FINITE-ELEMENT SURFACE-WATER MODELING SYSTEM: TWO-DIMENSIONAL FLOW IN THE HORIZONTAL PLANE— ADDENDUM TO THE USERS MANUAL

By Jonathan K. Lee

U.S. GEOLOGICAL SURVEY

Open-File Report 97-410

Prepared in cooperation with the

SOUTH FLORIDA WATER MANAGEMENT DISTRICT

Reston, Virginia

1999

U.S. DEPARTMENT OF THE INTERIOR
BRUCE BABBITT, Secretary

U.S. GEOLOGICAL SURVEY
Charles G. Groat, Director

For additional information write to:

U.S. Geological Survey
Branch of Regional Research
Mail Stop 430
Reston, Virginia 20192

Copies of this report may be purchased from:

U.S. Geological Survey
Branch of Information Services
Box 25286
Denver, Colorado 80225-0286

CONTENTS

Page

Abstract 1
Introduction 2
Addendum to section 4. Governing equations 3
  Depth-averaged flow equations 3
    Bed shear stress 3
    Surface shear stress 4
    Source/sink terms 5
    Ground-water inflow/outflow 5
  Flow through and under levees 7
Addendum to section 5. Finite-element equations 13
  Residual expressions 13
  Derivative expressions 13
  Application of boundary and special conditions—total flow across a boundary 13
Addendum to section 6. Modeling system operation 15
  Data collection 15
  Network design 16
    General network layout 16
    One-dimensional levees 16
Addendum to section 8. Input data description 18
  Depth-averaged flow module: FLOMOD 18
    Program-control data set 20
    Element data set 34
    Node data set 37
    Turbulence-model data set 39
    Friction data set 40
    Wind data set 42
    Vegetation data set 43
    Precipitation data set 44
    Evapotranspiration data set 45
    Ground-water-seepage data set 46
    Ground-water-head data set 47
    Initial-condition data set 48

CONTENTS (Continued)

Page

Boundary-condition data set 50
Total-flow-cross-section data set 54
Water-surface-elevation-cross-section data set 56
Weir data set 59
Culvert data set 61
Levee data set 63
Flux-line data set 65
Time-dependent data set 66
Input data files 68
Network data file 68
Wind data file 69
Ground-water-head data file 70
Initial-condition data file 70
Boundary-condition data file 72
Addendum to section 9. Output data description—depth-averaged flow module: FLOMOD 73
Printed output 73
Output data files—flow data file 73
References 74

FIGURES

Page

Figures 4.1–4.4. Sketches showing:
- 4.1. Conceptual representation of surface-water/ground-water interconnection 6
- 4.2. Finite-element network with levee segments, a weir segment, and a culvert 8
- 4.3. Levee cross section 9
- 4.4. Definition of levee-seepage parameters 11

TABLE

Page

Table 4.1. Definition of the function f 12

CONVERSION FACTORS

Multiply	By	To obtain
inch (in)	25.40	millimeter (mm)
inch per hour (in/h)	25.40	millimeter per hour (mm/h)
inch per day (in/d)	25.40	millimeter per day (mm/d)
foot (ft)	0.3048	meter (m)
foot per second (ft/s)	0.3048	meter per second (m/s)
foot per square second (ft/s^2)	0.3048	meter per square second (m/s^2)
square foot (ft^2)	0.09290	square meter (m^2)
square foot per second (ft^2/s)	0.09290	square meter per second (m^2/s)
cubic foot per second (ft^3/s)	0.02832	cubic meter per second (m^3/s)
second per foot (s/ft)	3.281	second per meter (s/m)
slug per cubic foot ($slug/ft^3$)	515.7	kilogram per cubic meter (kg/m^3)

FINITE-ELEMENT SURFACE-WATER MODELING SYSTEM: TWO-DIMENSIONAL FLOW IN THE HORIZONTAL PLANE—ADDENDUM TO THE USERS MANUAL

By Jonathan K. Lee

Abstract

This manual provides users of the Finite-Element Surface-Water Modeling System: Two-Dimensional Flow in the Horizontal Plane (FESWMS-2DH) the information necessary to use new additions and modifications to the original flow model, for which a users manual was published by Froehlich in 1989. This information includes descriptions of the basic logic of the changes, the necessary input data, and the output generated by the modeling system.

The modified or added features described in this manual include element source/sink terms for precipitation, evapotranspiration, and ground-water inflow, or outflow, or both; linearized friction in the momentum equations; sheltering coefficients added to surface-shear-stress terms to account for the reduced effect of wind due to emergent vegetation; seepage through and under levee segments; water-surface elevations outside the model domain for use with culverts, weir segments, and levee segments; modification with time of turbulence-model coefficients, friction coefficients, wind, vegetation characteristics, precipitation rate, evapotranspiration rate, ground-water-seepage parameters, ground-water heads, and boundary conditions, including total flows and water-surface elevations at specified cross sections, and external water-surface elevations at culverts, weir segments, and levee segments; optional formats for element incidence lists; and optional formats for the flow-data file.

Introduction

This manual provides users of the Finite-Element Surface-Water Modeling System: Two-Dimensional Flow in the Horizontal Plane (FESWMS-2DH) the information necessary to use new additions and modifications to the original flow model. The original users manual (Froehlich, 1989) describes three related FORTRAN programs: DINMOD, the data-input module; FLOMOD, the solution module; and ANOMOD, the output-analysis module. DINMOD is used to prepare a finite-element network, FLOMOD is used to obtain the water depth and depth-averaged flow velocity at the nodes of a finite-element network, and ANOMOD is used to create plots of model results.

The information presented in this manual includes descriptions of the basic logic of the changes made to the original version of FLOMOD, the necessary input data, and the output generated by the solution module. The modified or added features described in this manual include:

- Element source/sink terms for precipitation, evapotranspiration, and ground-water inflow, or outflow, or both
- Linearized friction in the momentum equations
- Sheltering coefficients added to surface-shear-stress terms to account for the reduced effect of wind due to emergent vegetation
- Seepage through and under levee segments
- Water-surface elevations outside the model domain for use with culverts, weir segments, and levee segments
- Modification with time of turbulence-model coefficients, friction coefficients, wind, vegetation characteristics, precipitation rate, evapotranspiration rate, ground-water-seepage parameters, ground-water heads, and boundary conditions, including total flows and water-surface elevations at specified cross sections, and external water-surface elevations at culverts, weir segments, and levee segments
- Optional formats for element incidence lists
- Optional formats for the flow-data file

Modifications and additions to the flow equations are presented in the addendum to Section 4. This information will help the user understand the new capabilities of the system, how required empirical coefficients are used, and how results are obtained. The addendum to Section 5 describes how new boundary conditions are incorporated into constraint equations at model boundaries. The addendum to Section 6 provides additional information on developing a finite-element network. A complete description of the input data needed to run FLOMOD is given in the addendum to Section 8. The addendum to Section 9 describes modifications to the printed output and the flow-data file.

The units of dimensioned parameters and variables are given in the text in terms of length (L), time (T), and mass (M). Equations referenced in the original users manual are written with a hyphen (for example, 4-14); those in this report are written with a period (for example, 4.14).

Addendum to Section 4. Governing Equations

The two-dimensional, depth-averaged surface-water-flow equations are modified by the addition of source/sink terms for precipitation, evapotranspiration, and ground-water inflow, or outflow, or both. The capability to use linearized bed-shear-stress terms is added, and the surface-shear-stress terms are modified to treat water-surface sheltering by emergent vegetation. Equations used to model one-dimensional flow through and under permeable levees are described.

Depth-Averaged Flow Equations

The depth-averaged flow equations consist of equations for the conservation of momentum in the x- and y-directions, equations 4-2 and 4-3 in the orginal manual (Froehlich, 1989), and an equation for conservation of mass, equation 4-4 in the original manual.

Bed Shear Stress

Linear bed-friction terms in the momentum equations are useful in testing models and in initiating simulations. The directional components, τ_x^b and τ_y^b, of the bed shear stress (M/LT^2), given in the original manual by equations 4-9 and 4-10, are replaced by

$$\tau_x^b = \rho c_f \sqrt{gH}\, U \left[1 + \left(\frac{\partial z_b}{\partial x}\right)^2 + \left(\frac{\partial z_b}{\partial y}\right)^2\right]^{1/2} \tag{4.1}$$

and

$$\tau_y^b = \rho c_f \sqrt{gH}\, V \left[1 + \left(\frac{\partial z_b}{\partial x}\right)^2 + \left(\frac{\partial z_b}{\partial y}\right)^2\right]^{1/2}, \tag{4.2}$$

where ρ = density of water (M/L^3), c_f = bed-friction coefficient (dimensionless), g = gravitational acceleration (L/T^2), H = flow depth (L), U = depth-averaged velocity in the x-direction (L/T), V = depth-averaged velocity in the y-direction (L/T), and z_b = bed elevation (L).

The linear bed-friction coefficient, c_f, is computed as

$$c_f = \sqrt{gH}\, k_f, \tag{4.3}$$

where k_f = linear friction coefficient (T/L). The expression $\sqrt{H}$ appearing in equations 4.1, 4.2, and 4.3 results because the momentum equations are solved in a conservative form, obtained in part by multiplying the nonconservative momentum equations by H. Appropriate values of k_f can be estimated by substituting representative values of n, H, U, and V into the equation

$$k_f = \frac{n^2}{\phi H^{4/3}}(U^2 + V^2)^{1/2}, \tag{4.4}$$

where n = Manning's roughness coefficient (T/L$^{1/3}$) and ϕ = 2.208 when inch/pound units are used and 1.0 when International System (SI) units are used. The coefficient k_f is allowed to vary in space and time.

Surface Shear Stress

The directional components, τ_x^s and τ_y^s, of the surface shear stress (M/LT^2) are given in the original manual by equations 4-13 and 4-14. To account for the sheltering effect of emergent vegetation, τ_x^s and τ_y^s are multiplied by ε.

dimensionless sheltering coefficient, S, defined by Reid and Whitaker (1976, p. 64) for $h_v > H$ as

$$S = \frac{1}{1 + \frac{c_v N_v w_v (h_v - H)}{c_s}}, \tag{4.5}$$

where c_v = vegetation drag coefficient (dimensionless); N_v = plant density, in stems or leaves per unit area ($1/L^2$); w_v = average stem or leaf width (L); h_v = average stem or leaf height (L); and c_s = surface-stress coefficient (dimensionless). The parameters c_v, N, w_v, and h_v are allowed to vary in space and time. Assigning a value to c_s is discussed in the original manual, and assigning a value to c_v is discussed in Reid and Whitaker (1976, p. 68). If $h_v \leq H$, S is assigned the value 1.0.

Source/Sink Terms

The symbol q, which is used to denote source/sink terms (L/T) in the mass-conservation equation, is subtracted from the left-hand side of equation 4-4 given in the original manual. This term can be expanded to represent precipitation, evapotranspiration, and ground-water inflow, or outflow, or both. Thus,

$$q = q_p - q_e + q_g, \tag{4.6}$$

where q_p = precipitation rate (L/T); q_e = evapotranspiration rate (L/T); and q_g = ground-water-inflow or -outflow rate (L/T), positive for inflow and negative for outflow. These rates are permitted to vary in space and time.

Ground-Water Inflow/Outflow

Seepage between surface water and ground water, q_g, is expressed as in McDonald and Harbaugh (1988, chap. 6). The surface water is assumed to be separated from the ground water by a low-permeability layer of material of thickness M_ℓ (L) (fig. 4.1). The hydraulic conductivity of this layer is denoted by K_ℓ (L/T). If it is further assumed that all head losses between the surface water and the aquifer are across the low-permeability layer and that the piezometric head of the ground water is above the bottom of the low-permeability layer, then the flow q_g between the surface water and the

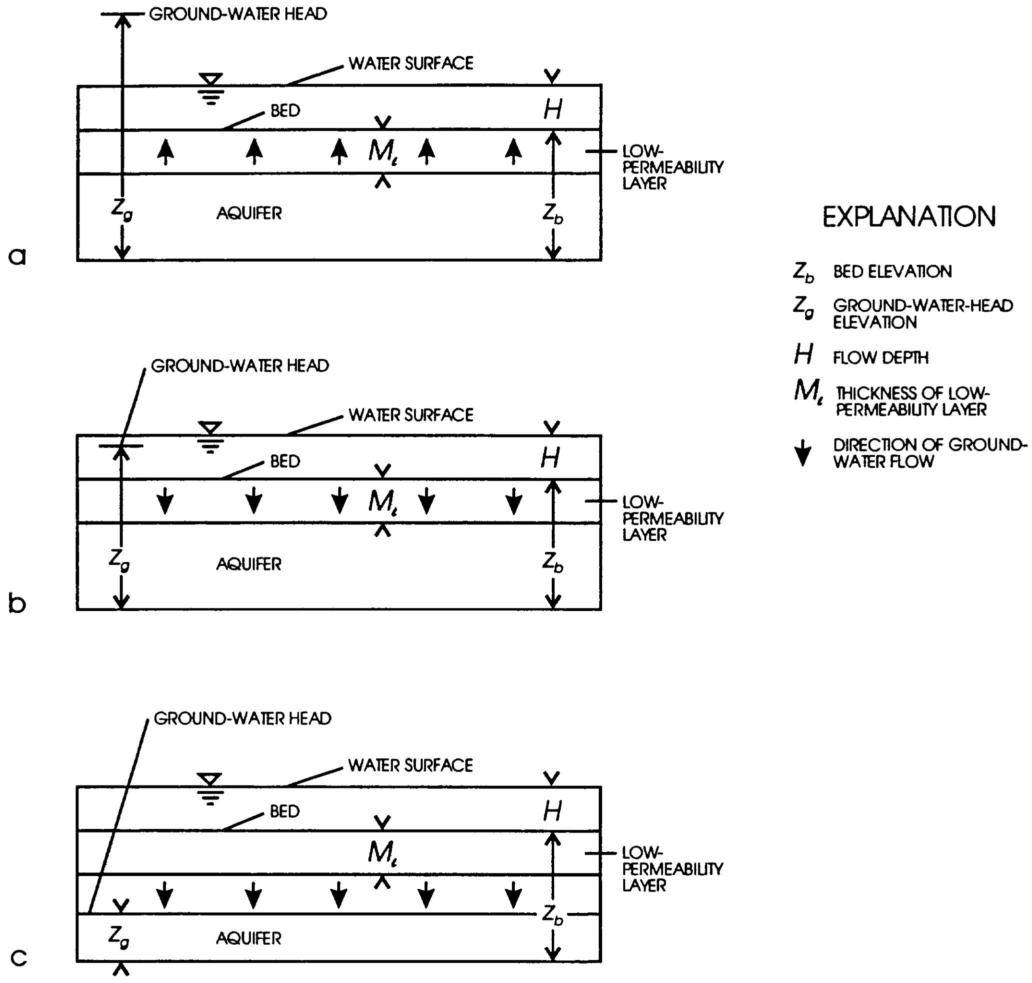


Figure 4.1: Conceptual representation of surface-water/ground-water interconnection with (a) the ground-water head above the surface-water elevation, (b) the ground-water head below the surface-water elevation but above the bottom of the low-permeability layer, and (c) the ground-water head below the bottom of the low-permeability layer.

aquifer is given by

$$q_g = \frac{K_\ell}{M_\ell}(z_g - z_b - H) = C_\ell(z_g - z_b - H), \tag{4.7}$$

where $C_\ell = K_\ell/M_\ell$ = hydraulic conductance of the low-permeability layer (1/T) and z_g = ground-water head (L). Note that q_g is positive (the flow is from the ground water to the surface water) if $z_g > z_b + H$ and q_g is negative if $z_g < z_b + H$. As discussed in McDonald and Harbaugh (1988, chap. 6), if the ground-water level falls below a certain point, seepage from the surface water to the ground water no longer depends on the ground-water head. If the water level in the aquifer falls below the base of the low-permeability layer and if it is assumed that the low-permeability layer remains saturated, then the head at its base is the elevation of that point. Because the elevation of the base of the low-permeability layer is $z_b - M_\ell$, the flow q_g from the surface water to the ground water is

$$q_g = C_\ell(z_b - M_\ell - z_b - H) = -C_\ell(M_\ell + H). \tag{4.8}$$

Equations 4.7 and 4.8 can be combined to give

$$q_g = \begin{cases} C_\ell(z_g - z_b - H) & \text{if } z_g > z_b - M_\ell, \\ -C_\ell(M_\ell + H) & \text{if } z_g \leq z_b - M_\ell. \end{cases} \tag{4.9}$$

Flow Through and Under Levees

Seepage flow through and under levees can have a appreciable effect on the mass balance in a wetland system. One-dimensional seepage flow through and under a levee is modeled by dividing the levee into sections called levee segments. Each levee segment is described by either one or two boundary nodes and the base width, top width, height, and length of the levee segment. If water is allowed to flow under the levee, the thickness of the levee-segment sublayer must be given. Additionally, the permeabilities of the levee segment and the sublayer are required. Two boundary nodes are needed, one on each side of the segment, if the areas on both sides of the levee segment are included in the finite-element network. Water that flows through or under a levee segment defined by two boundary nodes (fig. 4.2) leaves the network at the upstream node (the node with the higher water-surface elevation) and

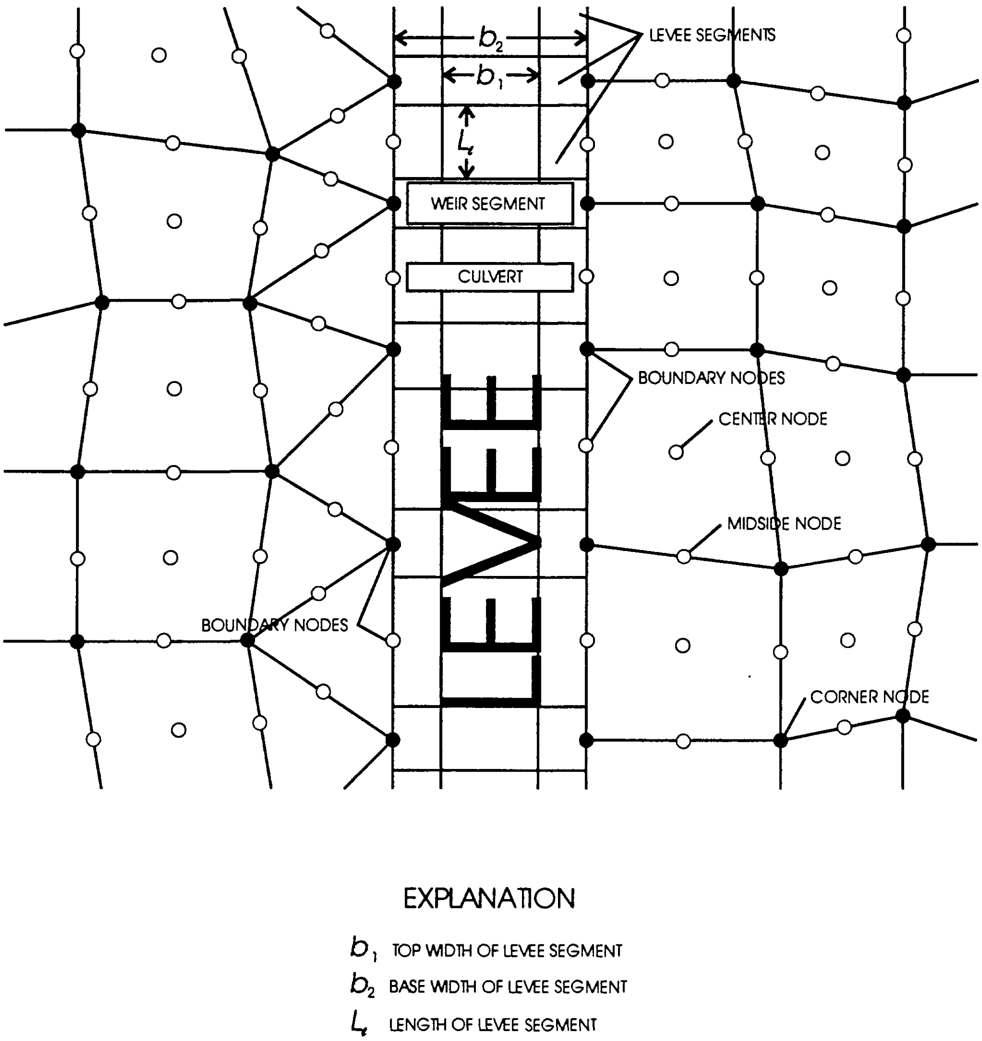


Figure 4.2: Finite-element network with levee segments, a weir segment, and a culvert.

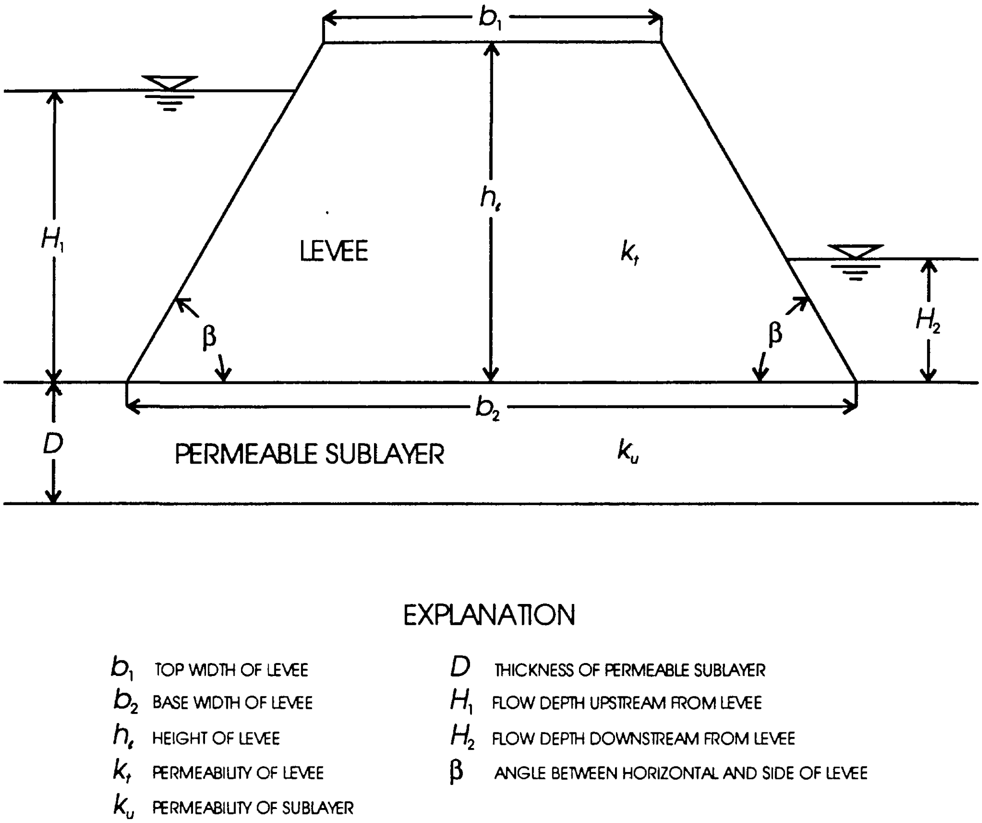


Figure 4.3: Levee cross section.

enters the network at the downstream node. If the area on only one side of a levee segment is included in the network, only one boundary node is needed for each levee segment. In this case, the water-surface elevation outside the network must be specified, and flow into or out of the network is determined on the basis of the interior and specified exterior water-surface elevations. Levee segments, weir segments, and culverts may share the same nodes, as shown in figure 4.2.

A definition sketch of the cross section of a permeable levee with a permeable sublayer is shown in figure 4.3. The angle, β, between the horizontal and the side of the levee is assumed to be the same for both sides. Then

$$\tan\beta = \frac{2h_\ell}{b_2 - b_1} \tag{4.10}$$

or

$$\beta = \arctan\left(\frac{2h_\ell}{b_2 - b_1}\right), \tag{4.11}$$

where h_ℓ = height of the levee (L), b_2 = base width of the levee (L), and b_1 = top width of the levee (L). The symbol D denotes the thickness of the permeable sublayer (L), and k_t and k_u denote the permeabilities of the levee and the sublayer, respectively (L/T).

Peter (1982, p. 123) presents the following equation for the total seepage flow, q, per unit length of levee segment (L^2/T):

$$q_\ell = q_t + q_u, \tag{4.12}$$

where

$$q_t = k_t \frac{H_1^2 - H_{2s}^2}{2d} \tag{4.13}$$

and (Department of the Army, 1978, p. B-13)

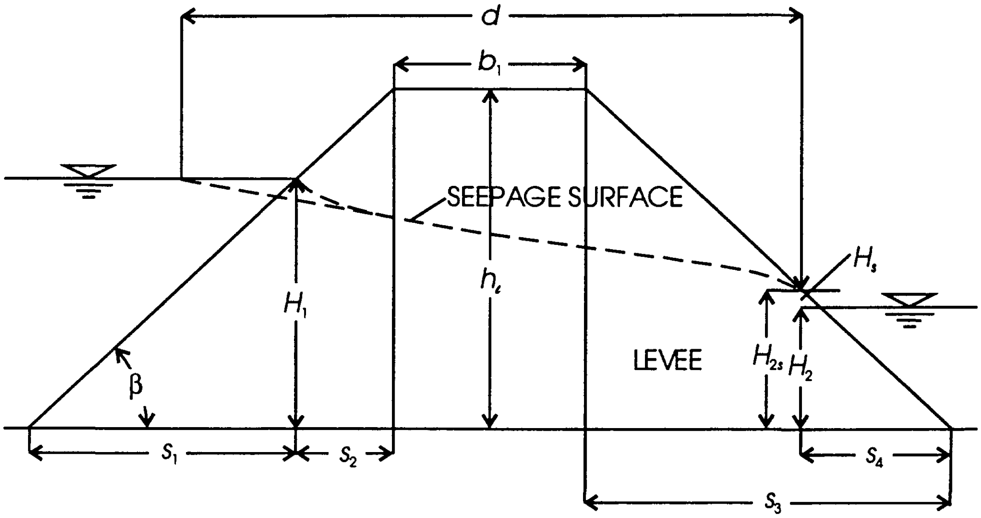
$$q_u = k_u \frac{(H_1 - H_2)D}{b_2\left(1 + 0.86\frac{D}{b_2}\right)}. \tag{4.14}$$

Here, q_t and q_u = seepage flow per unit length through and under the levee segment, respectively (L^2/T)); H_1 = depth of water on the upstream side of the levee segment (L); H_2 = depth of water on the downstream side of the levee segment (L); and

$$H_{2s} = H_s + H_2, \tag{4.15}$$

where H_s = height above the downstream water surface at which the seepage surface emerges on the downstream face of the levee (L) (fig. 4.4). It can be shown (Peter, 1982, p. 107) that the lower part of the seepage surface can be approximated by a parabola. The variable d in equation 4.13 is the length (L) of the projection on the horizontal of the line connecting the focus of the parabola and the point where the tangent of inflection to the parabola intersects the water surface (fig. 4.4). Thus, d can be approximated as

$$d = 0.3s_1 + s_2 + b_1 + s_3 - s_4. \tag{4.16}$$



EXPLANATION

b_1	TOP WIDTH OF LEVEE	S_4	VARIABLE DEFINED IN EQUATION 4.20
d	VARIABLE DEFINED IN EQUATION 4.16	H_1	FLOW DEPTH UPSTREAM FROM LEVEE
h_ℓ	HEIGHT OF LEVEE	H_2	FLOW DEPTH DOWNSTREAM FROM LEVEE
S_1	VARIABLE DEFINED IN EQUATION 4.17	H_{2s}	VARIABLE DEFINED IN EQUATION 4.15
S_2	VARIABLE DEFINED IN EQUATION 4.18	H_s	VARIABLE DEFINED IN EQUATION 4.21
S_3	VARIABLE DEFINED IN EQUATION 4.19	β	ANGLE BETWEEN HORIZONTAL AND SIDE OF LEVEE

Figure 4.4: Definition of levee-seepage parameters.

Table 4.1. Definition of the function f.

Angle β, in degrees	$f(\beta)$, dimensionless
10	10.45
20	4.14
25	2.78
30	2.11
40	1.60
50	1.26
60	1.00
70	.83
90	.63

The terms of equation 4.16 can be written as

$$s_1 = H_1 \cot \beta, \tag{4.17}$$

$$s_2 = (h_\ell - H_1) \cot \beta, \tag{4.18}$$

$$s_3 = h_\ell \cot \beta, \tag{4.19}$$

and

$$s_4 = (H_2 + H_s) \cot \beta = H_{2s} \cot \beta, \tag{4.20}$$

where

$$H_s = f(\beta) \frac{(H_1 - H_2)^2}{b_1 + b_2}, \tag{4.21}$$

and the function f (Peter, 1982, p. 111) is defined in table 4.1. Substituting equations 4.17 through 4.21 into equation 4.16 gives

$$d = b_1 + \cot \beta \left[2h_\ell - 0.7H_1 - H_2 - f(\beta) \frac{(H_1 - H_2)^2}{b_1 + b_2} \right]. \tag{4.22}$$

By equations 4.13, 4.15, 4.21, and 4.22,

$$q_t = k_t \frac{H_1^2 - \left[H_2 + f(\beta) \frac{(H_1 - H_2)^2}{b_1 + b_2} \right]^2}{2 \left\{ b_1 + \cot \beta \left[2h_\ell - 0.7H_1 - H_2 - f(\beta) \frac{(H_1 - H_2)^2}{b_1 + b_2} \right] \right\}}. \tag{4.23}$$

Addendum to Section 5. Finite-Element Equations

Residual Expressions

The residual at node i of the mass-conservation equation is given in equation 5-3 of the original manual (Froehlich, 1989). The source/sink terms given in equation 4.6 are subtracted from the terms inside the square brackets of equation 5-3.

Derivative Expressions

The only source/sink term contributing to a derivative expression for the mass-conservation equation is the ground-water inflow or outflow term. The term C_ℓ is added to the terms inside the first pair of square brackets of equation 5-20, the derivative of the residual of the mass-conservation equation at node i with respect to the depth, H_j, at node j.

Application of Boundary and Special Conditions—Total Flow Across a Boundary

Modifying equation 5-35 of the original manual (Froehlich, 1989), flow across a closed boundary (referred to as a solid boundary in the orginal manual) at node i can be represented as

$$Q_i^c = Q_{si}^c + Q_{wi} + Q_{ci} + Q_{\ell i}, \tag{5.1}$$

where Q_i^c = total flow across the closed boundary at node i (L^3/T), Q_{si}^c = specified flow normal to the closed boundary at node i (L^3/T), Q_{wi} = total weir flow across the closed boundary at node i (L^3/T), Q_{ci} = total culvert flow across the closed boundary at node i (L^3/T), and $Q_{\ell i}$ = total levee-seepage flow across the closed boundary at node i (L^3/T).

The constraint equation for flow normal to the closed boundary at node i is, from equation 5-39 in the original manual,

$$f'_{2i} = a_i^c U_i + b_i^c V_i - Q_i^c = 0, \tag{5.2}$$

where a_i^c, b_i^c = coefficients defined by equations 5-44 and 5-45 in the original manual (L^2) and U_i, V_i = depth-averaged velocity components at node i (L/T). Partial Newton-Raphson iteration is used to treat the constraint equation; that is, only one nodal depth at a time is treated as variable in the constraint equation; the other nodal depth, if it exists, is treated as constant and is given the value obtained during the previous iteration. However, when two nodes are involved, each constraint equation appears twice, and each nodal depth will be variable in one of the equations.

The derivatives of the constraint equation with respect to U_j and V_j are given by equations 5-51 and 5-52 of the original manual, and the derivative of the constraint equation with respect to H_j becomes

$$\frac{\partial f'_{2i}}{\partial H_j} = \frac{\partial a_i^c}{\partial H_j}U_i + \frac{\partial b_i^c}{\partial H_j}V_i - \frac{\partial Q_{wi}}{\partial H_j} - \frac{\partial Q_{ci}}{\partial H_j} - \frac{\partial Q_{\ell i}}{\partial H_j}, \tag{5.3}$$

where all the coefficients on the right-hand side of equation 5.3 except the last are given by equations 5-54 through 5-57 in the original manual. The expression $Q_{\ell i}$ in equations 5.1 and 5.3 can be written as

$$Q_{\ell i} = L_{\ell i} q_{\ell i} = L_{\ell i}(q_{ti} + q_{ui}), \tag{5.4}$$

where $L_{\ell i}$ = length of the levee segment at node i (L), $q_{\ell i}$ = flow per unit length through and under the levee segment at node i (L^2/T), q_{ti} = flow per unit length through the levee segment at node i (L^2/T), and q_{ui} = flow per unit length under the levee segment at node i (L^2/T). To simplify the notation, the subscript i is dropped in the remainder of this subsection. The numerator, $\mathcal{N}$, and the denominator, $\mathcal{D}$, of the expression for q_t are, by equation 4.23,

$$\mathcal{N} = k_t\left\{H_1^2 - \left[H_2 + f(\beta)\frac{(H_1 - H_2)^2}{b_1 + b_2}\right]^2\right\} \tag{5.5}$$

and

$$\mathcal{D} = 2\left\{b_1 + \cot\beta\left[2h_\ell - 0.7H_1 - H_2 - f(\beta)\frac{(H_1 - H_2)^2}{b_1 + b_2}\right]\right\}. \tag{5.6}$$

Now assume that the nodal variable $H_j = H_1$, the depth corresponding to the higher water-surface elevation at the levee segment. Then, by equations

5.4, 5.5, 5.6, and 4.14,

$$\frac{\partial Q_\ell}{\partial H_j} = \frac{\partial Q_\ell}{\partial H_1} = L_\ell \frac{\dfrac{\partial \mathcal{N}}{\partial H_1}\mathcal{D} - \mathcal{N}\dfrac{\partial \mathcal{D}}{\partial H_1}}{\mathcal{D}^2} + L_\ell k_u \frac{D}{b_2\left(1 + 0.86\dfrac{D}{b_2}\right)}, \tag{5.7}$$

where

$$\frac{\partial \mathcal{N}}{\partial H_1} = k_t \left\{ 2H_1 - 2\left[H_2 + f(\beta)\frac{(H_1 - H_2)^2}{b_1 + b_2}\right]\left[2f(\beta)\frac{H_1 - H_2}{b_1 + b_2}\right]\right\} \tag{5.8}$$

and

$$\frac{\partial \mathcal{D}}{\partial H_1} = 2\cot\beta\left[-0.7 - 2f(\beta)\frac{H_1 - H_2}{b_1 + b_2}\right]. \tag{5.9}$$

If $H_j = H_2$, the depth corresponding to the lower water-surface elevation at the levee segment, then

$$\frac{\partial Q_\ell}{\partial H_j} = \frac{\partial Q_\ell}{\partial H_2} = L_\ell \frac{\dfrac{\partial \mathcal{N}}{\partial H_2}\mathcal{D} - \mathcal{N}\dfrac{\partial \mathcal{D}}{\partial H_2}}{\mathcal{D}^2} - L_\ell k_u \frac{D}{b_2\left(1 + 0.86\dfrac{D}{b_2}\right)}, \tag{5.10}$$

where

$$\frac{\partial \mathcal{N}}{\partial H_2} = -2k_t \left[H_2 + f(\beta)\frac{(H_1 - H_2)^2}{b_1 + b_2}\right]\left[1 - 2f(\beta)\frac{H_1 - H_2}{b_1 + b_2}\right] \tag{5.11}$$

and

$$\frac{\partial \mathcal{D}}{\partial H_2} = 2\cot\beta\left[-1 + 2f(\beta)\frac{H_1 - H_2}{b_1 + b_2}\right]. \tag{5.12}$$

Addendum to Section 6. Modeling System Operation

Data Collection

Additional data may be required if the optional model features described in this addendum are to be used in model implementation. These data may include vegetation characteristics (for wind-sheltering effects), precipitation rates, and evapotranspiration rates. If ground-water seepage is to be considered, the thickness and conductance of the low-permeability layer and ground-water heads must be obtained. Modeling levee seepage requires collection of geometric data, permeabilities, and, possibly, water-surface elevations outside the network.

Network Design

General Network Layout

After model boundaries have been defined, as discussed in the orginal manual, a map of the water body to be modeled is obtained. The scale and detail of the base map depend on the detail and accuracy sought in the solution. A series of overlays of the base map are made. On each overlay, the model domain is subdivided into relatively large subdomains, in each of which a model parameter or set of parameters is approximately uniform. The parameters or parameter sets for turbulence, friction, and, optionally, vegetation, precipitation, evapotranspiration, and ground-water seepage are considered in this process. The subdivision lines between the regions are located, as much as possible, where abrupt changes in the selected parameter or parameter set occur. The union of all the subdivision lines defined by this series of overlays is used as the basis for further subdivision of the model domain into elements. The series of overlays can be used as the basis for defining element codes for turbulence, friction, vegetation, precipitation, evapotranspiration, and ground-water seepage.

One-Dimensional Levees

One-dimensional seepage flow through and under a levee is treated as a point flow on the boundary of a finite-element network. A point flow is the total flow that crosses the network boundary because of flow at a single node. A one-dimensional levee segment is described by a set of parameters and two nodes, one on each side of the levee, if the areas on both sides of the levee segment are included in the finite-element network, or one node, if only one side of the levee segment is included in the network. Flow through and under the levee segment is computed on the basis of the water-surface elevations at the two nodes (or the water-surface elevation at the interior node and a specified external water-surface elevation). The following items must be specified for each levee segment (see figs. 4.2 and 4.3): (1) the base width of the levee segment (b_2), (2) the top width of the levee segment (b_1), (3) the height of the levee segment (h_ℓ), (4) the length of the levee segment (L_ℓ), (5) the thickness of the levee-segment sublayer (D), (6) the permeability of the levee segment (k_t), and (7) the permeability of the levee-segment sublayer (k_u).

To model flow through and under a levee contained within the flow domain, the finite-element network must be designed so that closed boundaries are located on both sides of the levee. The levee is divided into a number of levee segments, and appropriate parameters are assigned to each segment. The number of segments used to divide the levee depends on the variation of the levee geometry and permeability and the spacing of the nodes on the closed boundaries that define the levee. Nodes that define the sides of a levee segment must be located approximately at the center of the levee segment. Flow will be allowed to leave the network at the upstream node (the node with the higher water-surface elevation) and reenter the network at the downstream node. If only one side of the levee is included in the finite-element network, flow through and under the levee will be determined on the basis of the interior water-surface elevation and a specified exterior water-surface elevation. The determination of levee segments is governed by the considerations discussed above. The location of levee segments must be considered during the initial design of a finite-element network. It is good practice to align nodes on opposite sides of a levee as shown in figure 4.2. A single point can be used to define the side or end of more than one weir segment, culvert, or levee segment, as shown in figure 4.2.

Assigning two-thirds of the side length of an element to the midside node and one-sixth of the side length to each vertex node of that side will result in equal flows across the network boundary at each of the nodes if water-surface elevations and levee parameters are constant along the element side. The lengths of levee segments assigned in this manner will alternate in size.

Addendum to Section 8. Input Data Description

Depth-Averaged Flow Module: FLOMOD

Several new features are found in the input data to FLOMOD, the FORTRAN solution module. Input element incidence lists can be read in either I5 or I6 format. The element-property code and data set are replaced with separate element-property codes and data sets for turbulence-model coefficients, friction coefficients, vegetation characteristics, precipitation rate, evapotranspiration rate, and ground-water-seepage parameters. A linearized friction coefficient is added to the friction data set. Levee and ground-water-head data sets are added. General wind and ground-water-head data are added to the time-dependent data set. Updated turbulence-model coefficients, friction coefficients, wind data, vegetation characteristics, precipitation rate, evapotranspiration rate, ground-water-seepage parameters, ground-water heads, and boundary conditions, including total flows and water-surface elevations at specified cross sections, and external water-surface elevations at boundary culverts, weir segments, and levee segments, can be read at any time step. Finally, several errors and omissions in the input data description of the original users manual are corrected.

Data sets read by FLOMOD are preceded by the following identification records.

DATA-SET IDENTIFICATION RECORD

Variable IDS	Data to be entered
SWMS[1]	Program-control data
ELEM	Element data
NODE	Node data
TURB	Turbulence-model data
FRIC	Friction data
WIND	Wind data
VEGE	Vegetation data
PREC	Precipitation data

Variable IDS	Data to be entered
EVAP	Evapotranspiration data
GRWT	Ground-water-seepage data
HEAD	Ground-water-head data
INIT	Initial-condition data
BOUN	Boundary-condition data
QSEC	Total-flow-cross-section data
ZSEC	Water-surface-elevation-cross-section data
WEIR	Weir data
CULV	Culvert data
LEVE	Levee data
FLUX	Flow-check data
TIME[2]	Time-dependent data
LAST[3]	Last record in the input data stream

[1]This must be the first data set in the input data stream.

[2]Time-dependent data sets must appear in chronological order at the end of the input data stream.

[3]This must be the last record in the input data stream.

Program-Control Data Set

Program-control data records immediately follow an SWMS-data-set identification record. The identification record contains the data-set identification code and codes that control the width of the printed output, the printing of screen messages, and the formats of input element data and output flow data. The other six records of the data set contain information that controls the overall operation of the program. Record 1 contains a job title, which will be used in printed output headings. Record 2 contains job option codes. Record 3 contains input/output file specifications. Record 4 contains iteration control data and time-dependent-solution parameters. Record 5 contains general system specifications. Record 6 contains general wind and ground-water data.

SWMS IDENTIFICATION RECORD—FORMAT(A4,6X,4E10.0)

Columns	Variable	Value	Description
1 to 4	IDS	SWMS	Program-control-data-set identification code
5 to 10	—	—	Not used
11 to 20	WIDE	0	80-column format will be used for printed output (default)
		1	132-column output will be used for printed output
21 to 30	SCREEN	0	Messages that describe program operations will not be written to the terminal screen (default)
		1	Messages that describe program operations will be written to the terminal screen

Columns	Variable	Value	Description
31 to 40	GRIDF	0	I5 format will be used for input element data (default)
		1	I6 format will be used for input element data
41 to 50	FLOWF	0	E10.3 format will be used for output flow data (default)
		1	E12.5 format will be used for output flow data

SWMS RECORD 1—FORMAT(A80)

Columns	Variable	Value	Description
1 to 80	TITLE	a/n	Alphanumeric characters to be used in printed output headings. Usually the title will include the name of the water body being modeled

SWMS RECORD 2—FORMAT(15I5)

Columns	Variable	Value	Description
1 to 5	IDRUN	0	A normal steady-state or time-dependent solution will be performed
		1	A restart/recovery run will be performed. Initial conditions will be read from a restart/recovery file. The default name of the restart/recovery file is RSRC.DAT
6 to 10	IPRNT	0 to 31	Sum of the following printed-output options that are desired:

Columns	Variable	Value	Description
			Options
		0	Control data, error messages, and solution results will be printed
		1	All input data read from data records will be echo printed
		2	Element and node data will be printed
		4	Initial-condition data will be printed
		8	Element assembly sequence will be printed
		16	Degree-of-freedom array that contains equation numbers that correspond to each nodal variable will be printed
11 to 15	IUNIT	0	Inch/pound units will be used in all computations and printed output
		1	International System (SI) units will be used in all computations and printed output
16 to 20	IWIND	0	Wind-induced surface stresses will not be considered
		1	Wind-induced surface stresses will be considered
21 to 25	IFRIC	0	Bottom stresses will be computed using the Manning equation

Columns	Variable	Value	Description
		1	Bottom stresses will be computed using the Chézy equation
		2	Bottom stresses will be computed using linearized friction
26 to 30	IVEGE	0	Vegetative sheltering of the water surface from wind will not be considered
		1	Vegetative sheltering of the water surface from wind will be considered
31 to 35	IPREC	0	Precipitation will not be considered
		1	Precipitation will be considered
36 to 40	IEVAP	0	Evapotranspiration will not be considered
		1	Evapotranspiration will be considered
41 to 45	IGRWT	0	Ground-water seepage will not be considered
		1	Ground-water seepage will be considered
46 to 50	ISLIP	0	Slip (tangential-flow/zero-tangential-shear) conditions will be applied automatically at all closed-boundary nodes
		1	No-slip (zero-flow) conditions will be applied automatically at all closed-boundary nodes

Columns	Variable	Value	Description
51 to 55	IHINT	0	Low-order numerical integration will be used on all elements
		1	High-order numerical integration will be used on all curve-sided elements
		2	High-order numerical integration will be used on all elements
56 to 60	INORM	0 to 3	Sum of the following continuity-norm options that are desired:
			Options
		0	Continuity norms will not be computed
		1	Continuity norms will be computed at the end of a steady-state solution
		2	Continuity norms will be computed at the end of every time step of a time-dependent solution
61 to 65	IONOFF	0	Elements will not be turned on and off during a run
		1	Elements will be turned on and off during a run

Columns	Variable	Value	Description
			Note: Use of the automatic-boundary-adjustment option allows elements that are not covered entirely by water to be included in a network. If IONOFF = 1, all "dry" elements will be excluded from the "active" network
66 to 70	ISAVE	0	Files that contain the upper and lower decompositions of the coefficient matrix will be deleted at the end of a run
		1	Files that contain the upper and lower decompositions of the coefficient matrix will be saved at the end of a run
71 to 75	NPVMIN	+	Minimum number of completed equations retained in the active matrix during a frontal solution. NPVMIN > 1 provides a choice of pivotal coefficients but will increase the number of computations and the frontwidth of the system of equations. The default value is 1

SWMS RECORD 3—FORMAT(9I5)

Columns	Variable	Value	Description
1 to 5	IGRID	0	All finite-element-network data will be entered on data records
		1	Finite-element-network data will be read from a data file. Additional network data may be entered on data records. The default name of the network data file is GRID.DAT

Columns	Variable	Value	Description
6 to 10	INITC	0	All initial-condition data will be entered on data records
		1	Initial-condition data will be read from a data file. Additional initial-condition data may be entered on data records. The default name of the initial-condition data file is INIT.DAT Note: An initial-condition data file usually will be a flow (solution-output) data file created by a previous run
11 to 15	ISOUT	0	Solution output will not be written to a data file
		±	Solution output will be written to a data file at the end of a steady-state run and at the end of every time step of a time-dependent run. If ISOUT > 0, data will be written in "text" form; if ISOUT < 0, data will be written in "binary" form. The default name of the flow data file is FLOW.DAT
16 to 20	IRSRC	0	A restart/recovery file will not be used

Columns	Variable	Value	Description
		±	Intermediate results will be written to a restart/recovery file after every iteration to allow a run to be restarted from the last successful iteration if the run terminates abnormally. If IRSRC > 0, data will be written in "text" form; if IRSRC < 0, data will be written in "binary" form. The default name of the restart/recovery file is RSRC.DAT
21 to 25	IBCIN	0	All boundary-condition data will be entered on data records
		+	Boundary-condition data will be read from a data file. Additional boundary-condition data may be entered on data records. The default name of the boundary-condition data file is BOUN.DAT
26 to 30	IWDIN	0	All wind data will be entered on data records
		+	Wind data will be read from a data file. Additional wind data may be entered on data records. The default name of the wind data file is WIND.DAT
31 to 35	IHDIN	0	All ground-water-head data will be entered on data records

Columns	Variable	Value	Description
		+	Ground-water-head data will be read from a data file. Additional ground-water-head data may be entered on data records. The default name of the ground-water-head data file is HEAD.DAT
36 to 40	ILCOF	+	Unit number of the "binary" file that will contain the lower triangular decomposition of the coefficient matrix. The default unit number is 98. The file is written only if quasi-Newton iterations are to be performed or if the upper and lower triangular matrices are to be saved at the end of a run. The default name of the lower-coefficient-matrix file is LOWER.DAT
41 to 45	IUCOF	+	Unit number of the "binary" file that will contain the upper triangular decomposition of the coefficient matrix. The default unit number is 99. The file is always written. The default name of the upper-coefficient-matrix file is UPPER.DAT

SWMS RECORD 4—FORMAT(4I10,4E10.0)

Columns	Variable	Value	Description
1 to 10	NITS	0	A steady-state solution will not be performed
		+	Steady-state-solution iteration code read as *KKJJII*, where: *II* = number of initial full-Newton iterations to be performed (usually 2 or 3)

Columns	Variable	Value	Description
			JJ = number of quasi-Newton iterations to be performed after all the initial full-Newton iterations and after each additional full-Newton iteration KK = number of additional full-Newton iterations to be performed Note: The maximum allowable number of steady-state iterations is 99. Therefore, $II + JJ \times (1 + KK)$ must be less than or equal to 99
11 to 20	`NITD`	0	A time-dependent solution will not be performed
		+	Time-dependent-solution iteration code read as $KKJJII$, where II, JJ, and KK for each time step are the same as for a steady-state solution. The maximum total number of iterations at a time step is 99
21 to 30	`NUPV`	0 to 5	Number of update vectors to be used in a quasi-Newton solution. If `NUPV` = 0, no update vectors are used, and modified-Newton iteration results. The maximum value is 5. The default value is 0 (that is, a modified-Newton iteration is the default)
31 to 40	`NITP`	+	Iteration print code read as $KKJJII$, where:

Columns	Variable	Value	Description
			II = number of iterations skipped between printed output during a steady-state solution JJ = number of iterations skipped between printed output during each time step of a time-dependent solution KK = number of time steps skipped between printed output during a time-dependent solution Note: The default value for II, JJ, and KK is 0 (that is, output will be printed at the end of every iteration of both steady-state and time-dependent solutions and for every time step of a time-dependent solution)
41 to 50	TSTRT	+	Starting simulation time, in hours
51 to 60	TMAX	0	A time-dependent solution will not be performed
		+	Maximum simulation time, in hours, for a time-dependent solution
61 to 70	DELT	+	Length, in hours, of each time step used in a time-dependent solution
71 to 80	THETA	+	Time-integration factor (dimensionless). The minimum value is 0.5; the maximum value is 1.0

SWMS RECORD 5—FORMAT(8E10.0)

Columns	Variable	Value	Description
1 to 10	WSEL	+	Water-surface elevation, in feet (meters), assigned to each node in a network that has not been assigned an initial water-surface elevation. WSEL can be used to assign a constant water-surface elevation for a cold start
11 to 20	OMEGA	0	Effect of the Coriolis force will not be considered
		±	Average local latitude, in degrees, of the surface-water body being modeled. OMEGA is positive in the northern hemisphere and negative in the southern hemisphere
21 to 30	ROWAT	+	Average water density, in slugs per cubic foot (kilograms per cubic meter). The default value is 1.937 slug/ft^3 (999.0 kg/m^3)
31 to 40	BETAO	+	Coefficient β_0 (dimensionless) in equation 4-8 (of the original users manual) used to compute the momentum-correction coefficient. The default value is is 1.0
41 to 50	CBETA	+	Coefficient c_β (dimensionless) in equation 4-8 (of the original users manual) used to compute the momentum-correction coefficient. The default value is 0

Columns	Variable	Value	Description
51 to 60	CFLAG	+	Continuity-norm flag value. Continuity norms greater than CFLAG will be flagged with an asterisk. Appropriate values are problem dependent. The default value is 1.0E+35 (that is, continuity norms will not be flagged)
61 to 70	DEPTOL	+	Depth tolerance, in feet (meters), used during automatic boundary adjustment to decide whether or not to turn on an element. The default value is 0.5 ft (0.15 m)
71 to 80	RELAX	0 to 2	Relaxation factor (dimensionless) used in equation solution (ω_r in equation 7-6 of the original users manual). The default value is 1.0

SWMS RECORD 6—FORMAT(7E10.0)

Columns	Variable	Value	Description
1 to 10	WVEL	+	Wind velocity, in feet per second (meters per second). This value will be assigned to each node in the network unless overridden
11 to 20	WDIR	+	Wind-direction angle, in degrees, measured counterclockwise from the positive x-axis. This is the direction toward which the wind is blowing. This value will be assigned to each node in the network unless overridden

Columns	Variable	Value	Description
21 to 30	ROAIR	+	Air density, in slugs per cubic foot (kilograms per cubic meter). The default value is 0.00237 slug/ft^3 (1.225 kg/m^3)
31 to 40	CSURF1	+	Coefficient c_{s1} (dimensionless) in equation 4-15 of the original users manual. It is used to compute the wind-stress coefficient. The default value is 1.0
41 to 50	CSURF2	+	Coefficient c_{s2}, in seconds per meter, in equation 4-15 of the original users manual. It is used to compute the wind-stress coefficient. The default value is 0 s/m
51 to 60	WVMIN	+	Minimum wind velocity, W_{min}, in meters per second, used in equation 4-15 of the original users manual. It is used to compute the wind-stress coefficient. The default value is 0 m/s
61 to 70	GWHEAD	+	Ground-water head, in feet (meters). This value will be assigned to each node in the network unless overridden

Element Data Set

Element data records immediately follow an ELEM-data-set identification record. One record is required for each element. An element data record contains the element number, the sequence of nodes connected to the element (the element connectivity list), the turbulence-model code, the friction code, the vegetation code, the precipitation code, the evapotranspiration code, the ground-water-seepage code, and the element assembly sequence. Two options for the ELEM-record format are available (see the variable GRIDF on the SWMS identification record of the program-control data set). The second option can be used to avoid the overlapping of node numbers when node numbers exceed 10,000. The data set is terminated with one or more blank records.

ELEM IDENTIFICATION RECORD—FORMAT(A4)

Columns	Variable	Value	Description
1 to 4	IDS	ELEM	Element-data-set identification code

ELEM RECORD—FORMAT((11I5,5I3,I5) or (10I6,I5,5I2,I5))

Columns	Variable	Value	Description
1 to 5 (1 to 6)	L	+	Element number. It must be less than or equal to the element-array dimension
6 to 50 (7 to 60)	NOP(L,K)	+	Element connectivity list, the sequence of nodes connected to the element: six node numbers for a triangular element, eight node numbers for a serendipity quadrangular element, or nine node numbers for a Lagrangian quadrangular element. The list starts at any corner node and proceeds in a counterclockwise direction around the element. For a nine-node quadrangular element, the center node is entered last

Columns	Variable	Value	Description
51 to 55 (61 to 65)	LTRB(L)	±	Turbulence code. The code corresponds to a set of turbulence-model coefficients that are entered in the TURB data set. If LTRB(L) $<$ 1, the element will be turned off (that is, the element will not be used in computations). If LTRB(L) $>$ 1000, pressure flow will be permitted at the element
56 to 58 (66 to 67)	LFRC(L)	+	Friction code. The code corresponds to a set of friction coefficients that are entered in the FRIC data set
59 to 61 (68 to 69)	LVEG(L)	+	Vegetation code. The code corresponds to a set of vegetation characteristics that are entered in the VEGE data set
62 to 64 (70 to 71)	LPRC(L)	+	Precipitation code. The code corresponds to a precipitation rate that is entered in the PREC data set
65 to 67 (72 to 73)	LEVP(L)	+	Evapotranspiration code. The code corresponds to an evapotranspiration rate that is entered in the EVAP data set
68 to 70 (74 to 75)	LGWT(L)	+	Ground-water-seepage code. The code corresponds to a set of ground-water-seepage parameters that are entered in the GRWT data set

Columns	Variable	Value	Description
71 to 75 (76 to 80)	LSEQ(L)	+	Element assembly sequence. The sequence number of the element for processing by the frontal method. An efficient element sequence can be generated by using the resequencing option of the preprocessing program

Terminate the ELEM data set with one or more blank records.

Node Data Set

Node data records immediately follow a NODE-data-set identification record. The identification record contains the data-set identification code and factors used to convert node-point coordinates and ground and ceiling (Froehlich, 1989, p. 1-4-21) elevations read from data records to the desired units (either feet or meters). A node data record contains the node number, the x- and y-coordinates of the node, and the ground-surface and ceiling elevations at the node. A node data record is required for each vertex node. Curved element sides are specified by also entering coordinate data for the midside node of the element side. Note that because bed and ceiling elevations are interpolated within an element using linear functions, values for these variables are needed only at vertex nodes. Therefore, bed and ceiling elevations specified at midside and center nodes will be ignored. The data set is terminated with one or more blank records.

NODE IDENTIFICATION RECORD—FORMAT(A4,6X,6E10.0)

Columns	Variable	Value	Description
1 to 4	IDS	NODE	Node-data-set identification code
5 to 10	—	—	Not used
11 to 20	XFACT	+	Multiplication factor used to convert x-coordinates read from data records to feet (meters). The default value is 1.0
21 to 30	YFACT	+	Multiplication factor used to convert y-coordinates read from data records to feet (meters). The default value is 1.0
31 to 40	ZFACT	+	Multiplication factor used to convert ground and ceiling elevations read from data records to feet (meters). The default value is 1.0

Columns	Variable	Value	Description
41 to 50	XZERO	+	Feet (meters) to be added to all x-coordinates read from data records after multiplication by XFACT. The default value is 0 ft (m)
51 to 60	YZERO	+	Feet (meters) to be added to all y-coordinates read from data records after multiplication by YFACT. The default value is 0 ft (m)
61 to 70	ZZERO	+	Feet (meters) to be added to all ground and ceiling elevations read from data records after multiplication by ZFACT. The default value is 0 ft (m)

NODE RECORD—FORMAT(I10,4E10.0)

Columns	Variable	Value	Description
1 to 10	N	+	Node number. It must be less than or equal to the node-array dimension
11 to 20	CORD(N,1)	+	The x-coordinate of node N
21 to 30	CORD(N,2)	+	The y-coordinate of node N
31 to 40	GRND(N)	+	Ground-surface elevation at node N
41 to 50	CEIL(N)	+	Ceiling elevation at node N
			Note: Coordinates and elevations are converted to feet (meters) by the factors specified on the data-set identification record. Therefore, any system of units can be used to record coordinates and elevations

Terminate the NODE data set with one or more blank records.

Turbulence-Model Data Set

Turbulence-model data records immediately follow a TURB-data-set identification record. One record is required for each set of turbulence-model coefficients. The coefficients in this record are applied to all elements that have been assigned the turbulence code of the record. The data set is terminated with one or more blank records.

TURB IDENTIFICATION RECORD—FORMAT(A4)

Columns	Variable	Value	Description
1 to 4	IDS	TURB	Turbulence-model-data-set identification code

TURB RECORD—FORMAT(I10,2E10.0)

Columns	Variable	Value	Description
1 to 10	M	+	Turbulence code Note: Elements can be effectively removed from the network by assigning the turbulence-model code a negative value. All elements having the negative code will be "turned off" during the computations Note: Pressure flow will be permitted at elements assigned a code greater than 1,000
11 to 20	PTRB(M,1)	+	Turbulence-model base kinematic eddy viscosity, $\hat{\nu}_0$, in square feet per second (square meters per second), used in equation 4-19 of the original users manual
21 to 30	PTRB(M,2)	+	Turbulence-model coefficient, c_μ (dimensionless), used in equation 4-19 of the original users manual

Terminate the TURB data set with one or more blank records.

Friction Data Set

Friction data records immediately follow a FRIC-data-set identification record. One record is required for each set of friction coefficients. A friction data record contains a friction code, a Manning roughness coefficient as a function of depth, a Chézy discharge coefficient, and a linear friction coefficient. The coefficients in this record are applied to all elements that have been assigned the friction code of the record. The data set is terminated with one or more blank records.

FRIC IDENTIFICATION RECORD—FORMAT(A4)

Columns	Variable	Value	Description
1 to 4	IDS	FRIC	Friction-data-set identification code

FRIC RECORD—FORMAT(I10,6E10.0)

Columns	Variable	Value	Description
1 to 10	M	+	Friction code
11 to 20	PFRC(M,1)	+	Manning roughness coefficient applied to all water depths less than or equal to the depth entered in the next field
21 to 30	PFRC(M,2)	+	Water depth, in feet (meters), below which the roughness coefficient entered in the previous field is applied
31 to 40	PFRC(M,3)	+	Manning roughness coefficient applied to all water depths greater than or equal to the depth entered in the next field
41 to 50	PFRC(M,4)	+	Water depth, in feet (meters), above which the roughness coefficient entered in the previous field is applied

Columns	Variable	Value	Description
			Note: For depths greater than the first depth and less than the second depth entered above, the roughness coefficient is linearly interpolated. If the second coefficient is zero or blank, the first coefficient is applied to all depths
51 to 60	PFRC(M,5)	+	Chézy discharge coefficient (dimensionless) used for all water depths. This coefficient is multipled by the square root of gravitational acceleration to obtain the dimensional value used in calculations
61 to 70	PFRC(M,6)	+	Coefficient of linear friction, in seconds per foot (seconds per meter), used in equation 4.3

Terminate the FRIC data set with one or more blank records.

Wind Data Set

An optional wind data set, identified as WIND, is used if wind is to be modeled. Wind data records immediately follow a WIND-data-set identification record. One record is required for each node at which conditions other than general wind specifications are desired. For a time-dependent (unsteady) run, only values that change from the previous time step need to be specified. A wind data record contains the node number and the specified wind velocity and direction. The data set is terminated with one or more blank records.

WIND IDENTIFICATION RECORD—FORMAT(A4)

Columns	Variable	Value	Description
1 to 4	IDS	WIND	Wind-data-set identification code

WIND RECORD—FORMAT(I10,2E10.0)

Columns	Variable	Value	Description
1 to 10	N	+	Node number. It must be less than or equal to the node-array dimension
11 to 20	SIGMA(N,1)	+	Wind velocity, in feet per second (meters per second), at node N
21 to 30	SIGMA(N,2)	+	Wind direction, in degrees measured counterclockwise from the positive x-direction, at node N. This is the direction toward which the wind is blowing

Terminate the WIND data set with one or more blank records.

Vegetation Data Set

An optional vegetation data set, identified as VEGE, is used if vegetative sheltering of the water surface from wind is to be modeled. Vegetation data records immediately follow a VEGE-data-set identification record. One record is required for each set of vegetation parameters. A vegetation data record contains a vegetation code, a vegetation density, an average stem or leaf width and height, and a vegetation drag coefficient. The coefficients in this record are applied to all elements that have been assigned the vegetation code of the record. The data set is terminated with one or more blank records.

VEGE IDENTIFICATION RECORD—FORMAT(A4)

Columns	Variable	Value	Description
1 to 4	IDS	VEGE	Vegetation-data-set identification code

VEGE RECORD—FORMAT(I5,4E10.0)

Columns	Variable	Value	Description
1 to 10	M	+	Vegetation code
11 to 20	PVEG(M,1)	+	Plant density, in stems or leaves per square foot (stems or leaves per square meter)
21 to 30	PVEG(M,2)	+	Average stem or leaf width, in feet (meters)
31 to 40	PVEG(M,3)	+	Average stem or leaf height, in feet (meters)
41 to 50	PVEG(M,4)	+	Vegetation drag coefficient (dimensionless)

Terminate the VEGE data set with one or more blank records.

Precipitation Data Set

An optional precipitation data set, identified as PREC, is used if precipitation is to be modeled. Precipitation data records immediately follow a PREC-data-set identification record. One record is required for each precipitation code. A precipitation data record contains a precipitation code and a precipitation rate. The precipitation rate in this record is applied to all elements that have been assigned the precipitation code of the record. The data set is terminated with one or more blank records.

PREC IDENTIFICATION RECORD—FORMAT(A4)

Columns	Variable	Value	Description
1 to 4	IDS	PREC	Precipitation-data-set identification code

PREC RECORD—FORMAT(I10,E10.0)

Columns	Variable	Value	Description
1 to 10	M	+	Precipitation code
11 to 20	PPRC(M)	+	Precipitation rate, in inches per hour (centimeters per hour)

Terminate the PREC data set with one or more blank records.

Evapotranspiration Data Set

An optional evapotranspiration data set, identified as EVAP, is used if evapotranspiration is to be modeled. Evapotranspiration data records immediately follow an EVAP-data-set identification record. One record is required for each evapotranspiration code. An evapotranspiration data record contains an evapotranspiration code and an evapotranspiration rate. The evapotranspiration rate in this record is applied to all elements that have been assigned the evapotranspiration code of the record. The data set is terminated with one or more blank records.

EVAP IDENTIFICATION RECORD—FORMAT(A4)

Columns	Variable	Value	Description
1 to 4	IDS	EVAP	Evapotranspiration-data-set identification code

EVAP RECORD—FORMAT(I10,E10.0)

Columns	Variable	Value	Description
1 to 10	M	+	Evapotranspiration code
11 to 20	PEVP(M)	+	Evapotranspiration rate, in inches per day (centimeters per day)

Terminate the EVAP data set with one or more blank records.

Ground-Water-Seepage Data Set

An optional ground-water-seepage data set, identified as GRWT, is used if ground-water seepage is to be modeled. Ground-water-seepage data records immediately follow a GRWT-data-set identification record. One record is required for each set of ground-water-seepage parameters. A ground-water-seepage data record contains a ground-water-seepage code, a conductance, and the thickness of the low-permeability layer. The parameters in this record are applied to all elements that have been assigned the ground-water-seepage code of the record. The data set is terminated with one or more blank records.

GRWT IDENTIFICATION RECORD—FORMAT(A4)

Columns	Variable	Value	Description
1 to 4	IDS	GRWT	Ground-water-seepage-data-set identification code

GRWT RECORD—FORMAT(I10,2E10.0)

Columns	Variable	Value	Description
1 to 10	M	+	Ground-water-seepage code
11 to 20	PGWT(M,1)	+	Conductance, per second
21 to 30	PGWT(M,2)	+	Thickness of the low-permeability layer below the bed, in feet (meters)

Terminate the GRWT data set with one or more blank records.

Ground-Water-Head Data Set

An optional ground-water-head data set, identified as HEAD, is used if ground-water seepage is to be modeled. Ground-water-head data records immediately follow a HEAD-data-set identification record. One record is required for each node at which a ground-water head other than the ground-water head given in SWMS record 6 is desired. For a time-dependent (unsteady) run, only values that change from the previous time step need to be specified. A ground-water-head data record contains the node number and the ground-water head. Values entered on data records will override those read from a ground-water-head data file. The data set is terminated with one or more blank records.

HEAD IDENTIFICATION RECORD—FORMAT(A4)

Columns	Variable	Value	Description
1 to 4	IDS	HEAD	Ground-water-head-data-set identification code

HEAD RECORD—FORMAT(I10,E10.0)

Columns	Variable	Value	Description
1 to 10	N	+	Node number. It must be less than or equal to the node-array dimension
11 to 20	HEAD(N)	+	Ground-water head, in feet (meters), at node N

Terminate the HEAD data set with one or more blank records.

Initial-Condition Data Set

Initial-condition data records immediately follow an INIT-data-set identification record. One record is prepared for each node at which initial conditions are specified. An initial-condition data record contains the node number, the initial velocities in the x- and y-directions, the depth of flow, and time derivatives of the x- and y-velocities and the depth of flow. Values entered on data records will override those read from an initial-condition data file. Two options for the INIT-record format are available (see the variable **FLOWF** on the **SWMS** identification record of the program-control data set). The second option can be used to retain additional significant digits. The data set is terminated with one or more blank records.

INIT IDENTIFICATION RECORD—FORMAT(A4)

Columns	Variable	Value	Description
1 to 4	IDS	INIT	Initial-condition-data-set identification code

INIT RECORD—FORMAT((I10,6E10.0) or (I7,6E12.5))

Columns	Variable	Value	Description
1 to 10 (1 to 7)	N	+	Node number. It must be less than or equal to the node-array dimension
11 to 20 (8 to 19)	VEL(1,N)	+	Initial x-velocity, in feet per second (meters per second), at node N
21 to 30 (20 to 31)	VEL(2,N)	+	Initial y-velocity, in feet per second (meters per second), at node N
31 to 40 (32 to 43)	VEL(3,N)	+	Initial depth of flow, in feet (meters), at node N
41 to 50 (44 to 55)	VDOT(1,N)	+	Initial x-velocity time rate of change, in feet per second per second (meters per second per second), at node N

Columns	Variable	Value	Description
51 to 60 (56 to 67)	VDOT(2,N)	+	Initial y-velocity time rate of change, in feet per second per second (meters per second per second), at node N
61 to 70 (68 to 79)	VDOT(3,N)	+	Initial flow-depth time rate of change, in feet per second (meters per second), at node N

Terminate the INIT data set with one or more blank records.

Boundary-Condition Data Set

Boundary-condition data records immediately follow a BOUN-data-set identification record. One record is prepared for each boundary node at which conditions other than slip/no-slip are specified. For a time-dependent (unsteady) run, only values that change from the previous time step need to be specified. A boundary-condition data record contains the number of the node to which the data apply, a boundary-condition code, and specified conditions. Either tangential-flow (slip) or zero-flow (no-slip) conditions (as specified by the variable ISLIP on SWMS data record 2) are applied automatically at all boundary nodes unless otherwise specified. Values entered on data records will override those read from a boundary-condition data file. The data set is terminated with one or more blank records.

BOUN IDENTIFICATION RECORD—FORMAT(A4)

Columns	Variable	Value	Description
1 to 4	IDS	BOUN	Boundary-condition-data-set identification code

BOUN RECORD—FORMAT(I10,5X,I5,3E10.0)

Columns	Variable	Value	Description
1 to 10	N	+	Node number. It must be less than or equal to the node-array dimension
11 to 15	—	—	Not used
16	NFIX(N)	1	Velocity in the x-direction at node N is specified
		2	Unit flow in the x-direction at node N is specified
		3	Velocity *tangent* to the boundary at node N is specified

Columns	Variable	Value	Description
		4	Unit flow *tangent* to the boundary at node N is specified
		5	Total flow *normal* to the *open* boundary resulting from flow at node N is specified
17	NFIX(N)	1	Velocity in the y-direction at node N is specified
		2	Unit flow in the y-direction at node N is specified
		3	Velocity *normal* to the boundary at node N is specified
		4	Unit flow *normal* to the boundary at node N is specified
		5	Total flow *normal* to the *closed* boundary resulting from flow at node N is specified
			Note 1: A zero-flow (no-slip) condition is applied at a boundary node by entering a "1" in columns 16 and 17 and specifying the x- and y-velocities to be zero. A tangential-flow (slip) condition is applied at a boundary node by entering a "0" in column 16, a "5" in column 17 and specifying the total flow normal to the boundary to be zero

Columns	Variable	Value	Description
			Note 2: Special consideration must be given nodes where closed and open boundaries meet. Prescribed unit flow or velocity must be parallel to the closed boundary at these nodes. If the water-surface elevation is prescribed, tangential or zero flow must be specified as described in Note 1 above
18	NFIX(N)	1	Water-surface elevation at node N is specified as an *essential* boundary condition
		2	Water-surface elevation at node N is specified as a *natural* boundary condition
		3	*Supercritical* flow exists at the *outflow* boundary node. Water-surface elevation is not specified
19	NFIX(N)	—	Not used. Enter zero
20	NFIX(N)	—	Not used. Enter zero
21 to 30	SPEC(N,1)	+	Specified velocity, in feet per second (meters per second), or unit flow, in square feet per second (square meters per second), in the x-direction or tangent to the boundary at node N

Columns	Variable	Value	Description
31 to 40	SPEC(N,2)	+	Specified velocity, in feet per second (meters per second), or unit flow, in square feet per second (square meters per second), in the y-direction or normal to the boundary at node N; or total flow, in cubic feet per second (cubic meters per second), normal to the boundary at node N Note: If total flow is specified, a positive value indicates flow into a network and a negative value indicates flow out of a network resulting from flow at node N
41 to 50	SPEC(N,3)	+	Specified water-surface elevation, in feet (meters), at node N

Terminate the BOUN data set with one or more blank records.

Total-Flow-Cross-Section Data Set

Total-flow-cross-section data records immediately follow a QSEC-data-set identification record. A set of records is required for each section of the open boundary of a network for which a total flow will be specified. For each cross section, the first record contains a cross-section identification number and the total flow. A list of the nodes that define the cross section is contained on up to five additional records. A maximum of 10 cross sections may be specified. The data set is terminated with one or more blank records.

QSEC IDENTIFICATION RECORD—FORMAT(A4)

Columns	Variable	Value	Description
1 to 4	IDS	QSEC	Total-flow-cross-section-data-set identification code

QSEC RECORD 1—FORMAT(I10,E10.0)

Columns	Variable	Value	Description
1 to 10	IXSQ	1 to 10	Cross-section identification number
11 to 20	XSQ(IXSQ)	±	Total flow, in cubic feet per second (cubic meters per second), normal to the cross section, to be distributed among the node points that define the cross section. A positive value denotes inflow

QSEC RECORD 2—FORMAT(16I5)

Columns	Variable	Value	Description
1 to 80	LXSQN(K)	+	List of node numbers that define a connected series of element sides of the network boundary that is open (that is, through which flow can enter or leave the the network). The list is terminated by a "-1" entry. Up to five records (79 node points plus the "-1" entry) may be used to define a cross section. The first entry for each record must be placed in the first field Note: During a time-dependent simulation, total flow is not changed until a new QSEC data set is read. Nodes that define the cross section can be changed at any time during a time-dependent run by entering a new list of nodes. If no nodes change, enter a "-1" in the first field

Terminate the QSEC data set with one or more blank records.

Water-Surface-Elevation-Cross-Section Data Set

Water-surface-elevation-cross-section data records immediately follow a ZSEC-data-set identification record. A set of records is required for each section of the open boundary of a network for which the water-surface elevation will be specified. For each cross section, the first record contains a cross-section identification number and the water-surface elevation. A list of the nodes that define the cross section is contained on up to five additional records. A maximum of 10 cross sections may be specified. The data set is terminated with one or more blank records.

ZSEC IDENTIFICATION RECORD—FORMAT(A4)

Columns	Variable	Value	Description
1 to 4	IDS	ZSEC	Water-surface-elevation-cross-section-data-set identification code

ZSEC RECORD 1—FORMAT(I10,2E10.0,I10)

Columns	Variable	Value	Description
1 to 10	IXSZ	1 to 10	Cross-section identification number
11 to 20	XSZ1(IXSZ)	±	Water-surface elevation, in feet (meters), at every node in the cross section if XSZ2 = 0; or water-surface elevation in feet (meters), at the first point of the cross section if XSZ2 $\neq$ 0
21 to 30	XSZ2(IXSZ)	±	Water-surface elevation, in feet (meters), at the last node in the cross section if the water surface slopes from one end of the cross section to the other. Enter "0" or leave blank if the water-surface elevation is constant across the cross section

Columns	Variable	Value	Description
			Note: If XSZ2 $\neq 0$, the water-surface elevation at nodes between the two end nodes is linearly interpolated between the two end nodes
31 to 40	IBCZ(IXSZ)	1	Water-surface elevation will be specified as an *essential* boundary condition at each node of the cross section
		2	Water-surface elevation will be specified as a *natural* boundary condition at each node of the cross section
		3	*Supercritical* flow exists at each node of the cross section. The cross section is assumed to form an outflow boundary, and the water-surface elevation is not specified

ZSEC RECORD 2—FORMAT(16I5)

1 to 80	LXSZN(K)	+	List of node numbers that define a connected series of element sides of the network boundary that is open (that is, through which flow can enter or leave the the network). The list is terminated by a "-1" entry. Up to five records (79 node points plus the "-1" entry) may be used to define a cross section. The first entry for each record must be placed in the first field

Columns	Variable	Value	Description
			Note: During a time-dependent simulation, the water-surface elevation is not changed until a new ZSEC data set is read. Node points that define the cross section can be changed at any time during a time-dependent run by entering a new list of nodes. If no nodes change, enter a "-1" in the first field

Terminate the ZSEC data set with one or more blank records.

Weir Data Set

Weir data records immediately follow a WEIR-data-set identification record. One record is required for each weir segment. A weir data record contains the numbers of the nodes on the upstream and downstream sides of the weir segment, the water-surface elevation outside the network if only one side of the weir is part of the network, a discharge coefficient, and the length and crest elevation of the weir segment. The data set is terminated with one or more blank records.

WEIR IDENTIFICATION RECORD—FORMAT(A4)

Columns	Variable	Value	Description
1 to 4	IDS	WEIR	Weir-data-set identification code

WEIR RECORD—FORMAT(2I5,3E10.0)

Columns	Variable	Value	Description
1 to 5	NOPW(J,1)	+	Number of the boundary node on one side of the weir segment
6 to 10	NOPW(J,2)	0	Water is allowed to leave the network at the previously specified node
		+	Number of the boundary node on the opposite side of the weir segment
11 to 20	WWS2(J)	+	Water-surface elevation outside the network, in feet (meters), used if NOPW(J,2) = 0
21 to 30	WCOF(J)	+	Discharge coefficient (dimensionless) for free-flow conditions at the weir segment (C_w in equation 4-21), usually about 0.53
31 to 40	WLEN(J)	+	Length of the weir segment (L_w in equation 4-21), in feet (meters)

Columns	Variable	Value	Description
41 to 50	WCEL(J)	+	Crest elevation of the weir segment (z_c in equation 4-20), in feet (meters)

Terminate the **WEIR** data set with one or more blank records.

Culvert Data Set

Culvert data records immediately follow a CULV-data-set identification record. One record is required for each culvert. A culvert data record contains the numbers of the nodes at the upstream and downstream ends of the culvert, a discharge coefficient, and the cross-section area, hydraulic radius, length, roughness coefficient, and entrance invert elevation of the culvert. The data set is terminated with one or more blank records.

CULV IDENTIFICATION RECORD—FORMAT(A4)

Columns	Variable	Value	Description
1 to 4	IDS	CULV	Culvert-data-set identification code

CULV RECORD—FORMAT(2I5,6E10.0)

Columns	Variable	Value	Description
1 to 5	NOPC(J,1)	+	Number of the boundary node on one side of the culvert. If the node number is negative, water will only be allowed to leave the network at this node, as if a flap-gate were installed at the other end of the culvert
6 to 10	NOPC(J,2)	0	Water is allowed to leave the network at the previously specified node
		+	Number of the boundary node at the other end of the culvert
11 to 20	CCOF(J)	+	Discharge coefficient (dimensionless) for the culvert (C_c in equation 4-23 or equation 4-24)
21 to 30	CARE(J)	+	Cross-section area of the culvert (A_c in equation 4-23 or equation 4-24), in square feet (square meters)

Columns	Variable	Value	Description
31 to 40	CHYR(J)	+	Hydraulic radius of the culvert (R_c in equation 4-23), in feet (meters)
41 to 50	CLEN(J)	+	Barrel length of the culvert (L_c in equation 4-23), in feet (meters)
51 to 60	CMAN(J)	+	Manning roughness coefficient of the culvert (n_c in equation 4-23)
61 to 70	CELV(J)	0	Type 4 flow is assumed
		+	Entrance invert elevation of the culvert (z_{inv} in equation 4-22), in feet (meters). Type 5 flow is assumed

Terminate the CULV data set with one or more blank records.

Levee Data Set

Levee data records immediately follow a LEVE-data-set identification record. One record is required for each levee segment. A levee data record contains the numbers of the nodes on the upstream and downstream sides of the levee segment, the water-surface elevation outside the network if only one side of the levee is part of the network, the base width, top width, height, and length of the levee segment, the thickness of the subsurface layer, and the permeabilities of the levee segment and the subsurface layer. The data set is terminated with one or more blank records.

LEVE IDENTIFICATION RECORD—FORMAT(A4)

Columns	Variable	Value	Description
1 to 4	IDS	LEVE	Levee-data-set identification code

LEVE RECORD—FORMAT(2I6,6E7.0,2E10.3)

Columns	Variable	Value	Description
1 to 6	NOPL(J,1)	+	Number of the boundary node on one side of the levee segment
7 to 12	NOPL(J,2)	0	Water is allowed to enter or leave the network at the previously specified node
		+	Number of the boundary node on the other side of the levee segment
13 to 19	XLWS2(J)	+	Water-surface elevation outside the network, in feet (meters), used if NOPL(J,2) = 0
20 to 26	XLBAS(J)	+	Base width of the levee segment (b_2 in equation 4.11), in feet (meters)
27 to 33	XLTOP(J)	+	Top width of the levee segment (b_1 in equation 4.11), in feet (meters)

Columns	Variable	Value	Description
34 to 40	XLHGT(J)	+	Height of the levee segment (h_l in equation 4.11), in feet (meters)
41 to 47	XLLEN(J)	+	Length of the levee segment (L_l in equation 5.4), in feet (meters)
48 to 54	XLTHK(J)	+	Thickness of the levee-segment sublayer (D in equation 4.14), in feet (meters)
55 to 64	XLPRL(J)	+	Permeability of the levee segment (k_t in equation 4.13), in feet per second (meters per second)
65 to 74	XLPRS(J)	+	Permeability of the levee-segment sublayer (k_u in equation 4.14), in feet per second (meters per second)

Terminate the LEVE data set with one or more blank records.

Flux-Line Data Set

Flux-line data records immediately follow a FLUX-data-set identification record. A flux-line data record contains node numbers that define a line of element sides across which total flow is to be computed. Flow across the first line is used as a base flow to which other calculated flows are compared. A flux line may be composed of both straight and curved element sides. The data set is terminated with one or more blank records.

FLUX IDENTIFICATION RECORD—FORMAT(A4)

Columns	Variable	Value	Description
1 to 4	IDS	FLUX	Flux-data-set identification code

FLUX RECORD—FORMAT(16I5)

Columns	Variable	Value	Description
1 to 80	LFLUXN(K)	+	List of node numbers that define a connected series of straight or curved element sides across which total flow is to be computed. The line is terminated by a "-1". The first entry for each line must be placed in the first field. Use as many records as necessary to complete the line Note: The only limit placed on the number of flux lines and the number of points that define a single line is that the total number of points in all lines may not exceed the flux-line-array dimension

Terminate the FLUX data set with one or more blank records.

Time-Dependent Data Set

The time-dependent data record immediately follows a TIME-data-set identification record. A time-dependent data set immediately precedes sets of turbulence-model, friction, wind, vegetation, precipitation, evapotranspiration, ground-water-seepage, ground-water-head, boundary-condition, total-flow-cross-section, water-surface-elevation-cross-section, and levee data used in time-dependent (unsteady) simulations. A time-dependent data record contains the simulation time in hours at which the following data become effective and wind and ground-water-head data that apply to all nodes in the network. Time-dependent data sets and their associated turbulence-model, friction, wind, vegetation, precipitation, evapotranspiration, ground-water-seepage, ground-water-head, boundary-condition, total-flow-cross-section, water-surface-elevation-cross-section, and levee data sets must appear in chronological order at the end of the input data stream.

TIME IDENTIFICATION RECORD—FORMAT(A4)

Columns	Variable	Value	Description
1 to 4	IDS	TIME	Time-dependent-data-set identification code

TIME RECORD—FORMAT(5E10.0)

Columns	Variable	Value	Description
1 to 10	TSNEW	+	Simulation time, in hours, at which the following turbulence-model, friction, wind, vegetation, precipitation, evapotranspiration, ground-water-seepage, ground-water-head, boundary-condition, total-flow-cross-section, water-surface-elevation-cross-section, and levee data become effective
11 to 20	WVEL	+	Wind velocity, in feet (meters) per second. This value will be assigned to each node in the network unless overridden

Columns	Variable	Value	Description
21 to 30	WDIR	+	Wind-direction angle, in degrees, measured counterclockwise from the positive x-axis. This value will be assigned to each node in the network unless overridden
31 to 40	ROAIR	+	Air density, in slugs per cubic foot (kilograms per cubic meter). The default value is 0.00237 slug/ft^3 (1.225 kg/m^3)
41 to 50	GWHEAD	+	Ground-water head, in feet (meters). This value will be assigned to each node in the network unless overridden

Input Data Files

Input data files that contain finite-element-network data, wind data, ground-water-head data, initial-condition (flow) data, and boundary-condition data can be read by FESWMS-2DH programs. Input data files may be either formatted or unformatted. Examples of FORTRAN statements that can be used to read the data files are presented in this subsection.

Network Data File

A network data file contains node, element, and property-code data that define a finite-element network. An unformatted network data file can be read using the following FORTRAN statements:

```
      READ (IUNIT) NP,NE
      READ (IUNIT) ((XCORD(N),YCORD(N),GRND(N),CEIL(N),N=1,NP)
      READ (IUNIT) ((NOP(L,K),K=1,9),LTRB(L),LFRC(L),LVEG(L),
     #  LPRC(L),LEVP(L),LGWT(L),LSEQ(L),L=1,NE)
```

A formatted network data file can be read using the following FORTRAN statements:

```
      READ (IUNIT,'(2I10)') NP,NE
      READ (IUNIT,'(I10,4E10.0)') (N,XCORD(N),YCORD(N),
     #  GRND(N),CEIL(N),I=1,NP)
      IF (GRIDF) THEN
        READ (IUNIT,'(11I5,5I3,I5)')
     #     (L,(NOP(L,K),K=1,9),LTRB(L),LFRC(L),LVEG(L),
     #     LPRC(L),LEVP(L),LGWT(L),LSEQ(L),I=1,NE)
      ELSE
        READ (IUNIT,'(10I6,I5,5I2,I5)')
     #     (L,(NOP(L,K),K=1,9),LTRB(L),LFRC(L),LVEG(L),
     #     LPRC(L),LEVP(L),LGWT(L),LSEQ(L),I=1,NE)
      END IF
```

Variables contained in the preceding FORTRAN statements are defined below:

IUNIT = FORTRAN unit number of the network data file,

NP = maximum node number,
NE = maximum element number,
N = node number,
K = array index,
L = element number,
XCORD = array of x-coordinates,
YCORD = array of y-coordinates,
GRND = array of ground-surface elevations,
CEIL = array of ceiling elevations,
NOP = array of element connectivity lists,
LTRB = array of turbulence codes,
LFRC = array of friction codes,
LVEG = array of vegetation codes,
LPRC = array of precipitation codes,
LEVP = array of evapotranspiration codes,
LGWT = array of ground-water-seepage codes,
I = index, and
GRIDF = format code (0 or 1).

Wind Data File

A wind data file contains wind velocity and direction at node points. An unformatted wind data file can be read using the following FORTRAN statements:

```
READ (IUNIT) TIME,NODES
READ (IUNIT) (N,WINDV(N),WINDA(N),I=1,NODES)
```

A formatted wind data file can be read using the following FORTRAN statements:

```
 READ (IUNIT,'(E10.0,I10)') TIME,NODES
 READ (IUNIT,'(I10,2E10.0)') (N,WINDV(N),WINDA(N),
#  I=1,NODES)
```

Variables contained in the preceding FORTRAN statements are defined below:

IUNIT = FORTRAN unit number of the wind data file,
TIME = simulation time, in hours,
NODES = number of nodes for which wind data (that apply to the specified simulation time) are to be read,
N = node number,
WINDV = array of wind velocities, in feet per second (meters per second),
WINDA = array of wind directions, in degrees measured counterclockwise from the positive x-direction (these are the directions toward which the wind is blowing), and
I = index.

Ground-Water-Head Data File

A ground-water-head data file contains the ground-water head at node points. An unformatted ground-water-head data file can be read using the following FORTRAN statements:

```
READ (IUNIT) TIME,NODES
READ (IUNIT) (N,HEAD(N),I=1,NODES)
```

A formatted ground-water-head data file can be read using the following FORTRAN statements:

```
READ (IUNIT,'(E10.0,I10)') TIME,NODES
READ (IUNIT,'(I10,E10.0)') (N,HEAD(N),I=1,NODES)
```

Variables contained in the preceding FORTRAN statements are defined below:

IUNIT = FORTRAN unit number of the ground-water-head data file,
TIME = simulation time, in hours,
NODES = number of nodes for which ground-water-head data (that apply to the specified simulation time) are to be read,
N = node number,
HEAD = array of ground-water heads, in feet (meters), and
I = index.

Initial-Condition Data File

An initial-condition (flow) data file contains values of depth-averaged velocity in the x- and y-directions, water depth, and the time derivative of these quantities at each node. These values are used as initial conditions for a simulation. An initial-condition data file is usually a flow data file that has been generated by a previous simulation. An unformatted initial-condition data file can be read using the following FORTRAN statements:

```
      READ (IUNIT) TIME,NP
      READ (IUNIT) (XVEL(N),YVEL(N),DEPTH(N),UDOT(N),VDOT(N),
     #  HDOT(N),N=1,NP)
```

A formatted initial-condition data file can be read using the following FORTRAN statements:

```
      READ (IUNIT,'(E10.0,I10)') TIME,NP
      IF (FLOWF) THEN
        READ (IUNIT,'(I10,6E10.0)') (N,XVEL(N),YVEL(N),
     #    DEPTH(N),UDOT(N),VDOT(N),HDOT(N),I=1,NP)
      ELSE
        READ (IUNIT,'(2X,I5,1X,E11.5,1X,E11.5,1X,E11.5,1X,
     #    E11.5,1X,E11.5,1X,E11.5)') (N,XVEL(N),YVEL(N),
     #    DEPTH(N),UDOT(N),VDOT(N),HDOT(N),I=1,NP)
      END IF
```

Variables contained in the preceding FORTRAN statements are defined below:

IUNIT = FORTRAN unit number of the initial-condition data file,
TIME = simulation time, in hours,
NP = maximum node number,
N = node number,
XVEL = array of x-velocity components, in feet per second (meters per second),
YVEL = array of y-velocity components, in feet per second (meters per second),
DEPTH = array of water depths, in feet (meters),
UDOT = array of derivatives of x-velocity components with respect to

time, in feet per second per second (meters per second per second),

VDOT = array of derivatives of y-velocity components with respect to time, in feet per second per second (meters per second per second),

HDOT = array of derivatives of depth with respect to time, in feet per second (meters per second),

I = index, and

FLOWF = format code (0 or 1).

Boundary-Condition Data File

A boundary-condition data file contains values of boundary-condition codes and boundary-condition specifications for nodes where boundary conditions are prescribed. During a time-dependent simulation, boundary-condition data for a node remain unchanged until new specifications are read. An unformatted boundary-condition data file can be read using the following FORTRAN statements:

```
      READ (IUNIT) TIME,NODES
      READ (IUNIT) (N,NFIX(N),XSPEC(N),YSPEC(N),ZSPEC(N),
     #   I=1,NODES)
```

A formatted boundary-condition data file can be read using the following FORTRAN statements:

```
      READ (IUNIT,'(E10.0,I10)') TIME,NODES
      READ (IUNIT,'(2I10,3E10.0)') (N,NFIX(N),XSPEC(N),
     #   YSPEC(N),ZSPEC(N),I=1,NODES)
```

Variables contained in the preceding FORTRAN statements are defined below:

IUNIT = FORTRAN unit number of the boundary-condition data file,

TIME = simulation time, in hours,

NODES = number of nodes for which boundary-condition data (that apply to the specified simulation time) are to be read,

N = node number,

NFIX = array of boundary-condition codes,

XSPEC = array of specified velocities, in feet per second (meters per second), or unit flows, in square feet per second (square meters per second), in the x-direction or tangent to the boundary,

YSPEC = array of specified velocities, in feet per second (meters per second), or unit flows, in square feet per second (square meters per second), in the y-direction or normal to the boundary; or total flows, in cubic feet per second (cubic meters per second), normal to the boundary,

ZSPEC = array of specified water-surface elevations, in feet (meters), and

I = index.

Addendum to Section 9. Output Data Description—Depth-Averaged Flow Module: FLOMOD

Printed Output

One-dimensional levee-seepage output is printed after the nodal solution values. For each levee segment, the levee-segment number, the node numbers and the water-surface elevations on each side of the levee, the flow through the levee segment, the flow under the levee segment, and the sum of the last two items are printed. Element inflows and outflows due to precipitation, evapotranspiration, and ground-water seepage are printed after weir, culvert, and levee-seepage output is printed.

Output Data Files—Flow Data File

Flow data written to the flow data file in text form is written in E12.5 format if FLOWF = 1 or E10.3 format if FLOWF = 0 (see the SWMS identification record).

REFERENCES

Department of the Army, Office of the Chief of Engineers, 1978, Design and construction of levees: Washington, D. C., Department of the Army, Office of the Chief of Engineers, Engineer Manual 1110-2-1913.

Froehlich, D. C., 1989, Finite element surface-water modeling system: two-dimensional flow in a horizontal plane. Vol. I. Users manual: McLean, Va., U. S. Department of Transportation, Federal Highway Administration, Publication No. FHWA-RD-88-177, 286 p.

McDonald, M. G., and Harbaugh, A. W., 1988, A modular three-dimensional finite-difference ground-water flow model: U. S. Geological Survey Techniques of Water-Resources Investigations, book 6, chap. A1, 586 p.

Peter, Pavel, 1982, Canal and river levees: New York, Elsevier, 540 p.

Reid, R. O., and Whitaker, R. E., 1976, Wind-driven flow of water influenced by a canopy: Proceedings of the American Society of Civil Engineers, v. 102, no. WW1, p. 61-77.