or free jets). Johnson et al. [1986] present an analysis of the separated data using a jet flow model and show that the most downstream portion of the plunge, $X_p$, is correlated well with the expression

   \[
   \frac{X_p}{B_0} = 0.52 \, F_p^4 \quad \text{(VI-2)}
   \]

   b. **Attached flow**

   The flows that remained attached were examined from the point of view of the theoretical results in Section II.C.3. The plunge line position was
Fig. VI-15. Flow situation with a separated flow.
Fig. VI-16. Plunge point densimetric Froude number, $F_p$, as a function of sidewall angle, $\delta$. 

\[ F_p = F_o \frac{B_o}{B_p} \]

\[ B_p = B_o + 2X_p \tan \delta \]
ascertained by introducing red dye in the downstream warm water and noting the upstream position at which the dye stabilized. Local lateral motions were observed in the plunge region in most flows, and the plunge line generally moved about over time though on an average it occupied a fixed position in the flow field.

The only flows which in any way approximated Type II flows occurred at low outlet densimetric Froude numbers (< 2.5) and large sidewall half-angles (> 10°), i.e. flow type A3. Under these conditions the sides of the flow plunged at the outlet but the central portion continued for a distance downstream before plunging so that the plunge line had a V or U shape as shown in Fig. VI-17. This configuration, which is no doubt due to the low velocities at the channel walls and to the inability of the parallel flow to convert instantly to a radial flow, prevented the association of a single $F_p$ value with the plunge region. The width of the diverging channel at the most downstream location of the plunge line was such that the densimetric Froude number there (calculated on the full channel width) was always less than unity. Apart from these flows there were no other indications to suggest the existence of Type II plunging, so it can be stated that there was no clear identification of this flow type.

All the other plunging flows occurred some distance down the diverging channel section. Type A2 flows had a skewed velocity profile (in plan) and a skewed plunge line so that it was not possible to associate a single radial location with the plunge line. Those experiments which had a well defined plunge region with little variation in radial distance to the plunge line over the channel width were selected for comparison with theory.

The data on the selected experiments is given in Table VI-5. The first four experiments in that table were carried out on an 8 ft long beach. The remainder were carried out on a 16 ft beach. The depth of flow was generally about 0.3 ft. The outlet densimetric Froude number and Reynolds number ($\text{Re} = \frac{\text{ud}}{\nu}$, where $u$ is the outflow velocity and $\nu$ the kinematic viscosity) for each experiment are shown in Table VI-5. The dimensionless parameters $\text{Re}_{\text{nd}} F_0^{-1}$ and $\text{G}_{\text{in}} F_0$ are also shown in Table VI-5. The parameter $\text{G}_{\text{in}} F_0$ is given as $\text{G}_{\text{in}} F_0/f$ since the friction factor $f$ is unknown.

The experimental data is plotted in the form $F_p$ against $\text{Re}_{\text{nd}} F_0^{-1}$ in Fig. VI-18. The low and high Reynolds number values (above or below 5000) are indicated by different symbols. The curves from Fig. II-19 corresponding to $\text{G}_{\text{in}} F_0$ values of 0.175 and 0.3 have been drawn through the data points. These correspond to average $f$ values of 0.09 for the higher Re points and 0.14 for the lower Re points. Apart from the single low Re point, the theoretical curves fit the data trends reasonably well though there is considerable scatter in the data particularly at high Reynolds numbers.

The interfaces in individual experiments were examined by numerically integrating Eq. (II-54) using a fourth-order Runge-Kutta scheme from the downstream critical section with the friction factors given by Fig. VI-18. These interfaces fitted well the vertical temperature profiles measured at locations downstream of the plunge line.
Fig. VI-17. Plunge region configuration in a type A3 flow.
<table>
<thead>
<tr>
<th>Experiment Date</th>
<th>Sidewall Divergence Angle</th>
<th>Inflow Channel Width</th>
<th>Densimetric Froude Nr. $F_o$</th>
<th>Reynolds Number, $Re$</th>
<th>Densimetric Froude Nr at Plunge Region, $F_p$</th>
<th>Dimensionless Parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>(*)</td>
<td>(mm)</td>
<td>(-)</td>
<td>(-)</td>
<td>(-)</td>
<td>(-)</td>
</tr>
<tr>
<td>12/21/83</td>
<td>4</td>
<td>704</td>
<td>0.58</td>
<td>1310</td>
<td>.47</td>
<td>2.6</td>
</tr>
<tr>
<td>1/31/84</td>
<td>4</td>
<td>173</td>
<td>0.98</td>
<td>1518</td>
<td>.47</td>
<td>2.9</td>
</tr>
<tr>
<td>2/3/84</td>
<td>4</td>
<td>173</td>
<td>1.01</td>
<td>2738</td>
<td>.53</td>
<td>2.9</td>
</tr>
<tr>
<td>2/10/84</td>
<td>4</td>
<td>173</td>
<td>0.85</td>
<td>2005</td>
<td>.46</td>
<td>3.4</td>
</tr>
<tr>
<td>5/8/84</td>
<td>4</td>
<td>173</td>
<td>1.13</td>
<td>2684</td>
<td>.43</td>
<td>4.3</td>
</tr>
<tr>
<td>5/11/84</td>
<td>4</td>
<td>173</td>
<td>1.18</td>
<td>1928</td>
<td>.36</td>
<td>4.1</td>
</tr>
<tr>
<td>5/15/84</td>
<td>4</td>
<td>173</td>
<td>1.37</td>
<td>3844</td>
<td>.46</td>
<td>3.6</td>
</tr>
<tr>
<td>5/22/84</td>
<td>4</td>
<td>173</td>
<td>1.06</td>
<td>2873</td>
<td>.41</td>
<td>4.6</td>
</tr>
<tr>
<td>6/22/84</td>
<td>4</td>
<td>173</td>
<td>1.26</td>
<td>5500</td>
<td>.59</td>
<td>3.9</td>
</tr>
<tr>
<td>7/24/84</td>
<td>2</td>
<td>173</td>
<td>1.33</td>
<td>12000</td>
<td>.83</td>
<td>2.2</td>
</tr>
<tr>
<td>7/24/84</td>
<td>3</td>
<td>173</td>
<td>1.31</td>
<td>12000</td>
<td>.77</td>
<td>3.0</td>
</tr>
<tr>
<td>7/24/84</td>
<td>4</td>
<td>173</td>
<td>1.43</td>
<td>12000</td>
<td>.87</td>
<td>3.5</td>
</tr>
<tr>
<td>8/8/84</td>
<td>4</td>
<td>173</td>
<td>1.86</td>
<td>12800</td>
<td>.78</td>
<td>2.6</td>
</tr>
<tr>
<td>8/9/84</td>
<td>4</td>
<td>173</td>
<td>2.51</td>
<td>12800</td>
<td>.79</td>
<td>2.0</td>
</tr>
<tr>
<td>8/9/84</td>
<td>3</td>
<td>173</td>
<td>2.45</td>
<td>12800</td>
<td>.79</td>
<td>1.6</td>
</tr>
<tr>
<td>8/24/84</td>
<td>4</td>
<td>173</td>
<td>2.73</td>
<td>12500</td>
<td>.68</td>
<td>1.8</td>
</tr>
<tr>
<td>8/27/84</td>
<td>4</td>
<td>173</td>
<td>1.85</td>
<td>12500</td>
<td>.84</td>
<td>2.7</td>
</tr>
<tr>
<td>8/27/84</td>
<td>7</td>
<td>173</td>
<td>1.90</td>
<td>12000</td>
<td>.68</td>
<td>4.2</td>
</tr>
<tr>
<td>10/19/84</td>
<td>5</td>
<td>45</td>
<td>2.25</td>
<td>12000</td>
<td>.56</td>
<td>9.0</td>
</tr>
<tr>
<td>10/24/84</td>
<td>5</td>
<td>45</td>
<td>3.18</td>
<td>11000</td>
<td>.55</td>
<td>6.3</td>
</tr>
<tr>
<td>11/5/84</td>
<td>5</td>
<td>45</td>
<td>5.0</td>
<td>9100</td>
<td>.54</td>
<td>4.0</td>
</tr>
<tr>
<td>11/5/84</td>
<td>7</td>
<td>45</td>
<td>4.9</td>
<td>8700</td>
<td>.68</td>
<td>5.7</td>
</tr>
<tr>
<td>1/9/85</td>
<td>3</td>
<td>45</td>
<td>4.8</td>
<td>8700</td>
<td>.83</td>
<td>2.6</td>
</tr>
<tr>
<td>1/11/85</td>
<td>3</td>
<td>45</td>
<td>1.5</td>
<td>20600</td>
<td>.64</td>
<td>8.4</td>
</tr>
<tr>
<td>2/13/85</td>
<td>3</td>
<td>45</td>
<td>4.1</td>
<td>3250</td>
<td>.62</td>
<td>3.1</td>
</tr>
<tr>
<td>2/25/85</td>
<td>3</td>
<td>45</td>
<td>3.9</td>
<td>10500</td>
<td>.68</td>
<td>3.2</td>
</tr>
<tr>
<td>4/10/85</td>
<td>3</td>
<td>45</td>
<td>2.1</td>
<td>11000</td>
<td>.63</td>
<td>6.0</td>
</tr>
<tr>
<td>10/2/85</td>
<td>0.5</td>
<td>45</td>
<td>1.4</td>
<td>9217</td>
<td>.64</td>
<td>2.1</td>
</tr>
<tr>
<td>10/2/85</td>
<td>1.5</td>
<td>45</td>
<td>1.4</td>
<td>9242</td>
<td>.57</td>
<td>4.9</td>
</tr>
<tr>
<td>10/2/85</td>
<td>3.1</td>
<td>45</td>
<td>1.5</td>
<td>9838</td>
<td>.60</td>
<td>8.9</td>
</tr>
<tr>
<td>10/4/85</td>
<td>1.4</td>
<td>173</td>
<td>1.4</td>
<td>9586</td>
<td>.82</td>
<td>2.7</td>
</tr>
<tr>
<td>10/4/85</td>
<td>2.1</td>
<td>173</td>
<td>1.4</td>
<td>9586</td>
<td>.68</td>
<td>2.2</td>
</tr>
<tr>
<td>10/4/85</td>
<td>3.1</td>
<td>173</td>
<td>1.4</td>
<td>9586</td>
<td>.68</td>
<td>2.9</td>
</tr>
</tbody>
</table>
Fig. VI-18. $F_p$ variation in attached flows.
Fig. VI-19. Initial mixing coefficients, \( \gamma \), as a function of sidewall angle, \( \delta \).
The friction factors quoted above appear high. The values may be influenced by sidewall friction and by entrainment. Furthermore a constant $f$ was assumed in the theoretical development. This may not be a good assumption as both $Re$ and the layer densimetric Froude numbers vary along the flow. The bottom friction factor is of course a function of the Reynolds number. Demissis and Partheniades [1984] show that the interfacial friction factor is a function of the Reynolds number and the layer densimetric Froude number.

4. **Initial mixing values**

Details on the initial mixing measurements are presented in Johnson et al. [1986]. The results are summarized here.

The total flow downstream of the plunge region (either separated or non-separated) was determined by direct measurement of the vertical velocity profiles. (A simplified flow measurement method, based on a two-layer flow approach, was in fact tested and was found to yield reasonable results particularly at low inflow densimetric Froude numbers.) At a section 0.5 ft from the beach end profiles were measured at a number of points across the channel width. These profiles were integrated vertically to yield local flow rates for ambient (proceeding upstream) and mixed (proceeding downstream) water. These local rates were then integrated across the channel width to yield total flow rates of ambient and mixed water.

The mixed flow rate, $Q_m$, consists of the flow rate just downstream of the plunge region, $Q_p$, and the flow due to layer entrainment between the plunge region and the measuring section. In the case of separated flow the plunge region was defined as the most downstream point of the plunge line as delineated by dye in the ambient water. Hence, in this case all mixing due to the separated flow was lumped together as initial mixing.

The layer entrainment flow was estimated using Equation (11-21) with the local layer densimetric Froude number of the lower layer and in most cases was found to be very small. This flow was then subtracted from $Q_m$ to yield $Q_p$. Having found $Q_p$, the initial mixing coefficient, $\gamma$, was computed as

$$\gamma = \frac{Q_p}{Q_o} - 1$$

where $Q_o$ is the river inflow rate.

Details of the experiments and the calculated initial mixing coefficient are shown in Table VI-6. Values of the initial mixing coefficient, $\gamma$, are shown plotted against channel side divergence angle in Fig. VI-19.

It can be seen that at small angles of divergence the initial mixing values are low and of the same order of magnitude as found in the sloping reservoir. As the sidewall angle increases, mixing increases markedly. Most of these high mixing values are associated with jet type separated flow phenomena.
### TABLE VI-6. Details of Initial Mixing Experiments in the Diverging Reservoir

<table>
<thead>
<tr>
<th>Experiment Date</th>
<th>Inflow Denisometric Froude Number, $F_o$</th>
<th>Inflow Reynolds Number, Re</th>
<th>Inflow Channel Aspect Ratio, AR</th>
<th>Sidewall Divergence Angle, $\theta$ (°)</th>
<th>Inflow Discharge, $Q_o$ (ft$^3$/sec)</th>
<th>Downstream Measured Layer Discharge, $Q_m$ (ft$^3$/sec)</th>
<th>Distance to Plunging, $X_p$ (ft)</th>
<th>Computed layer discharge at Plunge region, $Q_p$ (ft$^3$/sec)</th>
<th>Initial Mixing Coeff., $Y$</th>
<th>Flow Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>2/13/85</td>
<td>3.86</td>
<td>10,500</td>
<td>.47</td>
<td>3</td>
<td>.024</td>
<td>.0275</td>
<td>7.5</td>
<td>.0257</td>
<td>0.07</td>
<td>A1</td>
</tr>
<tr>
<td>4/10/85</td>
<td>2.10</td>
<td>11,000</td>
<td>.48</td>
<td>3</td>
<td>.0215</td>
<td>.0345</td>
<td>3.5</td>
<td>.0281</td>
<td>0.34</td>
<td>A1</td>
</tr>
<tr>
<td>9/18/85</td>
<td>3.99</td>
<td>10,300</td>
<td>.46</td>
<td>10</td>
<td>.017</td>
<td>.0267</td>
<td>3.5</td>
<td>.0243</td>
<td>0.46</td>
<td>C</td>
</tr>
<tr>
<td>9/25/85</td>
<td>1.88</td>
<td>11,200</td>
<td>.46</td>
<td>10</td>
<td>.019</td>
<td>.0370</td>
<td>3.5</td>
<td>.0338</td>
<td>0.78</td>
<td>C</td>
</tr>
<tr>
<td>4/01/85</td>
<td>3.87</td>
<td>10,500</td>
<td>.48</td>
<td>20</td>
<td>.021</td>
<td>.0603</td>
<td>11</td>
<td>.0583</td>
<td>1.77</td>
<td>C</td>
</tr>
<tr>
<td>12/10/85</td>
<td>2.75</td>
<td>10,100</td>
<td>.47</td>
<td>20</td>
<td>.0205</td>
<td>.0445</td>
<td>4.75</td>
<td>.0504</td>
<td>1.40</td>
<td>A2/A3</td>
</tr>
<tr>
<td>4/03/85</td>
<td>1.97</td>
<td>10,200</td>
<td>.48</td>
<td>20</td>
<td>.021</td>
<td>.0682</td>
<td>1.3</td>
<td>.0572</td>
<td>1.86</td>
<td>E/C</td>
</tr>
<tr>
<td>10/16/85</td>
<td>3.40</td>
<td>10,270</td>
<td>.48</td>
<td>45</td>
<td>.020</td>
<td>.0584</td>
<td>7.5</td>
<td>.0572</td>
<td>1.19</td>
<td>C</td>
</tr>
<tr>
<td>12/18/85</td>
<td>3.40</td>
<td>9,922</td>
<td>.47</td>
<td>45</td>
<td>.020</td>
<td>.0449</td>
<td>3.13</td>
<td>****</td>
<td>1.82</td>
<td>E</td>
</tr>
<tr>
<td>12/12/85</td>
<td>2.74</td>
<td>10,100</td>
<td>.47</td>
<td>45</td>
<td>.020</td>
<td>.0734</td>
<td>3.25</td>
<td>****</td>
<td>2.67</td>
<td>E/A3</td>
</tr>
<tr>
<td>12/17/85</td>
<td>1.97</td>
<td>10,200</td>
<td>.48</td>
<td>45</td>
<td>.0205</td>
<td>.0498</td>
<td>.87</td>
<td>****</td>
<td>1.43</td>
<td>A2</td>
</tr>
<tr>
<td>4/16/85</td>
<td>3.81</td>
<td>10,000</td>
<td>.48</td>
<td>90</td>
<td>.0205</td>
<td>.0635</td>
<td>12.0</td>
<td>.0627</td>
<td>1.99</td>
<td>E</td>
</tr>
<tr>
<td>12/04/85</td>
<td>2.74</td>
<td>9,980</td>
<td>.47</td>
<td>90</td>
<td>.0205</td>
<td>.0915</td>
<td>4.5</td>
<td>****</td>
<td>2.10</td>
<td>E</td>
</tr>
<tr>
<td>12/04/85*</td>
<td>2.72</td>
<td>10,000</td>
<td>.47</td>
<td>90</td>
<td>.0205</td>
<td>.0914</td>
<td>4.25</td>
<td>****</td>
<td>3.46</td>
<td>E</td>
</tr>
<tr>
<td>11/19/85</td>
<td>2.72</td>
<td>9,550</td>
<td>.46</td>
<td>90</td>
<td>.0215</td>
<td>.0786</td>
<td>1.5</td>
<td>1.0725</td>
<td>1.43</td>
<td>E</td>
</tr>
<tr>
<td>4/12/85</td>
<td>2.01</td>
<td>10,200</td>
<td>.48</td>
<td>90</td>
<td>.021</td>
<td>.0786</td>
<td>1.5</td>
<td>****</td>
<td>2.43</td>
<td>E</td>
</tr>
<tr>
<td>11/19/85*</td>
<td>1.80</td>
<td>9,500</td>
<td>.46</td>
<td>90</td>
<td>.0213</td>
<td>.0829</td>
<td>.75</td>
<td>****</td>
<td>2.83</td>
<td>E</td>
</tr>
<tr>
<td>4/19/85</td>
<td>1.98</td>
<td>10,600</td>
<td>1.81</td>
<td>3</td>
<td>.081</td>
<td>.084</td>
<td>12.5</td>
<td>.082</td>
<td>0.01</td>
<td>A2</td>
</tr>
<tr>
<td>4/24/85</td>
<td>1.79</td>
<td>10,100</td>
<td>1.78</td>
<td>3</td>
<td>.072</td>
<td>.0772</td>
<td>10.0</td>
<td>.075</td>
<td>0.04</td>
<td>A2</td>
</tr>
<tr>
<td>6/18/85</td>
<td>2.58</td>
<td>10,200</td>
<td>1.77</td>
<td>11</td>
<td>.064</td>
<td>.085</td>
<td>10.0</td>
<td>.083</td>
<td>0.28</td>
<td>C</td>
</tr>
<tr>
<td>6/12/85</td>
<td>1.70</td>
<td>10,200</td>
<td>1.79</td>
<td>11</td>
<td>.064</td>
<td>.085</td>
<td>4.0</td>
<td>.089</td>
<td>0.41</td>
<td>A2</td>
</tr>
<tr>
<td>5/08/85</td>
<td>1.78</td>
<td>10,600</td>
<td>1.79</td>
<td>10</td>
<td>.076</td>
<td>.1205</td>
<td>9.5</td>
<td>.1156</td>
<td>0.58</td>
<td>C</td>
</tr>
<tr>
<td>4/29/85</td>
<td>2.30</td>
<td>10,100</td>
<td>1.76</td>
<td>20</td>
<td>.073</td>
<td>.1243</td>
<td>3.5</td>
<td>.1101</td>
<td>0.51</td>
<td>E</td>
</tr>
<tr>
<td>9/22/85</td>
<td>2.30</td>
<td>9,900</td>
<td>1.76</td>
<td>45</td>
<td>.055</td>
<td>.0964</td>
<td>11.5</td>
<td>.0955</td>
<td>0.74</td>
<td>A3</td>
</tr>
<tr>
<td>8/02/85</td>
<td>1.80</td>
<td>10,700</td>
<td>1.79</td>
<td>45</td>
<td>.057</td>
<td>.111</td>
<td>3.0</td>
<td>.106</td>
<td>0.86</td>
<td>A3</td>
</tr>
<tr>
<td>7/03/85</td>
<td>1.67</td>
<td>10,000</td>
<td>1.76</td>
<td>45</td>
<td>.059</td>
<td>.111</td>
<td>2.5</td>
<td>.111</td>
<td>0.87</td>
<td>A3</td>
</tr>
<tr>
<td>5/13/85</td>
<td>1.75</td>
<td>10,300</td>
<td>1.79</td>
<td>90</td>
<td>.069</td>
<td>.1290</td>
<td>6.5</td>
<td>.1268</td>
<td>0.84</td>
<td>E</td>
</tr>
</tbody>
</table>

* Downstream constraint removed.

**** Layer flow correction ignored.
The highest initial mixing values on Fig. VI-19 arose when part of the flow confining walls downstream of the plunge region were removed. This relaxation in confinement appeared to allow more ambient water to flow upstream. Once the flow has separated (Type B or C) further increase in the sidewall angle has little effect on the initial mixing. The inflow densimetric Froude number does not appear to influence initial mixing strongly. When the flow is attached, inflow geometry as expressed by the inflow channel aspect ratio, AR, is not important. Once the flow separates, inflow geometry becomes important.

A surprising feature of the data is the relatively high mixing exhibited by the Type A3 flows. These are non-separated flows and low two-dimensional type initial mixing would be expected. The reason for the high mixing is not known. One possible source is the eddying motion set up in the warm water by the V-shaped plunge line.

D. Closure

The data from the sloping bottom reservoir indicate that plunging occurs at an average densimetric Froude number value of about 0.69. This finding is in line with the experimental results of Singh and Shah [1971]. This result refers to steep slopes. It was not possible to examine mild slopes in the experimental facility. Initial mixing rates were found to be small, and the data generally confirms the numerically generated relationship for initial mixing.

In the diverging reservoir, flow separation was found to be the dominant phenomenon. The flows were categorized into various broad types which were a function mainly of the side divergence angle, $\delta$, and the inflow densimetric Froude number, $F_o$. The plunge point densimetric Froude numbers in the attached flows followed the general trends of the theoretical predictions. Initial mixing rates in the attached flow types A1 and A2 were of the same order of magnitude as the sloping reservoir data. At high side divergence angles, mixing increased rapidly and was influenced primarily by local geometry and downstream conditions.
VII. SUMMARY AND CONCLUSIONS

The considerations in this report go some way, it is hoped, in advancing the understanding of the plunging flow phenomenon.

Various basic ideas have been clarified in Chapter II and a new theory has been presented for plunging flows in reservoirs with diverging walls. The considerations in this chapter indicate that the densimetric Froude number at the plunge point does not have a single point unique value but depends on individual reservoir downstream conditions. The idea is advanced that in non-separated flow the initial mixing is a function of downstream normal densimetric Froude number only. It is argued that use of the cooling pond relation of Jirka and Watanabe [1980] to estimate initial mixing is inappropriate.

The plunging phenomenon has been successfully modelled using a laminar flow model. It has been demonstrated that while the laminar model can qualitatively reproduce the features of a plunging flow field in both sloping and diverging reservoirs, bottom flow separation arising from purely physical reasons render the model results inapplicable to prototype situations. Laminar, or constant viscosity, flow models are much used in general reservoir modelling. The discussions in Chapter IV may help to highlight the inherent limitations in such models.

In Chapter V the k-ε turbulence model was used to construct a reservoir flow model with a variable eddy viscosity. While numerical diffusion was a problem in the model, the basic physics appeared to be well reproduced. This model yielded some useful insights into the development, growth and decay of a plunging flow field. Specific relations were derived for predicting the plunge point position and the initial mixing coefficient. The plunge data suggest that plunging occurs at a densimetric Froude number of about 0.5. The model confirms that initial mixing is independent of inflow parameters and varies only with the normal densimetric Froude number, $F_N$, of the underflow. A relation linking the initial mixing coefficient to $F_N$ is presented.

The model developed in Chapter VI is a general reservoir flow model applicable far beyond the specific use it has been put to here. Some suggestions are put forward as to how the model could be improved and used to study other reservoir flow situations.

The results of an extensive experimental program are presented in summary form in Chapter VI. Only steep slopes could be examined in the sloping reservoir study. The experimental data for the plunge point fell in the same range as the data of Singh and Shah [1971] and gave an average densimetric Froude number at the plunge point of 0.69. This simple relation is recommended for predicting plunge point position on steep slopes. A careful field study is probably required to study plunging on mild slopes. Initial mixing measurements confirmed the general order of magnitude
of the numerically generated relationship. The numerically generated relationship as given by Eq. (V-20) is thus recommended for initial mixing estimation in reservoirs in which the flow does not separate.

In the diverging reservoir experiments, flow separation was found to be the dominant phenomenon. The flows in diverging reservoirs were categorized according to defined flow types. Figure VI-12 is a useful summary chart for flow type categorization and shows that inflow densimetric Froude number and sidewall divergence angle are the important parameters.

If the reservoir sidewall divergence angle is greater than 4 or 5 degrees, and the inflow densimetric Froude number is greater than about 2.0, flow separation will occur. Initial mixing in this case will be a function of inflow geometry and downstream conditions but values of order 1.0 to 2.0 can be expected. The separated inflow will behave as a free or attached jet and Eq. VI-2 can be used to estimate the maximum distance the jet will penetrate before plunging.

If the reservoir sidewall angle is greater than about 10 degrees, and the inflow densimetric Froude number is less than 2.0, then type A3 plunging may occur close to the inflow point. Initial mixing in this case will be of the same order of magnitude as for separated inflows.

For small sidewall divergence angles (< 4.0 or 5.0 degrees) the flow remains attached and is quite similar to flow in a parallel sided reservoir. Initial mixing values may be estimated using the relation for the sloping reservoir situation. If the total friction factor can be estimated, Fig. II-19 can be used to estimate \( \beta \) and thus locate the plunge line in this case.
REFERENCES


Forel, F. A. (1892), Le Lèman: Monographie Limnologique, Vol. 1, F. Rouge, Lausanne, Switzerland.


APPENDIX 1

THE k-ε MODEL

1. Introduction
2. The k equation
   2.1. The exact equation
   2.2. Equation simplifications
3. The ε equation
   3.1. The exact equation
   3.2. Equation simplifications
4. Summary
1. Introduction

In recent years many models have appeared which can compute the variable Reynolds stress $u'u'_{ij}$. These models are called turbulence models but they really make no attempt to model the flow turbulence. Rather they simulate the effect of turbulence on a mean flow. The models all draw heavily on the basic understanding of turbulence that has emerged from the study of isotropic turbulence initiated by Taylor [1935]. The best known of these models, the $k$-$\varepsilon$ model was developed in the early 1970's and is well described and discussed by Rodi [1980].

In the $k$-$\varepsilon$ model, the Reynolds stress is computed using the eddy viscosity idea of Boussinesq [1877] i.e.

$$-u'_i u'_j = \nu_t \left( \frac{\delta u'_i}{\delta x_j} + \frac{\delta u'_j}{\delta x_i} \right) - \frac{2}{3} \kappa \delta_{ij} \quad (A1-1)$$

The second part of the expression is uncommon and is introduced to ensure that the contracted form of the expression reduces to the kinetic energy per unit mass, $k$, where

$$k = \frac{1}{2}(u'_1^2 + u'_2^2 + u'_3^2)$$

The eddy viscosity, $\nu_t$, is calculated from the expression

$$\nu_t = c'_\mu \sqrt{k} L \quad (A1-2)$$

suggested by Kolmogorov [1942] and Prandtl [1945]. The coefficient $c'_\mu$ is an empirical constant and $L$ is a turbulent length scale. If the rate of turbulent energy dissipation per unit mass, $\varepsilon$, is known, then dimensional considerations show that

$$L = \frac{k^{3/2}}{\varepsilon} \quad (A1-3)$$

where $c_d$ is an empirical constant. Combination of Equations (A1-2) and (A1-3) yields

$$\nu_t = \frac{k^2}{\mu \varepsilon} \quad (A1-4)$$

where $c$ is a new empirical coefficient incorporating $c'_\mu$ and $c_d$. The turbulent kinetic energy and dissipation rate $\varepsilon$ are obtained from differential equations for those quantities. The equations are discussed below.
2. The $k$ equation

2.1. Exact equation

The equation governing the kinetic energy of the turbulent velocity fluctuations can be obtained by two methods. In the first method the equation is obtained by multiplying the equation for the fluctuating velocity, $u'_i$, by $u'_j$. In the second method the equation is obtained by subtracting the mean flow energy equation from the total flow energy equation. The equation resulting from the application of either method is

$$
\frac{\delta}{\delta t} \left( \frac{u'_i u'_j}{2} \right) + u'_j \frac{\delta}{\delta x_j} \left( \frac{u'_i u'_j}{2} \right) = -\frac{1}{\rho} \frac{\delta}{\delta x_i} \left( p' u'_j \right) - \frac{\delta}{\delta x_j} \left( u'_i \right) \frac{u'_i u'_j}{2} + \nu \frac{\delta}{\delta x_j} \left( u'_i \left( \frac{\delta u'_j}{\delta x_j} + \frac{\delta u'_j}{\delta x_i} \right) \right)
$$

--Group 1--

$$
- \frac{u'_i u'_j}{\delta x_j} + \nu \frac{\delta u'_i}{\delta x_j} \left( \frac{\delta u'_j}{\delta x_j} + \frac{\delta u'_j}{\delta x_i} \right)
$$

--Group 2--

$$
- \beta g_i \frac{u'_i T'}{\delta x_j}
$$

--Group 3--

$$
- \beta g_i \frac{u'_i T'}{\delta x_j}
$$

--Group 4--

Equation (A1-5) is an exact equation derived directly from the Navier-Stokes equation. The equation must be simplified before it can be of use in practical situations. The simplifications introduced are set out below.

2.2. Equation simplification

The complex correlations appearing the Group 1 are removed by considering the turbulence and pressure terms together and simulating their effect with a simple gradient diffusion model. Thus, the group
The laminar part of the diffusion expression is usually neglected.

The buoyancy term is simplified by relating the velocity/temperature correlation, $u'_i T'$, to the local temperature gradient. Thus, $u'_i T'$, is replaced by $-\Gamma \delta T / \delta x_j$, where $\Gamma$ is the turbulent diffusivity of heat. The turbulent diffusivity is usually written in the form $\nu_t / \sigma_t$, where $\sigma_t$ is the turbulent Prandtl number. Thus, the buoyancy term becomes

$$+ g_1 \beta \frac{\nu_t}{\sigma_t} \frac{\delta T}{\delta x_j}$$

The energy dissipation term does not have to be modelled as in the $k-\varepsilon$ model a separate differential equation is solved for this quantity. Thus,
Using all the above expressions the final \( \varepsilon \) equation is, with \( \varepsilon = \frac{u''_1 u''_1}{2} \):

\[
\frac{\delta k}{\delta t} + u_j \frac{\delta k}{\delta x_j} = \frac{\delta}{\delta x_j} \left[ (v + \nu_t \frac{\delta}{\delta x_j}) \frac{\delta k}{\delta x_j} \right] + \nu_t \left( \frac{\delta u''_i}{\delta x_j} + \frac{\delta u''_j}{\delta x_i} \right) + \frac{\nu_t}{\sigma_t} \frac{\delta T}{\delta x_i} - \varepsilon \tag{A1-6}
\]

The assumptions used in developing this equation from the exact equation are modest and physically reasonable. Thus, it can be expected that the \( \varepsilon \) equation will behave well in use.

3. The \( \varepsilon \) equation

3.1. The exact equation

The exact expression for the rate of turbulent energy dissipation (per unit mass) is

\[
\varepsilon = v \frac{\delta u''_i}{\partial x_j} \left( \frac{\delta u''_j}{\partial x_i} + \frac{\delta u''_i}{\partial x_j} \right)
\]

When the Reynolds number of the flow is high, most of the energy dissipation takes place at very small length scales and the dissipation may be approximated by

\[
\varepsilon = v \frac{\delta u''_i}{\partial x_j} \frac{\delta u''_j}{\partial x_i} \tag{A1-7}
\]

The quantity on the right hand side of equation (A1-7) multiplied by the viscosity is the fluctuating vorticity. An equation for this quantity may be derived from the vorticity equation of the flow. Tennekes and Lumley [1972] derive this equation and discuss its various terms. The equation for
may also be obtained directly from the equation for the fluctuating velocity, \( u'_1 \), by differentiating this equation with respect to \( x'_2 \) and then multiplying by \( \delta u'_1 / \partial x'_2 \). On utilizing this latter approach and taking a mean the resulting equation is

\[
\frac{\delta u'_1 \delta u'_1}{\partial x'_2 \partial x'_2} \cdot
\]

\[
\delta \left[ \frac{\delta u'_1^2}{\partial x'_2^2} \right] + \bar{u}_{ij} \frac{\partial}{\partial x'_j} \left[ \frac{\delta u'_1^2}{\partial x'_2^2} \right] = -2 \frac{\delta^2}{\partial x'_2 \partial x'_j} (u'_{ij} \frac{\delta u'_1}{\partial x'_i}) - 2 \frac{\delta}{\partial x'_j} \left[ \frac{\delta u'_1}{\partial x'_2} \frac{\delta u'_1}{\partial x'_j} \right] - 2 \frac{\delta u'_1}{\partial x'_j} \frac{\delta u'_1}{\partial x'_2} \frac{\delta u'_1}{\partial x'_2}
\]

\[
\text{Group 1} \quad \text{Group 2} \quad \text{Group 3} \quad \text{Group 4} \quad \text{Group 5}
\]

This equation has been discussed by Daly and Harlow [1970].

The terms in this equation can be grouped as shown. The terms in Group 1 govern interactions between the mean flow field and the turbulence. The single term in group 2 represents production or addition to the moment by vortex stretching. The terms in group 3 represent the transport of the moment by pressure and turbulent and viscous diffusion. Term 4 is the viscous sink term and term 5 represents the effect of buoyancy.
Equation (A1-B) is an exact equation. It contains many complex higher order moments. These must be removed by modelling assumptions before this equation can be of use in practical situations. The simplifications introduced in this equation are considered below.

3.2. Equation simplifications

When the Reynolds number is high, it is considered that the dominant source mechanism for the moment

\[
\frac{\delta u'_i \delta u'_j}{\delta x_k \delta x_l}
\]

is that of vortex stretching and that the contribution of the mean flow interaction terms is negligible. Thus, as a first simplification, the terms in group 1 of Eq. (A1-B) are discarded.

The diffusion terms making up group 3 are treated in the same manner as the analogous terms in the \( k \) equation, i.e. the turbulent and pressure terms are modelled as a group by a gradient expression. The resulting diffusion expression is

\[
\frac{\delta}{\delta x_j} \left( (\nu + \frac{\nu}{\sigma} \frac{\delta \varepsilon}{\delta x_j}) \right)
\]

where \( \sigma \) is an empirical coefficient. The laminar part of the diffusion expression is usually neglected. Rodi [1971] argues that the vortex stretching term 2 and the viscous destruction term 4 should not be considered separately; only their difference needs to be modelled. Hanjalic and Launder [1972] and Launder, Reece and Rodi [1975] suggest that these terms, i.e.

\[
-2 \frac{\delta u'_i \delta u'_j \delta u'_l}{\delta x_k \delta x_l \delta x_j} - 2\nu \left[ \frac{\delta u'_i}{\delta x_k \delta x_l} \right]^2
\]

be replaced by the expression

\[
C_{\epsilon} \frac{\epsilon}{k} P - C_{\epsilon} \frac{\epsilon^2}{k}
\]

where \( P \) is the production term in the \( k \) equation and \( C_{\epsilon1}, C_{\epsilon2} \) are empirical coefficients.

Treatment of the buoyancy term 5 stems from the suggestion of Daly and Harlow (1970) that the behavior of the tensor
be approximated by that of $u'T'$. Thus the term

$$- 2g_1 \beta \frac{\delta u_i}{\delta x_i} \frac{\delta T}{\delta x_i}$$

is replaced by

$$- g_1 \beta \frac{\nu}{\sigma} C_{1e} C_3 \frac{\varepsilon}{k} T'u'_i$$

and, using the expression for $T'u'_i$ in the $k$ equation, this becomes

$$+ \left( g_1 \beta \frac{\nu}{\sigma} \frac{\delta T}{\delta x_i} \right) C_{1e} C_3 \frac{\varepsilon}{k}$$

The coefficient combinations used with this term vary in the literature. A more sophisticated formulation of the buoyancy term involving a flux Richardson number is discussed by Rodi [1980].

Using all the above expressions the final $\varepsilon$ equation becomes

$$\frac{\delta \varepsilon}{\delta t} + \frac{\nu}{\sigma} \frac{\delta \varepsilon}{\delta x_j} = \frac{\delta}{\delta x_j} \left( \frac{\nu}{\sigma} \frac{\delta \varepsilon}{\delta x_j} \right)$$

$$+ C_{1e} \frac{\varepsilon}{k} p - C_{2e} \frac{\varepsilon^2}{k} + C_{1e} C_3 \frac{\varepsilon}{k} \left( g_1 \beta \frac{\nu}{\sigma} \frac{\delta T}{\delta x_i} \right)$$

The simplifications required to reduce the $\varepsilon$ equation to manageable form are much more severe than those required on the $k$ equation. In fact the final $\varepsilon$ equation bears little resemblance to the original exact equation. The $\varepsilon$ equation could have been written down ab initio with a standard diffusion term and a simple sink-source formulation.
4. Summary

In summary, the \( k-\varepsilon \) model consists of the following set of equations:

The Reynolds stress equation

\[
-u_i'u_j' = \nu_t \left( \frac{\delta u_i}{\delta x_j} + \frac{\delta u_j}{\delta x_i} \right) - \frac{2}{3} k \delta_{ij}
\]

The eddy-viscosity equation

\[
\nu_t = \frac{k^2}{\mu \varepsilon}
\]

and the \( k \) and \( \varepsilon \) equations

\[
\frac{\delta k}{\delta t} + u_j \frac{\delta k}{\delta x_j} = \frac{\delta}{\delta x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_k} \right) \frac{\delta k}{\delta x_j} \right] + P + G - \varepsilon
\]

\[
\frac{\delta \varepsilon}{\delta t} + u_j \frac{\delta \varepsilon}{\delta x_j} = \frac{\delta}{\delta x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \frac{\delta \varepsilon}{\delta x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} P - C_{2\varepsilon} \frac{\varepsilon^2}{k} + C_{1\varepsilon} C_3 \frac{\varepsilon}{k} G
\]

where

\[
P = \nu_t \left( \frac{\delta u_i}{\delta x_j} + \frac{\delta u_j}{\delta x_i} \right) \frac{\delta u_i}{\delta x_j}
\]

is the turbulent kinetic energy production and

\[
G = g_{i\beta} \frac{\nu_t}{\sigma_t} \frac{\delta T}{\delta x_i}
\]

is the buoyancy energy source or sink term.

The model contains the six coefficients \( C_\mu, \sigma_k, \sigma_\varepsilon, C_{1\varepsilon}, C_{2\varepsilon}, \) and \( C_3 \). Apart from \( C_3 \), values have been allocated to these coefficients by reference to standard simple flow situations and by computer optimization as outlined by Rodi [1980]. The constant values given by Launder and Spalding [1974] have become the standard ones and are given in Table A1.1. The values of the quantities \( C_\mu, \sigma_t \), and \( C_3 \) are discussed in the main text.
<table>
<thead>
<tr>
<th>$c_\mu$</th>
<th>$c_k$</th>
<th>$c_\varepsilon$</th>
<th>$c_{1\varepsilon}$</th>
<th>$c_{2\varepsilon}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.09</td>
<td>1.0</td>
<td>1.3</td>
<td>1.44</td>
<td>1.92</td>
</tr>
</tbody>
</table>
APPENDIX 2

BASIC RELATIONSHIPS IN CYLINDRICAL COORDINATES

1. Introduction

2. Momentum Equations in Cartesian Coordinates

3. Momentum Equations in Cylindrical Coordinates
   3.1. General three-dimensional equations
   3.2. Equations in polar \((r, \theta)\) coordinates
   3.3. Equations in axisymmetric \((r, z)\) coordinates

4. Temperature Equations
   4.1. Equation in polar \((r, \theta)\), coordinates
   4.2. Equation in axisymmetric \((r, z)\) coordinates

5. Energy production

6. Buoyancy effects
1. Introduction

The equations in the polar \( r,\theta \) system and in the axisymmetric \( r,z \) system are subsets of the general three-dimensional equations in the cylindrical \( r,\theta,z \) coordinate system. In this appendix the general three-dimensional equations are examined. This avoids the duplication of the \( r \) terms which would occur in separate examination of the \( r,\theta \) and \( r,z \) systems. The \( r,\theta \) and \( r,z \) equations are recovered from the final form of the general equation.

Various types of terms appear in the cylindrical situation which may obscure the essential simple steps used in arriving at the various equations. Before going on to consider the cylindrical case, the various procedures involved are outlined in a cartesian situation with the compact tensor notation. This is effectively a summary of the procedures involved. Incompressible flow is considered throughout.

2. Momentum Equation in Cartesian Coordinate

Conservation of momentum considerations applied to a small element of fluid yields the equation

\[
\rho \left[ \frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} \right] = \sigma_{ij} \frac{\partial u_j}{\partial x_i} + \rho F_i \tag{A2-1}
\]

where \( \sigma_{ij} \) represents the surface stress in the \( i \) direction on the element face with normal in the \( j \) direction. \( F_i \) is the body force per unit mass and in this case represents gravity forces. For a Newtonian fluid the stress \( \sigma_{ij} \) is related to the velocity gradients in the fluid as

\[
\sigma_{ij} = -p \delta_{ij} + \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \tag{A2-2}
\]

where \( p = -1/3 \sigma_{ii} \) is defined as the hydrostatic pressure. The quantity

\[
\left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)
\]

is the deformation tensor and \( \mu \) is the coefficient of dynamic viscosity of the fluid. Substitution of expression (A2-2) for \( \sigma_{ij} \) in equation (A2-1) yields

\[
\rho \left[ \frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} \right] = \delta P \frac{\partial u_i}{\partial x_j} + \delta \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + \rho F_i \tag{A2-3}
\]
This is the basic Navier-Stokes equation. If the viscosity is assumed to be constant, equation (A2-3) can be written as

$$\frac{\delta u_i}{\delta t} + u_j \frac{\delta u_i}{\delta x_j} = - \frac{1}{\rho} \frac{\delta p}{\delta x_i} + \nu \frac{\delta^2 u_i}{\delta x_j \delta x_j} + F_i \tag{A2-4}$$

This is the Navier-Stokes equation for an incompressible fluid with constant viscosity. When the flow is turbulent, the instantaneous velocity, pressure, and body force may be written as a mean and fluctuating part as

$$u_i = \bar{u}_i + u'_i$$

$$p = \bar{p} + p'$$

and

$$F = \bar{F} + F'$$

Introducing these expressions into equation (A2-4) and taking a mean, yields

$$\frac{\delta \bar{u}_i}{\delta t} + \bar{u}_j \frac{\delta \bar{u}_i}{\delta x_j} + \frac{\delta}{\delta x_j} \bar{u}_i u'_j = - \frac{1}{\rho} \frac{\delta \bar{p}}{\delta x_i} + \nu \frac{\delta^2 \bar{u}_i}{\delta x_j \delta x_j} + \bar{F}_i \tag{A2-5}$$

The velocity correlation $u'_i u'_j$ arises because of the turbulent fluctuations. The nature of this term may be seen if equation (A2-5) is rewritten in the form

$$\frac{\delta \bar{u}_i}{\delta t} + \bar{u}_j \frac{\delta \bar{u}_i}{\delta x_j} = \frac{1}{\rho} \frac{\delta}{\delta x_j} \left( - \bar{p} \delta_{ij} + \nu \frac{\delta u_i}{\delta x_j} - \rho \bar{u}_i u'_j \right) + \bar{F}_i \tag{A2-6}$$

In this form the new quantities are quite clearly additional stresses. They are called Reynolds stresses after O. Reynolds [1901] who first analyzed the Navier-Stokes equation in this way.

Modelling the Reynolds stresses using the eddy-viscosity concept of Boussinesq [1877] as

$$- \bar{u}'_i u'_j = \nu \left( \frac{\delta \bar{u}_i}{\delta x_j} + \frac{\delta \bar{u}_j}{\delta x_i} \right) - \frac{2}{3} k \delta_{ij}$$
yields the equation

$$\frac{\delta \bar{u}_i}{\delta t} + \bar{u}_j \frac{\delta \bar{u}_i}{\delta x_j} = -\frac{1}{\rho} \frac{\delta p}{\delta x_1} + \nu \frac{\delta \bar{u}_i}{\delta x_j} \frac{\delta \bar{u}_i}{\delta x_j}$$

$$+ \frac{\delta}{\delta x_j} \left[ \nu \left( \frac{\delta \bar{u}_i}{\delta x_j} + \frac{\delta \bar{u}_i}{\delta x_i} \right) \right] - \frac{2}{3} \frac{\delta k}{\delta x_1} + F_i$$  \hspace{1cm} (A2-7)

Defining a new pressure as $\bar{\bar{p}} = \bar{p} + 2\rho k/3$ Eq. (A2-7) becomes

$$\frac{\delta \bar{u}_i}{\delta t} + \bar{u}_j \frac{\delta \bar{u}_i}{\delta x_j} = -\frac{1}{\rho} \frac{\delta \bar{\bar{p}}}{\delta x_1} + \nu \frac{\delta \bar{\bar{u}}_i}{\delta x_j} \frac{\delta \bar{\bar{u}}_i}{\delta x_j}$$

$$+ \frac{\delta}{\delta x_j} \left[ \nu \left( \frac{\delta \bar{\bar{u}}_i}{\delta x_j} + \frac{\delta \bar{\bar{u}}_i}{\delta x_i} \right) \right] + F_i$$  \hspace{1cm} (A2-8)

Returning to the laminar forms in equation (A2-3) before the application of the continuity condition, then equation (A2-8) can be written as

$$\frac{\delta \bar{u}_i}{\delta t} + \bar{u}_j \frac{\delta \bar{u}_i}{\delta x_j} = -\frac{1}{\rho} \frac{\delta \bar{\bar{p}}}{\delta x_1} + \frac{\delta}{\delta x_j} \left( \nu_{\text{eff}} \frac{\delta \bar{\bar{u}}_i}{\delta x_j} + \frac{\delta \bar{\bar{u}}_i}{\delta x_i} \right) + F_i$$  \hspace{1cm} (A2-9)

where $\nu_{\text{eff}} = \nu + \nu'$. This form can be obtained ab initio by noting that the total stress term, laminar and turbulent, is $(\sigma_{ij} - \rho \bar{u}_i \bar{u}_j)$ and can be written as

$$\frac{\sigma_{ij}}{\rho} - \bar{u}_i \bar{u}_j = -\frac{\rho}{\rho} \frac{\delta \bar{u}_i}{\delta x_j} + \nu \left( \frac{\delta \bar{u}_i}{\delta x_j} + \frac{\delta \bar{u}_i}{\delta x_i} \right) + \nu \left( \frac{\delta \bar{u}_i}{\delta x_j} + \frac{\delta \bar{u}_i}{\delta x_i} \right) - \frac{2}{3} k \delta_{ij}$$

$$= -\delta_{ij} \left( \frac{\bar{\bar{p}}}{\rho} \right) + \nu_{\text{eff}} \left( \frac{\delta \bar{\bar{u}}_i}{\delta x_j} + \frac{\delta \bar{\bar{u}}_i}{\delta x_i} \right)$$  \hspace{1cm} (A2-10)
3. Momentum Equation in Cylindrical Coordinate

3.1. General three-dimensional equations

The basic Navier Stokes equations with constant viscosity for the instantaneous velocities \( V_r, U_\theta, \) and \( V_z \) in the \( r, \theta, z \) coordinate system are

\[
\frac{\delta V_r}{\delta t} + V_r \frac{\delta V_r}{\delta r} + V_\theta \frac{\delta V_r}{\delta \theta} + V_z \frac{\delta V_r}{\delta z} - \frac{U_\theta^2}{r} \frac{\delta V_r}{\delta r} = - \frac{1}{\rho} \frac{\delta p}{\delta r} \\
+ \nu \left[ \frac{\delta}{\delta r} \left( r \frac{\delta V_r}{\delta r} \right) + \frac{1}{r^2} \frac{\delta^2 V_r}{\delta \theta^2} + \frac{\delta^2 V_z}{\delta z^2} \right] - \frac{\nu V_r}{r^2} - \frac{2\nu}{r^2} \frac{\delta U_\theta}{\delta \theta} + F_r \tag{A2-11}
\]

\[
\frac{\delta U_\theta}{\delta t} + V_r \frac{\delta U_\theta}{\delta r} + V_\theta \frac{\delta U_\theta}{\delta \theta} + V_z \frac{\delta U_\theta}{\delta z} + \frac{V_r U_\theta}{r} = - \frac{1}{\rho} \frac{\delta p}{\delta \theta} \\
+ \nu \left[ \frac{\delta}{\delta r} \left( r \frac{\delta U_\theta}{\delta r} \right) + \frac{1}{r^2} \frac{\delta^2 U_\theta}{\delta \theta^2} + \frac{\delta^2 U_\theta}{\delta z^2} \right] - \frac{\nu U_\theta}{r^2} + \frac{2\nu}{r^2} \frac{\delta V_r}{\delta \theta} + F_\theta \tag{A2-12}
\]

\[
\frac{\delta V_z}{\delta t} + V_r \frac{\delta V_z}{\delta r} + V_\theta \frac{\delta V_z}{\delta \theta} + V_z \frac{\delta V_z}{\delta z} = - \frac{1}{\rho} \frac{\delta p}{\delta z} \\
+ \nu \left[ \frac{\delta}{\delta r} \left( r \frac{\delta V_z}{\delta r} \right) + \frac{1}{r^2} \frac{\delta^2 V_z}{\delta \theta^2} + \frac{\delta^2 V_z}{\delta z^2} \right] + F_z \tag{A2-13}
\]

where the \( F \) term signifies the body force in the appropriate coordinate direction.

In these equations the continuity condition has been applied in the incompressible form

\[
\frac{1}{r} \frac{\delta}{\delta r} (r V_r) + \frac{1}{r} \frac{\delta U_\theta}{\delta \theta} + \frac{\delta V_z}{\delta z} = 0
\]

to simplify the viscous terms. Thus, these equations correspond to equation (A2-4) in the Cartesian case.

The instantaneous velocities, pressure, and body force may be broken down into mean and fluctuating parts as
\[
\begin{align*}
V_r &= \overline{V}_r + V'_r \\
U_\theta &= \overline{U}_\theta + U'_\theta \\
V_z &= \overline{V}_z + V'_z \\
p &= \overline{p} + p'
\end{align*}
\]

and
\[
F = \overline{F} + F'
\]

Introducing these expression into equations (A2-11) to (A2-13) and taking a mean yields the equations for the mean velocities. These are

\[
\begin{align*}
\frac{\delta \overline{V}_r}{\delta t} + \frac{\delta \overline{V}_r}{\delta r} + \frac{\delta \overline{U}_\theta}{\delta \theta} + \frac{\delta \overline{V}_z}{\delta z} - \frac{\overline{U}^2}{r} &= - \frac{1}{\rho} \delta \overline{p} \\
+ \nu \left[ \frac{1}{r} \frac{\delta \overline{V}_r}{\delta r} \left( \frac{r}{\overline{V}_r} \right)^2 + \frac{1}{r^2} \frac{\delta^2 \overline{V}_r}{\delta \theta^2} + \frac{\delta^2 \overline{V}_z}{\delta z^2} \right] - \overline{V}_r \frac{\delta \overline{U}_\theta}{\delta \theta} - \frac{2\nu}{r} \frac{\delta \overline{U}_\theta}{\delta \theta}
\end{align*}
\]

\[
- \frac{1}{r} \frac{\delta}{\delta r} \left( r \frac{\delta \overline{V}_r}{\delta \theta} \right) - \frac{1}{r} \frac{\delta}{\delta \theta} \frac{\overline{V}_r \overline{U}_\theta}{r} - \frac{\delta}{\delta z} \frac{\overline{V}_r \overline{U}_\theta}{r} + \frac{\overline{U'}^2}{r} + \frac{\overline{U'}^2}{r} + \overline{F}_r \] (A2-14)

\[
\begin{align*}
\frac{\delta \overline{U}_\theta}{\delta t} + \frac{\delta \overline{U}_\theta}{\delta r} + \frac{\delta \overline{U}_\theta}{\delta \theta} + \frac{\delta \overline{V}_r}{\delta \theta} + \frac{\delta \overline{V}_z}{\delta \theta} &= - \frac{1}{\rho} \frac{\delta \overline{p}}{\rho} \\
+ \nu \left[ \frac{1}{r} \frac{\delta \overline{U}_\theta}{\delta r} \left( \frac{r}{\overline{U}_\theta} \right) + \frac{1}{r^2} \frac{\delta^2 \overline{U}_\theta}{\delta \theta^2} + \frac{\delta^2 \overline{U}_\theta}{\delta z^2} \right] - \overline{U}_\theta \frac{\delta \overline{V}_r}{\delta \theta} + \frac{2\nu}{r^2} \frac{\delta \overline{V}_r}{\delta \theta}
\end{align*}
\]

\[
- \frac{1}{r} \frac{\delta}{\delta r} \frac{\overline{U'} \overline{V'}}{\theta} - \frac{1}{r} \frac{\delta}{\delta \theta} \frac{\overline{U'} \overline{V'}}{r} - \frac{\delta}{\delta z} \frac{\overline{U'} \overline{V'}}{r} - 2 \frac{\overline{U'} \overline{U'} \theta}{r} + \overline{F}_\theta \] (A2-15)

and
\[
\frac{\delta V_z}{\delta t} + V_r \frac{\delta V_z}{\delta r} + V_\theta \frac{\delta V_z}{\delta \theta} + V_z \frac{\delta V_z}{\delta z} = -\frac{1}{\rho} \frac{\delta p}{\delta z} \\
\]
\[
+ \nu \left[ \frac{1}{r} \frac{\delta}{\delta r} \left( r \frac{\delta V_z}{\delta r} \right) + \frac{1}{r^2} \frac{\delta^2 V_z}{\delta \theta^2} + \frac{\delta^2 V_z}{\delta z^2} \right] \\
- \frac{1}{r} \frac{\delta}{\delta r} \left( r V' r' z \right) - \frac{1}{r} \frac{\delta}{\delta \theta} \left( \frac{U'_\theta V'}{r} z \right) - \frac{\delta V'^2}{\delta z} + \frac{F}{z} \\
\] (A2-16)

The continuity equation being linear remains in the same form as previously, i.e.,

\[
\frac{1}{r} \frac{\delta}{\delta r} \left( r \frac{\delta V_r}{\delta r} \right) + \frac{1}{r} \frac{\delta U_\theta}{\delta \theta} + \frac{\delta V_z}{\delta z} = 0 \\
\] (A2-17)

The quantities \( V'^2, U'^2, V'^2, U' V', U' V', V' V' \) appearing in the above equations are the Reynolds stresses. Using the Boussinesq model these stresses can be written as:

\[
\frac{V'^2}{r^2} = -2 \nu \frac{\delta V_r}{\delta r} + \frac{2}{3} k \\
\]

\[
\frac{U'^2}{\delta \theta} = -2 \nu \left( \frac{1}{r} \frac{\delta U_\theta}{\delta \theta} + \frac{V_r}{r} \right) + \frac{2}{3} k \\
\]

\[
\frac{V'^2}{z^2} = -2 \nu \frac{\delta V_z}{\delta z} + \frac{2}{3} k \\
\]

\[
\frac{V'_r U'_\theta}{r} = -\nu \left( \frac{\delta U_\theta}{\delta r} + \frac{1}{r} \frac{\delta V_r}{\delta \theta} - \frac{U'_\theta}{r} \right) \\
\]

\[
\frac{U'_\theta V'_z}{z} = -\nu \left( \frac{1}{r} \frac{\delta V_z}{\delta \theta} + \frac{\delta U_\theta}{\delta z} \right) \\
\]

and

243
\[
\frac{\delta V}{\delta t} + \delta V_r \frac{\delta V}{\delta r} + \frac{U_0}{r} \delta V_{\theta} + \frac{V_z}{\delta z} - \frac{U_r}{r} = - \frac{1}{\rho} \frac{\delta p}{\delta r}
\]

Substitution of these expressions into equations (A2-14) through (A2-16) yields the general mean flow equations. These are:

\[
\frac{\delta U_\theta}{\delta t} + \frac{\delta U_\theta}{\delta r} + \frac{U_\theta}{r} \delta U_{\theta} + \frac{V_z}{\delta z} + \frac{U_r}{r} = - \frac{1}{\rho} \frac{\delta p}{\delta \theta}
\]

\[
+ \frac{1}{r^2} \frac{\delta U_{\theta}}{\delta \theta} + \frac{1}{r^2} \frac{\delta U_{\theta}}{\delta \theta} + \frac{\delta}{\delta z} (\nu \frac{\delta U_{\theta}}{\delta z}) - \nu \frac{U_r}{r^2} + \bar{F} \quad (A2-18)
\]

\[
\frac{\delta U_{\theta}}{\delta t} + \frac{\delta U_{\theta}}{\delta r} + \frac{U_\theta}{r} \delta U_{\theta} + \frac{V_z}{\delta z} + \frac{U_r}{r} = - \frac{1}{\rho} \frac{\delta p}{\delta \theta}
\]

\[
+ \frac{1}{r^2} \frac{\delta U_{\theta}}{\delta \theta} + \frac{1}{r^2} \frac{\delta U_{\theta}}{\delta \theta} + \frac{\delta}{\delta z} (\nu \frac{\delta U_{\theta}}{\delta z}) - \nu \frac{U_r}{r^2} + \bar{F} \quad (A2-19)
\]

and

244
In the above \( \nu_{\text{eff}} = \nu + \nu_t \). The continuity condition has been used on the constant viscosity laminar portions of the above terms. Hence, these equations correspond in form to equation (A2-8) in the cartesian case.

As in the Cartesian situation, the laminar and turbulent stresses can be grouped together in the form of the cartesian relationship (A2-10). This leads to the appearance of \( \nu_{\text{eff}} \) only in the above equations, and they then correspond in form to equation (A2-9) in the cartesian case.

3.2. Equations in Polar \((r, \theta)\) Coordinates

The equations in \(r, \theta\) coordinate are recovered from the general equations by setting

\[
\nu_z = 0 \quad \text{and} \quad \frac{\delta}{\delta z} = 0
\]

The resulting momentum equations are:

\[
\frac{\delta \overline{V}_r}{\delta t} + \overline{V}_r \frac{\delta \overline{V}_r}{\delta r} + \frac{\overline{U}_\theta}{r} \frac{\delta \overline{V}_r}{\delta \theta} - \frac{\overline{U}_\theta}{r} = - \frac{1}{\rho} \frac{\delta p}{\delta r} + \frac{1}{r} \frac{\delta}{\delta r} (\nu_{\text{eff}} \frac{\delta \overline{V}_r}{\delta r}) + \frac{1}{r^2} \frac{\delta}{\delta \theta} (\nu_{\text{eff}} \frac{\delta \overline{V}_r}{\delta \theta}) - \nu_{\text{eff}} \frac{\overline{V}_r}{r^2} - \frac{2\nu_{\text{eff}}}{r^2} \frac{\delta \overline{U}_\theta}{\delta \theta}
\]

\[
\frac{\delta \overline{U}_\theta}{\delta t} + \overline{V}_r \frac{\delta \overline{U}_\theta}{\delta r} + \frac{\overline{U}_\theta}{r} \frac{\delta \overline{U}_\theta}{\delta \theta} - \frac{\overline{U}_\theta}{r} = \frac{1}{\rho} \frac{\delta p}{\delta \theta} + \frac{1}{r} \frac{\delta}{\delta r} (\nu_{\text{eff}} \frac{\delta \overline{U}_\theta}{\delta r}) + \frac{1}{r^2} \frac{\delta}{\delta \theta} (\nu_{\text{eff}} \frac{\delta \overline{U}_\theta}{\delta \theta}) - \nu_{\text{eff}} \frac{\overline{U}_\theta}{r^2} + \frac{\overline{U}_\theta}{r} (A2-21)
\]

and
The continuity equation becomes

\[ \frac{\delta \bar{U}_\theta}{\delta t} + \frac{\bar{V}}{r} \frac{\delta \bar{U}_\theta}{\delta r} + \frac{\bar{U}_\theta}{r} \frac{\delta \bar{U}_\theta}{\delta \theta} + \frac{\bar{V} \bar{U}_\theta}{r} = - \frac{1}{\rho} \frac{\delta \rho}{\delta \theta} \]

\[ + \frac{1}{r} \frac{\delta}{\delta r} (v_{\text{eff}} r \frac{\delta \bar{U}_\theta}{\delta r}) + \frac{1}{r^2} \frac{\delta}{\delta \theta} (v_{\text{eff}} \frac{\delta \bar{U}_\theta}{\delta \theta}) - v_{\text{eff}} \frac{\bar{U}_\theta}{r^2} \]

\[ + \frac{2v_{\text{eff}}}{r^2} \frac{\delta V}{\delta \theta} - \frac{1}{r} \frac{\delta}{\delta r} (v_{\text{t}} \bar{U}_\theta) + \frac{\delta}{\delta r} (v_{\text{t}} \frac{\delta V}{\delta \theta}) + \frac{1}{r^2} \frac{\delta}{\delta \theta} (v_{\text{t}} \frac{\delta U_\theta}{\delta \theta}) \]

\[ + \frac{2}{r^2} \frac{\delta}{\delta \theta} (v_{\text{t}} \bar{V}) + \frac{v_{\text{t}}}{r} \frac{\delta U_\theta}{\delta r} + \bar{V} \theta \]

(A2-22)

The continuity equation becomes

\[ \frac{1}{r} \frac{\delta}{\delta r} (r \bar{V}) + \frac{\delta \bar{U}_\theta}{\delta \theta} = 0 \]

(A2-23)

3.3. Equations in axisymmetric \((r,z)\) coordinates

The equations in \(r,z\) coordinates are recovered from the general equation by setting

\[ \bar{U}_\theta = 0 \quad \text{and} \quad \frac{\delta}{\delta \theta} = 0 \]

The resulting momentum equations are:

\[ \frac{\delta V}{\delta t} + \frac{\bar{V}}{r} \frac{\delta V}{\delta r} + \bar{V} \frac{\delta V}{\delta z} = - \frac{1}{\rho} \frac{\delta \rho}{\delta z} + \frac{\delta}{\delta r} (v_{\text{eff}} \frac{\delta V}{\delta r}) + \frac{\delta}{\delta z} (v_{\text{eff}} \frac{\delta V}{\delta z}) \]

\[ - v_{\text{eff}} \frac{\bar{V}}{r^2} + \frac{1}{r} \frac{\delta}{\delta r} (v_{\text{t}} \frac{\delta V}{\delta r}) + \frac{\delta}{\delta z} (v_{\text{t}} \frac{\delta V}{\delta z}) - \frac{v_{\text{t}} \bar{V}}{r^2} + \bar{V} \]

(A2-24)

and
The continuity equation becomes

\[
\frac{1}{r} \frac{\delta}{\delta r} \left( r \overline{V}_r \right) + \frac{\delta \overline{V}_z}{\delta z} = 0 \tag{A2-26}
\]

4. Temperature Equations

The equations for the temperature are presented separately for the polar and axisymmetric situation.

4.1. Equation in polar \((r, \theta)\) coordinates

The equation for the instantaneous temperature is

\[
\frac{\delta T}{\delta t} + V_r \frac{\delta T}{\delta r} + \frac{U_\theta}{r} \frac{\delta T}{\delta \theta} = \nu Pr \left[ \frac{1}{r} \frac{\delta}{\delta r} \left( r \frac{\delta T}{\delta r} \right) + \frac{1}{r^2} \frac{\delta^2 T}{\delta \theta^2} \right]
\]

where \( Pr \) is the Prandtl number. Letting

\[
V_r = \overline{V}_r + U'_r
\]
\[
U_\theta = \overline{U}_\theta + U'_\theta
\]

and \( T = \overline{T} + T' \)

and taking a mean, the equation for the mean temperature \( \overline{T} \) becomes

\[
\frac{\delta \overline{T}}{\delta t} + \overline{V}_r \frac{\delta \overline{T}}{\delta r} + \frac{\overline{U}_\theta}{r} \frac{\delta \overline{T}}{\delta \theta} = \nu Pr \left[ \frac{1}{r} \frac{\delta}{\delta r} \left( r \frac{\delta \overline{T}}{\delta r} \right) + \frac{1}{r^2} \frac{\delta^2 \overline{T}}{\delta \theta^2} \right]
\]

\[
- \frac{1}{r} \frac{\delta}{\delta r} \left( r V'_r T' \right) - \frac{1}{r} \frac{\delta}{\delta \theta} \left( U'_\theta T' \right)
\]

247
The new correlations \( \bar{V}' \bar{T}' \) and \( u'_r \bar{T}' \) arise because of the turbulent fluctuations. By analogy to the treatment of the Reynolds stresses, these quantities are modelled as

\[
\frac{\bar{V}' \bar{T}'}{r^{1/2}} = -\frac{\nu}{\sigma_t} \frac{\delta T}{\partial r}
\]

and

\[
\frac{u'_r \bar{T}'}{\partial \theta} = -\frac{\nu}{\sigma_t} \frac{1}{r} \frac{\delta T}{\partial \theta}
\]

where \( \sigma_t \) is the turbulent Prandtl number. Using these expressions the equations for the mean temperature becomes

\[
\frac{\delta T}{\partial t} + \bar{V} \frac{\delta T}{\partial r} + \bar{U} \theta \frac{\delta T}{\partial \theta}
\]

\[
= \frac{1}{r} \frac{\delta}{\partial r} \left( \alpha_{\text{eff}} r \frac{\delta T}{\partial r} \right) + \frac{1}{r^2} \frac{\delta}{\partial \theta} \left( \alpha_{\text{eff}} \frac{\delta T}{\partial \theta} \right)
\]

where

\[
\alpha_{\text{eff}} = \frac{\nu}{Pr} + \frac{\nu_t}{\sigma_t}
\]

4.2. Equation in axisymmetric \((r,z)\) coordinate

The turbulent equation for the mean temperature is derived by letting

\[
\bar{V}_r = \bar{V}_r + V'_r
\]

\[
\bar{V}_z = \bar{V}_z + V'_z
\]

and

\[
\bar{T} = \bar{T} + T'
\]

in the equation for the instantaneous temperature. This leads to the appearance of new correlations \( \bar{V}'_r \bar{T}' \) and \( \bar{V}'_z \bar{T}' \). Modelling these as
\[ \nabla \cdot \nabla T' = - \frac{\nu_t}{\sigma_t} \frac{\partial T'}{\partial r} \]

and

\[ \frac{\partial}{\partial z} \nabla \cdot \nabla T' = - \frac{\nu_b}{\sigma_t} \frac{\partial T'}{\partial z} \]

yields the mean temperature equation as

\[ \frac{\partial T'}{\partial t} + \nabla \cdot \left( \nabla T' \right) = \frac{1}{r} \frac{\partial}{\partial r} \left( \alpha_{\text{eff}} r \frac{\partial T'}{\partial r} \right) + \frac{\partial}{\partial z} \left( \alpha_{\text{eff}} \frac{\partial T'}{\partial z} \right) \]

where \( \alpha_{\text{eff}} \) is as defined above.

5. Energy Production

The energy production terms in the cylindrical coordinate system can be obtained directly by constructing the equation for \( k(= 1/2(\nu_r^2 + \nu_\theta^2 + \nu_z^2)) \) and extracting these terms that relate the production of energy. Alternatively the function may be derived simply by noting that the production function is the direct analogue of the viscous dissipation function. Hence, the expression for the dissipation function in the required coordinate system can be adopted for use. This is what is done here. However, before doing this it is worthwhile to briefly review the origins of these functions. This review can be most easily and clearly carried out in a cartesian system.

The total energy equation is obtained by multiplying equation (A2-1) by \( U_i \). The energy dissipation function, \( W \), may then be identified as

\[ W = \sigma_{ij} \frac{\partial U_i}{\partial x_j} \]

and since \( \sigma_{ij} \) is a symmetric tensor, \( W \) becomes

\[ W = \sigma_{ij} \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \]

(A2-27)

This is the product of stress and the rate of strain tensor and is the general expression for energy dissipation. Under the assumption of a Newtonian fluid.
In the turbulent flow situation, the energy production per unit mass by shear is

$$W = \frac{u}{2} \left( \frac{\delta U_i}{\delta x_j} + \frac{\delta U_j}{\delta x_i} \right)^2$$  \hspace{1cm} (A2-28)

and may be written

$$p = -u_iu_j \frac{\delta U_i}{\delta x_j}$$

This is the product of the turbulent stress with the mean rate of strain tensor.

Using the Boussinesq model for the Reynolds stress, the energy production becomes

$$p = \frac{\nu_t}{2} \left( \frac{\delta U_i}{\delta x_j} + \frac{\delta U_j}{\delta x_i} \right)^2$$  \hspace{1cm} (A2-29)

It can be seen that equation (A2-29) is the direct analogue of equation (A2-28), with $\nu$ replaced by $\nu_t$, and mean velocity gradients in place of instantaneous ones.

The production function in cylindrical coordinates can thus be written down directly from an examination of the dissipation function. The production function is

$$p = \nu_t \left[ 2 \left( \frac{\delta V_r}{\delta r} \right)^2 + 2 \left( \frac{\delta U_\theta}{r \delta r} + \frac{\nu}{r} \right)^2 + 2 \left( \frac{\delta V_z}{\delta z} \right)^2 \right. $$

$$+ \left( \frac{\delta U_\theta}{r \delta r} - \frac{\nu_\theta}{r} \right)^2 + \left( \frac{\delta V_r}{\delta r} + \frac{\delta V_\theta}{\delta z} \right)^2 + \left( \frac{\delta V_z}{\delta z} \right)^2 $$

$$+ \left( \frac{\delta V_\theta}{\delta z} + \frac{\delta U_\theta}{\delta z} \right)^2 \right]$$

250
The equation can be particularized to two-dimensional situations by application of the rules outlined in sections 3.2 and 3.3.

Thus, the energy production function in Polar \((r, \theta)\) coordinates is

\[
P = \nu \left[ 2 \left( \frac{\delta V_r}{\delta r} \right)^2 + \frac{2}{r} \left( \frac{\delta U_\theta}{\delta \theta} + \frac{V_r}{r} \right)^2 \right.
\]

\[
+ \left( \frac{\delta U_\theta}{\delta r} - \frac{U_\theta}{r} + \frac{1}{r} \frac{\delta V_r}{\delta \theta} \right)^2 \]

The energy production function in axisymmetric \((r, z)\) coordinates is

\[
P = \nu \left[ 2 \left( \frac{\delta V_r}{\delta r} \right)^2 + \left( \frac{V_r}{r} \right)^2 + 2 \left( \frac{\delta V_z}{\delta z} \right)^2 + \left( \frac{\delta V_r}{\delta z} + \frac{\delta V_z}{\delta r} \right)^2 \right]
\]

6. **Buoyancy effects**

The turbulent energy production/destruction by buoyancy forces is most easily considered in the separate coordinate systems.

6.1. **Polar \((r, \theta)\) system**

The buoyancy term in the equation for the fluctuating velocity \(U'_\theta\) is \(\rho'g \cos \theta\) or using the relationship at Equation II-4, is \(-\beta g T' \cos \theta\). On multiplication of the equation by \(U'_\theta\) to give the equation for the energy component \(U'_\theta^2\), the above term becomes

\[-\beta g \overline{T'U'_\theta} \cos \theta\]

Similarly the buoyancy term in the \(V'^2_r\) equation is

\[-\beta g \overline{T'V'_r} \sin \theta\]

There is no buoyancy effect in the \(z\) direction so that the total buoyancy term in the energy equation is

\[G = -\beta g \left( \overline{T'U'_\theta} \cos \theta + \overline{T'V'_r} \sin \theta \right)\]

Modelling the temperature velocity correlations as in section 4.1 of this appendix yields \(G\) as
\[ G = \beta g \frac{\nu}{\sigma_t} \left( \frac{1}{r} \frac{\delta T}{\delta \theta} \cos \theta + \frac{\delta T}{\delta r} \sin \theta \right) \]

6.2. Axisymmetric \((r,z)\) system

In the axisymmetric situation buoyancy effects appear only in the \(z\) direction. The buoyancy term in the equation for \(V_z^2\) is

\[ G = -\beta g \overline{T'V'_z} \]

On modelling the correlation \(\overline{T'V'_z}\) as in Section 4.2 of this appendix, \(G\) becomes

\[ G = \beta g \frac{\nu}{\sigma_t} \frac{\delta T}{\delta z} \]