Full Equations (FEQ) Model for the Solution of the Full, Dynamic Equations of Motion for One-Dimensional Unsteady Flow in Open Channels and Through Control Structures

U.S. GEOLOGICAL SURVEY

Water-Resources Investigations Report 96–4240

Prepared in cooperation with the

DU PAGE COUNTY DEPARTMENT OF ENVIRONMENTAL CONCERNS and ILLINOIS DEPARTMENT OF NATURAL RESOURCES, OFFICE OF WATER RESOURCES

Full Equations (FEQ) Model for the Solution of the Full, Dynamic Equations of Motion for One-Dimensional Unsteady Flow in Open Channels and through Control Structures

By DELBERT D. FRANZ, Linsley, Kraeger Associates and
CHARLES S. MELCHING, U.S. Geological Survey

U.S. GEOLOGICAL SURVEY

Water-Resources Investigations Report 96–4240

Prepared in cooperation with the

DU PAGE COUNTY DEPARTMENT OF ENVIRONMENTAL CONCERNS and ILLINOIS DEPARTMENT OF NATURAL RESOURCES, OFFICE OF WATER RESOURCES

Mountain View, California
Urbana, Illinois
1997

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

U.S. GEOLOGICAL SURVEY
Gordon P. Eaton, Director

For additional information write to:

District Chief
U.S. Geological Survey
221 North Broadway Avenue
Urbana, Illinois, 61801

Copies of this report can be purchased from:

U.S. Geological Survey
Branch of Information Services
Box 25286
Federal Center
Denver, CO 80225

CONTENTS

FIGURES

TABLES

CONVERSION FACTORS

Multiply	By	To obtain
Length		
foot (ft)	0.3048	meter
mile (mi)	1.609	kilometer
Area		
acre	0.4047	hectare
square foot (ft^2)	0.09290	square meter
Volume		
cubic foot (ft^3)	0.02832	cubic meter
acre-foot (acre-ft)	1,233	cubic meter
Flow rate		
foot per second (ft/s)	0.3048	meter per second
foot per hour (ft/hr)	0.3048	meter per hour
cubic foot per second (ft^3/s)	0.02832	cubic meter per second
mile per hour (mi/h)	1.609	kilometer per hour
Pressure		
millibar	0.1	kilopascal
pound-force per cubic foot (lbf/ft^3)	157.1	Newtons per cubic meter

Temperature in degrees Fahrenheit (°F) may be converted to degrees Celsius (°C) as follows:

$$°C = (°F - 32) / 1.8$$

Full Equations (FEQ) Model for the Solution of the Full, Dynamic Equations of Motion for One-Dimensional Unsteady Flow in Open Channels and Through Control Structures

By Delbert D. Franz *and* Charles S. Melching

Abstract

The Full EQuations (FEQ) model is a computer program for solution of the full, dynamic equations of motion for one-dimensional unsteady flow in open channels and through control structures. A stream system that is simulated by application of FEQ is subdivided into stream reaches (branches), parts of the stream system for which complete information on flow and depth are not required (dummy branches), and level-pool reservoirs. These components are connected by special features; that is, hydraulic control structures, including junctions, bridges, culverts, dams, waterfalls, spillways, weirs, side weirs, and pumps. The principles of conservation of mass and conservation of momentum are used to calculate the flow and depth throughout the stream system resulting from known initial and boundary conditions by means of an implicit finite-difference approximation at fixed points (computational nodes). The hydraulic characteristics of (1) branches including top width, area, first moment of area with respect to the water surface, conveyance, and flux coefficients and (2) special features (relations between flow and headwater and (or) tail water elevations, including the operation of variable-geometry structures) are stored in function tables calculated in the companion program, Full EQuations UTiLities (FEQUTL). Function tables containing other information used in unsteady-flow simulation (boundary conditions, tributary inflows or outflows, gate settings, correction factors, characteristics of dummy branches and level-pool reservoirs, and wind speed and direction) are prepared by the user as detailed in this report. In the iterative solution scheme for flow and depth throughout the stream system, an interpolation of the function tables corresponding to the computational nodes throughout the stream system is done in the model. FEQ can be applied in the simulation of a wide range of stream configurations (including loops), lateral-inflow conditions, and special features. The accuracy and convergence of the numerical routines in the model are demonstrated for the case of laboratory measurements of unsteady flow in a sewer pipe. Verification of the routines in the model for field data on the Fox River in northeastern Illinois also is briefly discussed.

The basic principles of unsteady-flow modeling and the relation between steady flow and unsteady flow are presented. Assumptions and the limitations of the model also are presented. The schematization of the stream system and the conversion of the physical characteristics of the stream reaches and a wide range of special features into function tables for model applications are described. The modified dynamic-wave equation used in FEQ for unsteady flow in curvilinear channels with drag on minor hydraulic structures and channel constrictions determined from an equivalent energy slope is developed. The matrix equation relating flows and depths at computational nodes throughout the stream system by the continuity (conservation of mass) and modified dynamic-wave equations is illustrated for four sequential examples. The solution of the matrix equation by Newton's method is discussed. Finally, the input for FEQ and the error messages and warnings issued are presented.

1. INTRODUCTION

Most open-channel flows of interest to hydraulic engineers, hydrologists, and planners are unsteady and can be considered to be one-dimensional (1-D). In unsteady flow, some aspect of the flow (velocity, depth, pressure, or another characteristic) is changing with time. In 1-D flow, longitudinal acceleration is significant, whereas transverse and vertical accelerations are negligible. Many problems involving 1-D unsteady flows have been approximated by assumption of steady flows (for example, constant peak discharges in flood-plain delineation studies) or piecewise steady flows, wherein storage-outflow relations are derived for channel reaches from a steady-flow hydraulic model and used in simple hydrologic-routing methods. Piecewise steady-flow analysis typically does not consider all the forces acting on the flow and only partially accounts for channel-storage effects. The approximate solutions for steady-flow and piecewise steady-flow analysis are adequate for certain simplified planning or design problems but are inadequate for many others (for example, streams with rapidly rising and falling stage, flat slopes, and broad flood plains where storage and acceleration effects could be substantial). No criteria are available to guide engineers and hydrologists as to when steady-flow methods are acceptable and when a complete unsteady-flow analysis is necessary. Further, problems such as tidally affected flows and sudden releases from power-generation stations require 1-D unsteady-flow analysis.

With the recent increases in the calculation speed and storage capabilities of computers, simulation of 1-D unsteady flow in a complex stream system with many hydraulic structures has become practicable. Runoff response to rainfall in urban areas is rapid, and streams throughout Illinois have relatively flat slopes and broad flood plains. Thus, engineers with the Illinois State government and rapidly urbanizing counties surrounding Chicago, became interested in applying unsteady-flow analysis for flood-plain delineation, flood forecasting, flood-control reservoir operation, and other applications. Because a wide variety of hydraulic structures in the stream network could be simulated in the Full EQuations (FEQ) model, this model was selected by the U.S. Geological Survey (USGS) and cooperating agencies for documentation and extensive testing (Ishii and Turner, in press; Ishii and Wilder, 1993; Turner and others, 1996). The Illinois Department of Natural Resources, Office of Water Resources, and the County of Du Page, Department of Environmental Concerns, cooperated with the USGS and Linsley, Kraeger Associates to document the model schematization, governing equations, mathematical solution procedures, numerical characteristics, and input description for FEQ.

Development of FEQ, a numerical tool for the solution of the flow-governing equations for a system of interconnected channels, began in 1976. The structure of the program was designed to represent the general structure of a stream system, so the model is highly flexible and capable of efficiently simulating a wide variety of stream systems. Among the many hydraulic structures represented in the model are bridges, culverts, dams, level-pool reservoirs, spillways, weirs, sluice gates, pumps, side weirs, expansions, contractions, drop structures, and flows over roadways. Several options for the choice of the governing equations are available. Wind forces on the stream surface can be calculated and their effects on flow momentum simulated.

1.1 Purpose and Scope

The purpose of this report is to document the stream-network visualization and schematization, flow-governing equations, and solution procedures used in the FEQ model to simulate 1-D unsteady flow in a network of open channels and control structures. The FEQ model and example inputs and outputs may be obtained by electronic retrieval from the World Wide Web (WWW) at http://water.usgs.gov/software/feq.html and by anonymous File Transfer Protocol (FTP) from water.usgs.gov in the pub/software/surface_water/feq directory. Because flow in a network of open channels and control structures is complex, the documentation of FEQ involves detailed discussions of many hydraulic-engineering and numerical-analysis topics. These topics are discussed in the following order. The basic principles of 1-D unsteady-flow modeling and the relation between steady flow and unsteady flow are discussed to give readers who are familiar with steady-flow analysis points of reference for understanding unsteady-flow analysis. The schematization of the stream system and the conversion of the physical characteristics of the stream reaches, including the effects of curvilinearity, into function tables for model applications are described. The modified dynamic-wave equation used in FEQ is developed for unsteady flow in curvilinear channels with drag forces on minor hydraulic structures and channel constrictions determined from an equivalent energy slope. The equations approximating flow through various hydraulic-control structures are presented, and conversions of the stage-discharge relations for these structures into function tables are given. The matrix equation relating flows and depths at computational nodes throughout the river system by the continuity (conservation of mass) and modified dynamic-wave equations is illustrated for four sequential examples. The solution of the matrix equation by Newton's method is discussed. Finally, the input for FEQ and the error messages and warnings issued in model simulation are listed.

1.2 Classification of One-Dimensional Steady Flow and Unsteady Flow

A classification scheme for steady and unsteady flow is useful in describing the flows of interest. The simplest steady flow is uniform flow, in which no flow variable changes with distance. In a uniform steady flow, every flow variable is a constant with respect to distance and time. If the flow is not uniform, then it is classified as nonuniform and can be further divided into gradually varied and rapidly varied flow. In gradually varied steady flow, the flow variables may change with distance, but all variables are constant in time. Furthermore, the variations with distance are gradual, so vertical accelerations are small. The series of backwater profiles discussed in the typical open-channel hydraulics course or textbook (for example, Chow, 1959, p. 227-237) are all gradually varied flows. In rapidly varied flow, substantial variations are present in vertical and/or transverse flow. An extreme example is a hydraulic jump below a dam. This flow can still be analyzed as 1-D flow, but the rapidly varied zone of the flow must be recognized and isolated in the analysis. Additional examples of rapidly varied flow are flows through culverts and bridges and over weirs and spillways.

Unsteady uniform flow is impossible, so only nonuniform unsteady flow is of interest in hydraulic analysis. Both gradually varied and rapidly varied unsteady flows are possible, and the same general rules for analysis apply as for steady flow. The zones of rapidly varied flow must be isolated before analysis under the 1-D flow assumption; thus, the method of analysis for steady and unsteady flow is the same in this respect.

1.3 Selection of Conservation Principles

Three conservation principles—conservation of water mass, conservation of the mechanical-energy content of the water, conservation of the momentum content of the water—are available for analysis of 1-D unsteady flow. Conservation of thermal energy is not considered because temperature-change and heat-transfer effects do not affect flow depth and discharge.

The first principle selected is the conservation of water mass, which becomes the conservation of water volume if the density is constant. Equations derived from application of the conservation of mass principle are often referred to as “continuity equations.”

The choice of conservation of momentum instead of conservation of mechanical energy of water for FEQ was based on how well the various flow parameters and variables can be approximated and how well each particular principle works when only approximations to physical reality are possible. Both principles are exact given precise knowledge of all the flow parameters and variables; however, precise knowledge of these is never possible. Yen (1973) provides a detailed list of differences between the energy and momentum approaches. Many researchers, including Abbott (1974), Cunge and others (1980), and Liggett (1975), argue for combined application of the conservation of mass and conservation of momentum principles as the equations of motion because this combination gives the correct wave speed and height should abrupt waves (hydraulic bores) form during the modeling of rapidly increasing or decreasing flow. If the conservation of momentum principle is used with the continuity equation and the equations are properly approximated, then the correct wave speed and height will be computed. In contrast, application of the conservation of energy principle provides no simple approximation that can be applied to yield the correct wave speed and height.

In many applications, the flow in the stream channels is derived from runoff entering the channels either overland or from storm sewers, drainage ditches, and streams too small to be explicitly represented in the model.

These flows generally enter approximately at right angles to the main-channel flow, and complex interaction between these flows involves considerable turbulence. Application of the energy-conservation principle would require that the kinetic and potential energy of lateral flows be estimated, and such estimates are nearly impossible to make accurately. The turbulence results in an unknown increase in energy dissipation. Therefore, lateral inflows are better approximated by use of the conservation of momentum principle. Because these flows enter approximately at right angles to the main-channel flow direction, the effect is approximated in the momentum equation without an additional requirement for estimated losses. The applicability of the conservation of momentum principle to the solution of lateral inflow problems has been demonstrated in modeling of side-channel spillways and wash-water troughs (Henderson, 1966, p. 268), both of which cause much greater turbulence than normally results in unsteady flow.

Yen and others (1972) give further evidence for the choice of the momentum-conservation principle. Using artificial rainfall on a sloping glass flume, they computed resistance coefficients for steady, spatially varied flow for both the energy and momentum conservation principles and found that the resistance coefficient from the momentum principle was always closer to the coefficient estimated from steady flow without lateral inflow. Because use of Manning's equation for resistance losses yields a better estimate of the resistance coefficient for the momentum principle than for the energy principle, methods based on momentum conservation yield better estimates of the water-surface profile than do methods based on energy conservation, especially if Manning’s n is calibrated to measured water-surface profiles or historic high water marks. In addition, the resistance coefficient estimated from the momentum principle was insensitive to variations in the velocity of lateral inflow (many applications of unsteady flow involve a wide range of lateral inflow rates).

Finally, the equation obtained with the conservation of momentum principle is simpler than the equation obtained with the conservation of energy principle. The simplicity of the equation obtained with the conservation of momentum principle is twofold; the equation includes fewer terms, and less information is needed for each cross section.

1.4 Major Assumptions in Unsteady-Flow Analysis

Analysis of 1-D unsteady flow in open channels requires many assumptions. The major assumptions are the following:

1. The wavelength of the disturbance of the flow is very long relative to the depth of the flow. This “shallow-water wave assumption” implies that the flow is principally 1-D and basically parallel to the walls and bottom forming the channel. Thus, streamline curvature is small; lateral and vertical accelerations are negligible relative to the longitudinal accelerations; and, therefore, the pressure distribution is hydrostatic.
2. The channel geometry is fixed so that the effect of deposition or scour of sediment is small.
3. The bed of the channel has a shallow slope so that (a) the tangent and sine of the angle that the bottom makes with the horizontal have nearly the same value as the angle and (b) the cosine of the angle is approximately 1.
4. The effect of boundary friction force can be estimated with a relation derived from steady uniform flow. Nonuniformity and unsteadiness are assumed to have only a small effect on the frictional losses.
5. Channel alignment with respect to the effect of directional changes on the conservation of momentum principle may be treated as if it were rectilinear even though the channel is curvilinear. Thus, the water surface in any cross section of the stream is assumed to be horizontal. Super-elevation effects on the water surface in channel bends are not considered in the analysis and are assumed to have a small effect on the results.
6. The fluxes of momentum and energy along the cross section resulting from nonuniform velocity distribution may be estimated by means of average velocities and flux-correction coefficients that are functions of location along the stream and water-surface elevation.
7. The flowing fluid is homogeneous (constant density).

From these assumptions, formal statements of the conservation of water volume (mass) and conservation of water momentum can be developed. The conservation of volume (mass) principle relates to flows and changes in the quantity of water stored in the channels and reservoirs. No forces of any kind are considered in the conservation of mass. Forces, momentum fluxes, and the momentum of water in storage are related in the conservation of momentum principle. The factors involved in this equation are

1. gravity force on the water in the channel,
2. friction force on the wetted perimeter of the channel,
3. pressure force on the boundaries,
4. wind force on the water surface, and
5. inertia of the water.

Some of these factors can be omitted to simplify the unsteady-flow computations. If all these factors are included in the analysis, the equations are referred to as the complete, full, dynamic, Saint-Venant, or shallow-water equations. If the inertia of the water is ignored, the zero-inertia form of the motion equation is obtained. If, in addition, the variations of pressure force along the channel are ignored because they are thought to be small, the kinematic form of the motion equation is obtained. Reservoir routing also is a form of unsteady-flow analysis in which the motion equation is simplified to a relation between water-surface elevation and the flow. In a certain sense, reservoir routing ignores all four factors although some or all are implicit in the relation between flow and water-surface elevation. In each case, at least one of the factors is dropped from the motion equation. FEQ includes three of the four forms of motion equations for unsteady flow: (1) the full-equation form, including all four factors, (2) the zero-inertia form, in which the inertia of the water is omitted, and (3) the reservoir-routing form, in which the motion equation is reduced to a relation between water-surface elevation and flow.

1.5 Examples of Unsteady-Flow Analysis

Examples of unsteady-flow analysis are easily found, only a few are mentioned here.

1. **Passage of a Flood Wave.** Flood-wave movement is unsteady, but in flood-insurance studies an approximate maximum-elevation envelope resulting from a flood wave is computed under the assumption of steady flow. Little work has been done to evaluate the accuracy of this approximation. In addition, the effect of flood-plain filling and obstruction is often analyzed by means of steady-flow analysis. Changes in the ability of the stream to convey water are evaluated in steady-flow analysis, whereas changes in the capability of the stream to store water are not considered in steady-flow analysis. The changes resulting from storage may be large in some cases. Therefore, application of unsteady-flow analysis may substantially improve flood-insurance studies.
2. **Operation of Irrigation and Power Canals.** Unsteady-flow analysis is required to design these canals properly because the flow variations can often be abrupt. Allowance must be made for the wave heights that might result. Furthermore, the traveltime of transients becomes important in the design and operation of structures intended to reduce or control transients.
3. **Tidal Effects.** Analysis of the effects of tides on streams requires consideration of unsteady flow. Steady-flow analysis is often used to approximate the envelope of maximum elevations; but again, little work has been done to evaluate the accuracy of this approximation.
4. **Junctions.** The complex interactions at stream junctions often require unsteady-flow analysis. For example, a large flood or failure of a dam on a tributary to a second, larger stream can sometimes result in upstream flow at the junction in the receiving stream. This, in turn, can lead to a very rapid rise in water-surface elevation because the influx of water serves not only as a temporary dam but also as another source of inflow.
5. **Measures to Control Floods.** Evaluation of the effects of proposed measures to control floods in a stream must involve unsteady-flow analysis. Simplified methods often fail to give adequate solutions where stream-bottom slopes are flat enough to make flow reversals possible or where flow is strongly affected by water-surface elevations downstream.

2. RELATION BETWEEN ANALYSES OF STEADY AND UNSTEADY FLOW

Analyses of steady and unsteady flows are similar in many ways. Because steady-flow analysis is much more widely applied and because hydraulic engineers are generally familiar with steady flow, steady-flow concepts are discussed first to introduce some aspects of the analysis of unsteady flow.

2.1 Channel Segmentation

In steady-flow analysis, a governing equation is given that describes flow variation. This is most often written as an energy-conservation equation but a momentum-conservation equation also can be used. In either case, this equation is in differential or integral form; and the solution cannot be determined without application of numerical methods. Thus, only an approximate solution to the governing equation can be determined. To find this solution, the channel is subdivided into short pieces called computational elements. Then, for each computational element, the differential or integral terms in the governing equations are approximated algebraically to yield an algebraic equation that approximates the governing equation for that element. From these computational elements, the whole solution scheme proceeds.

The ends of the computational element, called nodes, are defined by cross sections either measured or estimated from field measurements. The cross section is at right angles to the direction of flow as best as can be determined. One way to visualize a computational element is as a slice of the channel whose ends are at right angles to the longitudinal axis (fig. 1). Adjacent computational elements have a cross section and, therefore, a node in common; thus, there will always be one more node (cross section) than the number of computational elements.

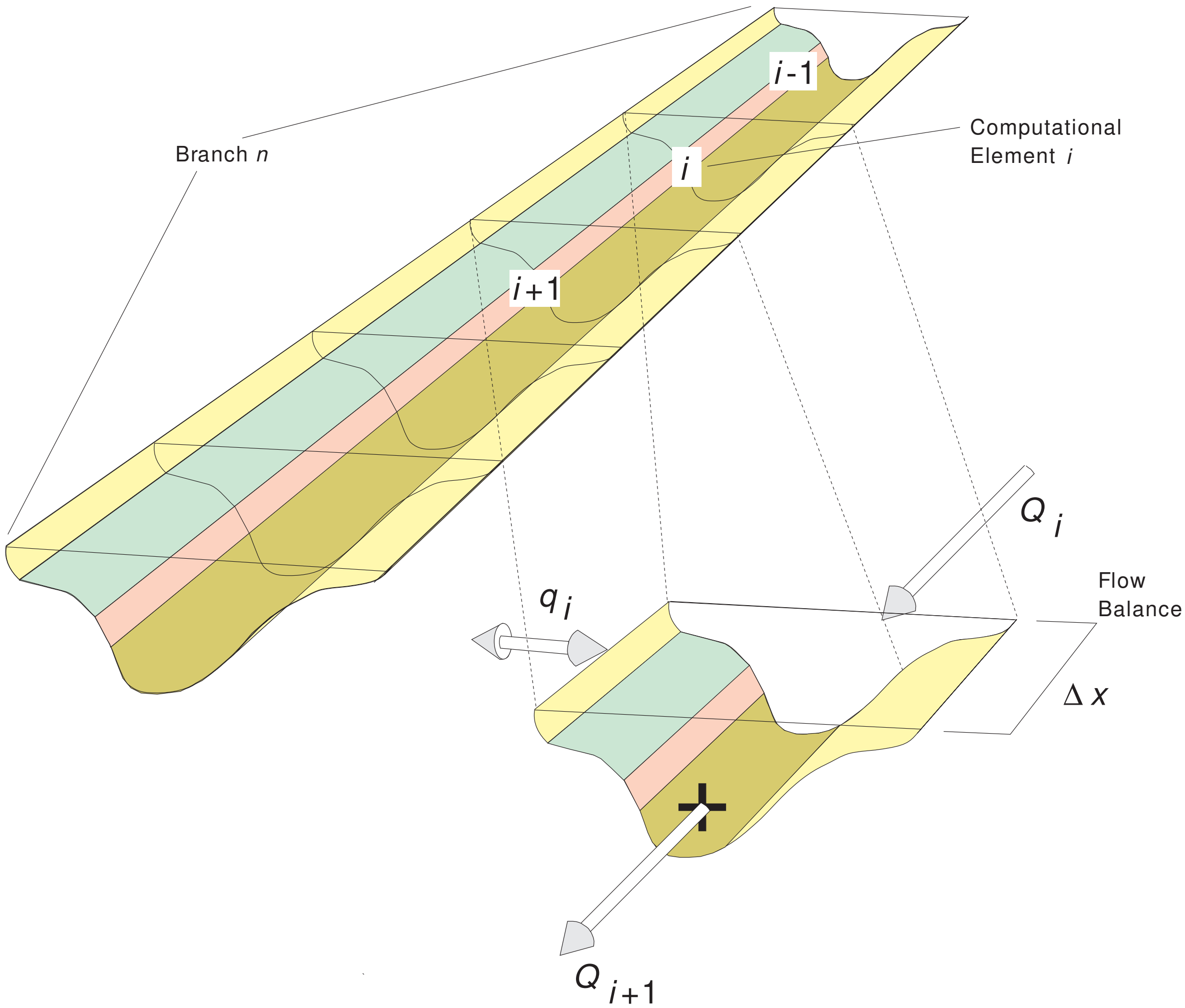
The values of interest in the cross sections are the width of the water surface, the flow area, the first moment of area, the conveyance, and, perhaps, an energy- or momentum-flux coefficient, all computed at any given elevation. The shape of the cross section does not appear explicitly in the governing equations, but only implicitly through these cross-sectional characteristics.

2.2 Review of Steady-Flow Analysis

In steady-flow analysis, the equation for conservation of water volume is trivial because the flow is known at all points in the channel unless flow over a side weir is simulated. The algebraic approximations of the conservation of flow momentum or energy are carefully written such that only water-surface elevation values at the ends of the computational element are needed. Consequently, there are two unknowns for each computational element; namely, the elevation of the water surface at each end. Given an initial elevation, the unknown elevations along the channel can be computed sequentially, one unknown at a time. The direction of solution must be from a point of known or assumed elevation to points of unknown elevation. In general, if the flow is subcritical, the direction of solution is upstream; if the flow is supercritical, the direction of solution is downstream.

One of the first steps in steady-flow analysis is to locate control points; that is, points along the stream where the elevation can be computed once the steady flow is selected. At least one point of known elevation is needed to start the computations. For subcritical flow, this point, called an initial condition, will be the downstream boundary of the region of interest.

The algebraic governing equation for steady flow does not apply to rapidly varied flow at bridges, culverts, falls, rapids, dams, and other special features. Furthermore, the governing equation does not apply to junctions of two or more channels or to abrupt changes in channel size or shape. These special features must be isolated and analyzed with equations other than those applied for each computational element. Each special feature forms internal control points in the stream system, one upstream and one downstream. Therefore, each stream segment between the boundary and a special feature or between special features is simulated by a separate steady-flow analysis that requires an initial condition. The necessary initial conditions can be computed with the equations relating flow and elevation from upstream to downstream for the special features.



EXPLANATION

Q_i	DISCHARGE FROM COMPUTATIONAL ELEMENT *i*
q_i	LATERAL INFLOW TO COMPUTATIONAL ELEMENT *i*
Δx	LENGTH OF COMPUTATIONAL ELEMENT

Figure 1. Branch of a stream system discretized into computational elements.

During the first run of a steady-flow analysis, one or more of the computational elements may prove to be too long, and computational failure results. The only recourse is to subdivide the computational element into two or more shorter computational elements and rerun the analysis.

Also during the solution process, an elevation may result such that the flow is supercritical at the upstream end of the computational element when the elevation at the downstream end is for a subcritical flow. This result is incorrect. In steady flow, such a pattern indicates a hydraulic jump somewhere in the computational element. The analyst may consider three possibilities. First, the incorrect solution may be purely a computational artifact resulting from the failure of the solution process to determine the subcritical solution at the upstream end of the computational element. If so, the solution process should be changed to seek a subcritical solution. Second, there may be no subcritical solution for the unknown in the computational element. This also can be a computational artifact if the computational element is too long and the errors in the approximation of the differential or integral terms are distorting the solution. If so, the computational element must be subdivided and the solution tried again. Third, the flow is physically supercritical near the computational element, yielding the incorrect solution. If so, the control point for the supercritical flow must be found, the profile downstream from the supercritical-flow control point must be computed, and the hydraulic jump can then be located by iterating between the downstream propagation of supercritical flow and the upstream propagation of subcritical flow.

2.3 Basic Principles of Unsteady-Flow Analysis

The previous discussion of steady-flow analysis gives background for some concepts of unsteady-flow analysis. Although some similarities can be expected because steady flow is a special case of unsteady flow, differences also can be expected because unsteady flow must describe conditions not included in the steady-flow governing equations.

In unsteady-flow analysis, two governing algebraic equations must be explicitly solved because the flow and the elevation of the water surface are both unknown. One of the governing equations is the conservation of water volume, and the other is the conservation of water momentum. In steady-flow analysis, the equation for conservation of water volume was trivial because the flows were constant and were used to solve for the flows everywhere in the channel (known elevations were unnecessary). In unsteady-flow analysis, however, a governing equation of conservation of water volume must be explicitly solved for flows and elevations.

In unsteady-flow analysis, computational elements and algebraic approximations to the differential or integral terms in the governing equations must be used to develop two algebraic equations for each computational element written in terms of elevations and flows at the ends of the element. These governing equations are more complex than those for steady-flow analysis. For unsteady flow, a computational element with respect to time also must be considered, but it is simple: the time axis is divided into finite increments that, ideally, will be short enough so that the algebraic approximations of the differential and integral terms will be sufficiently accurate. Because of this dependence on time, the algebraic governing equations involve not only the unknown flow and elevation at two points along the channel but also at two points in time.

Control points with known relations between elevation and flow must be identified, as well as points of rapidly varied flow or of interaction between channels not described by the algebraic governing equations. As in steady-flow analysis, these points establish the limits of applicability of the governing equations with respect to distance along the channel and provide known values for the analysis. In unsteady-flow analysis, however, a starting time for the computations when all the flow values are known at the computational nodes (ends of the computational elements) must be established. Flow is assumed to be steady everywhere in the system at the starting time. This is the first major difference between steady flow and unsteady flow: a steady-flow analysis must be completed to establish the initial condition for the unsteady-flow analysis.

A second major difference between unsteady-flow analysis and steady-flow analysis is the information needed at the boundaries of the stream system. In steady-flow analysis, knowledge of one elevation at the downstream boundary is needed to start the computations for subcritical flow or at the upstream boundary for supercritical flow. A cursory analysis of the number of equations available in unsteady flow shows that more information is needed for unsteady-flow analysis. For example, a single channel with no special features is divided into 9 computational elements yielding 10 nodes. With 2 unknowns at each node, there are 20 unknowns but only 18 equations (2 per computational element). Thus, the unknowns cannot be determined without some additional information at the boundaries of the system. When the flow is subcritical, information at both the upstream and the downstream boundary of the system is needed. This information can be in one of three forms: flow known as a function of time, water-surface elevation known as a function of time, or a relation between flow and water-surface elevation. The upstream boundary is commonly flow known as a function of time (a hydrograph), and the downstream boundary is commonly a known relation between flow and water-surface elevation (a rating curve). The information supplied at a boundary is called a boundary condition.

The information supplied at a special feature internal to the stream system is often called an internal boundary condition. In unsteady-flow analysis, internal boundary conditions are approximated as steady-flow relations because the special features generally are short enough that the changes in momentum and volume of water within the special features are small. The isolation and description of the special features is a major component of unsteady-flow analysis.

The same computational problems can arise for unsteady-flow analysis as steady-flow analysis because both analyses use algebraic approximations to the differential and integral terms. These approximations are developed for a computational element of finite length. If the computational element is too long, an incorrect solution results. The difference between the analyses is that in unsteady-flow analysis the computational problems are more complex and more frequent than in steady-flow analysis. The increased frequency is primarily because unsteady-flow analysis involves computations over a wide range of water-surface elevations, whereas most steady-flow analysis involves computations over a narrow range of water-surface elevations. Furthermore, the time dimension results in additional complications.

Similarities and differences between steady- and unsteady-flow analysis are summarized in table 1. The motion equation in this table is expressed by use of the principle of conservation of momentum.

Table 1. Similarities and differences between steady- and unsteady-flow analysis

[Q, flow rate; A, cross-sectional area; y height of water surface above the minimum point in the cross section; x, distance along the channel; t, time; g, gravitational acceleration; q, inflow into channel over or through the sides (lateral flow); S_0, bottom slope of the channel, positive with decline downstream; S_f, friction slope]

Item	Steady Flow	Unsteady Flow
Motion equation	$gA\frac{dy}{dx} + \frac{dQ^2/A}{dx} = gA(S_o - S_f)$	$\frac{\partial Q}{\partial t} + gA\frac{\partial y}{\partial x} + \frac{\partial Q^2/A}{\partial x} = gA(S_o - S_f)$
Mass equation	$\frac{dQ}{dx} = q$	$\frac{\partial A}{\partial t} + \frac{\partial Q}{\partial x} = q$
Exact solution	Not possible	Not possible
Approximate solution	At discrete points	At discrete points
Algebraic equations	Between nodes	Between nodes
Channel description	Cross sections at nodes	Cross sections at nodes
Unknowns	Water-surface elevations at nodes	Water-surface elevations and flows at nodes
Control points	Used to start solution	Isolated in advance
Initial conditions	At control points[1]	At all nodes
Boundary conditions	None	Required
Special features	Must be isolated	Must be isolated
Computational problems	Computational elements too long	Computational elements too long
Cross-section elements	Computed as needed	Placed in lookup tables

[1]These control points typically are at the boundaries of the stream system and special features. Therefore, the initial conditions for steady-flow analysis are often thought of as boundary conditions. In this report, the data required to begin the steady-flow analysis are called initial conditions; this usage is similar to Chow's definition (1959, p. 275) of an initial section for standard step-backwater computations of water-surface profiles in natural channels.

3. SCHEMATIZATION OF THE STREAM NETWORK

The first step in applying FEQ to the analysis of a stream system is the development of a schematic diagram that subdivides the stream system into a series of connected flow paths. A flow path conveys water between points in the stream system. Examples of flow paths, as applied in FEQ, are a stream channel, canal, storm sewer, or reservoir. Additional examples of flow paths on the stream are an overflow spillway, a swale that carries water overland during floods, or a breach in a levee or dam. In FEQ, these flow paths are connected by special features, which include culverts, bridges, dams, junctions, sluice gates, and other components of the stream system that do not fit the concept of flow paths considered in FEQ (branches, dummy branches, and level-pool reservoirs). The conceptual descriptions of each of the flow paths are given in sections 3.1.1–3.1.3. The primary hydraulic characteristic that distinguishes special features from flow paths in FEQ simulation is that the special features are potentially major flow transitions, either natural or constructed, small enough that changes in storage and momentum content can be neglected and relations between water-surface elevation and discharge can be derived from steady-flow principles. More than 20 special features are considered in FEQ. Thus, the stream system for unsteady-flow analysis with FEQ can be described in a schematic diagram that shows the branches, dummy branches, and level-pool reservoirs and the connections with the special features. The flow paths and special features are discussed below and in subsequent sections.

To draw the schematic diagram, the locations and types of special features in the stream system must be identified. Once the schematic diagram has been drawn, the flow paths must be labeled. The end nodes of a flow path have a special status because they connect the flow path to the rest of the stream system. A flow-path end node defines the end of the flow path. Each flow-path end node is labeled. A labeled schematic diagram is the basis for describing the various connections of the stream system as modeled in FEQ. The schematic diagram also defines how all parts of the stream system are to be modeled with the mathematical relations available in FEQ and the companion utility program, Full EQuations UTiLities (FEQUTL) (D. D. Franz and C. S. Melching, in press).

A map of an example stream system is shown in figure 2, and a sample schematic for this stream system is shown in figure 3. Each special feature has been isolated and the flow paths begin and end with a flow-path end node. Some of the flow-path end nodes on branches have been labeled in figure 3. The rule for labeling is very simple: The upstream flow-path end node on a branch is denoted by the letter “U” followed by the branch number, and the downstream flow-path end node is denoted by the letter “D” followed by the branch number. Because the rule is simple and is tied to the branch number, labels for flow-path end nodes are commonly omitted on branches in the FEQ schematic. The nodes on a level-pool-reservoir flow path or on a dummy-branch flow path must be labeled. The label is formed by the letter “F” followed by a number chosen by the user, as defined below. The stream system shown in figure 2 includes two run of the river dams that are represented by branches in FEQ simulation, as illustrated in figure 3.

3.1 Physical Features

The physical (geomorphologic) features of the stream system are divided into four categories in the stream-network schematization applied in FEQ: branches, dummy branches, level-pool reservoirs, and special features. Detailed descriptions and examples of each of these categories of physical features and the methods used to characterize these features in FEQ are given in the following sections.

3.1.1 Branches

A branch is the length of channel between special features (boundaries, junctions, and flow-control structures). Flow through a branch is described by the governing equations described in section 5, and in this sense every branch is identical. A branch is subdivided into computational elements for developing the approximate algebraic governing equations (see section 6). A branch also has nodes at the boundaries of these computational elements, each node representing an associated cross section. The nodes at the two ends of the branch are called flow-path end nodes, and those not on the ends are called interior nodes. The branch has an upstream end and a

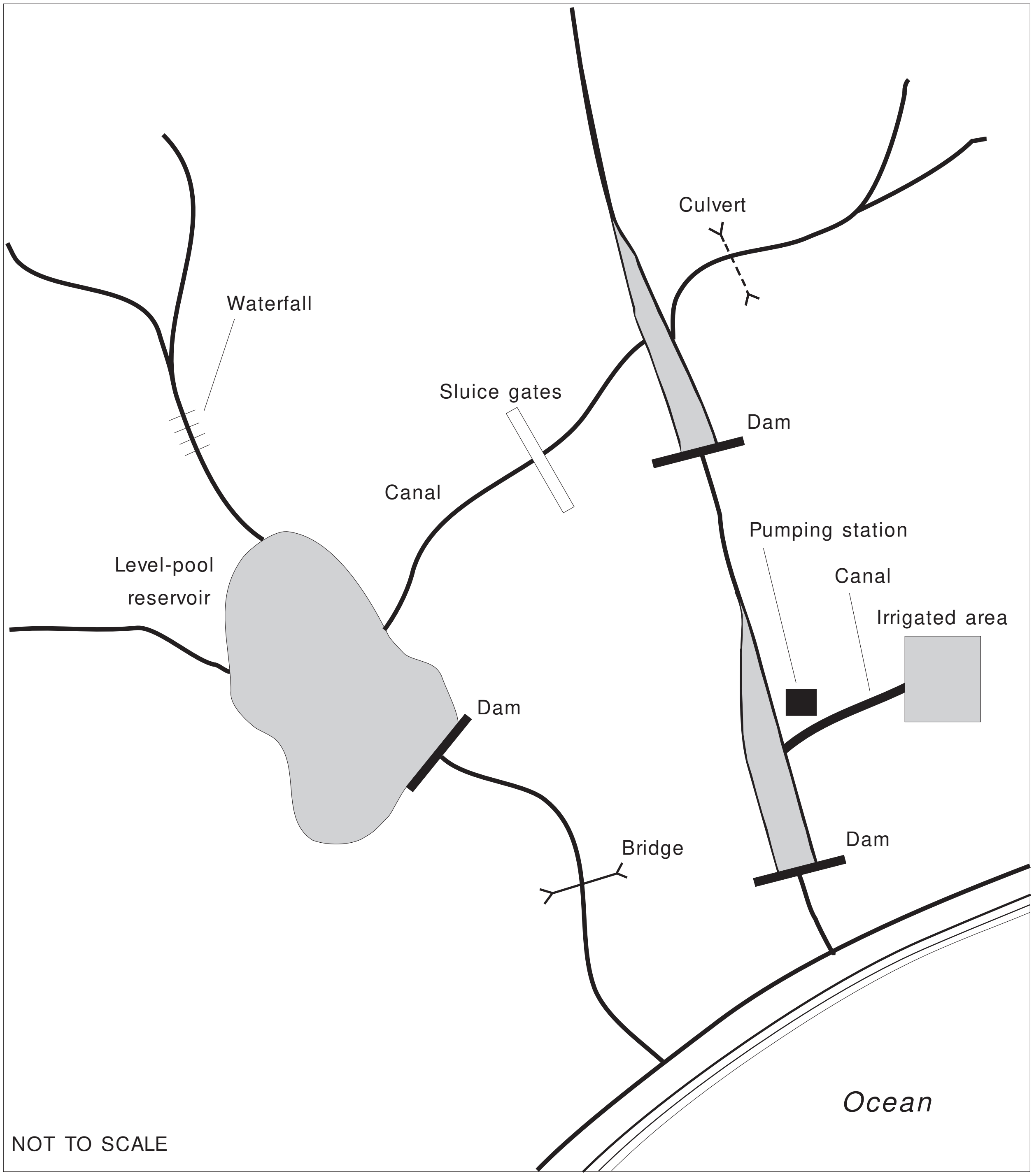


Figure 2. An example stream system for illustration of schematic-diagram preparation for the Full EQuations model.

downstream end that the user must assign. For example, the node at the upstream end is called the upstream flow-path end node. The nodes on a branch are numbered for identification, the numbers increasing and consecutive from the upstream end to the downstream end. A station—the distance measured along the stream from some convenient reference point—is assigned to each node on the branch. The elevation of the minimum point in the cross section is a key feature of each node on a branch. The station-elevation pair for each node defines the bottom

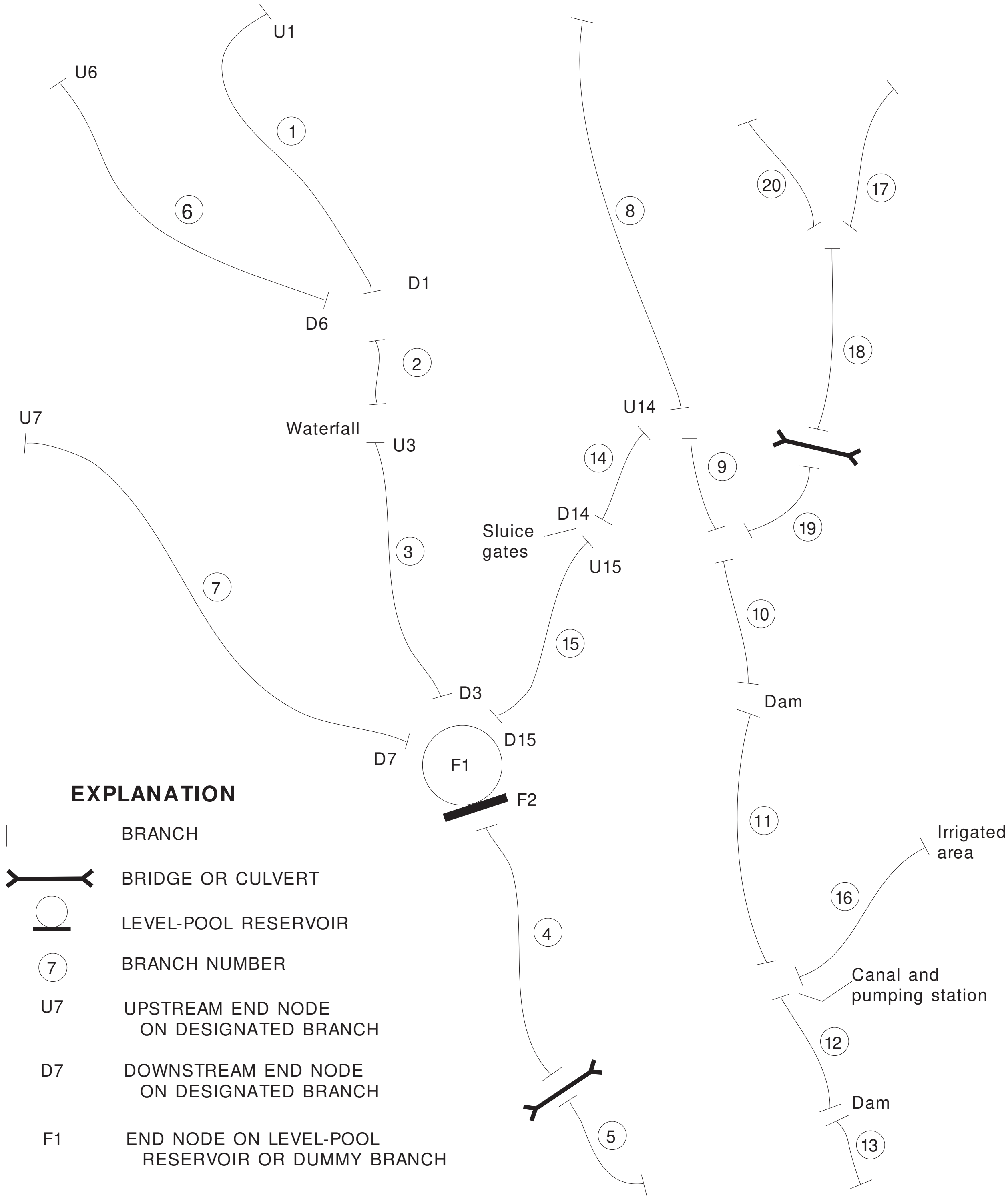


Figure 3. Schematic of example stream system for input to the Full EQuations model.

profile of the stream channel. The absolute value of the difference in the stations of two consecutive nodes on a branch gives the length of the computational element between the two nodes.

Water can enter a branch in three ways: as inflow at the two ends or inflow from the area tributary to the branch. Thus, an associated tributary area may be assigned to each branch and computational element. The tributary area for a computational element is the area that will contribute lateral inflow to the computational element. Lateral inflow to a computational element may include diffuse overland flow, seepage into (or out of) the channel, and point discharges from tributary streams or storm sewers too small to simulate explicitly. Most often the lateral inflow from a tributary area is estimated by a hydrologic model producing unit-area values of runoff intensity for one or more types of land cover. For example, the area tributary to a computational element may consist of agricultural, forest, and urban land. Each of these land covers would have a different rainfall-runoff relation in the hydrologic model. Therefore, it is convenient to allow the subdivision of the tributary area into the different land-cover types used in the hydrologic model.

An additional factor to the estimation of lateral inflow into the computational element is the gage; that is, the precipitation gage where rainfall was measured from which the runoff was computed. More than one gage may be available in a watershed. To consider multiple gages in a watershed in FEQ simulations, the tributary area for each computational element must be associated with the gage used to compute the unit-area runoff intensity.

A final feature of a branch required in FEQ simulation is a way of identifying and referencing the branch. Each branch defined in FEQ must be given a positive number for this purpose. Branch numbers can be nonconsecutive, but they have an upper limit as discussed below in the section on Flow-Path End Nodes (3.2.2).

3.1.2 Dummy Branches

A branch conveys water along a certain flow path whose characteristics include depth, area, and length. Other flows paths have these characteristics, but these details are not of interest in FEQ application. An example is the flow of water over the emergency spillway of a reservoir: depths in the spillway and the associated channel are of interest when these items are designed, but only flow over the emergency spillway is of primary interest in an unsteady-flow analysis. The water volume in the short, steep discharge channel associated with the spillway is too small to have an effect on the results, so a dummy branch is designed to represent such a flow path, as illustrated in figure 4.

A dummy branch has two flow-path end nodes but no associated cross sections. The only values of interest are the water-surface elevations and the flows at the nodes. For computational purposes, a small storage and friction loss must be assigned to the dummy branch, but these values are set so small that the flows and elevations at the two nodes are nearly equal. Other examples of the application of dummy branches include flows over a levee, multiple outflow paths through or around a dam, and intermittent flow of water over land connecting two streams or reservoirs.

3.1.3 Level-Pool Reservoirs

The final flow path as previously defined is a level-pool reservoir. Storage volume of a level-pool reservoir is large enough relative to the volume of flow entering and leaving the reservoir that the water surface can be treated as horizontal with only a small error in the results. A level-pool reservoir, like a branch and a dummy branch, has two flow-path end nodes (fig. 5). One node represents inflow to the reservoir, and the other node represents outflow. Long and narrow reservoirs or lakes often do not conform to the level-pool assumption and should be treated as branches in FEQ simulation because the flows result in an appreciable slope on the water surface.

3.1.4 Special Features

The identification and description of special features in a stream system is a major part of building a mathematical model of the stream system. A key hydraulic aspect of special features is their size; these features are so small that storage and momentum content changes may be neglected and relations between water-surface elevation and discharge may be derived from steady-flow principles. The variety of special features in streams systems is endless, especially in urban streams. The following are some examples of special features:

1. **Junctions between or among tributaries or distributaries.** Junctions are locations where two or more channels meet and combine to form a single channel (figs. 1 and 2). Locations where a single channel splits and forms two or more channels also are junctions. Multiple inflows for the inflow node of a level-pool reservoir

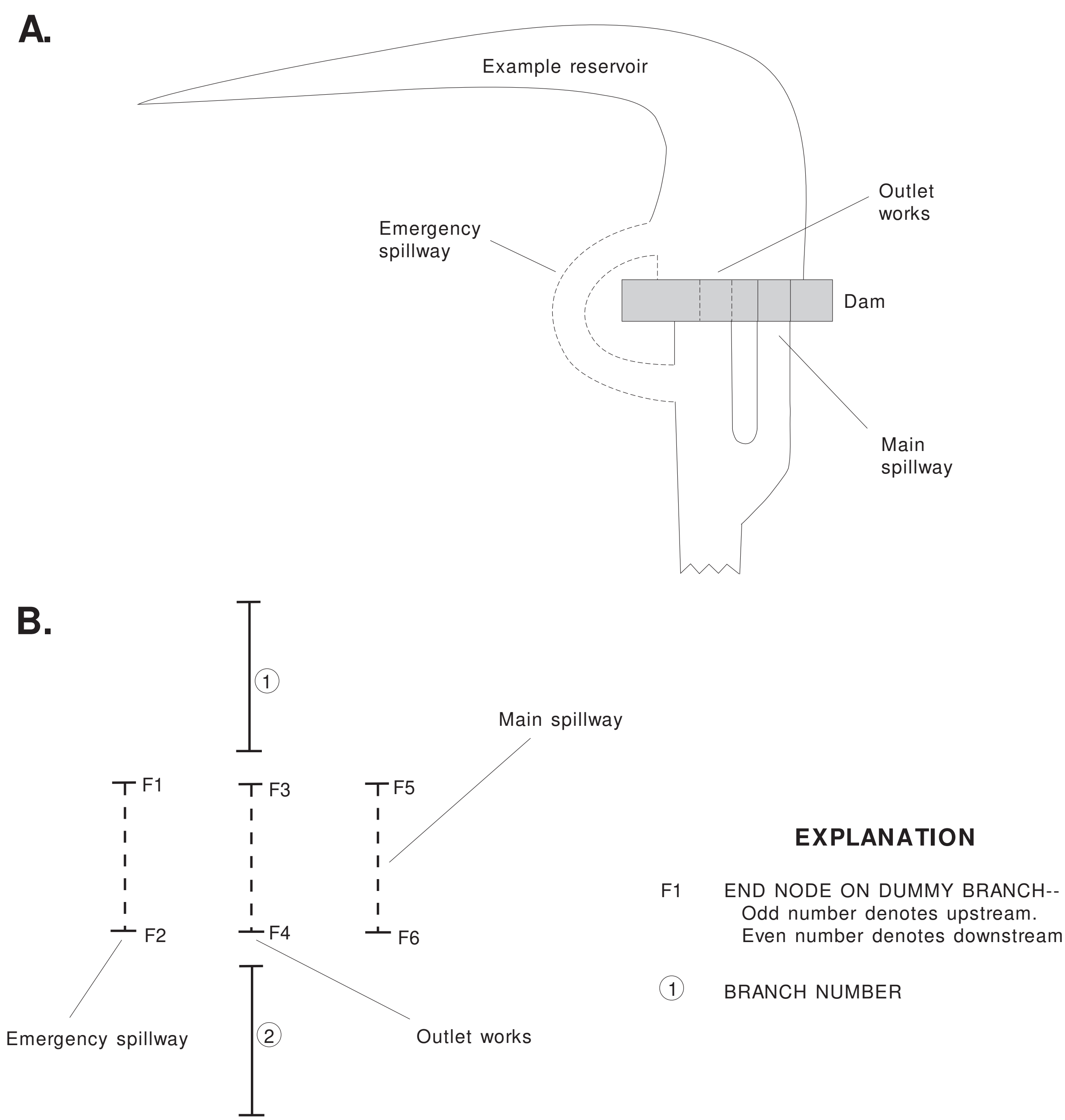


Figure 4. Example dummy branch: (A) dam with spillways and (B) schematic diagram of dam with spillways application in the Full EQuations model.

also can be represented by a junction (fig. 5). Junctions are always present at connections between flow paths; they establish the relation among the flows in the flow paths at the connections.

2. **Points of known water-surface elevation or of known or knowable flow.** These points are generally the logical places for boundary conditions. The values of water-surface elevation or flow can be functions of time and need not be constants.
3. **Points of known relations between water-surface elevation and flow rate, such as streamflow-gaging**

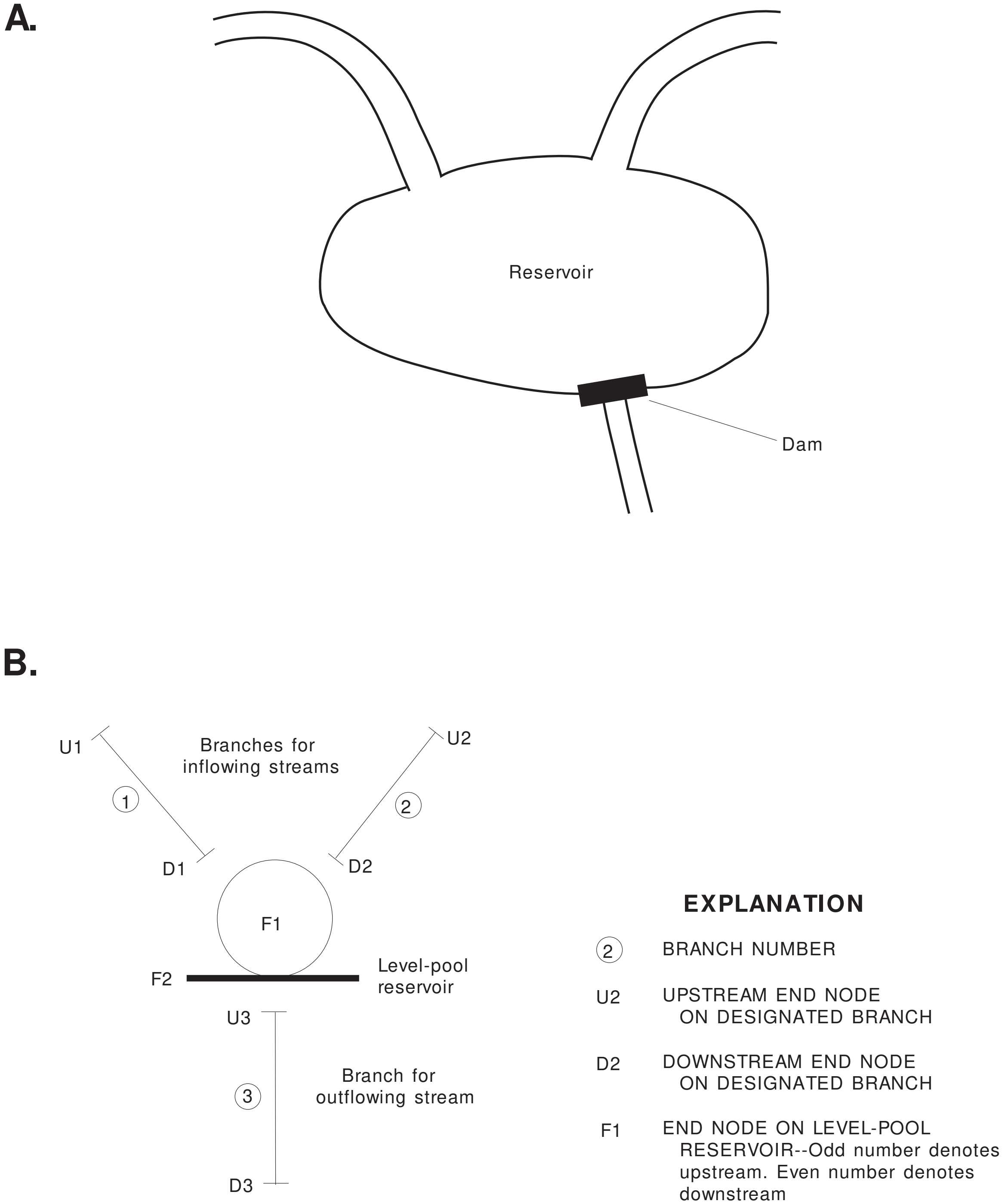


Figure 5. Example level-pool reservoir: (A) sketch of level-pool reservoir and (B) schematic diagram of a stream system with level-pool reservoir application in the Full EQuations model.

stations. These also make good boundary conditions especially at the downstream boundary.

4. **Any change in bottom slope that might be large enough to result in a critical control.** Critical controls must be isolated because branches that include supercritical flow must be treated differently than those characterized entirely by subcritical flow. Locations of potential critical flow must be isolated for proper analysis of steady or unsteady flow.

5. **Any abrupt change in channel shape or roughness.** These transitions must be isolated to account for the additional expansion or contraction losses.
6. **Dams, control weirs, and large pumping stations.** These and similar features can have a substantial effect on the water-surface elevation or the amount of water flowing in the stream.
7. **Drop structures, falls, or rapids.** These will be control points, at least at certain flow levels. These control points can be drowned out at high flows and reestablished at lower flows in FEQ simulation.
8. Bridges and culverts that are to be represented explicitly.
9. Points at which a special feature may be added to improve the control of the stream system.

Anything that is not a branch, a dummy branch, or a level-pool reservoir is a special feature. The list of special features can be further grouped into the general classes of junctions, boundary conditions, and control structures.

Control structures can be further grouped into several subclasses. A control structure is any physical feature that exerts a measure of control on the flow. If the control of flow is complete, so that a unique relation between flow and water-surface elevation is established by the structure, then the structure is called a one-node control structure because only the value of flow or elevation need be known at one flow-path end node to fully define the other value. If the control is incomplete, in that knowledge of the water-surface elevation at two flow-path end nodes is needed before the flow is defined, then the structure is called a two-node control structure. A major challenge of unsteady-flow analysis is often the identification and description of the control structures.

3.2 Computational Simplifications for Schematic Development

Several simplifications are applied in the FEQ schematization of the stream (open-channel) system so that the physical properties of the open-channel system and the general movement of flow between and among the physical features described earlier can be efficiently considered. These simplifications are described in the following sections.

3.2.1 Functions and Function Tables

A function is a mapping from one set of numbers, called the domain of the function, to another set of numbers, called the range of the function. For any number in the domain there must be only one number mapped in the range so that the function will be single valued. This definition is abstract, but it is the basis for the traditional function definition used by most engineers and scientists.

An important example from open-channel flow analysis is the top-width function for a cross section. The top width for a cross section is the width of the water surface at any elevation in the cross section from the minimum point to some user-established maximum point (the elevation domain). For an elevation in the elevation domain, a single value of top width will be determined from the top-width function. Other functions associated with a cross section include the area function, conveyance function, and the wetted-perimeter function. Characteristics of the cross section are viewed as a function because they are the features of a cross section considered in the governing equations.

Other functions of interest in flow analysis include stage-discharge relations at gaging stations, head-discharge relations for a wide variety of special features, elevation-area-storage relations for reservoirs, and inflow hydrographs. Defining these functions is one of the major tasks in the analysis of any stream. Hundreds of functions may have to be defined for even a small to medium-sized stream system.

Most of these functions of interest in flow analysis are not known as simple mathematical expressions, so solution of the governing equations requires description of various functions in a way that is both flexible and convenient. Function tables are used in FEQ simulation for nearly all functions needed in unsteady-flow analysis. A function table consists of a set of selected argument values (the tabulated argument set) and the corresponding set of function values, as well as a rule for defining the function values for arguments not in the tabulated argument set. This approach is taken because most functions of interest are known only approximately, and some error can be allowed in the function value and in the rule used to compute the values not found in the table. Consequently, the characteristics of the cross sections used in FEQ simulation are computed in the utility program FEQUTL (D. D. Franz and C. S. Melching, in press) and placed in specially designed function tables called cross-section tables. The cross section is not used in simulation except as reflected in the cross-section function table. (The need to store cross-section characteristics in function tables is another major difference between steady-flow and unsteady-flow analysis; the characteristics of the cross section are computed as needed from fundamental (raw) cross-section data in many steady-flow programs.)

The cross section is normally defined by a set of selected points on the periphery of the cross section in some convenient coordinate system; the points are measured in the field or taken from topographic maps with the assumption that adjacent points may be connected with straight lines. The cross section may be subdivided by vertical, frictionless, fictional walls to account for problems with application of the hydraulic radius to describe the shape of the cross section when computing the conveyance for compound and composite channels. A compound channel is a channel whose cross section consists of subsections of variously defined geometric shapes (Yen, 1992, p. 64). The most common example of a compound channel is one with flood plains. A composite channel is a channel whose wall roughness changes along the wetted perimeter of the cross section (Yen, 1992, p. 60). For compound and composite channels, each subdivision also may be assigned a separate value of Manning's n in FEQ simulation to account for variations in roughness along the periphery of the cross section.

The approach of computing the cross-sectional characteristics as required from the fundamental or raw cross-section data is not efficient for unsteady-flow analysis. In steady-flow analysis, cross-sectional characteristic values need be computed only a few times. In unsteady-flow analysis, however, values of cross-sectional characteristics may be needed many thousands of times; therefore, it is economical in terms of computer time to place the computed cross-sectional characteristics in a cross-section function table for later access.

Many types of function tables are supported in the FEQ model. Three broad classes of function tables are one-dimensional, 1-D (one argument, perhaps several functions), two-dimensional, 2-D (two arguments, several functions), and three-dimensional, 3-D (three arguments, several functions). Six options are available for cross-sectional characteristics (1-D table with several functions), three for 2-D tables, four for functions of time (1-D table with one function) such as hydrographs, and three for other 1-D tables. Details on the arguments, the values tabulated, and the methods applied for interpolation are given in Franz and Melching (in press).

3.2.2 Flow-Path End Nodes

Flow-path end nodes have already been defined as the nodes on ends of a branch, a dummy branch, or a level-pool reservoir. The function of flow-path end nodes requires labeling them so that they can be referenced later. For a branch, the flow-path end nodes will have two labels: the label for the node when it is referred to in the schematic description of the stream system and the label that is assigned to it in the computation of the governing equations for the branch. The label assigned for the computations for a branch must be a number, whereas the label assigned for the node as a flow-path end node in the stream-system schematic may be a number but need not be a number.

Two styles for the treatment of flow-path end nodes are available in FEQ because of changes and enhancements to the program. In the older style, derived from earlier versions of the program, the flow-path end nodes must be numbered in the range 1 to 1998. The numbers for flow-path end nodes on branches are limited to the range of 1 to 999. In contrast, nodes in the new style must be labeled with alphanumeric information to make stream-system modeling easier. In the new style (figs. 3 and 4) the upstream flow-path end node on a branch must be composed of the letter "U" followed by the branch number with no intervening spaces. For example, the upstream flow-path end node on branch 51 would be labeled by U51, and the downstream flow-path end node on that branch would be labeled D51. The flow-path end-node label gives both the branch number and the location on the branch. Under the new style, the branch numbers must be in the range of 1 to 999.

The flow-path end nodes on level-pool reservoirs and dummy branches are composed of the letter "F" followed by a number in the new style (figs. 3, 4, and 5). The letter "F" signifies that the node is free of a branch. The number must be in the range of 1 to 999. Consistent use of either odd or even numbers for downstream nodes on these flow paths is preferred. If all the downstream nodes are even and all the corresponding upstream nodes are the preceding odd number, more information about the location and function of each node is conveyed, and modeling is easier and less prone to errors.

3.2.3 Flow-Sign Convention

All possible stream systems can be described by the correct combination of branches, dummy branches, level-pool reservoirs, and special features. The special features form connections between and among end nodes on flow paths. The flow paths can be connected in any order and direction desired. The direction called downstream is defined by the user and need not be the physically downstream direction. In some cases, flow direction is unknown; thus, the meaning of downstream must be assigned by the user. The downstream direction on any path is always from the upstream node to the downstream node. This direction is assigned for a branch by the order in which the nodes on it are input to FEQ. For a dummy branch and a level-pool reservoir, the upstream and downstream flow-path end nodes are explicitly specified. Such specification also is done for the flow-path end nodes on branches for the older style of input.

A precise and simple sign convention is applied in FEQ for indicating the direction of flow at a node. If the flow is printed as a positive number, it is moving downstream. If the flow is printed as a negative number, it is moving upstream. This also indicates that a positive flow value at an upstream flow-path end node is flow into the path to which the node is attached. For a downstream flow-path end node, a positive flow is flow leaving the path to which the node is attached. A negative flow gives the opposite direction at a flow-path end node.

3.3 A Physical Analogy of the Schematization of the Stream Network

The schematization of a stream network applied in FEQ simulation is analogous to a child's Tinkertoys, a building toy composed of slender, pencil-like sticks and round knobs with holes on the periphery and a hole in the center of each knob. A wide variety of stick structures can be built by inserting the sticks into the holes of the knobs. In a sense, open-channel hydraulics are conceptualized like Tinkertoys in FEQ simulation. The branches, dummy branches, and level-pool reservoirs are like the sticks, and the special features are like the knobs. A model of an open-channel system can be built by use of these parts. Consequently, FEQ has few predefined limits. The limit to the complexity of the simulated stream system is usually set by the memory and computer-time requirements rather than by structure of the program. As a result, various stream systems and conditions can be described without requiring changes to the program.

In keeping with the Tinkertoys analogy, the parts of the system are described in FEQ simulation more or less separately. For example, the branches are completely described in terms of nodes, stations, elevations, and cross sections before any description of the special features is given. Furthermore, the cross sections are described only in terms of a table number, and the contents of the table are input later. All references to functions are given by the table number containing the description of the table. This is done to allow attention to be focused on how the various pieces are connected without concern for the location, size, or shape of the cross sections. This layered approach to describing the system reduces the number of details that must be comprehended simultaneously, simplifying management of the details in the application of FEQ.

4. DESCRIPTION OF THE CHANNEL GEOMETRY

Channel geometry—the description of the size and shape of the channels in which water flows—is often given cursory treatment in modeling documentation although it forms the foundation of any analysis of open-channel hydraulics. The description of the hydraulic geometry should be consistent with the demands of the analysis and the requirements of the governing equations. Use of 1-D flow equations is assumed throughout the analysis; therefore, an extensive review of implications for the description of the channel geometry is of great benefit.

4.1 The One-Dimensional Assumption

Because 1-D analysis ignores accelerations and velocities other than those in the longitudinal direction, 1-D analysis in even its most complex form in unsteady flow is approximate: no flow is really one dimensional. Nevertheless, analysts frequently lose sight of the approximate nature of 1-D methods, and focus too much analytical energy on what may prove to be trivial parts of the method. There are no easy answers in defining triviality. The only ultimate answer is comparison to experimental measurements and for some questions that may be difficult and expensive. In this report, judgment is used in defining what is important and in making general recommendations on approximations. The FEQ user should weigh these recommendations on the basis of new information either from the literature of hydraulics or measurements in the field or laboratory. Engineering judgment is a necessary part of all analysis because problems must be solved with the information at hand and with current tools.

Whitaker (1968, p. 212) makes a comment about this issue in a chapter on macroscopic balances (his term for 1-D analysis):

> The student should be forewarned that the methods to be studied in this chapter, and subsequent ones, are approximate; in general, there will be no "right" answers. There will often, however, be a "best" answer, and as often as possible we shall try to determine the best answer by comparing our results with experiments. In attacking this chapter, we should remember the macroscopic balances are perhaps the most powerful tool the engineer possesses for solving the often ill-defined problems of everyday practice. Judicious application of these equations comes only with experience and practice. At best, the student can hope to understand the development of the equations and gain some insight regarding the difficulties that may be encountered in their application.

In the methods used in FEQ and in applications of FEQ, 1-D analysis is pushed to its limits; therefore, simplifications involved in 1-D analysis need to be thoroughly understood. Consideration also must be given to the requirements on the hydraulic geometry to meet the assumptions of 1-D analysis and to development of a description of a stream channel that is consistent with these assumptions. An appreciation for the approximations of the hydraulic geometry is necessary to prevent overconfidence in the results of the FEQ analysis.

4.2 Directional Changes

The principles of conservation of mass and momentum are applicable to any flow, and simplification to a 1-D approximation is valid if the assumptions are met. These assumptions apply to linear channels that do not change shape rapidly. The classic example is the laboratory flume or a straight reach of a canal. A straight reach of a natural channel is an example of a gradual change in shape and a flow that follows a virtually linear path. However, many unsteady-flow model applications involve changes in direction of the channel. This change introduces curvilinear flow, which is a violation of the 1-D flow assumptions. This curvature is in the horizontal plane, however, and not in the vertical, so the hydrostatic-pressure-distribution assumption is still valid. The selection of the principle of conservation of momentum now causes a problem. This principle involves the momentum of the water and the forces exerted on the surfaces that confine it. Momentum and force are vectors, having both a magnitude and direction. Changes in direction can be as significant as changes in magnitude. Several questions need to be considered. First, is it valid to apply momentum conservation in a curved flow, ignoring the curvature and the nonhorizontal water surface in channel bends? Second, how should the distance axis be defined for a curved channel of varying size and shape? Third, how should the channel geometry be defined so that the equations are as simple as possible yet still retain physical meaning? For 1-D flow, distance and time are the only coordinates. Furthermore, the functions defined must be valid in terms of only distance and time. To properly integrate and differentiate with respect to distance along the stream, a reasonable definition of a distance axis must be established.

The definition of a distance axis includes mathematical and physical considerations. Mathematical considerations relate to the proper evaluation of integrals and derivatives. Therefore, careful mathematical and geometrical analysis will provide a definition. However, the effect of ignoring directional changes in the conservation of momentum principle is a physical question that only experimental results can answer. Miller and Chaudhry (1989) compared a physical-model result for a dam-break flood wave in a rectangular, prismatic channel with a 180-degree curve to the result obtained from a 1-D mathematical model that did not account for channel curvature. In this case, the definition of the distance axis was not a problem because the channel was rectangular and prismatic. For a prismatic channel, the distance along the centerline of the channel is the correct distance to use for the axis with the traditional equations for 1-D open-channel flow. Miller and Chaudhry (1989) found that the 1-D computational results with a postanalysis correction for super elevation of the transverse water surface in the bend derived for steady flow were acceptably accurate. The wave height for both the inner and outer bank was correctly estimated with the 1-D model if directional changes in its derivation were ignored but the super-elevation correction was applied. These experiments indicate that curved channels with compact cross sections and no overbank flow may be simulated with the conservation of momentum principle and that changes of direction can be ignored in the simulation.

The previously discussed experimental results likely apply to channels flanked by flood plains as long as the direction of flow in the channel and flood plain are nearly the same. If these directions differ markedly, then the effect of directional differences may be significant for certain water-surface elevations. A strongly meandering stream set in a broad flood plain is an example where flow directions differ markedly with changes in water level. Sometimes these can be represented by combining a 1-D model of the channel with a series of interconnected level-pool ponds to represent the flood plain (Cunge and others, 1980, p. 152-159). This approach also has been used with FEQ to model the complex flow paths around a series of diked islands in a braided estuary (Snohomish County Public Works, 1989).

If the directions of flow in the channel and flood plain are not too different and if concern about the effects of these directional differences on the momentum equation can be suppressed, then the 1-D equations can be generalized for application to curved open channels. To do this, careful definition must be given to how the channel size and shape should be measured and described.

4.2.1 Channel Geometry Requirements For One-Dimensional Analysis

For 1-D analysis to be valid, a three-dimensional stream system—where the flow variables are functions of time and three spatial dimensions—must be transformed to a simplified system where all variables are functions of distance and time. The selection of the cross-section locations and orientations and the distance axis must satisfy the requirements of this transformation. Picking the location and orientation of these cross sections becomes increasingly difficult as 1-D methods are applied to flow patterns that are increasingly complex. Many of these flow patterns require two- or three-dimensional (2-D or 3-D) methods; however, as the dimension of the analysis increases, the data requirements and computational effort also increase. The increase is not linear but steeply nonlinear, so that a 3-D analysis, if even possible, could require on the order of 10 times the personal time and 100 times the computational effort of a 1-D analysis. A 2-D analysis might require twice the personal time and 10 times the computational effort.

For the transformation to be valid, there must be an identifiable, predominant flow direction. The flow field in the predominant flow direction will be many times longer than the width or depth. In open-channel hydraulics, the velocity is commonly assumed to be the same at any point in a cross section normal to the predominant flow direction. This assumption on velocity restricts applications severely; therefore, most analyses allow for differences in the magnitude and direction of the velocity in a cross section.

4.2.2 Orientation of Cross Sections

The cross sections of the stream selected as computational nodes are oriented so that the computation of mass flux through a section is simplified. The mass flux through a small incremental area, ΔA, of a fluid of density, ρ, and with a velocity, v, is $\rho v \cos \theta \, \Delta A$, where θ is the angle between the direction of the water velocity and the direction of a line normal to the incremental area. The total flux through an area, A, is $\int_A \rho v \cos\theta dA$, where the angle and the magnitude of the velocity can vary over the area. To compute the mass flux, an estimate must be made of the angle that the local velocity vector makes with the normal to the cross-section surface at all points in the cross-section surface. The computation of the mass flux is simplified if the cross-section surface is defined to be normal to the local velocity vector. Then, $\theta = 0$ and $\cos \theta = 1$ everywhere in the cross section, so the mass flux becomes $\int_A \rho v dA$.

To select the cross sections, the flow directions must be known. The problem of needing flow directions before solving the flow equation is only apparent and is similar to the generalization from a uniform velocity in a plane cross section to a nonuniform velocity in a plane section. This generalization forces coefficients to be introduced so that expressions in which the velocity is averaged across the section give the correct flux of momentum or energy. For example, the total flux of momentum across an area in the flow is $\int_A \rho v^2 dA$. The momentum-flux correction coefficient, β, is then defined by $\beta A V^2 = \int_A v^2 dA$, where V is the cross-sectional average velocity and ρ is a constant. The local velocities appear to be needed before β can be defined, but that is not true; only knowledge of the distribution of the velocity in a relative sense is needed. The actual velocities do not need to be known and, in practice, the variation of conveyance across the stream channel is used to provide a surrogate estimate of the velocity distribution.

When the cross sections are curved, the momentum flux also should reflect the directional nature of momentum flux. The momentum per unit volume in the direction of the distance axis becomes $\rho v \cos \phi$, where ϕ is the angle between the direction of the velocity and the direction of the distance axis at the location of the cross section. The defining equation for β then becomes $\beta A V^2 = \int_A v^2 \cos\phi dA$. The effect of the direction on the momentum flux is small. A constant value of $\phi = 15^\circ$ yields an error in momentum flux of less than 4 percent. The effect of curvature on momentum flux is only substantial if the cross section is strongly curved. The same reasoning applied to the integrated hydrostatic pressure on a cross section analyzed in the plan as defined here, shows that the error in the downstream component of force also is less than 4 percent.

Although exact definition of the flow direction is impossible, reasonable assumptions can be made that provide satisfactory results. Approximate flow directions can be defined by use of a topographic map of the stream and flood plain, at a scale large enough to show sufficient detail. The general direction of flow when the depth of flow is not shallow is the central point of attention. Judgment is required to (1) select the features that are to be included in the definition of the cross-section boundary and which features are to be considered part of the roughness, and (2) determine the general direction of the flow from the pattern of contour lines over a region and not at a point. The goal is to assign the directions of flow that will be followed at each location when the flow is high enough to minimize the effect of local features.

Determination of appropriate flow directions may depend on the water-surface elevations assumed for the stream. An example is a strongly meandering stream with a broad, gently sloping flood plain, as sketched in figure 6. When the flow is within the banks of the main channel, the direction of flow is the same as the direction of the main channel. As the water level increases, water moves into the flood plain, and some water takes a shorter route downstream. At yet higher water levels, the main channel becomes deeply submerged and the direction of flow is affected primarily by the flood-plain topography. The principal difficulty in this example is deciding at what water levels the change in direction results. In some streams, the directional effect of the main channel will always be significant; in others, it will not.

The stream shown in figure 6 is the most general case, and lines showing the direction of flow sketched on a topographic map, called flow lines, would intersect. Lines that intersect would be those applicable to the flow at various water levels. Cross sections drawn locally perpendicular to these flow lines (orthogonal to the flow lines)

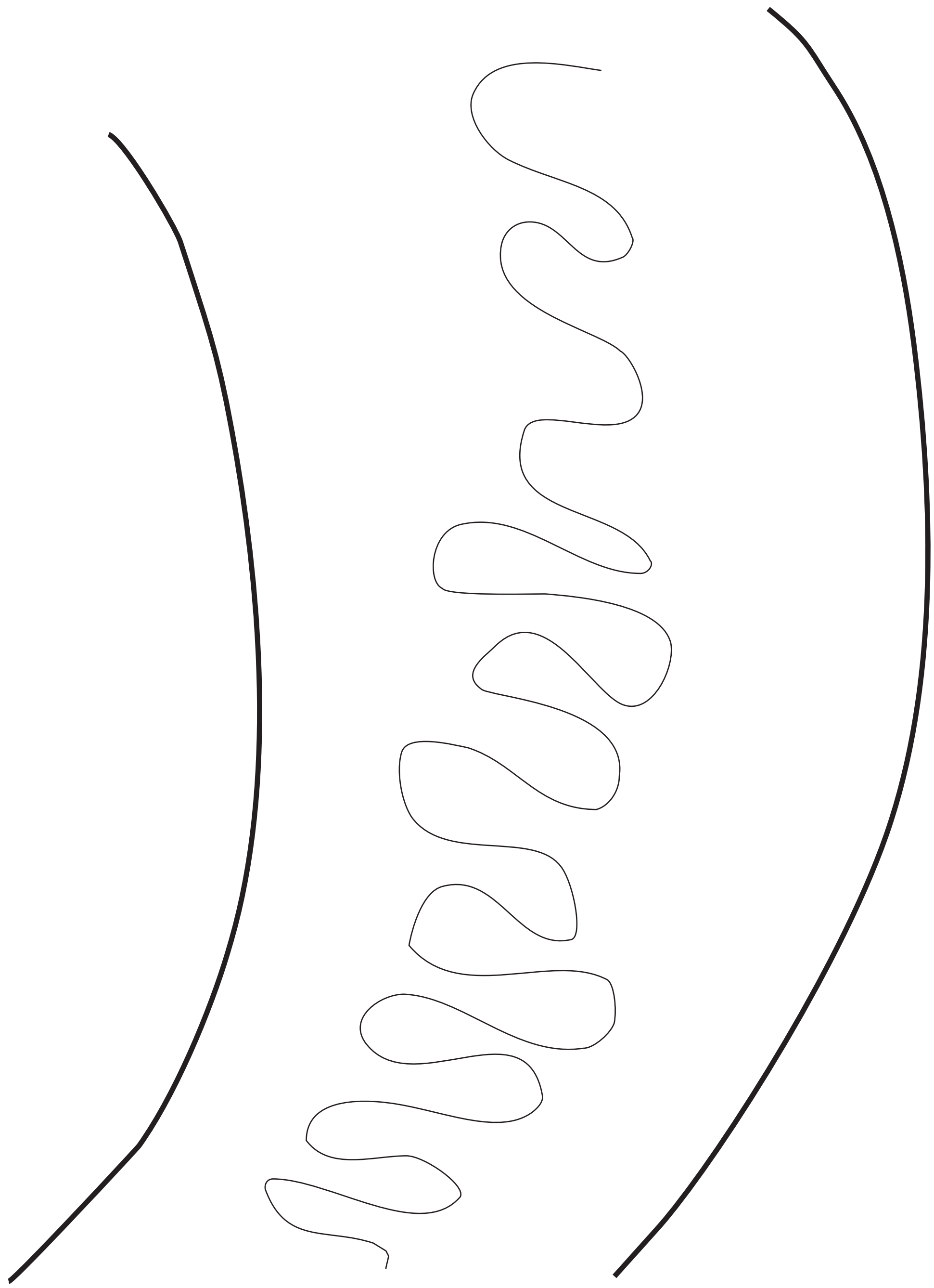


Figure 6. Example of a strongly meandering stream for illustration of the approximations of curvilinearity used in the Full EQuations model.

would then have curvature in the horizontal plane that changes with the water level; that is, the cross sections would change orientation or twist as the water level increases. Hence, they become 3-D cross sections instead of the 2-D cross sections currently used in available modeling systems. Measuring such sections in the field would be difficult, and a detailed topographic map of the stream and flood plain would be needed to complete the cross sections.

Describing these cross sections so that the cross-sectional characteristics could be computed would be difficult. The flow lines between sections also must be described in some way to specify the geometry. With these practical difficulties and the theoretical difficulties of major shifts in direction and the corresponding interaction between the flow in the main channel and the flood plain, representing a strongly meandering stream with a combined cross section for the main channel and flood plain seems unwise and impractical. The interaction between the main channel and flood plain in the computation of conveyance cannot be identified in a combined cross section. Better and perhaps simpler is to isolate this interaction and approximate this interaction explicitly by creating a system of two or more channels, each with the cross sections orthogonal to the corresponding flow lines, and by providing explicit interchange of flows between channels through use of flow relations that depend on differences in water-surface elevations between channels. These relations may be difficult to select, but the difficulty is explicit instead of implicit; this is a conceptual improvement because the uncertainties are more explicitly recognized and, thus, more caution can be exercised in presenting and using the results.

In figure 6, one or two channels would represent flood-plain flows and one would represent the main-channel flow. Direction of flow in each individual channel would be virtually independent of changes in water level. Thus, sketching of the flow lines becomes easier and the cross sections can be tailored to the needs of each channel with the cross-section spacing reflecting that channel only. The flow lines for a multiple-channel model would not intersect and flow-line intersection might be a reasonable criterion for deciding if a multiple-channel model is needed. If flow lines cannot be sketched that are, to a large degree, independent of changes in water level, then a multiple-channel model should be considered. Each channel would have a distinct distance axis and cross-section spacing and orientation. Computational difficulties may result for zero or near zero flow in the flood-plain channel for multiple-channel models during low flows. These difficulties are addressed later.

Interchange of water and momentum between interacting channels would be at interaction points placed at periodic intervals along the channels. These would approximate the continuous interchange in the stream system by a series of discrete weirs. By careful selection of the interaction points and definition of the flows, the theoretical basis for the results can be improved by making the interchange of momentum explicit. This multiple-channel model, although not much different in complexity than the single-channel model with variable-curvature twisted cross sections, nonetheless represents the strongly meandering stream more accurately with 1-D approximations than a single-channel model does. The approximation of the system can be improved only by use of a 2-D model at a considerable increase in time and effort.

Given the restriction that the flow lines should be independent of water-surface elevation, the cross sections in curvilinear 1-D flow can be defined by means of the following steps:

1. Sketch several flow lines on a topographic map showing the direction that water would flow if it were sufficiently deep to inundate minor topographic features. At least five flow lines are necessary: one along the thalweg of the channel, one near each bank of the main channel, and two showing the extent of the flood plain. Intermediate flow lines may be needed to reflect local variations within the main channel or on the flood plain.
2. Sketch the cross sections such that they are approximately orthogonal to the flow lines. If $\theta = 15$ degrees everywhere, then the error in the flow rate estimate is less than 4 percent, and if $\theta = 30$ degrees, the error is about 15 percent. (Moreover, in estimating differences between the mass inflow and the mass outflow, there is some opportunity for compensation of errors in defining the cross-section locations.) The cross sections will be curved surfaces, not planar. Ideally, the cross section measured in the field should follow this curved path as closely as is practical. The curved cross sections sketched on the topographic map may be replaced by plane cross sections if the plane sections do not intersect within the flow field and if the angles of intersection with the flow lines are approximately perpendicular.

A hypothetical example of this process is shown in figure 7. The main-channel boundaries and the flood-plain limits are considered to be approximate flow lines (although rigorously this is not true). The edge of the water in the channel is a flow line, but this line depends on the variation of water-surface elevation with distance that is unknown. Thus, the flow lines must be sketched in a general sense, with the assumption that they will apply approximately to flow at a wide range of water-surface elevations. (That is why the lines showing the direction of flow are referred to as “flow lines” instead of “streamlines.” To call them streamlines would make the analysis seem more exact than is really possible.) For any real flow, some of the sketched flow lines will likely be only partially under water; that is, one or more of the approximate flow lines will intermittently be included in the wetted part of the channel as viewed on the horizontal plane. This should not be a problem if the direction of the flow line is within an acceptable angle of the direction of flow when the flow line, as sketched, is inundated. What is

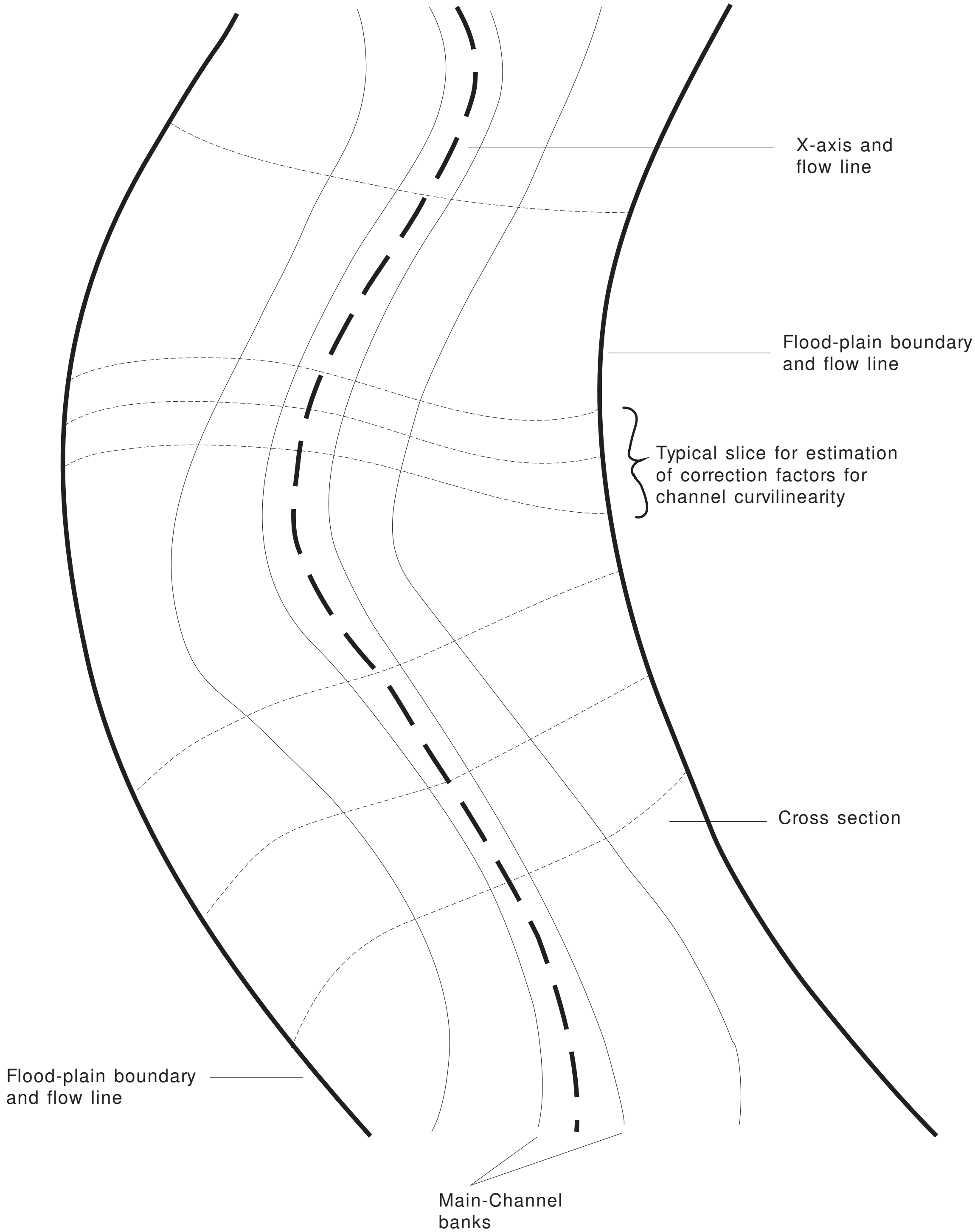


Figure 7. Example definition of flow lines and cross sections as used in the Full EQuations model.

acceptable depends on the desired accuracy of the analysis. Clearly, a difference of direction of 45 degrees is suspect, but a deviation of 30 degrees should not be of concern.

4.2.3 Selection of Distance Axis

The flow lines and intersecting cross sections define the channel geometry in three dimensions. Transformation of this geometry to one dimension requires definition of a curvilinear distance axis. The thalweg of the channel—that is, the locus of the minimum elevation points in the main channel—is a convenient choice for the distance axis. For some stream reaches, this definition applies to more than one part of the channel, but a reasonable choice can be made for those cases where the cross sections have multiple and equal points of minimum elevation. In the following discussion, the distance axis is assumed to be the thalweg of the channel or near to it.

Cross sections at any location along the distance axis must be defined such that they will not intersect and will approximately meet the requirements of a simple mass-flux computation. To apply the conservation of mass principle to the water in the stream, the volume of water between any two cross sections along the stream must be defined. Let x denote the distance variable, $y(x)$ denote the height[1] of the water surface above the minimum point in the cross section at the location given by x, and $A[x,y(x)]$ denote the area of the cross section at the location along the distance axis given by x. As x varies, the curvature of the cross section will vary. In the 1-D equations, all cross sections must be referenced by location along the distance axis; information regarding any other distances between cross sections is unavailable. The following is a development of a systematic method for summarizing the effects of this distance variation on the 1-D equations.

The apparently natural formula for computing the volume of water, S, in the channel between a section at x_1 and one at x_2 is

$$S = \int_{x_1}^{x_2} A\,[x, y\,(x)\,]\,dx\,, \tag{1}$$

where $y(x)$ is assumed to be a known function. Equation 1 is incorrect when applied to curvilinear channels. To obtain the correct volume by integrating in one dimension only, the integrand must represent a volume per unit length. The unit length is defined by the variable of integration. If the cross sections are plane and parallel, then $A\Delta x$ is a valid volume increment where Δx is the distance between cross sections at x_1 and x_2. For curvilinear channels, however, the cross sections are neither plane nor parallel. Thus, multiplication of a small distance increment by an area yields a volume increment, but not the correct volume increment. The volume increment would be correct if two cross sections Δx apart at the x-axis would be the same distance apart, measured parallel to the x-axis, at all other points in the cross section. This is true if the cross sections are plane or curved. If the cross sections are plane surfaces and the x-axis follows the path traced by the centroid of the cross sections, then it also is valid though the distance between cross sections varies with position in the section. These are special cases, although the latter is important because equation 1 is correct in this case if the cross sections are plane and the distance axis is the same as the path of the centroid of the areas. This can only result for varying heights of water, $y(x)$, if the cross sections are symmetrical about a vertical line and the distance axis coincides with the trace made by the line of symmetry on the horizontal plane.

In general, stream cross sections are not symmetrical, especially in bends, and the path formed by their centroids shifts with the water-surface profile. Thus, an approach is needed to represent the channel volume in a set of 1-D governing equations. Using DeLong's approach (1989), a weight coefficient is introduced so that the product of the weight coefficient and the area will yield a valid volume per unit length along the distance axis. This weight coefficient will vary with distance, water height, and the choice of the distance axis.

---

[1]The water height also is called the depth. Both terms are used interchangeably in this report. Height denotes measurement from some reference point to some other point above it. In some contexts this is the appropriate direction of measurement. In other contexts, depth, which denotes measurement downward, is more appropriate.

The weight coefficient, M_A, that will result in a valid volume per unit length when multiplied with the cross-sectional area is defined as

$$M_A(x, y_0) = \lim_{\Delta x_S \to 0} \frac{S_h(x - \Delta x_s/2, x + \Delta x_s/2)}{A(x, y_0)\Delta x_s}, \tag{2}$$

where Δx_S is a small increment along the distance axis of the channel, y_0 is a constant water-surface height, x is the cross-section location, $A(x, y_0)$ is the flow cross-sectional area at location x for water-surface height y_0, and $S_h(x_1,x_2)$ is the correct volume of water between cross sections at locations x_1 and x_2.

The computation of M_A can be done at enough points at the cross-section locations in a 1-D model so that the coefficient is defined as accurately as the area or top width. Because the weight coefficient is defined at any point, the volume integral can be written as

$$S_h = \int_{x1}^{x2} M_A[x, y(x)]\, A[x, y(x)]\, dx. \tag{3}$$

The evaluation of the exact volume of the slice needed in the definition is presented in the development of the actual means of computing the cross-section characteristics.

The conservation of momentum principle requires the evaluation of the change in momentum content of the water in the stream. This also requires an integration along the distance axis and over the flow field (cross section). This integration is complicated because the flow is not uniformly distributed along the cross section, the momentum content is a vector quantity, and the direction for the vector is taken as that given by the distance axis. Thus, to be rigorous, the analysis should include the cosine of the angle between the direction of the distance axis and the assumed direction of the velocity at each point in the volume of water for which momentum content is estimated. However, the results of the experiments of Miller and Chaudhry (1989), mentioned previously in this report, indicate this refinement is not needed. Thus, the momentum content is considered in FEQ as if the angle were zero everywhere.

The weight coefficient, M_Q, that will result in a valid momentum content per unit length when multiplied with the total flow rate through the cross section is defined as

$$M_Q(x, y_0) = \lim_{\Delta x_S \to 0} \frac{S_q(x - \Delta x_s/2, x + \Delta x_s/2)}{Q(x, y_0)\Delta x_s}, \tag{4}$$

where $Q(x,y_0)$ is the total flow rate through the cross section at location x for water-surface height y_0, and $S_q(x_1,x_2)$ is the correct momentum content of the flow between cross sections at locations x_1 and x_2. The momentum-content integral can be written as

$$S_q = \int_{x1}^{x2} M_Q[x, y(x)]\, Q[x, y(x)]\, dx. \tag{5}$$

The means for defining the flow per unit width is described in section 5.

These two weights, M_A and M_Q, represent the effect of curvature of the channel on the application of the 1-D principles of conservation of mass and momentum, respectively, to open-channel flow. The methods applied to determine the exact volume by means of multidimensional integration are described later, after descriptions of how to develop other cross-section characteristics that depend only on the cross section and not on its orientation or the location of the distance axis.

4.3 Characteristics of a Cross Section

The characteristics of a cross section can be placed into three classes: static, dynamic, and curvilinear. For curvilinear characteristics, weight coefficients for integrands are derived in the previous section. The static and dynamic characteristics of a cross section are described in the following sections.

4.3.1 Static Characteristics

The static characteristics are fixed for a given water depth at a given location along the channel. These characteristics are the top width, the wetted perimeter, the area, and the first moment of area about the water surface.

4.3.1.1 Area

A typical cross section in outline form is shown in figure 8. If the cross section was curved, the figure would show true length along the cross section, not the projection on a plane surface. The top width, $T[x,y(x,t)]$, is a function of the distance along the channel, x, and the height, y, of the water in the channel. The water surface is assumed to be horizontal as required for 1-D open-channel flow. The top width is the horizontal distance across the cross section at a given height in the plane (possibly curved) of the cross section. The area of flow in the cross section is defined as the integral of the top-width function, resulting in

$$A[x, y(x, t)] = \int_0^{y(x, t)} T[x, z]\, dz\,, \tag{6}$$

where z is height above the thalweg. The integrand, $T[x,z]$, varies only with the height, z, from the minimum point in the cross section because the location along the channel, x, and the time, t, are held constant during the integration.

4.3.1.2 First Moment of Area With Respect to the Water Surface

The hydrostatic pressure force on the narrow horizontal strip at height z in figure 8 is approximately $\rho g\{y(x,t)-z\}T[x,z]\Delta z$, where g is the acceleration of gravity. Thus, the pressure force, F_P, on the cross section below y is given by the integration of the pressure forces on many small horizontal strips as

$$F_P = \rho g \int_0^{y(x, t)} \{y(x, t) - z\}\, T[x, z]\, dz\,. \tag{7}$$

Dividing equation 7 by ρg gives the first moment of area about the water surface as

$$J[x, y(x, t)] = \int_0^{y(x, t)} \{y(x, t) - z\}\, T[x, z]\, dz\,. \tag{8}$$

Expansion of equation 8 and integration by parts yields

$$J[x, y(x, t)] = \int_0^{y(x, t)} A[x, z]\, dz\,, \tag{9}$$

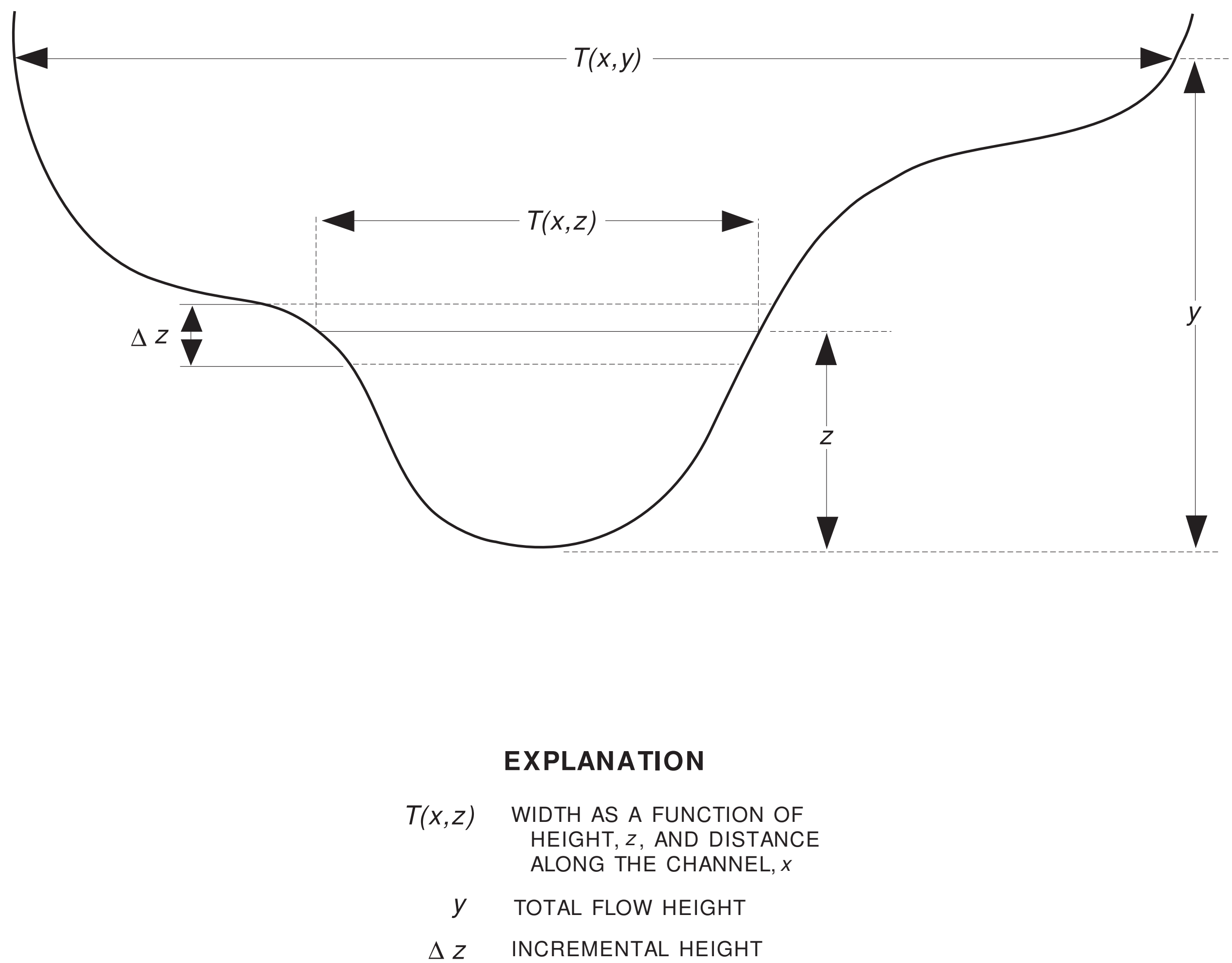


Figure 8. Definition of cross-section elevation characteristics.

as a simpler relation for the first moment of area. The qualifier that the first moment should be about the water surface is now dropped, because this is the only axis where moments are determined.

The directional aspect of the hydrostatic pressure force has been ignored in keeping with the previous discussion. If direction were to be included, the cosine of the angle between the normal to the cross-section surface and the x-axis would have to be included in equations 7–9. However, the effect of omitting the direction is reduced to negligible levels with the cosine function, as was the result for the computation of the mass flux through a cross section in the flow.

4.3.1.3 Wetted Perimeter

The wetted perimeter is the length of the boundary of the cross section that is under water for a given height of water, y. It can be defined in terms of an integral involving derivatives of the boundary shape. (The mathematics will not be discussed here because the characteristic can be simply described.) The wetted perimeter, $P[x,y(x,t)]$, is never less than the top width and is often nearly equal to the top width. However, there are cross sections for which the difference between top width and wetted perimeter is substantial. Therefore, the conveyance, which includes the wetted perimeter implicitly, is used in FEQ and FEQUTL (D. D. Franz and C. S. Melching, in press) simulations of a channel. The conveyance is described in section 4.3.2.1.

4.3.1.4 Derivatives of Area and First Moment of Area

Partial derivatives of the area and the first moment of area are needed for some derivations and for an understanding of some of the terms in the equations of motion. Among these necessary partial derivatives are the rate of change of area with distance at a fixed water-surface height and the rate of change of the first moment of area with respect to the water surface with distance for a fixed water-surface height. The notation used should make clear which variable is held constant. For example,

$$\frac{\partial}{\partial x} T\,[x, y\,(x, t)\,]\,\Big|_y$$

indicates that the height, y, is held constant and that the time variable is suppressed. This does not mean that time is held constant; on the contrary, time is ignored and the top-width function is defined by the distance along the channel, x, and the height above the minimum point, y. A shorthand form for this notation is T_x^y, where the subscript denotes the variable used in taking the derivative and the superscript denotes the variable held constant. On the other hand,

$$\frac{\partial}{\partial x} T\,[x, y\,(x, t)\,]$$

indicates that only t is held constant. The height, y, can vary so long as the time is held constant.

The derivatives of area and first moment of area with respect to the water surface with distance along the channel may be determined by application of the Leibniz rule, resulting in

$$\frac{\partial}{\partial x} A\,[x, y\,(x, t)\,] \;=\; T\,[x, y\,(x, t)\,]\,\frac{\partial y}{\partial x} + A_x^y \tag{10}$$

and

$$\frac{\partial}{\partial x} J\,[x, y\,(x, t)\,] \;=\; A\,[x, y\,(x, t)\,]\,\frac{\partial y}{\partial x} + J_x^y\,. \tag{11}$$

The terms A_x^y and J_x^y are not needed if the channel is prismatic. The last term in equation 11, J_x^y, is related to the downstream component of the pressure force on the sides of the channel, which is given by the product of ρg and the derivative of the first moment of area at constant depth with respect to distance along the channel, that is $\rho g J_x^y$.

The effects of the curvature of the cross section and the flow in the channel are ignored in these derivatives. Addition of the directional effect substantially increases the complexity of the analysis.

4.3.2 Dynamic Characteristics

The dynamic characteristics of channels relate to concepts of water in motion. These include the conveyance, the momentum-flux correction coefficient, the kinetic-energy-flux correction coefficient, and the critical flow. The calculation of each of these dynamic characteristics is described in the following sections.

4.3.2.1 Conveyance

The conveyance is the simplest of the dynamic elements, at least if the Manning friction-loss relation is applied. A compact channel is shaped such that the ratio of the flow area to the wetted perimeter (that is, the hydraulic radius) adequately describes the effect of channel shape on the friction losses. The conveyance for a compact channel is

$$K(x, y) = \frac{1.49}{n} A(x, y) R(x, y)^{2/3}, \tag{12}$$

where $R(x,y)$ is the hydraulic radius, which equals $A(x,y)/P(x,y)$; and n is Manning's roughness coefficient. If the cross section is noncompact, it must be subdivided. The subdivision of compound and composite cross sections is discussed in Franz and Melching (in press).

4.3.2.2 Flux Coefficients

The effects of nonuniform velocity distributions are corrected with momentum- and kinetic-energy-flux coefficients. In 1-D flow analysis, the average velocity is used to compute the flux of momentum and kinetic energy; however, these fluxes involve powers of the velocity at each point of the cross section (local velocities) so that an error results if the average velocity is used. The square of the average velocity does not equal the sums of the squares of the local velocities used to define the average.

The average velocity is defined so that continuity is preserved. That is, the flow rate Q for the cross section is defined by

$$Q = \int_A v\,dA, \tag{13}$$

where v is the velocity at each point in the cross section. The average velocity is then simply defined as $V = Q/A$. The momentum flux through a small area, ΔA, is $\rho v^2 \Delta A$. The sum of these fluxes for the cross section becomes

$$M_F = \rho \int_A v^2\,dA. \tag{14}$$

For ease of computation, the momentum flux computed by use of the average velocity should be the same as the momentum flux computed from the point velocities in the cross section. Thus, a coefficient, β, is introduced to correct for the errors resulting from use of the average velocity instead of the local-velocity field in the cross section. The defining equation for, β, the momentum-flux correction coefficient, is then

$$\beta V^2 A = \int_A v^2 dA \,, \tag{15}$$

where ρ is assumed to be constant and therefore is deleted from the relation. Solving for β yields

$$\beta = \frac{1}{QV}\int_A v^2 dA \,. \tag{16}$$

The kinetic-energy-flux correction coefficient, α, is defined in a similar manner. Kinetic energy replaces momentum in the concept development, and the defining equation is

$$\alpha = \frac{1}{QV^2}\int_A V^3 dA \,, \tag{17}$$

where again ρ is deleted.

For the case where the velocities are unidirectional (all downstream) but nonuniform across the section, Jaeger (1956, p. 115) found that

$$\alpha - 1 = 3(\beta - 1) + \frac{1}{QV^2}\int_A (\delta v)^3 dA \,, \tag{18}$$

where $\delta v = v - V$ and is the deviation of the point velocity from the average velocity. Equations 16 and 17 indicate that α is greater than β.

Stream analyses have provided considerable evidence that values of α and β are significantly and frequently different from 1. For example, 36 of 62 values of β computed by the U.S. Geological Survey from current-meter measurements[2] were substantially greater than 1 for natural trapezoidal-shaped channels without overbank flow, bridge piers, or other manmade obstructions. Thus, these channels were compact, yet β was greater than 1.1 in more than one-half of the channels; in fact, β was greater than 1.2 for 8 of the 62 channel measurements. Further, α was greater than 1.3 and greater than 1.5 in 30 and 13 channel measurements, respectively. The average value of α for the 62 measurements was 1.36, and the average for β was 1.12. These measurements show that the flux correction coefficients may be substantially different from 1 in compact natural channels. Consideration of extensive overbank flow could make the values of α and β much higher. Therefore, the effect of velocity distribution may have to be included in the governing equations.

Recent research by Xia and Yen (1994) indicates that the effects of flow nonuniformity and of approximations of β may not substantially affect the calculated water-surface profile. Xia and Yen (1994) compared the relative accuracy of the Saint-Venant equations ($\beta = 1$) with the results from the nearly exact momentum equations (Yen, 1973), including pressure correction coefficients (*k* and *k*') and $\beta \neq 1$. A series of numerical experiments was done for various values of *k*, *k*', and β (parameter interaction was only partially considered) for flow subject to various downstream backwater conditions. These experiments involved routing a sinusoidal stage

[2]The values of β were computed by Harry Hulsing and others in 1966, but the results were not published. A copy of the results was obtained from U.S. Geological Survey personnel in Menlo Park, Calif.

hydrograph with a peak 2.25 times the base stage, h_0, through a 54-mile long channel of rectangular, wide rectangular, or trapezoidal geometry. The maximum error in the computed depth was found to be 0.36 percent for $\beta = 1.33$ and 1.11 percent for $\beta = 2$ for a channel with a bed slope of 0.00019 and downstream backwater ranging from 0 to 2.53 times h_0. Thus, a reasonable approximation of β should not result in substantial error in the computed water-surface profile.

4.3.2.3 Critical Flow and Critical Depth

Critical flow and critical depth are important concepts in open-channel hydraulics in establishing the boundary between two broad classes of flow that must often be distinguished to understand hydraulic effects and compute estimates of the flow variables. Critical flow is adequately defined in steady flow, and unsteady flow only complicates the derivations. Thus, steady flow is used in all derivations in this section. Traditionally, critical depth is defined as the depth that minimizes the specific energy at a cross section when the flow is constant. The specific energy, $E_s(Q,y)$, is defined as the sum of the velocity head and the water-surface height of flow as

$$E_s(Q, y) = \frac{Q^2}{2gA(y)^2} + y. \tag{19}$$

(Explicit functional notation is applied in equation 19 to emphasize dependence on water-surface height and flow rate. Subsequent equations will include explicit arguments only when necessary to show the functional dependence. Otherwise, any cross-sectional characteristic in an equation is a function of the water-surface height in the cross section.) If the partial derivative of specific energy with respect to water-surface height is set to zero, the result is

$$Q_c = \sqrt{\frac{(gA^3)}{T}} = A\sqrt{\frac{gA}{T}}, \tag{20}$$

for the critical flow. Here, Q_c is the critical flow rate producing a minimum in the specific energy at a given water-surface height, y. Hereafter, for convenience, critical flow is the basis for discussion rather than the water-surface height at critical flow. Most introductory treatments of critical flow go on to develop the concept of force plus momentum, M, called specific force by Chow (1959, p. 53). Specific force is the sum of the hydrostatic pressure force on a cross section and the momentum flux for the section (treating the density of water as 1 because it is constant). Thus, specific force is defined as

$$M(Q, y) = gJ + \frac{Q^2}{A}. \tag{21}$$

If the partial derivative of specific force with respect to water-surface height is set equal to zero, the result is

$$Q_c = \sqrt{\frac{(gA^3)}{T}} = A\sqrt{\frac{gA}{T}}, \tag{22}$$

which is the same as the result obtained from minimization of specific energy.

Much is made of the equivalence of critical flow defined from specific momentum and specific energy in some introductory hydraulics texts and with good reason. If the cross sections are compact and the velocity distribution is virtually uniform, then equivalence of critical flow determined from specific energy and specific force follows. However, when the flow is sufficiently nonuniform to require values of $\beta > 1$ and $\alpha > 1$ be included in the analysis, mathematical inconsistencies can arise for steady, gradually varied, nonuniform flow. In most discussions, the effects of nonuniformity are not considered, and the inconsistencies resulting from $\alpha \neq \beta \neq 1$ are not recognized.

Generalizing the specific energy and the specific force values to include the flux-correction coefficients yields

$$E_s(Q, y) = \alpha(y)\frac{Q^2}{2gA^2} + y \tag{23}$$

and

$$M(Q, y) = gJ + \beta(y)\frac{Q^2}{A}. \tag{24}$$

As indicated in equations 23 and 24, the flux coefficients vary with the water-surface height in the cross section (Chow, 1959, p. 43). Again, if the partial derivatives with respect to water-surface height are set to zero and the equations are solved for the critical flow, the result is

$$Q_E = A\sqrt{\frac{gA}{\alpha T - \frac{A}{2}\frac{\partial\alpha}{\partial y}}} \tag{25}$$

and

$$Q_M = A\sqrt{\frac{gA}{\beta T - A\frac{\partial\beta}{\partial y}}}, \tag{26}$$

where Q_E is the critical flow defined from specific energy and Q_M is the critical flow defined from specific force. These results clearly show that the functions representing the flux-correction coefficients must follow certain restrictions for these two values to be the same at all depths.

Jaeger (1956, p. 93-119) extensively discusses the equivalence of these two definitions of critical flow. He includes coefficients for potential energy and hydrostatic-pressure force to correct for the effect of streamline curvature and he is able to show that the two values of critical flow are the same. This equivalence is based on the assumption that the appropriate correction-coefficient values have been used. In the above derivation, substantial inconsistency may result for some flows if only the flux-correction coefficients are applied. Experiments with steady uniform flow in a laboratory compound channel with flood plains indicate that minimization of specific energy or minimization of specific force yielded the same values for critical flow and water-surface height (Blalock and Sturm, 1981, 1983). In these experiments, many point measurements of velocity were made to accurately estimate the values of α and β. If the flux-correction coefficients are included in the analysis and if they are estimated by the usual means (applying subsection conveyances as a surrogate for the velocity distribution), then the difference in Q_E and Q_M can be greater than 30 percent. It should be recognized immediately that the flux-correction coefficients cannot be computed exactly because the calculated velocity distribution is only approximate. Thus, part of the large differences in Q_E and Q_M reflects the approximate values of α and β.

Direct observation of critical flow in a stream is impossible. Thus, inferences about flow values must be made from the mathematical description of the flow. Consequently, the value computed for critical flow will depend on the governing equation selected to describe the flow. A change in the choice of the governing equation or in the terms to be included in such an equation will change the computed value of critical flow. Furthermore, critical flow results in mathematical problems (singularities) in the governing equations, such as division of a quantity by zero. These singularities are commonly a direct result of ignoring certain terms in the governing equations.

The classic example is the flow of water over a brink. The 1-D governing equations indicate that the water-surface slope at the brink should be vertical and the depth should be the critical depth; however, as the flow approaches the brink, the streamlines become strongly curved and the pressure distribution deviates appreciably from hydrostatic. This deviation violates the assumption of the governing equation. Thus, the depth at the brink is not the critical depth nor is the water surface vertical. The critical depth as computed from equation 20 is some distance upstream from the brink, and the depth at the brink is appreciably less than the critical depth given by equation 20 (Henderson, 1966, p. 191).

Steady flow has been assumed in equations 19-26. Blalock and Sturm (1983), however, show that equations 25 and 26 also are obtained from unsteady-flow governing equations. These equations include the effect of velocity distribution, but deviations from hydrostatic pressure distribution are ignored; these are the typical assumptions made in current applications of steady and unsteady flow. The unsteady-flow governing equations applied by Blalock and Sturm (1983) are of the same form as those developed in section 5.

Although the equations for critical flow derived from steady and unsteady flow are the same, the velocity distribution in a channel at a given stage may not be the same for steady flow as for unsteady flow. The difference could be substantial in a compound channel. If the stage is rising, water will be moving from the main channel into the flood plains; whereas if the stage is falling, the water will move in the opposite direction. This flow interchange with the flood plains must have an effect on the velocity distribution and, therefore, on α and β. This interchange also indicates that the simple assumption that the flux coefficients depend only on the water-surface height is only an approximation. In a compound channel, a more rigorous analysis might indicate that a rate of change of water-surface height also must be included. No studies that examine this problem are known. Therefore, the simple assumption is retained.

Critical flow is a function of the governing equation selected to represent the flow. The critical-flow value used must be consistent with the governing equation to avoid improper solutions. The physical meaning and interpretation of the computed critical flow must be established by observation and practice, as has been done for compact channel shapes in steady flow. In these cases, critical flow clearly defines a boundary in the physical system that has proven useful in describing various flow phenomena. The physical basis for critical flow computed for noncompact channels is less clear. Blalock and Sturm (1981) and Petryk and Grant (1978) have made some general observations, but no extensive body of experience is available to validate the physical interpretation of these estimates.

Because the momentum-conservation principle is used in FEQ to represent the 1-D flows, Q_M is the best estimate of critical flow. This estimate will be as consistent with the governing equations as possible. The effects of unsteadiness, however, cannot be included. Furthermore, representations of special features may be inconsistent (as described previously). The flow equations for some special features are based on energy-conservation principles. In most cases, however, the cross sections considered will be compact and simple, such as the barrel of a culvert, so that taking $\alpha = \beta = 1$ will be reasonable. For flexibility, the option of applying either equation 25 or 26 is available in the utility program, FEQUTL (Franz and Melching, in press). For most flow conditions, equation 26 should be selected.

Another practical problem with the critical-flow estimates with flux-correction coefficients is that the critical flow may become undefined. In equations 25 and 26, the estimated rate of change of the flux-correction coefficient with respect to water-surface height may be such that the argument for the square root becomes negative. If the rate of change in the flux-correction coefficient is positive and large enough, the numerator in these equations can become zero or negative. The critical flow then becomes a complex number that is physically undefined. This can be a result of large inconsistencies between the estimated value of the flux-correction coefficient and the estimated rate of change with respect to water-surface height.

5. FULL, DYNAMIC EQUATIONS OF MOTION FOR ONE-DIMENSIONAL, UNSTEADY FLOW IN OPEN CHANNELS

The equations presented in section 5 include the major physical factors affecting shallow-water flows and are thus called full equations. Various forms of the equations are shown. All are mathematical restatements of the same physical principles, each having advantages and disadvantages as detailed in the subsections that follow. The integral form of the equations is basic to all forms, so it is presented first. The integral form is used as a basis for defining numerical approximations to shallow-water flows applied in the FEQ model and described in the section 6, "Approximation of the Full Equations of Motion in a Branch."

Detailed derivations of the unsteady-flow equations are given in Cunge and others (1980, p. 7-24), Strelkoff (1969), and Yen (1973). Abbott and Basco (1989, p. 1-43) present a detailed mathematical and philosophical discussion of these equations. The major assumptions applied in the derivation of the full, dynamic equations of motion are listed in the section 1.4.

5.1 Integral Form of the Equations

The integral form of the full equations is a macroscopic statement of the principles of conservation of mass and momentum for what is called a control volume. A control volume is a conceptual device for clearly describing the various fluxes and forces in open-channel flow. A conceptual control volume for open-channel flow is shown in figure 9. The upstream face of the control volume at station x_L is assumed to be orthogonal to the flow direction at that point in the channel, as outlined previously. The downstream face of the control volume at station x_R also is assumed to be orthogonal to the flow direction. The sides and bottom of the control volume are formed by the sides and bottom of the channel. The top of the control volume is formed by the water surface. The length of the control volume, $\Delta x_{CV} = x_R - x_L$, does not have to be small and is measured along the distance axis defined previously.

The integral form of the equations can be explained simply in a 1-D approximation; therefore, the equations are presented here without derivation, followed by discussion of what each major term or set of terms represents in the conservation principle. To keep the details as simple as possible, the weight coefficients used to correct certain integrands in the integral form for the effects of curvilinear flow are omitted at first. These weights are added later when the full form of the equations is developed for curvilinear channel alignments.

5.1.1 Conservation of Mass

The conservation of mass principle for a control volume is

$$\int_{X_L}^{X_R} [A(x, t_U) - A(x, t_D)]\, dx = \int_{t_D}^{t_U} [Q(x_L, t) + I(t) - Q(x_R, t)]\, dt\,. \qquad (27)$$

The time interval of integration is defined by two points in time, t_U and t_D, such that $t_U > t_D$. (The meaning of the subscripts on these time points is explained in more detail in subsequent sections.) The term $I(t)$ denotes the inflow of water that enters the control volume over or through the sides of the channel. Density is constant and is not shown in equation 27 because each term would have a constant multiplier that cancels from the relation. Thus, the conservation of mass is equivalent to conservation of water volume in FEQ simulation. Equation 27 is a precise mathematical statement of a simple concept. The left-hand side of equation 27 is the change in volume of water contained in the control volume during the time interval (t_D, t_U). The integral of flow area with respect to distance at a fixed time defines the volume of water in the control volume at that time. The right-hand side of equation 27 is the net volume of inflow to the control volume (inflow minus outflow) during the time interval. Water enters from upstream, $Q(x_L, t)$, leaves downstream, $Q(x_R, t)$, and enters over or through the sides of the channel, $I(t)$. Thus, equation 27 indicates that the change in volume of the water in the control volume during any time interval is equal to the difference between the volume of inflow and the volume of outflow during that time interval. The

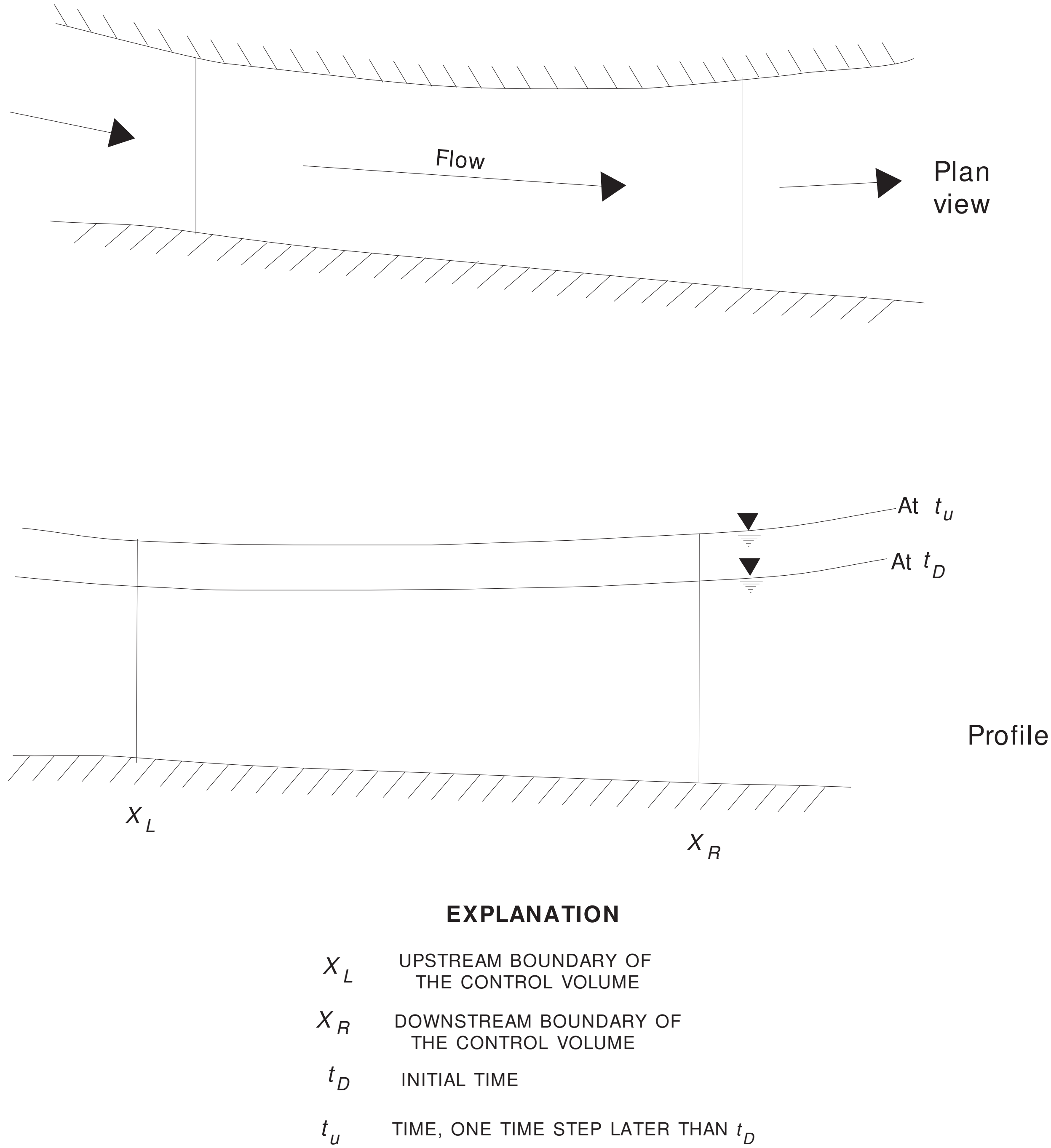


Figure 9. Control volume in a stream for unsteady-flow equations.

term $I(t)$ represents what is commonly called the lateral inflow, which comes from several sources: runoff from the land surface, discharges from sewers, outflows of water from pumping, and others. If the lateral flow is out of the channel, then $I(t)$ is negative.

5.1.2 Conservation of Momentum

The principle of conservation of water volume includes only the flows and changes in volumes. The conservation of momentum includes the momentum flux and various forces on the boundaries of the control volume.

Because forces are vectors, the momentum equation is vectorial; therefore, the terms are written relative to the direction assigned to downstream flow in the channel. As discussed previously, preliminary evidence from laboratory studies shows that the vector nature of momentum does not substantially affect 1-D flows; therefore, flow is treated as if it were all in the same direction.

To satisfy the conservation principle, the change in momentum of the water in the control volume over any time period must be equal to the net downstream impulse during the time period plus the net flux of momentum during the time period (that is, influx minus efflux). Impulse is a time integral of a force. In most basic fluid mechanics texts (for example, Streeter and Wylie 1985, p. 117), the conservation of momentum for a control volume in one dimension, x, is expressed as

$$\sum F_x = \frac{\partial}{\partial t}\int_{CV} \rho v_x d\forall + \int_{CS} \rho v \mathbf{V} \bullet d\mathbf{A}\,, \tag{28}$$

where

F_x are the forces acting on the control volume, CV;
v_x is the velocity in the x-direction;
$d\forall$ is the volume differential;
$\boldsymbol{V}$ is the velocity vector; and
dA is the differential area taken as a vector normal to the control surface, CS, of the control volume.

The first term on the right-hand side of equation 28 is the rate of change in momentum stored in the control volume, and the second term is the momentum flux through the control volume.

For open-channel flow, the forces included are pressure forces on the upstream and downstream faces, downstream component of the pressure force on the sides of the channel, gravity force, channel friction, and wind-shear stress on the water surface. By moving the momentum stored in the control volume to the left-hand side and the sum of forces to the right-hand side and expanding the sum of forces, the conservation of momentum for the control volume becomes

$$\begin{aligned}
\rho\int_{x_L}^{x_R} [Q(x, t_U) - Q(x, t_D)]\,dx = {} & \rho\int_{t_D}^{t_U} [\beta Q V(x_L, t) - \beta Q V(x_R, t)]\,dt \\
& +\rho g\int_{t_D}^{t_U}\left[J(x_L, t) + \int_{x_L}^{x_R} J_x^y\, dx - J(x_R, t)\right]dt \\
& +\rho g\int_{t_D}^{t_U}\int_{x_L}^{x_R} S_0 A\,dt\,dx \\
& -\int_{t_D}^{t_U}\int_{x_L}^{x_R} \bar{\tau} P\,dt\,dx \\
& +\int_{t_D}^{t_U}\int_{x_L}^{x_R} C_{D^{(w)}}\rho_a U^2 T\cos\psi\,dt\,dx,
\end{aligned} \tag{29}$$

where

S_0 is the bottom slope of the channel,
$\bar{\tau}$ is the average shear stress on the water from the channel boundary,
$C_{D^{(w)}}\rho_a U^2$ is the wind-induced shear stress on the water surface in the direction of the wind-velocity vector,
ρ_a is the density of air, U is the wind velocity,
$C_{D^{(w)}}$ is the dimensionless drag coefficient for wind shear stress, and
ψ is the angle between the downstream flow direction in the channel and the velocity of the wind.

The integral on the left-hand side of equation 29 is a rewritten form of the change in momentum stored in the control volume. The downstream component of the pressure force on the sides of the channel also is given by integration with respect to distance of the force per unit length discussed previously. The gravity force is represented by the integration of the product of fluid density and area (the mass per unit length of channel) and the bottom slope of the channel. The friction force per unit length is given by the product of the shear stress and the wetted perimeter. The friction force per unit length is then integrated over the length of the control volume to yield the friction force at any time. The wind-shear stress per unit length of the channel is given by the product of the wind-shear stress in the direction of the channel and the top width of the channel. Integration with respect to distance yields the total wind-shear force at any time. The influx and efflux of momentum are represented by the first integral to the right of the equal sign in equation 29. The time integration of the distance integrals for the forces on the control volume yield the impulses from these sources. The arguments affecting the elements integrated with respect to both time and distance in equation 29 are not shown to simplify the notation. The arguments of time and of distance are both implicit in these instances.

Although complicated, the integral equation (eq. 29) is a precise mathematical statement of the conservation of momentum principle; specifically, that the change in momentum of the water in the control volume is given by the net influx of momentum and the net downstream impulse from all forces acting on the water in the control volume. Given the selection of forces and other assumptions, equation 29 is an exact statement that applies for any length of control volume.

The friction-force term simplifies if it is assumed that the relation between slope and boundary friction from steady-uniform flow,

$$S_0 = \frac{\bar{\tau} P}{\rho g A}, \tag{30}$$

can be generalized to unsteady flow by replacing the bottom slope, S_0, with the friction slope, S_f. Applying this definition of the friction slope and dividing equation 29 by ρ results in

$$\begin{aligned}
\int_{x_L}^{x_R} [Q(x, t_U) - Q(x, t_D)]\, dx = {} & \int_{t_D}^{t_U} [\beta Q V(x_L, t) - \beta Q V(x_R, t)]\, dt \\
& + g \int_{t_D}^{t_U} \left[J(x_L, t) + \int_{x_L}^{x_R} J_x^{\,y} dx - J(x_R, t) \right] dt \\
& + g \int_{t_D}^{t_U} \int_{x_L}^{x_R} A\,(S_0 - S_f)\, dt dx \\
& + \int_{t_D}^{t_U} \int_{x_L}^{x_R} C_{D^{(w)}} \frac{\rho_a}{\rho} U^2 T \cos\psi\, dt dx
\end{aligned} \tag{31}$$

as the integral form for the conservation of momentum equation for open-channel flow. This and related equations are called motion equations. In equation 31, the momentum contribution from the lateral inflow is ignored. Reliable information is rarely available regarding the velocities and depths of lateral inflows, and lateral inflows are often nearly orthogonal to the flow in the channel. Thus, omitting the effects of lateral inflow in equation 31 should not result in substantial error.

The friction slope must be estimated from the cross-sectional characteristics and the flow. In terms of the total channel conveyance, K, the friction slope is computed from

$$S_f = \frac{Q|Q|}{K^2}\,. \tag{32}$$

Use of the product $Q|Q|$ instead of Q^2 as normally seen in steady-flow analysis gives the result that the friction is a retarding force on the water in the control volume for either direction of flow. Therefore, the possibility of reversing flows is simulated in FEQ.

5.2 Differential Form of the Equations

The integral form of the equations (eqs. 27 and 29 or 31) is a basis for all other forms of the governing equations for unsteady open-channel flow. These other forms involve differential equations derived by manipulating the integral form or an approximation of it by taking limits as the time and distance intervals approach zero. The wind-stress terms are omitted in these developments to simplify the equations because these terms are not necessary for the general development of the differential equations of motion. Furthermore, the momentum-flux correction coefficients are assumed to be 1.

5.2.1 The Conservation Form

Approximating the integrals in equations 27 and 31 by finite differences and taking limits yields

$$\frac{\partial A}{\partial t} + \frac{\partial Q}{\partial x} = q\,, \tag{33}$$

and

$$\frac{\partial Q}{\partial t} + g\frac{\partial J}{\partial x} + \frac{\partial}{\partial x}(\mathrm{QV}) = gA\,(S_0 - S_f) + gJ_x^y\,, \tag{34}$$

where q is the lateral inflow per unit length along the channel, defined as a function of distance and time such that

$$I(t) = \int_{x_L}^{x_R} q\,(x, t)\,dx\,. \tag{35}$$

Equations 33 and 34 are in conservation form because the basic variables are explicitly expressed.

The area in equation 33 should be considered the volume per unit length of channel. Thus, the time derivative of area gives the rate of change of volume per unit length. The derivative of the flow rate in the channel with respect to distance should be considered the channel outflow per unit length of channel. All of the quantities in equations 33 and 34 are algebraic expressions and can be positive or negative; therefore, a negative outflow is an inflow. Equation 33 is a statement of the conservation of mass principle (with ρ constant) on a per-unit-length basis.

Similarly, equation 34 is a statement of the principle of conservation of momentum per unit length. In the time derivative of flow, the flow rate is the momentum per unit length. The terms involving derivatives of J on the right-hand side of the equal sign represent the net downstream pressure force per unit length. The derivative of QV, when moved to the right of the equal sign, represents the net efflux of momentum per unit length. Finally, the term

$gA(S_0\text{-}S_f)$ is the net downstream force per unit length from gravity and friction forces. Thus, equation 34 (with all terms but the time derivative of flow moved to the right-hand side) defines the time rate of change of momentum per unit length as the sum of the net downstream forces and the net efflux of momentum.

5.2.2 The Saint-Venant Form

Expanding the derivatives in the conservation form and simplifying the equations yields

$$A\frac{\partial V}{\partial x} + VT\frac{\partial y}{\partial x} + T\frac{\partial y}{\partial t}VA_x^y = q \tag{36}$$

and

$$\frac{\partial V}{\partial t} + V\frac{\partial V}{\partial x} + g\frac{\partial y}{\partial x} = g(S_o - S_f) - \frac{Vq}{A}, \tag{37}$$

which is often called the Saint-Venant form of the equations of motion (Chow, 1959, p. 528). This and other similar forms of the equations are the most common forms in the hydraulic literature. The relation between the principles of conservation of mass and momentum and the terms in the equations has been obscured in equations 36 and 37.

5.2.3 The Characteristic Form

The final form of the equations to be presented here is obtained by transforming the Saint-Venant form so that derivatives taken in the proper directions, called characteristic directions, can be written as ordinary derivatives and not partial derivatives. The result of this transformation is

$$\left[\frac{\partial V}{\partial t} + \frac{dx}{dt}\frac{\partial V}{\partial x}\right] \pm \frac{g}{c}\left[\frac{\partial y}{\partial t} + \frac{dx}{dt}\frac{\partial y}{\partial x}\right] = g(S_o - S_f) - \frac{Vq}{A} \mp \frac{c}{A}\left(VA_x^y - q\right), \tag{38}$$

and

$$\frac{dx}{dt} = V \pm c, \tag{39}$$

where c is wave celerity, the speed of an infinitesimal disturbance in the channel relative to the water. If the flux correction coefficients are taken to be unity, then the celerity is equal to $Q_c|A$, where Q_c is given by equation 22. If the characteristic form is derived from a mass-energy formulation, the celerity is given by $Q_E|A$, where Q_E is given by equation 25; if it is derived from a mass-momentum formulation, the celerity is given by $Q_M|A$, where Q_M is given by equation 26. This relation between steady flow and unsteady flow is expected because the steady-flow equations are special cases of the unsteady-flow equations.

The bracketed terms in equation 38 represent the ordinary derivatives of velocity and water-surface height when these derivatives are taken in the directions given by equation 39. Then equation 38 becomes

$$\frac{dV}{dt} \pm \frac{g}{c}\frac{dy}{dt} = g(S_o - S_f) - \frac{Vq}{A} \mp \frac{c}{A}\left(VA_x^y - q\right). \tag{40}$$

Equation 40 is best understood as representing the rate of change of velocity and water-surface height that an observer moving along the stream channel in the characteristic direction and with the velocity given by equation 39 would measure. The characteristic form of the equations of motion is not applied in FEQ simulation; however, the characteristic form of the equations is presented here because understanding the movement of waves along the characteristic direction provides valuable insight on several aspects of unsteady-flow analysis, such as boundary conditions, initial conditions, and solution methods, as discussed in the next section.

5.3 Nature of Shallow-Water Waves

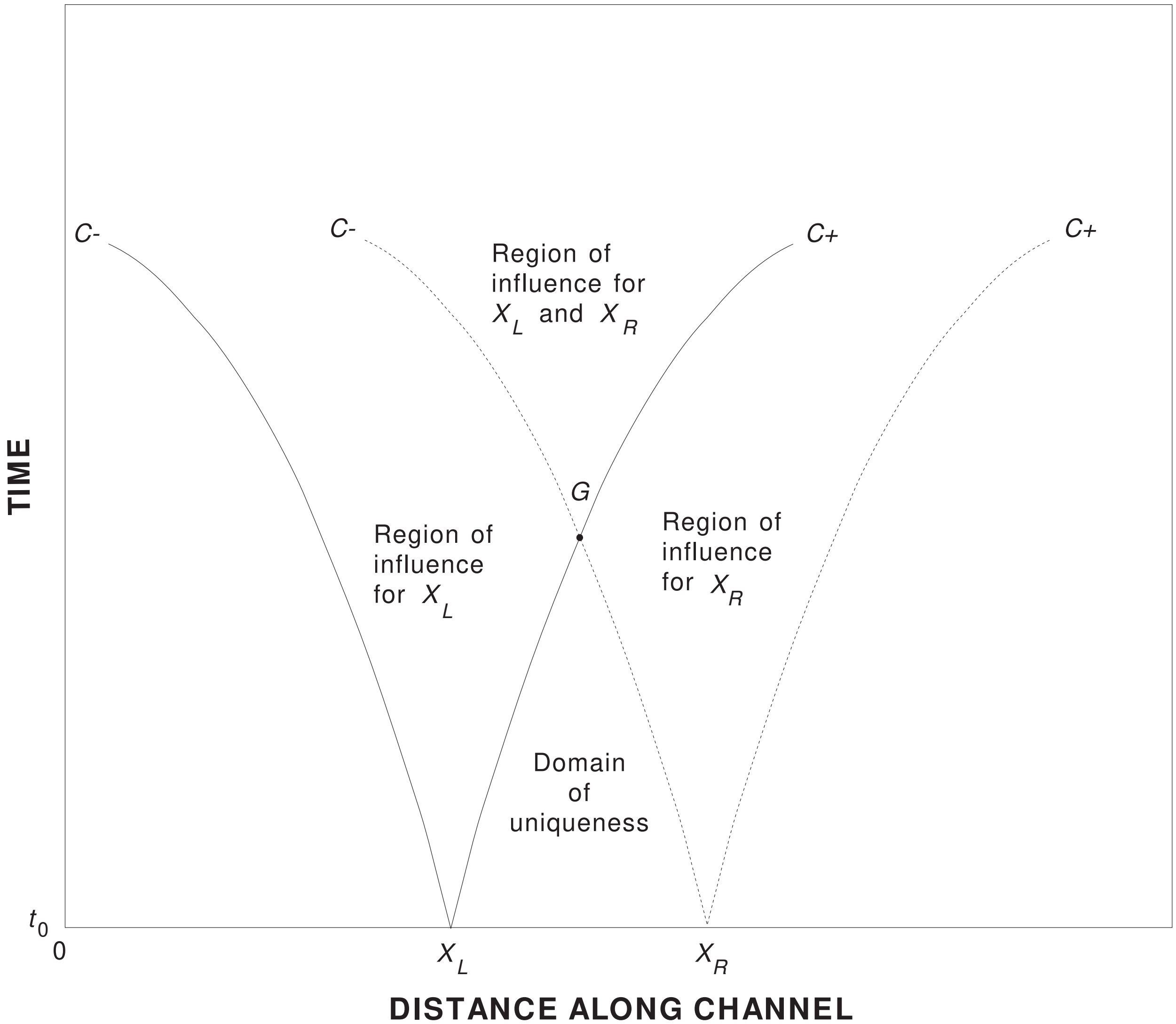
The various forms of governing equations progress from physical to mathematical forms. The integral form and the conservation form relate closely to the fluxes and forces acting on the flow. In contrast, the relation to the fluxes and forces is missing in the Saint-Venant and characteristic forms. The characteristic form has lost almost all reference to the forces and fluxes included in the equation for the conservation of momentum; however, the characteristic form provides insight into shallow-water wave motion, which is not evident in the other forms. This insight is vital to understanding the requirements (boundary and initial conditions) that must be met as approximate solutions to these equations are sought. Thus, no single form of the governing equations is adequate for understanding unsteady 1-D open-channel flow.

The insight to be gained from the characteristic form is best visualized by tracking small but identifiable disturbances in a stream channel. Consider a long rectangular stream channel with no special features, a branch in the FEQ network schematization. Also, imagine that the flow is steady and subcritical but nonuniform. To introduce a small shallow-water-wave disturbance, a short segment of the channel bottom is made of some flexible material that can be given a sudden sharp but small upward displacement. This displacement disturbs the whole column of water above the location of the flexible strip and is analogous to the mechanism thought to initiate tsunamis in the Pacific Ocean. Because the flow is subcritical, a shallow-water wave will move upstream and downstream. To track each of these small waves, the location of the waves along the channel is measured periodically. The path or trajectory of the wave or waves can then be depicted by use of a coordinate system in which the distance along the channel, x, is shown on the horizontal axis and the time, t, is shown on the vertical axis, as in figure 10. This coordinate system defines the x-t plane.

Suppose that the disturbance was introduced at station X_L at time $t = t_0$. Small shallow-water waves will travel upstream and downstream from this station. The upstream wave will have a velocity $V - c$ and the downstream wave a velocity of $V + c$. The trajectory of the upstream wave is denoted as $C-$ and of the downstream wave is denoted as $C+$ in figure 10. The region of the x-t plane between these two trajectories is the region of influence of the disturbance at point X_L at time $t = t_0$. Outside this region, the disturbance has no effect on the flow. Another disturbance has been introduced at a station X_R, some distance downstream from X_L. The $C+$ and $C-$ trajectories for this disturbance also are shown on figure 10. The region of the x-t plane between the $C+$ trajectory of the disturbance at X_L and the $C-$ trajectory of the disturbance at X_R is called the domain of uniqueness because the flows in this region cannot be affected by disturbances upstream from X_L or downstream from X_R originating at any time $t \geq t_0$. The distance interval from X_L to X_R is called the interval of dependence for point G because the flow at point G is dependent on knowledge of the flow on this interval at time $t = t_0$.

These features of shallow-water-wave motion are important in designing methods to compute approximations to the motion. In these methods, a known condition in the stream channel at a time $t = t_0$ is applied, and the conditions in the channel at some later time are computed with the equations of motion. The known conditions at which the computations start are called initial conditions. For practical reasons, the initial flow condition is almost always assumed to be a steady flow. To adequately estimate conditions at point G from information at $t = t_0$, information must be available about the flow on the interval of dependence for point G. If that information is not available, meaningful estimation cannot be made because unknown conditions have the potential to affect the values at point G. Thus, to estimate the conditions at any point on the x-t plane, information must be available about the interval of dependence for that point.

This requirement for information on the interval of dependence for each point in the channel has implications at the boundaries of the channel. Every channel is of finite length; at some point the analysis starts and at another it ends, so boundaries must be defined. Possible conditions at the boundaries of a channel are shown in figure 11. If the flow is subcritical, the interval of dependence for the upstreammost point on the channel is



EXPLANATION

$C+$	DOWNSTREAM WAVE TRAJECTORY
$C-$	UPSTREAM WAVE TRAJECTORY
X_L, X_R	POINTS ALONG THE CHANNEL THAT ARE DISTURBED AT TIME t_0
G	FIRST POINT ON THE x-t PLANE AFFECTED BY DISTURBANCES AT X_L AND X_R AT TIME t_0; LOWER BOUNDARY IN TIME OF THE REGION OF INFLUENCE FOR X_L AND X_R

Figure 10. Schematic diagram of regions of influence and domain of uniqueness for shallow-water waves in an open channel.

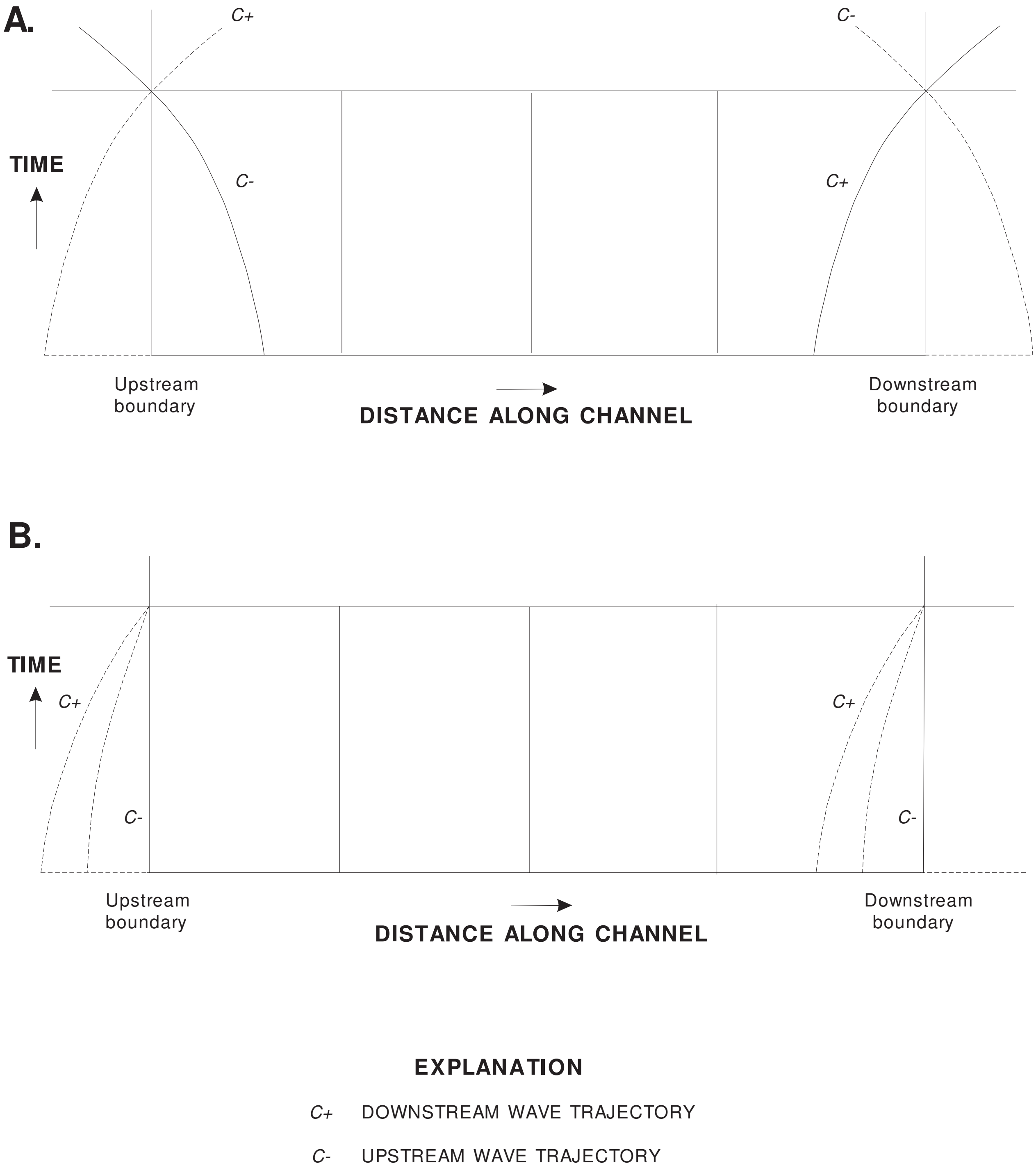


Figure 11. Required boundary conditions for unsteady flow in open channels for (A) subcritical flow and (B) supercritical flow at the boundaries.

somewhat upstream from the boundary point. Thus, estimation of flow conditions at this boundary point requires information about the flow conditions upstream of the boundary. The *C*+ trajectory from upstream points affects the flow at that point in the *x*-*t* plane, yet the analysis for this channel stops at the boundary. Therefore, a single condition must be specified at the boundary point. This condition, called a boundary condition, may be one of three types: flow as a function of time, water-surface elevation as a function of time, or a relation between flow and

water-surface
elevation. This condition supplies the information that is lacking along the upstream boundary for the channel. The same applies at a downstream boundary if the flow is subcritical. Again, part of the interval of dependence falls outside the channel length being analyzed, and a downstream-boundary condition must be supplied.

If the flow is supercritical at a boundary, the required number of conditions changes. At an upstream boundary, the flow and the elevation of water must both be supplied because, as shown in figure 11, the interval of dependence of points on the upstream boundary in the *x-t* plane are outside the length of channel analyzed. At the downstream end, no boundary conditions are required because the interval of dependence falls within the length of channel analyzed. Thus, the number of boundary conditions required depends on whether the flow is subcritical or supercritical at the boundary.

5.4 Integral Form for Curvilinear Alignment

To develop the integral form of the equations with weight coefficients to correct for channel curvilinearity, equations 27 and 31 are restated to explicitly show the dependence on the water-surface height, y, as

$$\int_{x_L}^{x_R} \{A[x, y(x, t_U)] - A[x, y(x, t_D)]\}\, dx = \int_{t_D}^{t_U} [Q(x_L, t) + I(t) - Q(x_R, t)]\, dt. \tag{41}$$

Dropping the wind-stress term, the integral form of the momentum-conservation equation for the same control volume is

$$\begin{aligned}
&\int_{x_L}^{x_R} [Q(x, t_U) - Q(x, t_D)]\, dx \\
&= \int_{t_D}^{t_U} \left[\frac{\beta[x_L, y(x_L, t)]\, Q(x_L, t)^2}{A[x_L, y(x_L, t)]} - \frac{\beta[x_R, y(x_R, t)]\, Q(x_R, t)^2}{A[x_R, y(x_R, t)]} \right] dt \\
&+ g\int_{t_D}^{t_U} \left[J[x_L, y(x_L, t)] + \int_{x_L}^{x_R} J_x^y\, [x, y(x, t)]\, dx - J[x_R, y(x_R, t)] \right] dt \\
&+ g\int_{t_D}^{t_U} \int_{x_L}^{x_R} A[x, y(x, t)]\, \{S_0(x) - S_f[x, y(x, t)]\}\, dt dx,
\end{aligned} \tag{42}$$

All the variables in equation 42 have been defined previously; but in this case, the dependence of these variables on location, x, time, t, and (or) water-surface height, y, is explicitly shown.

Equations 41 and 42 are in conservation form. If the water surface is discontinuous, these equations are still valid. However, the pressure-force terms involving the first moment of area and its rate of change at fixed water-surface height are cumbersome. Therefore, one can deviate slightly from the conservation form by requiring that the water-surface profile be continuous. In practice, this change has had negligible effects on results in simulating dam-break flood waves. By expressing the difference in the first moment of area at the ends of the control volume in differential form and by using the Leibniz rule to obtain an alternative form of the rate of change of the first moment of area with respect to distance, the momentum equation for the case of a continuous water-surface profile can be expressed without reference to the first moment of area as

$$\begin{aligned}
&\int_{x_L}^{x_R} [Q(x, t_U) - Q(x, t_D)]\, dx \\
&= \int_{t_D}^{t_U} \left[\frac{\beta[x_L, y(x_L, t)]\, Q(x_L, t)^2}{A[x_L, y(x_L, t)]} - \frac{\beta[x_R, y(x_R, t)]\, Q(x_R, t)^2}{A[x_R, y(x_R, t)]} \right] dt \\
&- \int_{t_D}^{t_U} \int_{x_L}^{x_R} gA[x, y(x, t)] \left[\frac{\partial y}{\partial x} - S_0(x) + S_f[x, y(x, t)] \right] dt dx.
\end{aligned} \tag{43}$$

Taking into account that $\frac{\partial y}{\partial x} - S_0(x) = \frac{\partial z_w}{\partial x}$, where z_w = water-surface elevation, simplifies equation 43 to

$$\begin{aligned}
&\int_{x_L}^{x_R} [Q(x, t_U) - Q(x, t_D)]\, dx \\
&= \int_{t_D}^{t_U} \left[\frac{\beta[x_L, y(x_L, t)]\, Q(x_L, t)^2}{A[x_L, y(x_L, t)]} - \frac{\beta[x_R, y(x_R, t)]\, Q(x_R, t)^2}{A[x_R, y(x_R, t)]} \right] dt \\
&- \int_{t_D}^{t_U} \int_{x_L}^{x_R} gA[x, y(x, t)] \left[\frac{\partial z_w}{\partial x} + S_f[x, y(x, t)] \right] dt dx.
\end{aligned} \tag{44}$$

Any term in equations 41 and 44 requiring integration with respect to distance may require a weight coefficient when these equations are applied to a curvilinear stream channel. The weight coefficients for computing volumes and momentum content have been developed previously. Weight coefficients have been developed for equations similar to equations 41 and 44 by DeLong (1989) and Froehlich (1991). DeLong presented a momentum equation that did not include a weight coefficient on any term except the momentum-content term. Froehlich questioned this result and derived a correction factor for the friction term given by

$$M_f = \frac{1}{A}\left(\frac{K}{Q}\right)^2 \int_{S_B}^{S_E} h(s)\, \sigma(s) \left(\frac{q_w(s)}{k(s)}\right)^2 ds, \tag{45}$$

where

s is the offset distance across the cross section as measured from a reference point on the channel bank,

$h(s)$ is local depth in a cross section at cross-section offset s,

$q_w(s)$ is the flow per unit width in a cross section at offset s,

$k(s)$ is the conveyance per unit width in a cross section at offset s,

$\sigma(s)$ is the rate of change of distance along the flow line at offset s to the rate of change of distance along the channel axis (sinuosity at offset s), and

s_B and s_E are the offset at the beginning and end of the wetted top width for the cross section.

DeLong (1991) pointed out that the total conveyance applied by Froehlich (1991) was not adjusted for sinuosity and was computed assuming that the friction slope was the same for each flow line across the section. If the total conveyance is adjusted for sinuosity and the local conveyance adjusted for sinuosity is substituted into equation 45, then $M_f = 1$ and, therefore, does not appear in the equations.

The omission of a weight factor on the pressure and gravity force terms in equations 41 and 44 may be surprising; however, as equation 44 indicates, the downstream force from these two terms is determined by the difference in water-surface elevation between two sections (that is, in the term $\partial z_w / \partial x$). One of the fundamental assumptions for curvilinear and linear flow indicates that this difference is constant for each flow line between the two cross sections considered. Thus, the variation in the lengths of the flow-line distances makes no difference in solving these integrals. The assumptions applied to derive estimates of $q(s)$, $k(s)$, $\sigma(s)$, and the adjusted total conveyance are discussed in detail in Franz and Melching (in press).

Use of the results for the distance integrals in equation 44 and subsequent simplification yields

$$\int_{x_L}^{x_R} [M_Q Q(x, t_U) - M_Q Q(x, t_D)]\,dx$$
$$= \int_{t_D}^{t_U} \left[\frac{\beta [x_L, y(x_L, t)]\, Q(x_L, t)^2}{A[x_L, y(x_L, t)]} - \frac{\beta [x_R, y(x_R, t)]\, Q(x_R, t)^2}{A[x_R, y(x_R, t)]} \right] dt$$
$$- \int_{t_D}^{t_U} \int_{x_L}^{x_R} gA[x, y(x, t)] \left[\frac{\partial z_w}{\partial x} + S_f[x, y(x, t)] \right] dt\,dx, \qquad (46)$$

as the equation expressing momentum conservation in 1-D unsteady flow in curvilinear channels.

The mass conservation equation becomes

$$\int_{x_L}^{x_R} \{ M_A A[x, y(x, t_U)] - M_A A[x, y(x, t_D)] \}\,dx$$
$$= \int_{t_D}^{t_U} [Q(x_L, t) + I(t) - Q(X_R, t)]\,dt. \qquad (47)$$

In equations 46 and 47, the arguments for the weight coefficients, M_A and M_Q, are omitted for simplicity. Definitions of M_A and M_Q are given in section 4.2.

5.5 Special Terms in the Equations of Motion

One-dimensional unsteady flow in an open channel can be simulated with the governing equations as given here; however, certain conditions in open channels require additional terms in the equations of motion. These conditions include the effects caused by diverging or converging channels and the effects of isolated obstructions, such as bridge piers or a pipe, in the flow. The latter effects can be isolated and treated separately from the governing equations, but it is often convenient to add to the governing equations the effect of simple submerged obstructions that do not exert appreciable control on the flow.

5.5.1 Drag for a Submerged Body

Because the momentum equation involves forces, any terms added to the motion equation must involve forces. The drag for a submerged body is

$$F_D = \frac{1}{2}\rho C_D A_p V_a^2, \tag{48}$$

where

- C_D is dimensionless drag coefficient depending on the nature of the submerged object and the flow;
- A_p is the area of the submerged object projected on a cross section orthogonal to the approach velocity; and
- V_a is the velocity of approach upstream from the submerged object.

In practical computations, V_a is taken as the average cross-sectional velocity in the computational element. The drag coefficient given earlier for the wind-stress term on the water surface included the factor $\frac{1}{2}$ for consistency with the source for wind-stress drag coefficients. Here, the traditional definition of the drag coefficient is retained. The projected area of the submerged object is a function of the water-surface elevation near the object. Because the density, ρ, is constant, it does not appear in the final forms of the equations of motion in equations 31 and 46. Therefore, the density is also dropped from the drag equation for consistency with the final forms of the equation of motion. Thus, the adjusted drag becomes

$$F_D' = \frac{1}{2} C_D A_p V_a |V_a|, \tag{49}$$

where V_a^2 has been replaced with $V_a|V_a|$ so that the drag will always be opposite the flow.

The drag is considered to apply to a point within the control volume. The location of the drag-producing object, at x_D, is needed to define the water-surface height for estimating the projected area and the drag coefficient from the approaching flow. For simplicity, a single location is assumed to be adequate for both directions of flow. The approach-velocity source may vary with changes in direction of flow in the control volume; this is denoted by placing a subscript a on the location of the velocity. Thus, the full description at the adjusted drag is

$$F'_D = \frac{1}{2} C_D [x_D, y(x_D, t)] A_p [x_D, y(x_D, t)] V[x_a, y(x_a, t)] \left| V[x_a, y(x_a, t)] \right|. \tag{50}$$

5.5.2 Transformation of Energy-Head Losses to Drags for Control Structures

Representing the effect of a complex structure—a large trash rack, for example—may be necessary or desirable. Typically, the effect of such structures is given as a head loss (that is, in terms of mechanical energy) and is estimated by the product of a loss coefficient and the approach-velocity head. A head-loss term is incompatible with equation 46 because all terms in that equation refer to momentum content, momentum flux, or forces on the water in the control volume. The problem of incompatibility between momentum equations and energy losses arises because most past work in open-channel flow has used the principle of conservation of mechanical energy for control structures, in part because simpler equations result for certain steady flows. (This is not true for unsteady flow, as stated earlier.) The basic data used to derive the mechanical-energy-loss coefficients also could have been developed for the conservation of momentum principle. Thus, the mechanical-energy losses that are available must be transformed to yield reasonable estimates of drags that are suitable for use in the equation of motion. For convenience, the term here should be similar to that for an explicit drag so that both can be represented in the solution process in the same manner.

In applications of the mechanical-energy balance, head losses are commonly applied at points along a 1-D flow path even though these losses may take place over an appreciable length of the flow path. This is done for several reasons. First, the length over which the losses take place is only approximately known. Second, the manner in which the losses are distributed over this length is unknown. Third, the head losses are often small relative to the total head and other losses. Fourth, the length scales for the flow fields of interest are many times longer than the structures or channel features causing the head loss. Therefore, the distortion in concentrating a loss at a point is negligible. The transformation of the head loss to an equivalent drag needs to be located at a point in the control volume. In the derivation of this transformation in the equations of motion, the integral of the product of a slope and a cross-sectional area with respect to distance (when multiplied by g) gives a force term that applies to both the friction slope and the bottom slope. Thus, a slope, S_p, is introduced to represent the loss of head, Δh_p, in the control volume where

$$\int_{x_L}^{x_R} S_p dx = \Delta h_p . \tag{51}$$

If equation 51 is satisfied, then the drag, F_{DE}, that represents the effect of the head loss computed for the structure is given by

$$F_{DE} = g\int_{x_L}^{x_R} A S_p dx . \tag{52}$$

F_{DE} is similar to the forces (gravity and boundary shear) computed with the other slopes in the equation of motion. By isolating the head-loss slope effect to the immediate vicinity of the obstruction at x_p and expressing the head loss as a fraction of the approach velocity head, the drag, F_{DE}, exerted on the flow by the obstruction at x_p may be expressed as

$$F_{DE} = \frac{1}{2} k_p [x_a, y(x_a, t)] A [x_p, y(x_p, t)] V [x_a, y(x_a, t)] \left| V [x_a, y(x_a, t)] \right| , \tag{53}$$

where k_p is the head-loss coefficient for the obstruction at x_p. $V|V|$ is used instead of V^2 in equation 53 because the drag is always opposite to the motion of the water.

A computational element in a channel may contain more than one simple structure, either constructed or natural, for which an estimated drag must be included in the momentum equation. For one simple structure (structure of type 1), it may be possible to estimate the drag from measured drag coefficients and the projected area of the structure, F_D, computed with equation 50. For another simple structure (structure of type 2), it may be necessary to estimate the drag from an equivalent energy slope determined from a head-loss relation, F_{DE}, computed with equation 53. The total drag for the computational element is given by a single integral term over time (because equations 50 and 53 are of the same form) as

$$-\frac{1}{2}\int_{t_D}^{t_U}\left\{\sum_{i=1}^{m_D} C_{D_i}A_{p_i}\left[x_{D_i}, y\left(x_{D_i}, t\right)\right] + \sum_{j=1}^{m_p} k_{p_j}A\left[x_{p_j}, y\left(x_{p_j}, t\right)\right]\right\}$$

$$\{V[x_a, y(x_a, t)]\,|V[x_a, y(x_a, t)]|\}\,dt,$$

where m_D is the number of structures of type 1 present and m_p is the number of structures of type *2* present. The total drag given in the above expression is added to equation 46 to obtain the momentum equation for the computational element accounting for the localized drags resulting from simple structures.

If this transformation seems inappropriate, consider steady flow in a rectangular, prismatic, horizontal channel. Furthermore, assume that the friction losses are small relative to the local losses resulting from a large trash rack in the channel. The losses for trash racks are most often given in terms of a fraction of the approach-velocity head. If section 1 upstream from the rack and section 2 downstream from the rack are placed such that the 1-D assumptions are valid, then the principle of conservation of mechanical energy yields

$$y_1 + \frac{Q^2}{2g\,(y_1T)^2} = y_2 + \frac{Q^2}{2g\,(y_2T)^2} + k_p\frac{(Q^2)}{2g\,(y_1T)^2}, \tag{54}$$

as the equation relating the flow and water-surface height at sections 1 and 2, y_1 and y_2, respectively. Application of the principle of conservation of momentum yields

$$\frac{g}{2}y_1^2T + \frac{Q^2}{y_1T} = \frac{g}{2}y_2^2T + \frac{Q^2}{y_2T} + F_D, \tag{55}$$

as another equation relating the flow and water-surface height at sections 1 and 2. For a given flow and approach water-surface height, equation 54 gives the water-surface height at section 2. Substitution of these water-surface heights into equation 55 gives a value for the drag. Equation 53 also gives an estimate of the drag.

How well does equation 53 represent the drag computed for this simple but physically realistic example? To answer this question, the Froude number for the approaching flow,

$$\mathbf{F}_a = \left(V_a/(gy_a)^{\frac{1}{2}}\right); \tag{56}$$

the ratio of downstream water-surface height to upstream water-surface height, $r = y_2 / y_1$; and the drag rescaled by the upstream hydrostatic-pressure force, $\tilde{F} = 2F_D / \left(g y_1^2 T \right)$ are used to convert the equations to a dimensionless form,

$$\frac{\mathbf{F}_a^2}{2r^2} + r = 1 + \frac{1}{2}\mathbf{F}_a^2 (1 - k_p) , \tag{57}$$

for the energy relation, and

$$1 + 2\mathbf{F}_a^2 = r^2 + \frac{2\mathbf{F}_a^2}{r} + \tilde{F}_{D1} \tag{58}$$

for the momentum relation. An estimate of drag is computed by solving equation 56 for the water-surface height ratio given the Froude number and the loss coefficient and substituting the ratio into equation 57 to estimate the dimensionless drag, $\tilde{F}_{D1}$. If equation 53 is applied in equation 55, the dimensionless form becomes

$$\tilde{F}_{D2} = \frac{1}{2}(1 + r) k_p \mathbf{F}_a^2 , \tag{59}$$

provided that the proper area is the arithmetic average of the area upstream and downstream from the trash rack or other structure. Equation 58 defines another dimensionless drag. How does the value of $\tilde{F}_{D2}$ compare with $\tilde{F}_{D1}$, the best estimate of the drag? The absolute value of the relative error in dimensionless drag force, $\tilde{F}_{D2}$, taking $\tilde{F}_{D1}$ as the correct value is shown in table 2.

The largest error in table 2 is about 4 percent when the approach Froude number and loss coefficient are such that the Froude number at the downstream section is between 0.95 and 1. When the downstream flow is clearly subcritical, the error is less than 1 percent for a wide range of flow conditions. Therefore, application of equation 53 provides a close estimate of the drag in any reasonable flow, and the area to use is the arithmetic average of the areas in the cross sections bounding the location of the loss.

Table 2. Error in drag estimate for an example trash rack made in simulation with the Full EQuations model to approximate drags for simple control structures

[nc, not computed]

Approach Froude number	Error, in percent, for values of loss coefficient				
	0.05	0.10	0.30	0.50	0.70
0.100	0.00	0.00	0.00	0.00	0.00
.200	.00	.00	.00	.00	.00
.300	.00	.00	.00	.00	.00
.400	.00	.00	.00	.01	.03
.500	.00	.00	.03	.09	.20
.587	nc	nc	nc	nc	4.02
.600	.00	.01	.17	.83	nc
.625	nc	nc	nc	2.80	nc
.680	nc	nc	.00	nc	nc
.700	.02	.08	nc	nc	nc
.780	nc	.93	.00	.01	.03
.800	.16	nc	nc	nc	nc
.830	.57	nc	nc	nc	nc

5.5.3 Sudden Cross-Sectional Expansions or Contractions

The effect of converging and diverging channels on the flow is complex. Again, these effects are typically expressed in terms of a head loss. The losses for diverging channels are larger than those for converging channels. When the velocities are decreasing in the direction of the flow, turbulence is increased, and the water adjacent to the sides of the channel may separate from the sides and form large eddies. The distribution of velocity becomes highly nonuniform and is no longer predominantly affected by the shape of the boundary of the channel. When the velocities are increasing in the direction of flow, the velocity becomes more nearly uniform and turbulence is reduced. Estimation of these losses is difficult, and empirical relations are used for loss estimation. See the discussion of the EXPCON command in the FEQUTL documentation report (Franz and Melching, in press) for a more extensive discussion of these difficulties.

The empirical relation applied to estimate head losses in expansions or contractions takes a fraction of the absolute value of the velocity-head difference over the length of the divergence or convergence. This relation was developed for designed and constructed transitions in canals where transitions are relatively short and infrequent. The same relation has been used in natural channels with irregular variations in size and shape. These irregular natural variations reduce the accuracy of the empirical relation for head-loss estimation, but no practical alternatives are available. Therefore, the velocity-head difference over the length of the control volume is taken in the derivation here.

The head losses resulting from flow separation from the sides of the channel, sometimes called eddy losses, are in excess of head losses resulting from boundary friction. The values of Manning's *n* given in various tabulations are presumed to exclude these eddy losses and, therefore, represent only the frictional effects in mostly straight prismatic channel sections. This means that estimates of Manning's *n* in natural channels derived from measurements in the channel must be treated carefully. The value of Manning's *n* could differ in response to how the eddy losses were estimated. Available steady-flow water-surface-profile computer programs, supported by various Federal agencies, differ in treating these losses. Thus, if each of the computer programs was used to estimate Manning's *n* from a set of careful measurements in a natural channel, different results would be produced merely because of differences in the treatment of minor losses.

Further issues must be addressed before the head-loss equations can be presented. The standard head-loss equation for expansions and contractions was developed for channels that are compact in shape; that is, channels where $\alpha \approx 1$. Therefore, potential effects of large values of α for natural channels are not discussed. Inclusion of α values will require additional values in the cross-section tables when the governing equations for a branch are approximated. In addition, the variation in the energy-flux-correction coefficients, α, over the distance of a typical computational element is generally small. Furthermore, uncertainties in the estimation of the eddy losses are large. Therefore, it seems reasonable to exclude α in the equations applied to estimate the eddy losses to simplify the computation. The effect of α can be partially reflected by increasing the fraction of the velocity-head difference used to estimate the loss. Alternatively, the roughness values in the cross-section description can be changed to reflect the eddy losses (for an example, see Chow, 1959, p. 267). If the change in cross section is large over a short distance, then the transition should be isolated and treated as a special feature so that a more comprehensive evaluation of the losses can be computed.

On the basis of the assumptions discussed above, the head loss for accelerating or decelerating flow is

$$\Delta h_{ad} = sign(Q_R + Q_L) f(V_R, V_L) \left| \frac{V_R^2}{2g} - \frac{V_L^2}{2g} \right|, \tag{60}$$

where

$$sign\,(Q_R + Q_L) = \begin{cases} 1, \text{ if } (Q_R + Q_L) \geq 0; \\ -1, \text{ otherwise} \end{cases} \tag{61}$$

$$f(V_R, V_L) = \begin{cases} k_a, \text{ if } V_R - V_L > 0; \\ k_d, \text{ otherwise} \end{cases} \tag{62}$$

k_a is coefficient for accelerating flow, k_d is the loss coefficient for decelerating flow, and the subscripts L and R refer to the left and right ends of the control volume. Again, the loss must change sign so that the flow will always be retarded. Because this loss is assumed to apply over the control volume, another assumption is required to determine the direction of flow for the control volume. The direction of flow is determined in equation 59 on the basis of the sum of flows at each end of the control volume.

In cases where flow reversal is possible (for example, in tide-affected reaches) the sum of flows for a flow element may be zero because flow is either entering both ends or leaving both ends. When the flow is entering the control volume from both ends, a loss for deceleration is assigned, whereas when the flow is leaving from both ends of the control volume, a loss for acceleration is assigned. This convention seems reasonable because the turbulence present when the flow is entering from both ends would be larger than when it is leaving from both ends.

The slope of the eddy loss distributed over the length of the control volume, S_{ad}, can be defined as

$$S_{ad} = \frac{\Delta h_{ad}}{x_R - x_L}. \tag{63}$$

This slope is analogous to the friction slope and must be added in a similar position and with the same sign.

5.6 Extended Motion Equation

The terms for the minor losses, together with the wind-stress terms added to equation 46, result in the extended-motion equation for curvilinear channels as

$$\begin{aligned} &\int_{x_L}^{x_R} [M_Q Q(x, t_U) - M_Q Q(x, t_D)]\, dx \\ &= \int_{t_D}^{t_U} \Bigg\{ \frac{\beta Q(x_L, t)^2}{A[x_L, y(x_L, t)]} - \frac{\beta Q(x_R, t)^2}{A[x_R, y(x_R, t)]} \\ &-g\int_{x_L}^{x_R} A[x, y(x, t)] \left[S_f[x, y(x, t)] + S_{ad} + \frac{\partial z_w}{\partial x} \right] dx \\ &+\int_{x_L}^{x_R} C_{D^{(w)}} \frac{\rho_a}{\rho} U^2 M_T T[x, y(x, t)] \cos\phi\, dx \\ &-\frac{1}{2}\left\{ \sum_{i=1}^{m_D} C_{D_i} A_{P_i}\left[x_{D_i}, y\left(x_{D_i}, t\right)\right] + \sum_{j=1}^{m_P} k_{P_j} A\left[x_{p_j}, y\left(x_{p_j}, t\right)\right] \right\} \\ &\{ V[x_a, y(x_a, t)] \left| V[x_a, y(x_a, t)] \right| \} \Bigg\}\, dt. \end{aligned} \tag{64}$$

Some arguments for coefficients have been deleted to make the equation more compact. The value M_T is a weight coefficient on the top width to give the correct water-surface area per unit length. It is related to volume per unit length just as the top width is to the area; that is,

$$M_T = \frac{1}{T}\frac{\partial}{\partial y}M_A A = M_A + \frac{A}{T}\frac{\partial M_A}{\partial y}. \tag{65}$$

6. APPROXIMATION OF THE FULL EQUATIONS OF MOTION IN A BRANCH

The integral form of the full equations of motion representing flow in a branch, derived previously, must be converted to an approximate algebraic form for numerical solution. Approximate solutions must be obtained by applying numerical methods because exact solutions of the equations are limited to a few special cases. The following sections describe the numerical methods available for solution of the approximated integral equations of motion, and the four options available in FEQ for approximating the equations of motion. General rules are given for approximate integration by weighted algebraic computation of the flow and water-surface elevation derivatives with respect to time and space. The approximations of the equations of motion are developed for the most general of the four approximation options. Finally, the selection of the appropriate weights for the time and distance integrals are discussed with respect to the selected approximation option in FEQ.

6.1 Methods of Mathematical Approximation

Computation of solutions for the equations of motion is necessarily limited to evaluating them at a finite number of points along the branch. The approaches to computing approximate solutions to these equations can be divided into two broad classes: those that fix the location of the points along the channel in advance and those that adjust the locations as needed in the solution. The latter class includes the broad range of the method of characteristics that solve the characteristic form of the equations by explicitly tracing the trajectories defined by equations 38 and 39 in whole or in part on the x-t plane. In the method of characteristics, the locations and times at which flows and elevations are computed are irregular and vary as the flow conditions vary. This method has some advantages in accuracy, but it becomes complex and impractical for use with prototype stream systems. Therefore, methods based on the characteristic form of the equations will not be considered further here because they are not used in FEQ.

The class of methods that uses a fixed set of locations (nodes) along the stream channel also is broad. This class divides into two subclasses: explicit methods and implicit methods. These methods offer practical advantages in that the locations of solution values in space and time are fixed. Thus, analysis and presentation of the results is simplified. In these methods, flow and elevation are assumed to be known at all locations at some initial time t_0, and the equations are solved for the values at some time $t_1 > t_0$. Thus, the solutions for the time period of interest are developed stepwise in these methods.

In explicit methods, the solution at time t_1 at each node is computed in sequence with information at t_0 within one or two distance intervals of the node. A pattern of points on the x-t plane used in some simple explicit methods is shown in figure 12. The values are known at points x_j, x_{j+1}, and x_{j+2} at time t_i, and the method is applied to compute values at point G_1 or point G_2. The dashed characteristic trajectories show that the interval of dependence for point G_1 is contained within the interval defined by x_j and x_{j+2}. Therefore, the flow conditions at point G_1 can be computed. However, the interval of dependence for point G_2 is larger than the interval defined by x_j and x_{j+2}. Thus, it is impossible to compute the flow conditions at G_2 given the information on the interval from x_j to x_{j+2} at time t_i. The time step in the latter case is too large. In general, explicit methods have limitations on time steps. If these limitations are exceeded, the methods and (or) the computations become unstable, and the resulting flows and elevations will develop large nonphysical oscillations and eventually the oscillations become so large that the computations fail because of negative depth, square root of a negative number, numerical overflow, and (or) other causes. The method is conditionally stable, however, if the time step meets certain constraints. The condition that generally applies is that the distance travelled by an infinitesimal wave in one time step must never exceed the distance between computational nodes. This is the Courant condition, given by

$$\Delta t \le \frac{\Delta x}{V \pm c}, \tag{65}$$

where Δt is the computational time step.

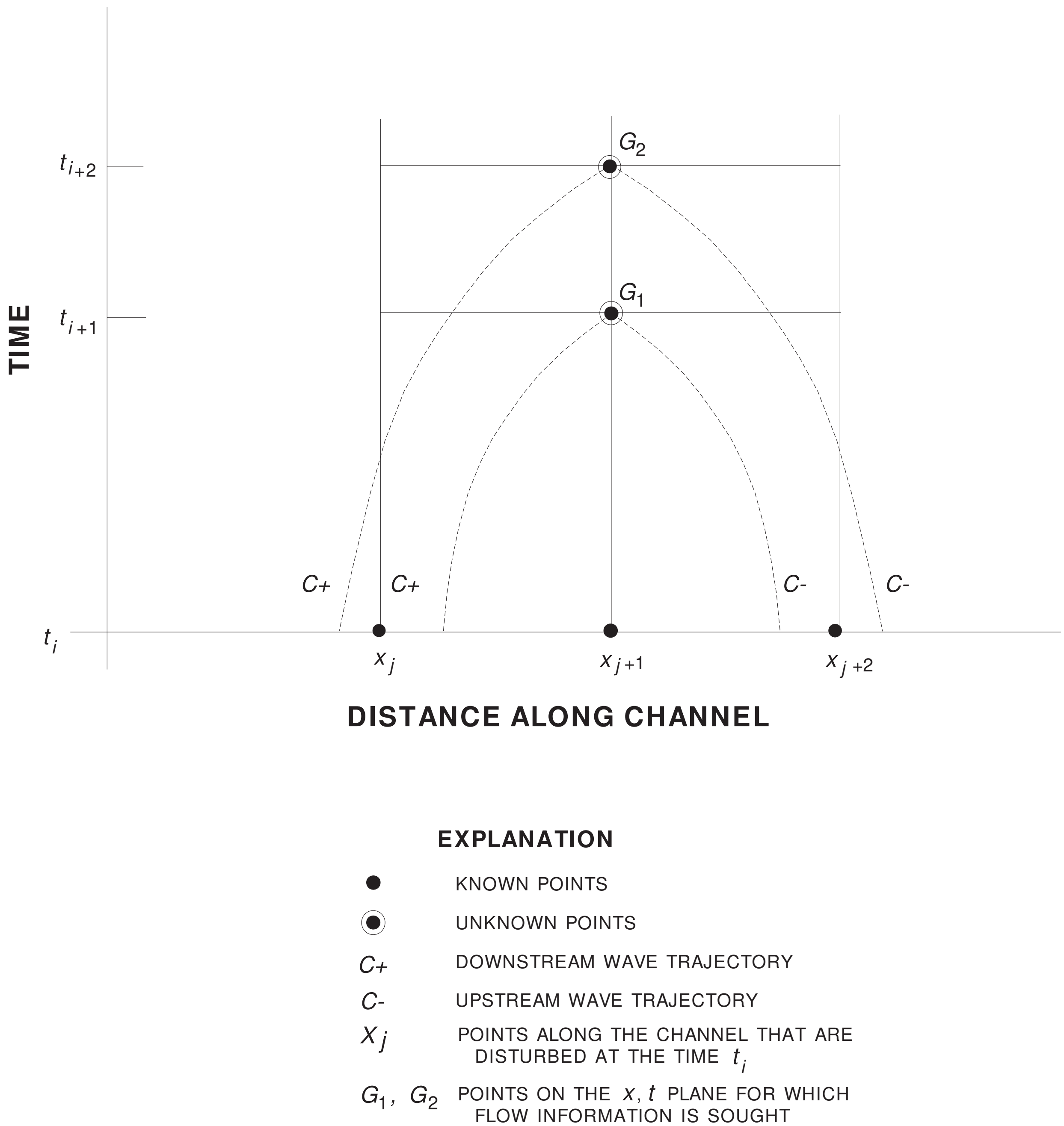


Figure 12. Explicit finite-difference method on the space-time plane.

This constraint proves to be restrictive in that the time step is often limited to a few seconds, making extensive unsteady-flow computations with explicit methods impractical if long time periods (greater than a few hours) are simulated. Consequently, an explicit method is not used in FEQ.

Implicit methods solve for all of the unknowns at time t_{i+1} simultaneously. They are thus much more complex than explicit methods, which solve each nonboundary point independently of any other nonboundary point. The characteristic trajectories defined for an implicit method are shown in figure 13. The interval of dependence is larger than the interval of known conditions at time t_i, but the boundary conditions supply the needed information. Thus, all the unknown points fall within the domain of uniqueness established by the characteristic trajectories beginning from the boundary points. Therefore, no restriction on the computational time step results from the

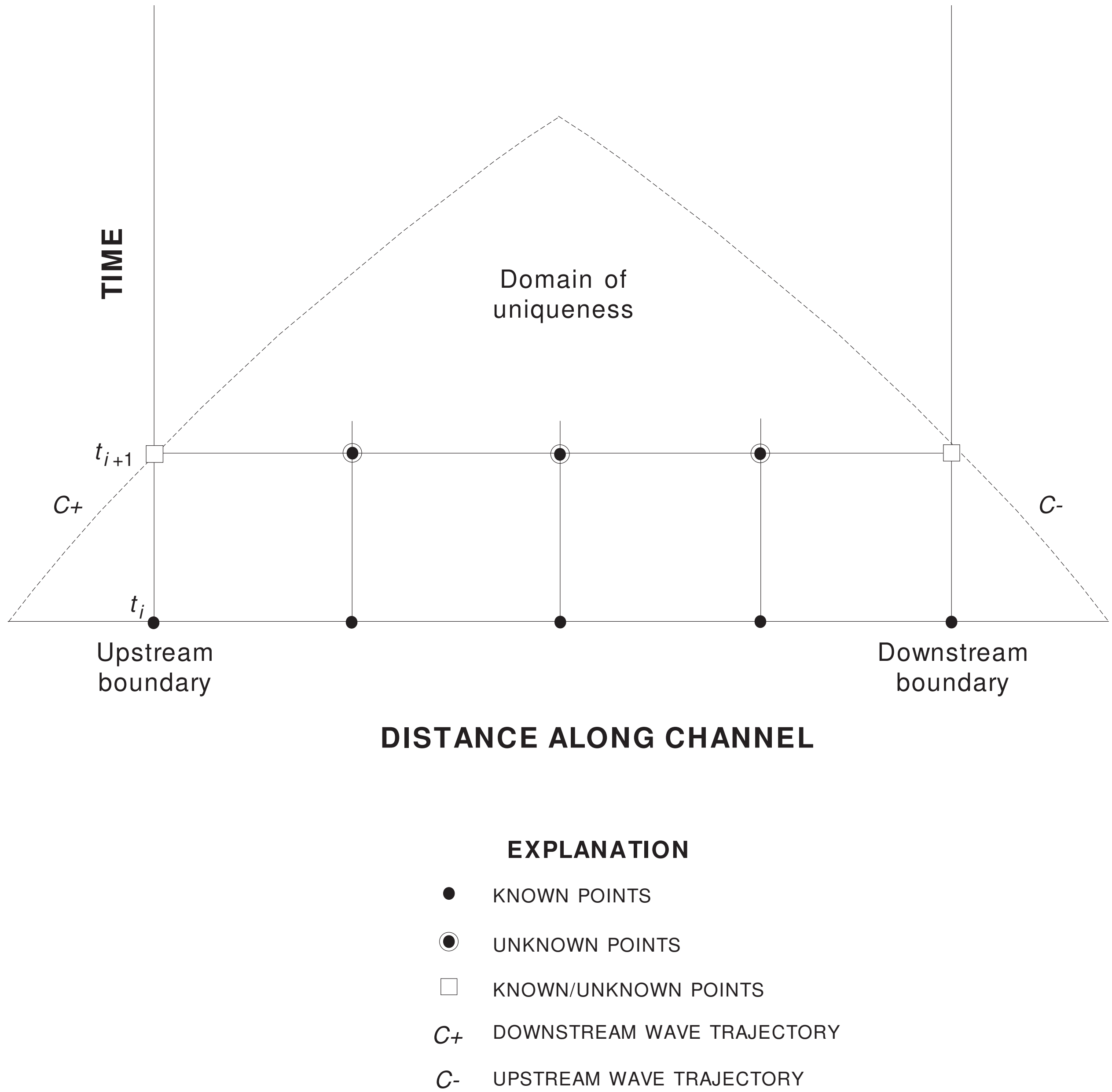


Figure 13. Implicit finite-difference method on the space-time plane.

nature of the shallow-water waves as in an explicit method. Implicit methods are not subject to the Courant condition for stability, so they are sometimes called unconditionally stable.

The stability of implicit methods allows use of large time steps in the solution. The time step may still be limited in terms of accuracy of results, but the time step can be adjusted to simulate varying flow conditions and not be restricted by the Courant condition. Implicit methods make it possible to simulate long time periods economically, with acceptable accuracy. Therefore, an implicit method called the Preissmann (1961) four-point scheme or method is applied in FEQ. This scheme has been used extensively with variations from the original form. Thus, the scheme as implemented in FEQ used only some of the concepts of the Preissmann method and might better be described as a weighted four-point scheme.

6.2 Equations of Motion Options

Equations 47 and 63 are the most general form for a branch in FEQ; however, four options for the governing equations for a branch are available. In the simplest option, denoted by STDX, the weight coefficients for curvilinearity are assumed to always be equal to 1. In the second option, denoted by STDW, a special user-controlled variable weight for certain distance integrals is added to the equations applied in STDX. This variable weight is important for simulating shallow flows. In the third and fourth options, STDCX and STDCW, the weights to account for channel curvilinearity are added to the previous two options. These options can be selected on a branch basis. The discussion below will be for the most general option, STDCW, because the other options correspond to special values of selected coefficients. These equations are solved approximately by converting them to algebraic equations using approximate integration over a computational element.

6.3 Rules for Approximate Integration

Four points in the *x-t* plane are used in the weighted four-point scheme to define the rectangle on the *x-t* plane over which equations 47 and 63 are approximately integrated to produce the system of algebraic equations that must be solved for a branch. These four points are shown in figure 14. The subscripts denoting these points are selected to assist in coding the computer program. The subscript L denotes the station on the left end of the control volume, and the subscript R denotes the station on the right end. The subscript U denotes the time point that is up, and the subscript D denotes the time point that is down relative to the center of the rectangular box on the *x-t* plane used to define the scheme. These subscripts are applied in the computer-program code by composing variable names for the various elements and values needed. For example, the Fortran name "ALU" specifies the area of flow at the left end of the control volume and at the upper time. The last two letters of the variable name are reserved to refer to the point on the *x-t* plane. This convention proves to be much simpler than use of the cumbersome indices for subscripts that may lead to many errors. The equations are written for a typical control volume corresponding to the computational element on the branch between adjacent cross sections.

The approximate rule of integration used in the weighted four-point scheme is the weighted-trapezoidal rule wherein

$$\int_a^b f(x)\,dx \approx (b-a)\ [\,(1-W)f(a) + Wf(b)\,]\ , \tag{66}$$

where a and b are the boundaries of the function region integrated and W is the weight on the function at the upper limit of the integral. The weight must satisfy $0 \le W \le 1$. According to traditional error analysis, the most accurate approximation results when $W = 0.5$; however, the most accurate value for W is not always the best value. Other considerations enter into the choice of weight when approximating the integrals in the governing equations for unsteady, open-channel flow. For integrals with respect to time the weight is W_T. The integrals with respect to distance will have two weights, W_X and W_A. The use of these weights will be discussed later. Equation 66 is recast so as to be more convenient in computation to

$$\int_a^b f(x)\,dx \approx (b-a)\ [f(a) + W\{f(b) - f(a)\}]\ , \tag{67}$$

thereby reducing the number of multiplications by one.

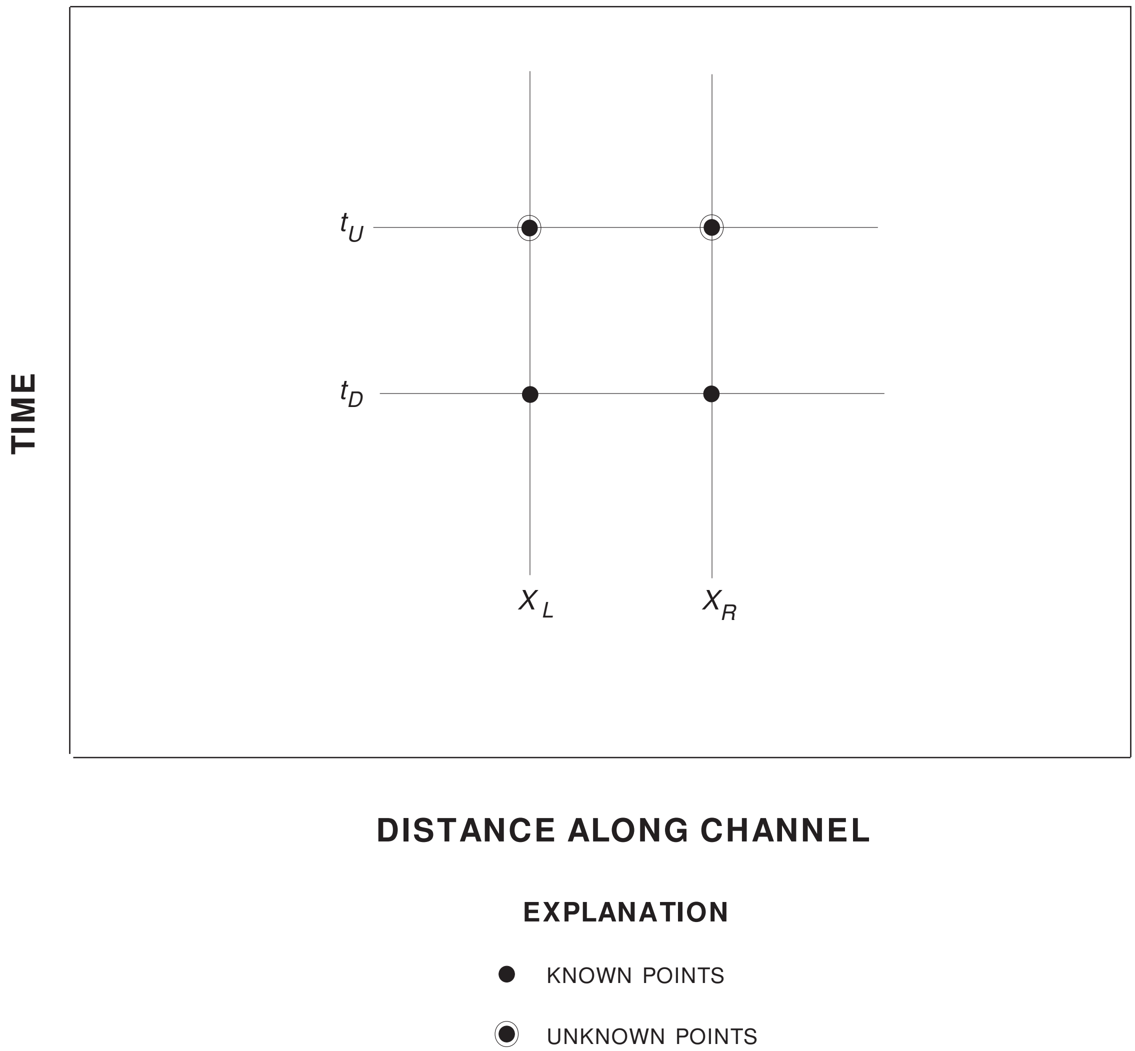


Figure 14. Points on the space-time plane for the weighted four-point solution method in the Full EQuations model.

6.4 Conservation of Mass

Let Δx be the length of a typical computational element and Δt be the length of a typical time increment. If the approximate integration rule on equation 47 is applied, then the conservation of mass equation becomes

$$\Delta x\left[\left(1-W_{A_U}\right)M_{A_{LU}}A_{LU}+W_{A_U}M_{A_{RU}}A_{RU}\right]-\Delta x\left[\left(1-W_{A_D}\right)M_{A_{LD}}A_{LD}+W_{A_D}M_{A_{RD}}A_{RD}\right]$$
$$-\Delta t\,[Q_{LD}+W_T(Q_{LU}-Q_{LD})]+\Delta t\,[Q_{RD}+W_T(Q_{RU}-Q_{RD})]-\Delta t\,I_M=0, \qquad (68)$$

where the subscript M denotes a middle or mean value and all terms were moved to the left-hand side of the equation before making the approximations. The average area at time t_U is determined by the weight W_{A_U}, and the average area at time t_D is determined by a different weight W_{A_D} because the weight applied to distance integrals of area, W_A, can vary with time. The algebraic form for the conservation of mass equation on a typical rectangle on the x-t plane is given in equation 68. The lengths of the computational elements may differ along the branch, and the time increment or step may differ with time. These variations between elements and between times are important neither to understanding the approximations nor to writing the equations. Thus, those variations are not described.

The lateral inflow term, I_M, requires special treatment for storm sewers; in this case, the lateral inflow truly refers to the surface runoff entering the sewer through catch basins and curb inlets along the path of the sewer. Surface runoff reaching these inlets is usually estimated with a hydrologic model. The calculations done with the hydrologic model are independent of the hydraulic routing in the storm sewer. The need for special treatment arises when surface runoff reaches the entrance of a sewer that is flowing under pressurized conditions. In the prototype system, water entry to the sewer is restricted, and water forms temporary ponds in streets, basements, and other depressions or flows to the stream by some overland flow path. The ponding may further add to the pressurization of the sewer downstream from the inlet. To consider tributary area connected to a storm sewer, an approximation to the storage of surface runoff that exceeds the capacity of the pressurized sewer is provided in FEQ.

In the prototype system, water enters the storm sewer at discrete points, usually at curb inlet structures. It is possible to simulate each inlet structure and specify its detailed hydraulic behavior, but this level of detail is rarely required, and providing it would prove expensive. Some simpler representation is needed to simulate the major factors affecting the entry of water into the storm sewer. These factors are the height of the ground surface above the storm-sewer invert, denoted by y_G, and an effective inlet area, denoted by A_{si}. It is assumed that the water is stored at the height y_G above the invert and that the ponding capacity is unlimited in FEQ simulation. Excess water is stored until the conditions in the storm sewer at that point allow the water to enter.

A current maximum rate of inflow for each computational element (control volume) on a branch is computed in FEQ simulation. Let y_M be the mean value for the water-surface height in the control volume for a time step defined as

$$y_M = \frac{1}{2}[W_T(y_{LU} + y_{RU}) + (1 - W_T)(y_{LD} + y_{RD})] . \tag{69}$$

For a storm sewer, a hypothetical thin slot is added to the top of the closed conduit to maintain a free surface, and y_M is measured in this slot to account for pressurization.

The head difference between the water-surface elevation in the assumed ponding area and in the control volume is

$$\Delta h = y_G - y_M . \tag{70}$$

The maximum current rate of inflow, Q_{MAX}, then is

$$Q_{MAX} = sign(\Delta h) A_{si} \sqrt{2g|\Delta h|} , \tag{71}$$

where sign (Δh) is +1 if $\Delta h > 0$ and −1 otherwise.

Derivatives of equation 71 are used in the solution process in FEQ with respect to the water-surface height in the control volume. This derivative becomes large as Δh becomes small and in the limit becomes infinite. These large derivatives can result in failure of the solution process. Therefore, equation 71 is linearized whenever $|\Delta h| < \Delta h^*$, where Δh^* is a small value of the head difference equal to 0.5 ft. In this case, the maximum inflow is

$$Q_{MAX} = \frac{(\Delta h)}{\Delta h^*} A_{si} \sqrt{2g\Delta h^*} . \tag{72}$$

If $I_M > Q_{MAX}$, then a rate of lateral inflow of Q_{MAX} enters the sewer and the excess is added to the ponding volume for the computational element. If $I_M < Q_{MAX}$, then the rate of lateral inflow, I_M, is augmented from the ponded water, if any is available, to bring the rate of inflow to Q_{MAX}. If ponded water is not available, then the rate of inflow is just I_M. The maximum rate of inflow from equations 71 or 72 can be negative. This means that water is leaving the storm sewer at this computational element and is being stored.

The approximation to the process of sewer surcharging is simple and requires a minimum of additional information about the storm sewer. The ponding volume in each computational element can be only a rough approximation to the process, and the ponding volume for the entire branch is probably a better index of the surcharging of the storm sewer. A more detailed representation of storm-sewer surcharging requires that the alternative flow paths be defined and that the storage capacity of the ponding areas also be defined. The surcharging process can be simulated in detail, but the data requirements are demanding.

6.5 Conservation of Momentum

For equation 63, the conservation of momentum equation, the derivation will proceed in steps to simplify the notation. Consider an approximation to the distance integral representing boundary shear (with S_f) and gravity and pressure forces (with $\partial z_w / \partial x$) in the first double integral in equation 63,

$$\int_{x_L}^{x_R} A \left[S_f + \frac{(\partial z_w)}{\partial x} \right] dx \approx A_M \left(\Delta x S_{f_M} + \Delta Z_W \right), \tag{73}$$

where the subscript M denotes a middle or mean value and the equivalent slope for accelerating and decelerating flow, S_{ad}, has been omitted. The averages given in equation 73 are all with respect to distance at a fixed time.

To further simplify the equations, let P_{GF} denote the approximation to the integral of the pressure, gravity, and friction terms in equation 73; then equation 73 becomes

$$P_{GF} = A_M \left(\Delta x S_{f_M} \Delta z_w \right). \tag{74}$$

In this equation, the average area is given by

$$A_M = A_L + W_A (A_R - A_L) , \tag{75}$$

and the average friction slope is

$$S_{f_M} = \frac{(Q_M|Q_M|)}{K_M}, \tag{76}$$

where average discharge Q_M is $Q_L + (Q_R - Q_L)/2$ and average conveyance K_M is $K_L + W_X(K_R - K_L)$. In this notation, the integral of the pressure, gravity, and friction terms over the length of the computational element at t_U is P_{GF_U}; that is, the time-point designator is appended to the subscript to indicate the time point of interest.

Applying a simple arithmetic mean over time to the momentum stored in the control volume and applying the time approximations described previously to the forces, the approximation to equation 63 is

$$\begin{aligned}
&\frac{\Delta x}{2}\left[M_{Q_{LU}}Q_{LU} + M_{Q_{RU}}Q_{RU}\right] - \frac{\Delta x}{2}\left[M_{Q_{LD}}Q_{LD} + M_{Q_{RD}}Q_{RD}\right] \\
&-\Delta t\ [\beta_{LD}Q_{LD}V_{LD} + W_T(\beta_{LU}Q_{LU}V_{LU} - \beta_{LD}Q_{LD}V_{LD})] \\
&+\Delta t\ [\beta_{RD}Q_{RD}V_{RD} + W_T(\beta_{RU}Q_{RU}V_{RU} - \beta_{RD}Q_{RD}V_{RD})] \\
&+g\Delta t\left[P_{GF_D} + W_T\left(P_{GF_U} - P_{GF_D}\right)\right] + \Delta t\left(F_{DEC_M} - W_{S_M}\right) = 0.
\end{aligned} \tag{77}$$

In equation 77, the time-averaged wind-stress term W_{S_M} is approximated by

$$W_{S_M} = \frac{\Delta x}{2}\left(M_{T_{LD}}T_{LD} + M_{T_{RD}}T_{RD}\right)C_{DU}^{(w)}\frac{\rho_a}{\rho}U_U^2\cos\psi_U. \tag{78}$$

This is the product of the wind stress in the downstream direction for the element at time t_U and the water-surface area of the element at time t_D. The term F_{DEC_M} is the time-averaged equivalent drag that represents the effect of eddy losses at expansions and contractions and the effect of submerged objects in the control volume (computational element).

A definition of the location of the velocity of approach and the projected area of a submerged object (if a drag coefficient is specified) or a cross-sectional area (if an approach-velocity head-loss factor is applied) are required in equation 53 to obtain an equivalent drag. A rigorous approximation could apply a different approach-velocity section depending on the direction of the flow past the obstruction; however, such rigor would complicate the implementation unnecessarily. The approach for the equivalent drag is designed to approximate smaller obstructions in the flow. If the obstruction is large, then the approximation is invalid because critical flow possible near the obstruction is not considered. Thus, the part of the flow area obstructed must remain small for equation 53 to be valid. The control volume defined by a computational element should be short, so that the velocity at any point and time results in a reasonable approximation to the approach velocity. If these conditions are met, then the average velocity in the control volume can reasonably be used for the approach velocity, the average water-surface height to define the projected area, and the average area for the velocity-head loss factor.

An additional convenience is to combine the two drag terms into a single equivalent term, because both forms of expression for the effects of a submerged object will probably not be needed in a single control volume. The integrand for the combined drag terms is

$$\frac{1}{2}\{C_{D_M}A_{P_M} + k_P A_M\}\, V_M|V_M|\,,$$

where the assumptions about areas and approach velocities have been substituted. It is important in this expression that the mean area be computed in model simulation from the cross-section description. The drag coefficient, C_D, the velocity-head coefficient, k_P, and the projected area of the obstruction will be given as functions of average water-surface height in the control volume (computational element). Thus, the drag expression must be rearranged so that the mean area will be separated from the other terms. This is done by normalizing the product of the drag coefficient and projected area by the mean value of area for the computational element determined in the four-point scheme of numerical integration; the result is

$$\frac{1}{2}A_M\left\{\frac{\left(C_{D_M}A_{P_M}\right)}{A_M} + k_P\right\} V_M|V_M|\,.$$

The bracketed term, $\frac{C_{D_M}A_{P_M}}{A_M} + k_P$, can be computed as a function of the average water-surface height and placed in a table. This term indicates that the area of the obstruction relative to the mean area and multiplied by a drag coefficient is equivalent in effect to a loss coefficient that is applied to a velocity head. In the normalized expression for the drag terms, the value of the integrand is evaluated at a given time t, and the averages are taken with respect to distance at this time.

The slope term for the effect of accelerating or decelerating flow (that is, for eddy losses), S_{ad}, was not included above. It is included here by extraction from equation 63 as

$$g\int_{x_L}^{x_R} A\,[x, y\,(x, t)]\, S_{ad}\, dx\,.$$

Replacing S_{ad} with equations 59 and 62 results in

$$g\frac{1}{\Delta x}sign\,(Q_R + Q_L)f(V_R, V_L)\left|\frac{\left(V_R^2\right)}{2g} - \frac{\left(V_L^2\right)}{2g}\right|\int_{x_L}^{x_R} A\,[x, y\,(x, t)\,]\,dx\,.$$

In this expression, neither the velocity-head difference nor the sum of flows is a function of the integration variable, and the area is the only variable subject to integration. Approximating the integral of the area by the product of the control-volume length and a mean area results in

$$\frac{1}{2}A_M sign\,(Q_R + Q_L)f(V_R, V_L)\left|V_R^2 - V_L^2\right|,$$

as the approximation for the term representing the effect of eddy losses resulting from channel expansions or contractions on the motion equation. This force term is similar in form to the drag resulting from a small obstruction in the flow. Thus, the eddy-loss term may be combined with drag as a single, equivalent drag term. The factor of $1/(2g)$ for the velocity heads was moved from within the absolute value because it is always positive.

The expressions for the normalized drag terms resulting from submerged objects or obstructions and eddy losses resulting from channel expansions or contractions are combined to represent the special terms in the motion equation as

$$F_{DEC} = \frac{1}{2}A_M\left[\left\{\frac{\left(C_{D_M}A_{P_M}\right)}{A_M} + k_P\right\}V_M|V_M| + \text{sign}\,(Q_R + Q_L)f(V_R,V_L)\left|V_R^2 - V_L^2\right|\right], \tag{79}$$

where F_{DEC} is the equivalent drag expressed in similar units as in equation 63. Again, F_{DEC} is a value at a given time that applies to the control volume. The time-averaged equivalent drag is then

$$F_{DEC_M} = F_{DEC_D} + W_T\left(F_{DEC_U} - F_{DEC_D}\right). \tag{80}$$

Equations 68 and 77 are evaluated in FEQ for each computational element in a branch to develop the equation system for a branch. These equations and the internal and external boundary-condition equations define the unknowns at each node in the stream system.

6.6 Selection of Weights

The equations of motion have as many as three weight coefficients that must be specified. The first weight, present in all four of the governing equation options, is W_T, the weight for integration with respect to time. The user has direct control over W_T with the FEQ input parameters of BWT and DWT. The value of W_T must satisfy $0.5 \leq W_T \leq 1$ for stability in model simulation. For a linearized form of the equations of motion, $W_T = 0.5$ results in the exact numerical result; however, the nonlinear terms in streamflow conditions necessitate the use of a higher value of W_T (Lai, 1986). These nonlinear terms affect the solution most if friction losses are relatively small. If the friction losses are small enough, then persistent, nonphysical oscillations in the solution will appear

because the
solution process, with convective terms, tends to move truncation and roundoff errors to the shorter wavelengths in the solution. In a stream, the energy contained in these shorter wavelengths would be dissipated by processes that take place at a level of detail far smaller than can be resolved with the shallow-water-wave approximations. Furthermore, there is a minimum wave length that can be resolved with the numerical solution. Therefore, the solution contains oscillations at the minimum wave length that cannot be resolved. These oscillations often persist but do not become large enough to result in computational failure as those arising from instability will. By increasing the value of W_T, a small amount of dissipation is introduced into the numerical solution. A number of researchers have studied the optimum value for W_T. From the standpoint of combining accuracy and stability, Fread (1974) found $W_T = 0.55$ to be the best value for slowly varied unsteady flow such as flood waves, Chaudhry and Contractor (1973) recommended the use of $W_T = 0.6$, and Schaffranek and others (1981) recommended the use of a W_T value between 0.6 and 1. In general, values of 0.67 or less are often sufficient to eliminate the oscillations without seriously affecting the accuracy of the final results. The additional dissipation introduced is generally much smaller than the dissipation from boundary friction. In some cases, boundary friction is large enough so that a value of $W_T = 0.5$ can be applied.

The second weight, W_X, also is present in all of the governing-equation options. It is a weight that is computed internally to adjust the computation of friction losses at small depths to avoid convergence problems in the numerical solution of the system of nonlinear equations. Following the work of Cunge and others (1980, p. 175-178), the value of this weight is computed to obtain a single-valued solution at shallow depths. To illustrate this approach, a simple approximation to the motion equation when all inertial terms are suppressed is considered, resulting in

$$Q = K_M\left(\frac{\left(z_{w_L} - z_{w_R}\right)}{\Delta x}\right)^{\frac{1}{2}}, \tag{81}$$

where K_M is a mean value of conveyance in the control volume, z_{w_L} is the water-surface elevation at the left (upstream) end of the control volume, and z_{w_R} is the water-surface elevation at the right end of the control volume. If the elevation on the left is held constant and the elevation on the right is allowed to decrease, the flow will increase for a time because the increase in slope more than compensates for the decrease in the conveyance. However, when the elevation on the right decreases below some level that depends on the two water-surface heights and the variation of conveyance, the flow will no longer increase. Thus, for a given flow, two values of water-surface height could satisfy the equation. Thus, the numerical solution may not converge because of this duplication of solutions at shallow depths. This convergence problem can be solved by changing the definition of K_M so that more weight is given to the upstream point and so that the partial derivative of the flow with respect to the right-hand elevation is always negative.

If $\Delta z_w = z_{w_R} - z_{w_L}$ is the change in water-surface elevation, $K_M = K_L + W_X(K_R - K_L)$ is the average value of the conveyance, and $S_w = -\Delta z_w/\Delta x$ is the water-surface slope with a decline in elevation given a positive slope, then

$$\frac{\partial Q}{\partial y_R} = \frac{\partial K_M}{\partial y_R} S_w^{1/2} - \frac{1}{2}\frac{K_M}{\Delta x} S_w^{-1/2} . \tag{82}$$

Multiplying equation 82 by $S_w^{1/2}$, requiring that the rate of change of flow with change in the right-hand water-surface elevation always be negative, and substituting the value for K_M results in

$$W_x \frac{\partial K_R}{\partial y_R} S_W - \frac{K_L + W_x(K_R - K_L)}{2\Delta x} < 0 , \tag{83}$$

as the defining inequality. If this inequality is not satisfied when $W_X = 0.5$, then the value of W_X must be changed so that the inequality will be satisfied when $W_X < 0.5$. The value of W_X that will satisfy the inequality is

$$W_x < \frac{(K_L/2)}{-\frac{\partial K_R}{\partial y_R}\Delta z_w - \frac{K_R - K_L}{2}} . \tag{84}$$

The denominator in this equation is always positive, given that

$$-\frac{\partial K_R}{\partial y_R}\Delta z_w - \frac{K_L + K_R}{2} > 0 . \tag{85}$$

The inertial terms have been ignored in the analysis up to this point. These are usually small when the water-surface height is small enough for this correction to be required. To allow for some consideration of these effects and to prevent the rate of change from becoming zero, a value of 0.8 of the right-hand side of the inequality in equation 84 is applied as the value for W_X in the FEQ simulation. The flow of water is assumed to be from left to right in the analysis. Similar relations are used when the flow of water is from right to left.

The third and last weight, W_A, is applied only in the options STDW and STDCW. This weight applies to certain integrals with respect to distance to allow computation of unsteady flows at small depths. The preceding small-depth adjustment allows receding flows to be computed to depths that are near zero; however, the computations may fail when the flows begin to increase from small depths unless the rate of increase with time is much smaller than for most stream rises. When the depths are small and an increase in flow begins, the errors of approximation in the distance integrals can become large. If W_{A_D} and W_{A_U} in the conservation of mass equation 68 are given the value 0.5, the distance integrals are approximated with the trapezoidal rule, and the integrand is assumed to vary linearly in the integration interval. When the depths are large, a linear approximation to the variation of flow area with distance results in a relatively small error. Equation 68 is exactly applied in the solution process, but this may require that either the left-hand or the right-hand area be negative whenever the flow increases from a small value. A negative area is impossible, so the computations fail. However, if the value of W_A is increased, giving more weight to the downstream point, equation 68 can be satisfied without resulting in a negative area. The value of W_A will approach 1.0 when the depths are very small. High values of W_A have larger apparent integration errors than does a value of 0.5, but in most applications, the details of the flow at small depths are not of great interest. Thus, some increase in error at these depths is acceptable in order to allow the computations to continue. If the small depths are of interest, then the user must reduce the length of the computational elements so that the small depths can be simulated without applying a variable W_A.

The values for W_A are defined by the user for each branch in the system. A typical depth below which the value of W_A becomes virtually 1 and a typical depth above which the value of W_A becomes 0.5 must be given for each branch. Values of the weight for depths intermediate to these two are determined by either linear or cubic interpolation at the user's request.

7. EQUATIONS OF MOTION FOR DUMMY BRANCHES AND LEVEL-POOL RESERVOIRS

The equations of motion that apply to dummy branches and level-pool reservoirs are presented and approximated in this section. Examples of conditions that may be simulated with dummy branches and level-pool reservoirs are described in sections 3.1.2 and 3.1.3, respectively. Changes in water volume and the momentum content of that water can be important factors in a branch; however, changes in water volume but not momentum content are important in a level-pool reservoir, and neither changes in water volume nor changes in momentum content are important in a dummy branch.

7.1 Approximation of the Equations of Motion in a Dummy Branch

The equations of motion for a dummy branch are simple because the storage and momentum content of a dummy branch are negligible, so flows and elevations at the two flow-path end nodes defining the dummy branch are virtually identical. The dummy branch is treated as a reservoir with constant surface area, no lateral inflow, and a linear friction-loss relation between flow and elevation difference. Therefore, the conservation of mass equation becomes

$$\frac{(A_s)}{2}\left(z_{w_{LU}} + z_{w_{RU}} - z_{w_{LD}} - z_{w_{RD}}\right) = \Delta t\,(Q_{LU} - Q_{RU})\ , \tag{86}$$

where A_S is a constant surface area and the weight for integration with respect to time has been set to 1. The left-hand node is taken to be the upstream node for the dummy branch. The default surface area applied in FEQ is 2 but this value may be overridden by the user.

The relation between the flow and the elevation difference is

$$z_{w_{LU}} - z_{w_{RU}} = \frac{(K_F)}{2}\,(Q_{LU} + Q_{RU})\ , \tag{87}$$

where the node on the left is taken to be the upstream node. The friction loss coefficient, K_F, is set by default to 2×10^{-8}, but different values can be input. The average water-surface elevation is used in equation 86 so that the partial derivatives of this equation will be nonzero in the nonlinear solution for the unknowns.

7.2 Approximation of the Equations of Motion in a Level-Pool Reservoir

A level-pool reservoir is similar to a dummy branch except that the storage relation is more complex and lateral inflows are possible. The elevation of the water surface in a level-pool reservoir is, by definition, the same at all points across the reservoir. However, it is useful to allow a small difference in elevation between the upstream and downstream flow-path end nodes, just as in a dummy branch. Thus, equation 87 applies as the relation between flow and elevation change for a level-pool reservoir. The conservation of mass equation is

$$ST\left(z_{w_{MU}}\right) - ST\left(z_{w_{MD}}\right) = \Delta t\ [Q_{LD} + W_T(Q_{LU} - Q_{LD} + I_M - Q_{RD} - W_T(Q_{RU} - Q_{RD})]\ , \tag{88}$$

where

$z_{w_{MU}} = \left(z_{w_{LU}} + z_{w_{RU}}\right)/2$;
$z_{w_{MD}} = \left(z_{w_{LD}} + z_{w_{RD}}\right)/2$; and
$ST(z_w)$ is storage capacity of the reservoir at water-surface elevation z_w.

Again, the average water-surface elevation of the two nodes is used to define the storage so that all the partial derivatives will be nonzero for equation 88 in the nonlinear solution for the unknowns. The lateral inflow, I_M, is computed for level-pool reservoirs in the same manner as for branches. One or more outflow relations can be associated with a level-pool reservoir. These relations are, however, viewed as internal boundary conditions for the stream system simulated with FEQ and not as part of the reservoir. The level-pool reservoir equations describe only the storage of water in the reservoir.

8. BOUNDARY AND INITIAL CONDITIONS

Simulation of unsteady flow in a stream segment (branch) requires information on the discharge and water-surface elevation at the boundaries of the segment throughout time and at all points in the system at some initial time, as described in section 5.3 "Nature of Shallow-Water Waves." For a network of open channels, the flow paths—branches, dummy branches, and level-pool reservoirs—are separated by special features such as junctions and hydraulic-control structures. Therefore, the system requires internal boundary conditions at the flow-path end nodes connected to special features, as well as external boundary conditions. Internal boundary conditions, which are typically determined from the hydraulic characteristics of the special features, are available for a wide variety of special features simulated in FEQ. This section details the internal and external boundary conditions and the initial conditions simulated in FEQ.

The special features in the stream system are considered small relative to the branches and level-pool reservoirs, so the storage and momentum content in the special features are negligible. Therefore, the equations of motion describing the hydraulic characteristics of flow through the special features that compose the internal boundary conditions are all steady-flow relations. For some special features—a long culvert is a simple example—this assumption is not valid. This culvert may have to be represented as a branch in the form of a storm sewer, not as a culvert, even though it has the physical form of a culvert. The concern in simulation of unsteady flow, however, is not on the physical form but on the effect that the feature has on the flows of interest.

8.1 Internal Boundary Conditions

The flow paths, each having two equations that describe the motion of water, are connected by special features. These special features compose the internal boundary conditions for the solution of the matrix describing the flow and connections in the stream system. Each internal boundary connects two or more flow paths because a special feature comprises a junction among at least two branches, dummy branches, and (or) level-pool reservoirs.

In the input to FEQ, the various internal and external boundary-condition options are referenced by a combination of a Network Matrix Control Block code number and a type number as listed in table 3. The details of the Network Matrix Control Block input are given in section 13.6, but table 3 is presented here to provide an overview of the following sections. In each heading in table 3, a listing of the codes and types is given for convenience in cross-referencing to the input description, the input, or the source code of the FEQ program.

The internal boundary conditions all reference flow-path end nodes on branches, dummy branches, and level-pool reservoirs. On the flow path, each end node has a nominal designation of upstream or downstream; although the water may flow in either direction, the designation of the end node stays the same. The same rules apply to internal boundary conditions; for some conditions, nominal designation of one end node as upstream and another as downstream is appropriate for convenience in referring to the end nodes and the application of values at those nodes. If it is necessary to refer to an end node in relation to the direction of water flow, then an adjective can be used to distinguish the nominal use of the node from the actual use. Finally, the designation attached to the node also is context relative. When referring to an internal boundary condition, the designation of a node is relative to the special feature of the stream system for which the condition is required. For example, consider a junction between two branches consisting of two flow-path end nodes, one from each of the branches. The junction is used in simulating an overflow dam at the junction. In describing the flow over the dam, one of the nodes in the junction is designated as the upstream node and the other as the downstream node. It is not important what node is selected for each use; but once selected, that designation must be applied consistently for the dam. Typically, the upstream node for the dam also will be the downstream node on the branch bringing water to the dam. In the same way, the downstream node for the dam also will be the upstream node for the branch taking water away from the dam. In some cases, to simplify the equations, this order must be maintained for a special feature. In other cases, however, the order is not prescribed.

The mathematical notation for the hydraulic characteristics in this section on internal boundary conditions becomes complex because many concepts must be distinguished. To simplify the notation, the arguments for the

Table 3. Description of Network Matrix Control Block used to represent codes and types of internal and external boundary conditions in the Full EQuations model

[--, not applicable]

Code	Type	Description	Remarks (explanation and examples)
1	--	Branch	Two externally generated equations for a branch are placed into the matrix.
2	--	Number of nodes at a junction	Continuity equation for flows at internal boundary conditions (junctions).
3	--	Equality of water-surface elevation between nodes	The water-surface elevation set equal for all nodes at a simple junction. This option can lead to improper energy relations in flow contractions.
4	--	--	One-node head-discharge relations
4	1	Flow over a weir	Standard weir flow equation; user-input weir coefficient, weir length, and weir-crest elevation.
4	2	Table relating discharge and head	Most general option allowing the user to compute a table of flows that could represent the combination of flow through several parts of a control structure—orifice and weir or spillway.
4	3	Channel control of flow	Normal-depth rating curve among other conditions.
4	4	Structure capacity as a function of time	Variable-geometry one-node control structure for which the opening fraction (that is, proportion of maximum flow rate) is known beforehand. This option is applied in model calibration.
4	5	Structure capacity varies dynamically	Variable-geometry one-node control structure for which the opening fraction (that is, proportion of maximum flow rate) is determined iteratively in model computations. This option is applied in the design of gate-operation rules.
4	6	Pump capacity limited by tail water	Flow rate is determined as a function of only upstream head and the maximum flow allowed for the given tail water level. Actual flow rate is the lesser of these two flows.
5	--	--	Two-node head-discharge relations
5	1	Expansion or contraction with critical flow	Locations along the stream where the change in cross section is abrupt enough to potentially result in critical flow. This option is used only for expanding flow and (or) contractions that do not result in critical flow.
5	2	Bidirectional flow in tables plus pumping	The pump in this option is constant flow independent of head with simple rules for starting and stopping. This option is used for modeling evacuation of offline flood-control reservoirs.
5	3	Variable-speed variable-head pump	The pump-characteristic curve is used to simulate the variation of head that the pump can supply as the flow varies.
5	4	Bridge with flow over roadway	Bureau of Public Roads bridge-loss routines, from Bradley (1970). These routines include numerous problems, so use of the bridge-loss routines in the Water Surface PROfile (WSPRO) model is preferred (see Franz and Melching, in press).
5	5	Abrupt expansion with inflow or outflow	Increase in width, drop in bottom, or both with possible inflow or outflow.
5	6	Two-dimensional tables	Tables with two independent variables. For example, piezometric head at two nodes yielding flow rate at flow node, or piezometric head at downstream node and flow at flow node yielding piezometric head at the upstream node. Examples include culvert flow, bridge flow (WSPRO), embankment flow, weir flow, channel rating, and transitions.
5	7	Variable-height weir	--
5	8	Sluice gates at McHenry Dam, Ill.	--
5	9	Underflow gates tables	Multiple two-dimensional tables are used to provide maximum generality.
6	--	--	Boundary conditions

Table 3. Description of Network Matrix Control Block used to represent codes and types of internal and external boundary conditions in the Full EQuations model—Continued

Code	Type	Description	Remarks (explanation and examples)
6	1	Flow boundary	Flow as a function of time.
6	2	Head boundary	Water-surface elevation as a function of time.
7	--	Level-pool reservoir	Two externally generated equations for a level-pool reservoir are placed into the matrix.
8	--	Critical depth	--
11	--	Conservation of momentum or constant elevation	When this code is applied, momentum is conserved if there is an inflow of water between two nodes, and water-surface elevation is constant if there is an outflow of water between two nodes.
12	--	Match average elevation at two nodes	When this code is applied, a weighted average of the upstream and downstream elevations is used as the elevation that affects the flow in a side channel.
13	--	Conservation of momentum or energy	When this code is applied, momentum is conserved if there is an inflow of water between two nodes, and energy is conserved if there is an outflow of water between two nodes.
14	--	Side-weir flow	--
15	--	Dummy branch	Two externally generated equations for a dummy branch are placed in the matrix.

cross-section characteristic functions—area, conveyance, and so forth—are often omitted when the subscript on the element symbol makes the argument clear. In all cases, the subscript on the symbol denotes the cross-section location for the characteristic denoted by the symbol. For contexts where the argument-value location differs from the location given by the subscript on the symbol, the argument is given.

The internal boundary conditions can conveniently be divided into two classes: those that relate to the conservation of mass and those that relate to water-surface elevations and flows. This classification is done because the conservation of mass must be satisfied at all junctions. In contrast, numerous choices are available for the relations between water-surface elevations and flows.

8.1.1 Conservation of Mass: Code 2

As mentioned previously, the physical size of the special feature represented with the internal boundary condition is small relative to the physical size of the branches, so changes in volume of water can be ignored in model simulations. Thus, in accordance with the conservation of mass relation, the sum of flows of water at any internal boundary must be zero if the flows are properly signed. By convention, a sign is given to each flow-path end node in a schematized stream system. The downstream end node is positive, and the upstream end node is negative. The conservation of mass (continuity) equation at each internal boundary is then

$$\sum_{i=1}^{n_j} sign_i Q_{EX_i} = 0 \,, \tag{89}$$

where Q_{EX_i} is the flow at the ith flow-path end node, n_j is the number of flow-path end nodes at the junction, and the sign function for the flow-path end node, $sign_i$, is taken as -1 for an upstream end and $+1$ for a downstream end. Equation 89 is applied at each internal boundary.

8.1.2 Elevation-Flow Relations

Only one conservation of mass relation or equation is needed at each internal boundary, but many water-surface elevation-flow relations are possible. The number of these relations depends on the number of flow paths that form the junction. If n_j flow-path end nodes are at the junction, then $n_j - 1$ relations between elevation and flow must be given. Adding the conservation of mass relation to these yields a total of n_j relations for a junction connecting n_j flow paths. All that these relations have in common is that a relation between water-surface elevations at an internal boundary is provided. The flows for one or more connecting flow paths also are involved in many of the relations. For convenience the relations can be divided into groups based on how many flow-path end nodes are involved and whether fixed- or variable-geometry relations are involved.

8.1.2.1 Fixed Geometry

A fixed-geometry elevation-flow relation results at a structure or natural feature in the stream at which the relation between flow and elevation is independent of time. In FEQ, these are described as control structures even though no physical structure may be present. This is done because the hydraulic effects are the same whether the feature is natural or constructed.

8.1.2.1.1 One-Node Control Structures: Code 4, Types 1-3; Code 8

The simplest relations are those involving the elevation at a single flow-path end node and the flow at that node or, in some cases, at another flow-path end node. Use of a one-node relation implies that conditions at other end nodes in the junction have no effect on this relation. An example of a one-node relation is flow over the spillway on a dam that is so high that the tail water does not affect the discharge. Thus, the control is complete, meaning that knowledge of the water-surface elevation at the flow-path end node completely specifies the value of the flow at that node. In the general case, the flow-path end node specifying the elevation of the water surface and the flow-path end node specifying the flow can be distinct. To denote these nodes clearly, q is the subscript for the flow-path end node at which the flow is defined, and h is the subscript for the flow-path end node that defines the head for the relation between elevation and flow. The generic equation for all one-node relations is

$$Q_q - f_q\left(z_{w_h}\right) = 0\,, \tag{90}$$

where $f_q\left(z_{w_h}\right)$ is the function defining the flow for each water-surface elevation. The source of this function depends on the situation. It could represent flow over a weir, flow over a spillway, a stream-gage rating curve, a normal-depth rating curve, a critical-flow rating curve, or some other condition. The source for defining the function is not needed in equation 90. As outlined later, various options for the source of the function in equation 90 are provided in FEQ.

Equation 90 does not include information on the direction of the flow. Information on direction of flow in a branch, a dummy branch, or a level-pool reservoir is contained in the sign of the flow at the end nodes on the flow path. If the sign of the flow is positive, then the flow is from upstream to downstream. If the sign of the flow is negative, then the flow is from downstream to the upstream. However, the sign must be specified in the model when the flow is leaving or entering the discharge node for the control structure. The numbers tabulated in a 1-D function table (sections 3.2.1 and 11.1) to represent the function f_q in equation 90 do not specify a direction relative to the discharge node. The user must specify the direction of flow that the numbers in the table represent. Direction is given as -1 if a positive number in the flow table represents flow out of the flow path where the discharge node is located and as +1 if a positive number in the flow table represents flow into the flow path on which the discharge node is located. If D_D is the direction of flow specified by the user and D_q is the flow-node sign computed in FEQ simulation, then the sign the flow at the discharge node must have is $D_F = -D_D D_q$, and equation 90, expanded to represent the flow direction as well as the flow magnitude, becomes

$$Q_q - D_F f_q\left(z_{w_h}\right) = 0\,. \tag{91}$$

8.1.2.1.2 Two-Node Control Structures: Codes 3 and 5

For one-node control structures, the relation between elevation and flow is not affected by changes downstream from the special feature (no backwater effects from the tail water). In many real-world cases, however, backwater does affect flow at control structures, so simulated control structures are provided in FEQ that are subject to effects from a downstream water level. One of the two flow-path end nodes included in the structure must specify the flow through the structure. This end node is called the discharge node because the flow that passes through, under, or over the structure is specified at this node. The upstream node and the downstream node for two-node control structures are specified by the user. If D_C is the control-structure sign, then D_C is +1 if the discharge node is the upstream node and D_C is −1 if the discharge node is the downstream node. A flow sign is defined in the model for two-node control structures, but it differs slightly from the flow sign for a one-node control structure. If D_F is +1, then the flow is from the upstream node to the downstream node; otherwise, the flow is from the downstream node to the upstream node.

8.1.2.1.2.1 Same Elevation: Code 3

In the simplest two-node relation, the water-surface elevation must be the same at the flow-path end nodes at all times and for all flows; that is,

$$z_{w_L} - z_{w_R} = 0\,, \tag{92}$$

where the subscripts L and R denote the two flow-path end nodes. This relation is useful for a simple junction.

Forcing the water-surface elevations to be the same at the flow-path end nodes with this code must be done with care. If the velocity head at the node downstream from the junction is greater than that at the node upstream from the junction, then energy is not conserved. Normally, the downstream flow area should be greater than the upstream flow area, and, if the flows do not differ markedly at the two nodes, a small amount of energy will be lost between the two nodes; this computational result is physically realistic. Otherwise, if the downstream flow area is smaller, then mechanical energy will be added to the system; this result is physically unrealistic. If this addition is large enough, the computations will fail. In other cases, incorrect answers will result because the internal boundary condition is unrealistic.

8.1.2.1.2.2 Flow Expansion. Code 5, Type 1

In the motion equation for a branch, the loss terms that relate to the additional losses (in excess of those from boundary resistance) represent the effects of gradual changes in cross section. The possibility of critical depth within a branch is not checked. Locations along the stream where change in section size is large enough to potentially result in critical flow must be isolated and represented as special features. If the flow expands in area as it moves from the upstream node to the downstream node of the control structure, the transition sign, D_T, is set to +1; otherwise, D_T is −1. It is convenient to define yet another sign for two-node control structures; this is the system sign, D_S, which equals $D_q D_C$. If Q_q is set equal to the flow value at the discharge node, then D_F equals $sign(Q_q)D_S$. This establishes the direction of flow relative to the upstream and downstream nodes of the control structure.

If the flow is expanding, the losses will be larger than if the flow is contracting. Therefore, the user must supply the transition sign, D_T, and the values of the loss coefficients to be applied for each direction of flow in the control structure. Let K_+ equal the head-loss coefficient when the flow is from the upstream node to the downstream node for the control structure (D_T = +1) and K_- equal the head-loss coefficient when the flow is from the downstream node to the upstream node for the control structure (D_T = -1). Further, let K_E equal $D_F D_T K_+$ if $D_F > 0$ and K_E equal $D_F D_T K_-$ otherwise. With these terms defined, the equation defining the relation between variables at the upstream node and the downstream node for the control structure when the flow is subcritical is

$$z_{w_L} + y_L + \alpha_L \frac{\left(V_L^2\right)}{2g} = z_{w_R} + y_R + \alpha_R \frac{\left(V_R^2\right)}{2g} + K_E\left(\alpha_L \frac{\left(V_L^2\right)}{2g} - \alpha_R \frac{\left(V_R^2\right)}{2g}\right). \tag{93}$$

Here the subscripts L and R refer to the upstream node and downstream node for the control structure, respectively. Recasting equation 93 into a more symmetrical form results in

$$z_{w_L} + y_L + (1 - K_E)\,\alpha_L \frac{\left(V_L^2\right)}{2g} = z_{w_R} + y_R + (1 - K_E)\,\alpha_R \frac{\left(V_R^2\right)}{2g}, \tag{94}$$

the equation used in FEQ simulation when the flow is subcritical. The left-hand side of equation 94 is denoted by z_{lhs} and the right-hand side by z_{rhs}. Values for these variables are used later in this section in checking for submergence of critical flow.

Critical flow can result in a transition at the smaller section. If the flow is expanding, then the critical section will be at the inflow cross section; and if it is contracting, the critical section will be at the outflow cross section. The following is done in simulation, given the information at only two flow-path end nodes:

1. The critical flow is detected, and the relation defining the flow is changed to represent critical flow.
2. The condition at which critical flow is drowned by downstream conditions is detected, and the relation is changed to represent subcritical flow.

If the flow is expanding, then two flow-path end nodes are sufficient for the functions listed above. The critical control is at the smaller section, and critical flow can be forced at the inflow end node. The outflow end node, at the larger section, reflects the conditions downstream and is used to detect submergence of the critical control. No expansion losses result when critical control is present because the water-surface elevations at the two flow-path end nodes are independent.

A classic example of flow expansion is an abrupt drop in the channel, such as at a drop structure, a waterfall, or a steep but short rapids. At low flows, critical control is present at the head of the drop; however, as the flow increases, tail water may eventually drown the critical flow section. When the critical control is present, the elevation of the water at the foot of the drop is determined by conditions downstream. Thus, only two flow-path end nodes are needed to both detect the formation of the critical control and detect its submergence.

If the flow is contracting, however, two flow-path end nodes are not sufficient for simulation of critical control. In a contracting flow, the water-surface elevation may decline substantially as the water accelerates to the smaller section. Substantial energy losses also may occur in the contraction. To include both of these effects, both nodes must be used: one for the conditions at the larger section and the other for the conditions at the smaller section. However, once the flow becomes critical at the outflow end node, there is no way to detect when the flow will be drowned because both flow-path end nodes are already in use and no node is available to represent tail water at the critical control. Rather than add some special nodes, the critical control is represented in both cases with only two flow-path end nodes. The representation of the critical flow in a contracting flow does not include the contraction losses or the decline in water-surface elevation. Therefore, contractions where critical flow may be present should not be represented with equations 93 and 94. An alternative representation that applies an explicit description of the flow through the transition for all conditions of flow is provided in FEQ and is discussed below in section 8.1.2.2.2.1. The alternative representation of contracting flow should be applied for all cases where critical flow is possible, and it is generally recommended for contracting flows subject to a wide range of flows.

To detect critical flow, the critical flow is estimated in simulation as equation 94 is solved. If the flow at the discharge node is greater than the estimated critical flow, a critical control is applied in the model at the node upstream from the control structure, using the cross-sectional characteristics from the smaller of the two cross sections. If the subscript c denotes the values at critical flow, then application of

$$T_c Q_q^{\,2} - g A_c^{\,3} = 0 \tag{95}$$

results in the flow at the discharge node being equal to the critical flow in the critical section. In computing the critical flow, it is assumed that $\beta = 1$.

The state of the transition (critical or subcritical) is recorded internal to the model simulation. If the state is subcritical, flow conditions are checked to determine whether the transition should be switched to critical as just outlined. If the state is critical, then the critical control is checked to determine whether it will be drowned from downstream. Let z_{w_c} be the water-surface elevation at the critical control. If $D_F > 0$ (meaning flow from upstream node to downstream node for the control structure), then the control is drowned if $z_{rhs} > z_{w_c}$. For flow in the other direction ($D_F < 0$), the control is drowned if $z_{lhs} > z_{w_c}$.

The equations outlined here function properly only if the transition does not change; that is, if the user denotes the transition as an expansion, then the flow must expand for all flows. This assumption is violated for some transitions. For example, consider the flow in the departure reach of a culvert. At low flows, the channel downstream from the culvert may be smaller than the cross section of the culvert barrel. Thus, a flow contraction results at low flows, whereas a flow expansion results at higher flows and defines the dominant transition at this location. In such cases, computational convergence problems may result, or the computed solution may not be valid. As already noted, this approach should not be used if a contracting flow becomes critical or if the variation of β affects the value of critical flow. For these cases, an alternative is provided in section 8.1.2.2.2.1.

8.1.2.1.2.3 Bi-Directional Flow with Pump or Simple Conveyance: Code 5, Type 2

This option is used to represent the flow of water between the main stream channel and storage areas adjacent to the stream. These storage areas may be natural slack-water areas that fill during high flows or constructed offline flood-control reservoirs.

Constructed offline flood-control reservoirs are typically connected to the stream by flow over a side weir, spillway, roadway embankment, or similar structure. The flow over this structure is approximated by a weir equation. During a flood, flow at this structure may be out of the stream as stage increases and into the stream as stage decreases. The bottom of the reservoir typically will be deeper than the normal water-surface elevation in the stream to maximize flood-control effectiveness. Therefore, pumps must be used to completely drain the flood-control reservoir after the flood has passed. This flood-control method can be represented with the bidirectional flow with pump option. In this option, only pumps that yield constant flow independent of head with simple rules for starting and stopping can be simulated.

Flow over the weir, Q_w, is specified by use of tables given by the user. Four tables must be given; two for each direction of flow. The first of the two tables lists the flow as a function of head, and the second lists the submergence factor as a function of submergence ratio. If z_m is the minimum elevation of the weir, h_w is the piezometric head on the weir, and d is the piezometric head of the tail water on the crest of the weir, then for flow from upstream node to downstream node for the weir (left to right), $h_w = z_{w_L} - z_m$ and $d = z_{w_R} - z_m$. The defining equation for flow over a weir is

$$Q_w - D_s f_{Qud}(h_w) f_{Sud}\left(\frac{d}{h_w}\right) = 0\,, \tag{96}$$

where f_{Qud} denotes the table of flow and f_{Sud} denotes the table of submergence factors for flow from upstream node to downstream node. For flow in the opposite direction the equation changes to

$$Q_w + D_s f_{Qdu}(h_w) f_{Sdu}\left(\frac{d}{h_w}\right) = 0, \tag{97}$$

where f_{Qdu} denotes the table of flow and f_{Sdu} denotes the table of submergence factors for flow from downstream node to upstream node. In this latter case, the heads become $h_w = z_{w_R} - z_m$ and $d = z_{w_L} - z_m$.

Pumping is specified when a pumping rate is input, $Q_p \neq 0$. If Q_P is greater than zero, then the pumped flow is from the upstream node to the downstream node of the pump; otherwise, the pumping is in the opposite direction. If the pump is on, then the pumped flow, with the proper sign, is added to the flow computed over the weir. If the pump is off, then it is turned on whenever water is available at its source node and capacity is available at the destination node to accept water. If the pump is on, then it is turned off if the water is no longer available at the source node or if capacity is no longer available at the destination node to accept water. The values controlling the pump are supplied by the user and are described in the "Input Description for Full Equations Model" (section 13.6).

Natural storage areas may be represented by level-pool reservoirs where the inflow and outflow is controlled primarily by boundary friction in the channels connecting the storage areas to the stream. The effects of inertia in the connecting channels are negligible. Thus, a connecting channel can be represented with the simple conveyance option.

A table describing the conveyance function and the distance for converting the difference in elevation between the upstream and downstream nodes for the conveyance channel to a water-surface slope must be input in the simple-conveyance option. If Δx_c is the distance between the two nodes, $S_w = \left(z_{w_L} - z_{w_R}\right)/\Delta x_c$ is the negative of the water-surface slope between the two nodes, $\bar{z}_w = 0.5\left(z_{w_L} + z_{w_R}\right)$ is the average water-surface elevation between the two nodes, and $f_K(z_w)$ is the conveyance function for the flow path between the two nodes, then the flow is defined by

$$Q_q - sign(S_w) D_s f_k(\bar{z}_w) \sqrt{|S_w|} = 0, \tag{98}$$

if $|S_w| > 1 \times 10^{-4}$ and

$$Q_q - 100 D_s f_k(\bar{z}_w) S_w = 0 \tag{99}$$

otherwise. Equation 99 is applied to linearize the near-zero flows to avoid convergence problems at small water-surface slopes.

8.1.2.1.2.4 Abrupt Expansion with Lateral Inflow/Outflow: Code 5, Type 5

Abrupt expansions are sometimes used to reduce the velocity of water in a stream channel so that water can be diverted more easily. Because the expansion is abrupt, the conservation of momentum principle can be applied to describe the flows. The abrupt expansion can be either an increase in width, a drop in the bottom of the channel, or both. The following conditions apply to the relations used in FEQ simulation:

1. No reverse flow is possible at the upstream node; however, the flow can reverse at the downstream node.
2. End nodes on branches must be specified such that the upstream node of the control structure is the downstream end node on the branch bringing water to the structure, and the node specified as the downstream node of the control structure is the upstream end node on the branch taking water from the control structure.
3. Friction and gravity forces are ignored in the control volume describing the abrupt expansion for the momentum balance.
4. The node designated as the discharge node must be the upstream node for the structure.
5. The diversion channel is assumed to be perpendicular to the flow through the control structure. The mouth of the diversion channel is designed to minimize separation of the water from the wall of the channel as water flows into it.

As the water enters the diversion channel, it must change flow direction by 90 degrees. This change is accomplished over some distance in the source and diversion channels. In this region, the velocity is constantly changing direction as well as magnitude, so estimating the streamwise momentum flux that enters the diversion channel cannot be done simply. Therefore, the boundaries of the control volume (fig. 15) for application of the momentum balance must include some part of the diversion channel. The control volume must include sufficient length of the diversion channel so that the velocity in the channel is again adequately described as 1-D. One problem is exchanged for another in this control-volume choice because the water in the diversion channel will have a cross slope on the surface for some distance downstream from the mouth. The cross slope results in an upstream component of hydrostatic-pressure force in the direction of application of the momentum balance. Including this force in the balance is difficult because no methods have been established for estimating it. Therefore, in this simple analysis, this force is omitted. A well-designed entrance will reduce the length of flow required for the velocity to become virtually parallel to the diversion walls and, as a consequence, will reduce the magnitude of the neglected hydrostatic-pressure force.

Application of the momentum balance to the control volume shown in figure 15 yields

$$gJ_R\left(z_{w_L}\right) + \beta_L \frac{Q_L^2}{A_L} = gJ_R\left(z_{w_R}\right) + \beta_R \frac{Q_R^2}{A_R}, \tag{100}$$

where J_R is the first moment of area function at the downstream node (cross section). If water is being diverted or added, $Q_L \neq Q_R$. Equation 100 is applied when no critical control is present. A critical control, if present, will be at the upstream node. In all cases, the user must supply a 1-D table (section 11.1) defining a function specifying critical flow at the upstream node. This function is denoted by $f_c\left(z_{w_L}\right)$.

If Q_L is greater than $f_c\left(z_{w_L}\right)$, then critical flow is established at the upstream node by requiring that $Q_L - f_c\left(z_{w_L}\right) = 0$ and the state of the flow is changed to critical. The critical flow is drowned in model simulation if the water-surface elevation at the downstream node becomes too high. Two criteria are used for determining whether critical flow is drowned: one criterion is quick and easy but not always correct, and a second that is more complex but is always correct (given the assumptions above). The first criterion is that if z_{w_L} is greater than or equal to z_{w_R}, then critical flow is retained; that is, the tail water elevation must become higher than the elevation of the water surface at critical depth before the critical control is drowned. If the tail water is higher than the critical-flow water surface at the upstream node, equation 100 is applied to define the residual function

$$F\left(z_{w_L}, z_{w_R}, Q_L, Q_R\right) = g\left[J_R\left(z_{w_L}\right) - J_R\left(z_{w_R}\right)\right] + \beta_L \frac{\left(Q_L^2\right)}{A_L} - \beta_R \frac{\left(Q_R^2\right)}{A_R}. \tag{101}$$

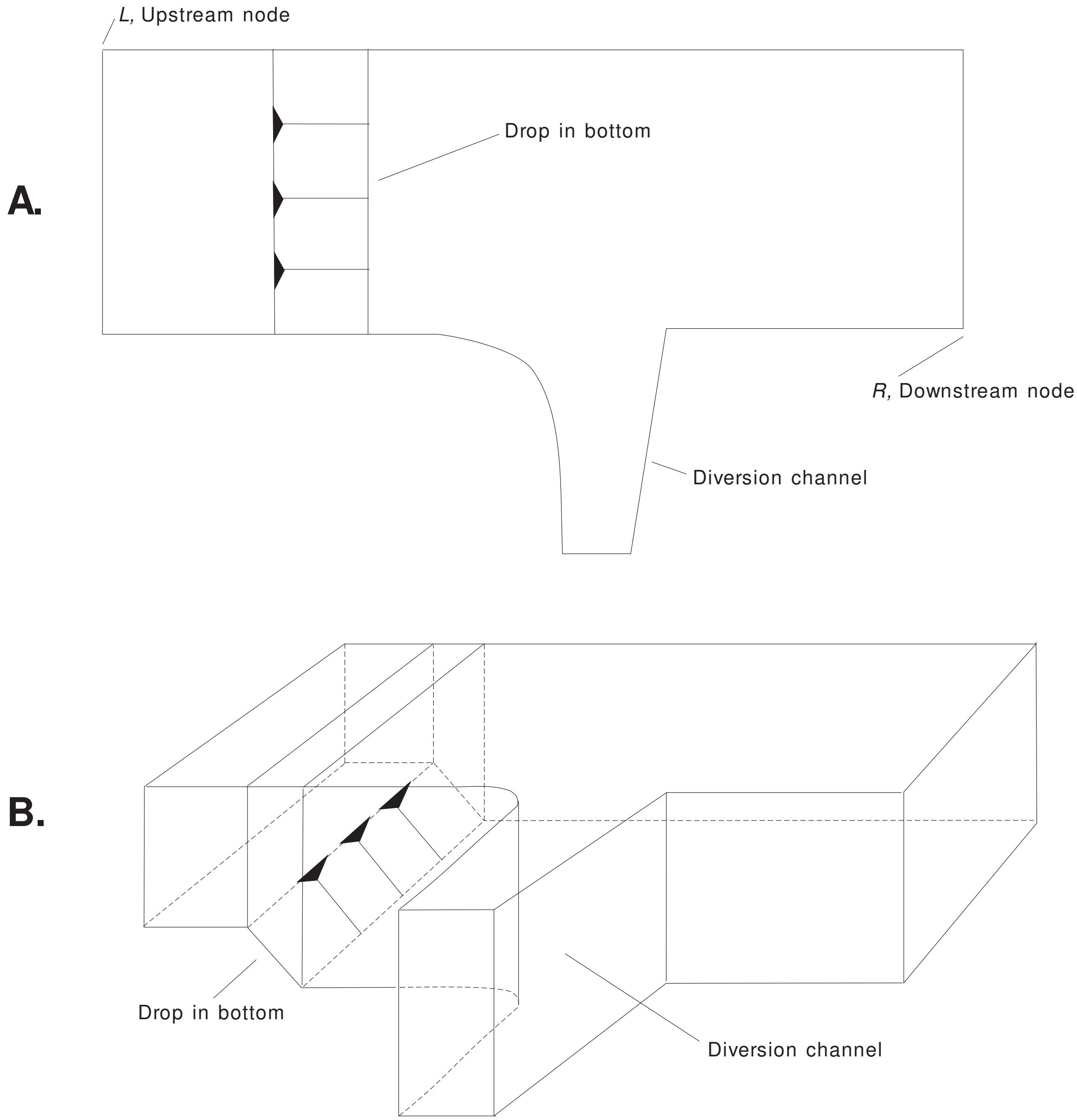


Figure 15. Control volume (A) plan view and (B) three-dimensional view for abrupt expansion with a diversion channel in a hypothetical stream.

The second criterion is that if $F\left(z_{w_{Lc}}, z_{w_R}, Q_{Lc}, Q_R\right) < 0$, where the subscript c denotes the critical state, the critical flow is drowned and the flow state is changed to subcritical in the FEQ calculations.

8.1.2.1.2.5 Explicit Two-Dimensional Flow Tables: Code 5, Type 6

Certain special features are difficult to represent within an unsteady-flow analysis because the hydraulics of those features are not well understood. The values of flow computed with the appropriate equations at a boundary between two flow types often are substantially different. This difference presents little problem for manual computations because the analyst merely smooths over the discontinuity. Such smoothing, however, is difficult to do properly within the computational scheme of a computer program for simulation of unsteady flow. The relations used for special features in FEQ are derived from steady flow. In principle, these relations can be computed over the necessary range of the independent variables to define the relation in tabular form. Thus, the computations are done only once instead of possibly thousands of times during the computation of the unsteady-flow results.

In the approach applied in FEQ simulation, FEQUTL or some other means is used to compute tables of numerical values, arranged in a predefined order, so that the values needed in the computations can be quickly found. The resulting tables are called 2-D tables because they include two independent variables. The tables referenced earlier are 1-D tables because they include only one independent variable. Two types of 2-D tables are supported in FEQ: the first, given the table type number 13, uses piezometric head at two nodes to estimate the flow rate at the discharge node, and the second, given the table type number 14, uses piezometric head at the downstream node and the flow at the discharge node to estimate the piezometric head at the upstream node.[3] The goal of using explicit 2-D tables is to circumvent the difficult task of providing the smooth flow relations needed to avoid convergence problems in the iterative solution process. Thus, the source of the information in these tables is not needed in FEQ simulation. The tables can be computed in FEQUTL, in some other computer program, or even manually.

Any smoothing of the transition between flow types or classes is done at the time the 2-D table is computed. The facility to create these tables is available in FEQUTL (Franz and Melching, in press) for flow over complex weirs, flow through culverts, flow through expansions and contractions, and flow through prismatic channel segments. Details of these computations and of the definition of the format and logical structure of 2-D tables is contained in the “Input Description for the Full Equations Model” (section 13).

The defining relation for the tables having arguments of piezometric head (type 13) is

$$Q_q - D_s \sum_{i-1}^{m_c} f_i\left(h_{L_i}, h_{R_i}\right) = 0\,, \tag{102}$$

where

f_i is the function giving flow through structure i that conveys flow between the upstream and downstream nodes of a special feature in the stream system;

$h_{L_i} = z_{w_L} - z_{h_i}$ is upstream piezometric head relative to the head-reference point, z_{h_i}, for structure i;

$h_{R_i} = z_{w_R} - z_{h_i}$ is downstream piezometric head; and

m_c is the number of control structures conveying water between the two nodes.

The flow paths through the structures at the special features are in parallel and the flow in any specific path is not retained in the simulation. All flows are automatically summed to the value of flow at the discharge node. One application of multiple flow paths in parallel is the hydraulic representation of a multibarrel culvert with barrels of different diameter, invert elevation, and composition. Thus, one barrel could be flowing full while another is flowing partly full. One way of representing such a culvert in FEQ would be to compute a 2-D table having arguments of piezometric head (type 13) for each barrel, prepared by use of the options in FEQUTL. Each table can be computed independently. In the model, the flows will be allocated to each path according to the characteristics of each table. Care must be taken in applying this method if the approach velocity to the culvert is a significant factor in the capacity of the culvert.

The defining relation for tables having arguments of downstream piezometric head and flow at the discharge node (type 14) is

---

[3]Two-dimensional tables of type 6 are supported in the FEQ computer code; however, support of such tables is maintained in the FEQ computer code for the benefit of users who have prepared models of a given stream system with earlier versions of the FEQ computer code. Tables of type 6 are equivalent to tables of type 13 but have a different format. Users of FEQ are encouraged to adopt tables of type 13 for all new applications of FEQ to stream systems. No information on the preparation or formatting of tables of type 6 is given in this report or in the documentation report for FEQUTL (Franz and Melching, in press).

$$f_{14}(h_R, Q_q) + z_h - z_{w_L} = 0\,, \tag{103}$$

where f_{14} is the function that yields the head at the upstream node given the piezometric head at the downstream node and the flow at the discharge node. In this option, only one structure can connect the two flow-path end nodes because head, unlike flow, is not additive. This option is complicated by a special case that must be isolated and treated separately; neither piezometric head may be above the head-reference point. This head relation indicates that the flow at the discharge node is zero, but this situation is not described in equation 103. In this special case, the relation becomes simply, $Q_q = 0$.

8.1.2.1.2.6 Conservation of Momentum/Constant Elevation: Code 11

In this option, momentum is conserved if there is an inflow of water between the two flow-path end nodes, and water-surface elevation is constant if there is an outflow of water. The assumptions for this option are the following:

1. The cross sections at the two flow-path end nodes are identical.
2. The bottom-profile elevations for the cross sections at the two flow-path end nodes are identical.
3. The inflow of water is perpendicular to the flow direction between the two flow-path end nodes. Any forces in the flow direction originating from the channel providing inflow are ignored.
4. Friction and gravity forces are ignored in the momentum balance.

If there is inflow, the defining relation is

$$J_L\left(z_{w_L}\right) + \beta_L \frac{Q_L^2}{A_L^2} - J_R\left(z_{w_R}\right) - \beta_R \frac{Q_R^2}{A_R^2} = 0\,, \tag{104}$$

and if there is outflow, the relation is

$$z_{w_L} - z_{w_R} = 0\,. \tag{105}$$

This option could be used to represent an inflow of water to the stream that flows over a high spillway or waterfall and then enters the stream. As stated in section 1.3, conservation of momentum is the preferred method in this case.

8.1.2.1.2.7 Conservation of Momentum/Energy: Code 13

In this option, momentum is conserved if there is an inflow of water between the two flow-path end nodes, and energy is conserved if there is an outflow. The assumptions are the same as in the preceding section. The momentum balance is equation 104. The energy-balance relation is

$$z_{w_L} + \alpha_L \frac{Q_L^2}{2gA_L^2} - z_{w_R} - \alpha_R \frac{Q_R^2}{2gA_R^2} = 0\,. \tag{106}$$

Flow over a side weir is commonly approximated by assuming that the specific energy is unaffected by the outflow of water because the outflow is smooth. Therefore, flow over a side weir is simulated with this option, in conjunction with codes 2 and 14.

8.1.2.1.3 Three-Node Control Structures

Many physical features in a stream system can be represented with one- and two-node control structures, but certain features will require access to more than two flow-path end nodes at one time. Simulation of these features is discussed in this section.

8.1.2.1.3.1 Average Elevation at Two Nodes: Code 12

When water enters a stream perpendicular to the flow, use of the option for conservation of momentum/elevation or momentum/energy may result in differences in the water-surface elevation at the two flow-path end nodes bounding the special feature. If the inflow enters from a channel that is affected by the water-surface elevation in the receiving stream, then a choice must be made as to which of the two elevations should be used. An average value of these two elevations may be used in this code as the elevation that affects the flow in the side channel. The average is specified by a user supplied weight, W_E, that applies to the elevation at the upstream node. The weight must be $0 \le W_E \le 1$ to yield a valid average. In the following discussion, the upstream node and the downstream node for the special feature continue to be denoted by the subscripts L and R, respectively. The third node is denoted by the subscript M, indicating a middle location. The equation used for this code is then

$$z_{w_M} - W_E z_{w_L} - (1 - W_E)\, z_{w_R} = 0\,. \tag{107}$$

8.1.2.1.3.2 Flow Over a Side Weir: Code 14

Side weirs are sometimes used to divert water from streams during high flows to reduce flood damages downstream. The diverted water may enter a reservoir or an additional channel that conveys the water around the protected area before rejoining the stream. The general character of these flows is understood (Henderson, 1960, p. 273-275); however, quantitative values for key coefficients are not well known. The only side-weir flows of interest in FEQ simulation are those that maintain subcritical flow throughout all flow conditions. (Cases of supercritical flow and of hydraulic jumps along the side weir are excluded from analysis because of their complexity.)

Flow of water over a side weir is generally smooth and gradual, so energy loss is minimal. In addition, if the channel slope is near zero and the channel resistance is small, the energy content per unit volume of water in the channel is changed little by the loss of water over the side-weir crest. This observation indicates that the specific energy of the water in the channel is virtually constant along the weir. Because the flow is decreasing in the downstream direction, the water surface must rise to maintain the constant specific energy with a decreasing velocity head. The velocity distribution in the source channel undergoes a change along the weir. As water is removed, the velocity on the bank opposite the weir is retarded; and, if the diversion over the weir is more than about 50 percent of the approaching flow, separation occurs near the downstream end of the weir and a reverse eddy forms, at least in the surface layers of the flow (El-Khashab and Smith, 1976; Tynes, 1989). Measurements made by El-Khashab and Smith (1976) indicated that α and β increase along the side weir in the direction of flow. They also noted that the higher velocity zones of flow in the source channel, which carry greater energy content per unit volume over the weir, are removed by the weir. This preferential removal of high-energy-content water leads to some reduction in the specific energy of the flow remaining in the source channel. Therefore, El-Khashab and Smith (1976) argued for the application of momentum conservation as the preferable alternative to specific-energy conservation. The momentum content removed by the water leaving the channel must be estimated for application of the momentum-conservation principle. El-Khashab and Smith developed simple means for estimating the momentum flux over the weir, but the results were for a sharp-crested weir and are of limited utility in applications to prototype stream systems.

Most of the work reported on flow over side weirs is for sharp-crested weirs, whereas most prototype systems use broad-crested weirs, at least for flood-control structures. Only the report by Tynes (1989) contains measurements for flow over a broad-crested side weir; however, his analysis was limited, and most of his results are specific to the particular configuration tested. Data on the performance of broad-crested weirs of the type most likely to be represented in applications of FEQ are scarce. Therefore, the techniques outlined here are best used in preliminary analyses. Whenever possible, physical-model tests should be made on side weirs to ensure that they will function as planned. The work by Tynes (1989) also could be used if the characteristics of the side weir are in the range of variables tested.

This brief discussion indicates that flow over a side weir is one of the more complex flows to simulate in 1-D, unsteady-flow analysis. Various approximations have been applied, but no consensus is apparent in the literature about the best method. Head will vary along the weir, but in FEQ simulation, a detailed integration along the side weir cannot be done; however, the major features of side-weir flow for subcritical flow along the weir are included in FEQ simulation.

In theory, the flow over the side weir, Q_{SW}, may be computed as

$$Q_{SW} = \int_O^L q_{SW}(l)\,dl = \int_O^L C_{SW}(l)\,h_{SW}(l)^{3/2}dl\,, \tag{108}$$

where l is the distance measured along the side weir with $l = 0$ at the upstream end of the weir; $h_{SW}(l)$ is the head on the side weir given by $y(l) - H_{SW}(l)$, where $y(l)$ is height of the water surface above the channel bottom and $H_{SW}(l)$ is the height of the weir crest above the channel bottom; $C_{SW}(l)$ is the side-weir coefficient, which varies with distance; $q_{SW}(l)$ is the rate of outflow per unit length of side weir; and L is the length of the side weir.

In steady-flow analysis, equation 108 is combined with some form of the energy- or momentum-conservation equation to solve for the water-surface profile along the weir and the flow over the side weir. Estimating the momentum content of the water flowing over the side weir is difficult, especially with broad-crested side weirs. Therefore, conservation of specific energy is probably the best choice for estimation of flow over a side weir. The integral in equation 108 is approximated by dividing the length of the weir into intervals. Within each interval, the flow is estimated by use of the midpoint head value in that interval. For example, if a single interval is used, the head at the midpoint of the weir is used to estimate the flow over the weir. To obtain the value of the head at the midpoint, a junction between branches is placed there so that the flow over the weir is simulated in the junction. (Side-weir flows are not permitted to enter the interior of a branch in model simulation.) This process can be refined so that the weir is divided into many intervals. In this way, the variation of head along the weir can be approximated. This process proceeds as follows:

1. The number of equal-length segments applied to represent the side weir is selected. The number selected should be a small integer power of 2. More than eight segments makes the model input too complicated. No guidelines are available for deciding the best number of segments.
2. The midpoint of each segment of the weir is marked.
3. A junction is placed between branches at each midpoint. For a single interval, there will be two branches: one coming from upstream to the midpoint and the other leading downstream from the midpoint. For two intervals, there will be three branches: one coming from upstream to the upstream midpoint, one connecting the two midpoints, and one leading downstream from the downstream midpoint. The branches represent the channel shape and frictional characteristics. At each junction between branches, code 13 (conservation of momentum or energy) is applied to include the effect of water leaving or entering the channel. If water is leaving the channel, the specific energy is assumed to be conserved, so an increase in the elevation of the water surface will result. Although this increase results over the length of each weir interval, the increase is approximated as taking place at the midpoint of the weir interval in FEQ simulation. The head on the weir is computed from the average water-surface elevation at the ends of the two branches at each weir interval midpoint. The flow over the weir is given in a 2-D table that contains flows over the weir interval, computed under the assumption that the water surface is horizontal and that the weir is perpendicular to the flow. The velocity of approach is forced to be small in this computation. The flow over the side weir is different from the flow over a normal weir for the same water-surface elevation because the component of velocity along the side weir retards water from flowing over the

weir. Thus, a correction must be made that depends on the flow characteristics at the midpoint of the weir interval.

4. Code 14, for side weirs, is used to compute the flow for each weir interval and collect these flows by means of dummy branches in a manner appropriate to the application. Code 13 (conservation of momentum or energy) and code 2 (conservation of mass) are used to complete the description at each junction.

In FEQ simulation, the correction developed by Hager (1987) is applied to the normal weir coefficient to approximate the coefficient for each location along a side weir. This factor, ω, is

$$\omega = \left\{\frac{\left(\mathbf{F}_w^2 + 2\right)}{3\mathbf{F}_w^2 + 2}\right\}^{1/2}, \tag{109}$$

where $\mathbf{F}_w$ is the Froude number of the channel flow relative to the head on the side weir, $V(l)/\sqrt{gh_{SW}(l)}$. The correction was derived and tested by Hager on a sharp-crested weir, but Hager felt that it also would apply to a broad-crested weir. Tynes (1989) did a rough test of the Hager correction factor and concluded that results were erratic when applied to broad-crested weirs; however, Tynes found that the performance of the correction factor averaged over many tests reliably estimated the average value.

Application of the correction factor computed in equation 109 is most accurate when the bottom slope of the channel is small and the channel along the weir is prismatic. The 2-D tables for flow over a normal weir reflect the nature of the weir crest and possible weir crest slope along the channel. The correction factor adjusts for the state of the flow at the midpoint of each weir interval. Continuous variation of the hydraulic characteristics along the side weir is approximated in FEQ in a series of steps. The correction factor is always less than 1.0, and, therefore, the flow over the side weir is always less than the flow over a normal weir for the same upstream head.

Let $\bar{z} = W_E z_{w_L} + (1 - W_E) z_{w_R}$ be the average water-surface elevation in the source channel defined by the user supplied averaging factor, W_E, $\bar{h} = \bar{z} - z_{hsw}$ be the average piezometric head in the source channel, and z_{hsw} be the elevation of the datum for defining heads. The middle node is the node on the flow path that receives the water flowing over the weir. Therefore, the middle node becomes the discharge node, as in a two-node control structure. Let the sign of this node be D_q in keeping with the notation used earlier. Also, let $h_M = z_{w_M} - z_{hsw} =$ the head at the middle node. The relation defining flow leaving the source channel is

$$\varpi f_w\left(\bar{h}, h_M\right) + D_q Q_{swM} = 0, \tag{110}$$

where ϖ is the Hager correction factor for side-weir flow computed at the average values at the upstream and downstream nodes and $f_w\left(\bar{h}, h_M\right)$ is the function defined by the 2-D table for outflow over the weir.

The equation for flow in the opposite direction is

$$f_I\left(h_M, \bar{h}\right) - D_q Q_{swM} = 0, \tag{111}$$

where $f_I\left(h_M, \bar{h}\right)$ is the function defined by the 2-D table for inflow over the weir. No correction for side-weir flow is given in equation 111 because flow over the weir from the diversion channel is perpendicular to the weir.

8.1.2.2 Variable Geometry

For variable-geometry control structures, the relation between flow and water-surface elevation or the elevation of the reference point may change with time. This variation is specified in some cases before simulation. In other cases, the variation is determined during simulation on the basis of conditions detected at nodes within the modeled stream system. Some control-structure options support both modes of operation, but not simultaneously. It is convenient to divide the options on the basis of the number of flow-path end nodes needed to specify the flow.

Variation of the geometry of control structures can take many forms. A control structure on a stream may consist of several devices for regulating the flow of water; for example, turbines, sluice gates, and overflow gates at a single structure. In some cases, each of these devices would have to be modeled separately; however, specification of the rules of operation for all of these devices would be complex and would require that special-purpose programming be added to the software. To avoid this, simpler approaches producing adequate results are applied in FEQ simulation.

Often, the control of the flow in the model needs merely to be similar to what is possible or feasible in the prototype stream. Devices used to regulate the flow do not need to be specifically described in the model. The opening of sluice gates or overflow gates commonly has the same hydraulic effect and attains the same flow-regulation goal. Therefore, the kind of gate opened is a detail that can often be ignored in the analysis without negative effects on the results. For proper simulation, the operational rules must be stated in terms of the objectives to be met, not in terms of rules for specific gates. Given such operational rules, the flows that are possible must be determined in the simulation of the control structure. At least one configuration of gate openings that will match the flow at the control structure computed in the stream model must be possible.

8.1.2.2.1 One-Node Control Structures

To represent one-node control structures of variable geometry, the flow through the control structure is approximated as the product of two functions. The maximum-capacity function, $f_m(z_w)$, gives the maximum flow that can pass the control structure for a given upstream water-surface elevation. This maximum usually results when all gates are at the limiting position to pass maximum flow or when a pump is operating at maximum speed. The maximum-capacity function is the summation of all flows through the control structure at a common value of upstream water-surface elevation. A second function provides the proportion of maximum capacity presently used. This is the opening-fraction function, $p(t)$, with an argument of time and a value that ranges from zero (when no flow results) to 1 (when the flow is at maximum capacity). The flow at the control structure, Q_{cs}, at any time, t, and any upstream water-surface elevation, z_w, is $p(t)f_m(z_w)$.

A simple pump with rate limited by tail water (section 8.1.2.2.1.3) is an exception to the general approach to simulating variable-geometry structures as the product of two functions. For these pumps, the maximum-capacity function is compared to a flow-limit function resulting from tail-water effects, and the smaller flow from the two functions is applied in FEQ simulation.

8.1.2.2.1.1 Opening Fraction Given Beforehand: Code 4, Type 4

In this code, the opening-fraction function must be supplied as part of the input. The control-structure capacity will be varied as a function of time without regard to the flow conditions anywhere in the stream-system model. Consequently, this option should be used only when simulating the prototype system if the actual gate operation is known; for example, in simulation of historic floods. The defining equation is a simple modification of the equation for fixed geometry,

$$Q_q - D_F p\,(t)\, f\left(z_{w_h}\right) = 0\,. \tag{112}$$

8.1.2.2.1.2 Opening Fraction Computed in Full Equations Model: Code 4, Type 5

The applications in which the opening-fraction function for the control structure is known before simulation are limited to calibration of the stream model. Examination of design alternatives or operation-rule evaluations require that the opening fraction be computed in FEQ. This option is identical to code 4, type 4, except that the opening fraction is computed internally according to a set of rules given in an Operation Control Block (section 8.1.2.2.3). These blocks are discussed in sections that follow.

The governing equation for this option is

$$Q_q - pD_F f\left(z_{w_h}\right) = 0, \tag{113}$$

where the opening fraction, p, is not shown as an explicit function because it is computed in FEQ from rules given in the input.

8.1.2.2.1.3 Simple Pump with Rate Limited by Tail Water: Code 4, Type 6

The last variable-geometry one-node control structure consists of a simple pump. The pump capacity is a function of upstream water-surface elevation only, but the pumping rate may be limited by tail-water elevation. Local drainage to a leveed stream often must be pumped into the stream because the levees or the current water level in the stream prevents gravity drainage. The rate of pumping into the stream may be regulated to prevent increases in the flooding. If this is the case, one or more detention ponds will hold the water from local drainage until it can be pumped.

In this option, the tail water is not simulated; rather, it is specified from some other source. The information on the tail water must be specified as a time series giving the tail-water condition used to control the rate of pumping. The tail-water condition can be flow, water-surface elevation, or water-surface height. Let $f_{tw}(t)$ be the function that gives the tail-water condition at any time t, and let Q_p equal the allowed pumping rate. The tail-water-conversion function, $f_{twc}(x_{tw})$, where x_{tw} is the tail-water condition, also must be supplied in the input. The tail-water-conversion function represents the limits on pumping rate resulting from tail-water effects as a maximum flow rate for a given tail-water condition. If the flow capacity for a given water-surface elevation at the head node, $f_m\left(z_{w_h}\right)$, is greater than $f_{twc}[f_{tw}(t)]$, then $Q_p = f_{twc}[f_{tw}(t)]$; otherwise $Q_p = f_m\left(z_{w_h}\right)$. Then, the flow at the discharge node must be equal to the given pumping rate,

$$Q_q - Q_p = 0. \tag{114}$$

8.1.2.2.2 Two-Node Control Structures

Variable-geometry structures whose flow properties are affected by both headwater and tail water (conditions at two flow-path end nodes) are common in river systems. Examples of such structures are sluice gates, variable-height weirs, and low-head spillways. The operation of variable-speed pumps is similar to that of variable-geometry control structures. The basic equations describing flow through these devices are presented in the following sections.

8.1.2.2.2.1 Explicit Two-Dimensional Flow Tables: Code 5, Type 6

Two-dimensional flow tables have already been discussed under “Fixed Geometry” because they are most often applied for that condition. However, two optional input items are provided in FEQ that can vary the geometry as a function of time. The first optional input specifies a 1-D table defining a function of time giving a multiplying factor to apply to all flow values derived from the 2-D table denoted by $p(t)$. This optional input is appropriate only for 2-D tables of type 13 that return a value of flow. The second optional input specifies a 1-D table defining the elevation of the reference point for head as a function of time denoted by $z_h(t)$. The same governing equations are used as for fixed geometry except that the heads are computed with $z_h(t)$ as the reference level and the flow from the table is multiplied by $p(t)$ before it is used in the equation.

Explicit 2-D tables with geometric variations as a function of time are useful for simulating conditions during a flood. The failure of parts of a levee can be approximated by varying the elevation of the reference point for head. The variation can be estimated from reports of the details of the flooding. The flow over the levee will be subject to submergence effects. The computation of the 2-D table must reflect the geometry of the levee and the assumed breach characteristics. Emergency measures during a flood can often involve installation of pumps and the installation of new flow paths, such as openings in levees, the addition of culverts to levees, and so forth. These special features change the stream, and the changes take place during the time period simulated. The function, $p(t)$, allows these features to be placed in the model but keeps them inactive until they have been installed. The status and condition of these special features can be varied by properly defining $p(t)$.

8.1.2.2.2.2 Variable-Height Weir: Code 5, Type 7

Several types of overflow gates can be approximated by a variable-height weir. The characteristics of a particular gate including the length, L_G, and five functions describing certain characteristics (described below) must be specified in this option.

The first function is the gate-position fraction, denoted by $p_G(t)$, where $0 \leq p_G(t) \leq 1$ specifies the gate position. When $p_G(t) = 0$, the gate is fully raised, and the minimum value of flow is simulated for a given upstream condition. When $p_G(t) = 1$, the gate is fully lowered, and the maximum value of flow is simulated for a given upstream condition.

The second function is the gate-crest-elevation function, $f_{gc}(p_G)$, which specifies the elevation of the gate crest for each value of the gate-position fraction. When $p_G = 1$, the gate is at the minimum crest elevation, and when $p_G = 0$, the gate is at the maximum crest elevation.

The third and fourth functions specify the weir coefficients for each gate-position fraction. $C_{ud}(p_G)$ is the weir coefficient when the flow is from the upstream node to the downstream node, and $C_{du}(p_G)$ is the weir coefficient when the flow is from the downstream node to the upstream node.

The fifth function specifies the submergence factor, $f_S(r_h)$, for flow over the gate, where r_h is the ratio of upstream piezometric head to downstream piezometric head.

For low weirs, the importance of the velocity head of the approach flow on discharge over the weir is greatly increased. A user-specified multiplier, K_{vh}, is applied to the velocity head, computed by taking $\alpha = 1$ in the approach channel to account for the increase in the energy head because of converging flow in the immediate vicinity of the weir. Streeter and Wylie (1985, p. 374) state that a multiplier value of about 1.4 is typically assumed in hydraulic engineering.

The governing equation for flow from upstream node to downstream node for the special feature (where upstream and downstream denote the dominant flow direction) is

$$Q_q - D_s C_{ud}(p_G) L \left(d_L + K_{vh} \frac{V_L^2}{2g} \right)^{3/2} f_s\left(\frac{d_L}{d_R} \right) = 0 , \tag{115}$$

where $d_L = z_{w_L} - f_{gc}(p_G)$, $d_R = z_{w_R} - f_{gc}(p_G)$, and V_L is the mean velocity in the approach channel. The value of the gate-position fraction is either specified in a user-defined table or by a user-specified Operation Control Block (described in section 8.1.2.2.3). The governing equation remains the same for either source. For flow in the opposite direction, the governing equation becomes

$$Q_q + D_s C_{du}(p_G) L \left(d_R + K_{vh} \frac{V_R^2}{2g} \right)^{3/2} f_s \left(\frac{d_R}{d_L} \right) = 0 , \tag{116}$$

where V_R is the mean velocity in the approach channel.

8.1.2.2.2.3 Sluice Gates at Stratton Dam at McHenry, Ill.: Code 5, Type 8

FEQ includes a special purpose set of subroutines for simulation of the hydraulic performance of the five sluice gates at Stratton Dam at McHenry, Ill., which are 13.75 ft wide and have a maximum opening of 9 ft. The gates at the dam close on a sill that is 1 ft above the approach and departure aprons. The relations used are a combination of empirical equations fit to measurements; conservation of energy and momentum are both applied to compute submerged-flow values.

8.1.2.2.2.4 Underflow Gates: Code 5, Type 9

Complex problems can result when representing the hydraulic performance of sluice gates. Multiple 2-D tables are used in FEQ simulation to provide maximum generality in hydraulic performance. There are four flow types with sluice gates and the transitions between them are often difficult to define; therefore, engineering judgement in the development of the 2-D tables is critical for obtaining adequate results. The four flow types for sluice gates are given below (Fisk, 1988).

1. Free-orifice (FO) flow, determined from the upstream head only.
2. Submerged-orifice (SO) flow, in which the gate opening is submerged on both sides and the flow is determined from upstream and downstream head.
3. Free-weir (FW) flow, in which the gate lip is free of the water surface and the water flows through the sluice openings. The flow is determined from the upstream head.
4. Submerged-weir (SW) flow, in which the gate lip is free of the water surface and the flow is determined from upstream and downstream heads.

If $f_{qs}(h_L, h_R, w_g)$ is the function defining the flow through the sluice-gate opening, where w_g is the opening distance of the sluice gate, then the governing equation is

$$Q_q - D_s f_q(h_L, h_R, w_g) = 0 . \tag{117}$$

Again, the gate opening is specified by a gate-position fraction given either in an input function of time or in an Operation Control Block (section 8.1.2.2.3).

If w_g is held constant, then $f_q(h_L, h_R, w_g)$ is the same as for a 2-D table in which the flow rate at the discharge node is estimated from piezometric head at two flow-path end nodes (type 13). As shown in the documentation for FEQUTL (Franz and Melching, in press), the variation of flow with gate opening, w_g, is approximately linear. Furthermore, once the gate lip is free of the water, the gate opening no longer affects the flow. Therefore, careful use of linear interpolation on w_g between a series of 2-D tables using piezometric head at two nodes (type 13) will result in a reasonable approximation to a wide variety of sluice gates and other underflow gates (for example, Tainter gates). Other means can be used for preparing a set of 2-D tables of type 13 in the proper format in addition to application of FEQUTL.

8.1.2.2.2.5 Variable-Head Variable-Speed Pump: Code 5, Type 3

A constant-flow pump for all heads is represented with the Bidirectional Flow with Pump option (code 5, type 2). A constant-flow pump is of limited usefulness if the flow varies appreciably with the pumping head. The pump-characteristic curve is used in code 5, type 3 to include the variation of the head that the pump can supply as the flow varies.

Let $f_Q(\Delta h_1)$ equal the flow rate delivered by the pump operating at relative speed, n_r, which is equal to 1, against the head difference across the pump, Δh_1, which is equal to $h_L - h_R$. Relative speed is taken with respect to the standard operating speed of the pump. Thus, setting n_r equal to 1 indicates that the pump is operating at standard speed. For a constant-speed pump, this is the only speed. For a variable-speed pump, however, useful efficiency over a range of speeds is possible, so it is convenient to define the function, f_Q, at the maximum speed of the pump. Then, the maximum relative speed is 1.0, the minimum is 0.0, and the entire speed range is normalized to the interval (0.0, 1.0). In representing the performance of the pump by means of a function, a unique value of flow is assumed to result for every head through the pump. Certain pumps operate outside this assumption, but they are normally operated in a head range such that a unique flow results for each head.

The head change across the pump when the flow is from upstream node to downstream node and the outlet is submerged is

$$\Delta h_1 = f_f(|Q_q|) + K_o\left(\frac{Q_q^2}{2gA_o^2} - \frac{\alpha_R Q_R^2}{2gA_R\left(z_{w_R}\right)^2}\right) + E\left(Q_R, z_{w_R}\right) - E\left(Q_L, z_{w_L}\right), \tag{118}$$

where $f_f(|Q_q|)$ are head losses resulting from separation at the entrance to the pump intake conduit, from flow resistance in the intake conduit, and from the flow resistance in the outlet conduit; K_o is the exit-loss coefficient; A_o is area of pump outlet conduit at the exit; and $E(Q,z_w)$ is elevation of the total energy line for water-surface elevation z_w and flow Q. The equation for free flow is defined such that $K_o = 1.0$ and z_{w_R} is at a level corresponding to free discharge at the pump outlet conduit. The equation for the head difference for flow in the opposite direction is obtained by an appropriate change in subscripts and order of terms. The absolute value appears in the argument to f_f because the argument must always be positive for this function. The flow at the discharge node may be negative depending on the relation of the pumping direction to the upstream and downstream nodes.

The governing equation for the pump then becomes

$$Q_q - D_p D_s n_r f_Q\left(\frac{\Delta h_1}{n_r^2}\right) = 0, \tag{119}$$

where D_p is pumping direction such that $D_p = 1$ denotes flow from upstream node to downstream node and $D_p = -1$ denotes flow from downstream node to upstream node. The head and flow for a variable-speed pump are estimated by use of the similarity relations outlined by Daugherty and Franzini (1977, p. 443–445).

Flow opposite the direction given by D_p is permitted if defined by f_Q; however, the entrance and exit losses will not be properly represented in equation 118. Reverse flow can result whenever the head across the pump is larger than the shutoff head; but normally, pumps have a check valve or flap gate to prevent reverse flow. To represent the condition of no reverse flow, f_Q is set to zero for all heads above the shutoff head for the pump. Negative heads—that is, operation as a turbine—also can be represented if f_Q is defined for negative values of head difference.

The control of the pump is similar to the control of a gate, but differences between the two are important. Pumps are most often constant speed, less commonly multispeed, and seldom variable speed. Efficiency of a variable-speed pump is often low when the pump is operated far from the optimum speed. Therefore, the speed of a pump is often constant or follows a series of discrete values. Furthermore, the pump can operate only when water is present to pump and when the destination node is able to accept the flow. These and other problems are discussed in the next section.

8.1.2.2.3 Operation Control Blocks

The Operation Control Block is a part of the input to FEQ (described in section 13.12) where the user specifies rules for the operation of variable-geometry control structures that refer to such a block. Operational rules in practice become quite complex; often, functions of information external to the flow in the stream system are used. Measured or predicted rainfall, for example, are used in planning the operation of control structures in many stream systems. Current operational rules simulated in FEQ are functions of information computed for the stream-system model. Changes to the software would be required to apply external information. Such changes could be implemented, but they would likely be specific to a particular stream system. The operational rules supplied here have been applied with success in several applications to determine the general nature of how a structure should be operated to obtain maximum flood-control benefits.

It is assumed that each control structure will be affected by one or more control points established at flow-path end nodes in the stream system. Either the flow or the water-surface elevation is monitored at each control point (a node in the stream system). A recommended action is determined on the basis of the value at the control point. If different actions are determined from values at different control points, the action simulated is determined on the basis of priorities assigned to each control point by the user. At the start of each time step, the control points are checked in the model simulation for each control structure to determine the appropriate action in changing the control-structure setting. Once the action is selected, the control-structure setting is changed and held constant throughout the following time step. No changes in control-structure setting are made during the computations for the time step. Changes in settings can be completed only between time steps.

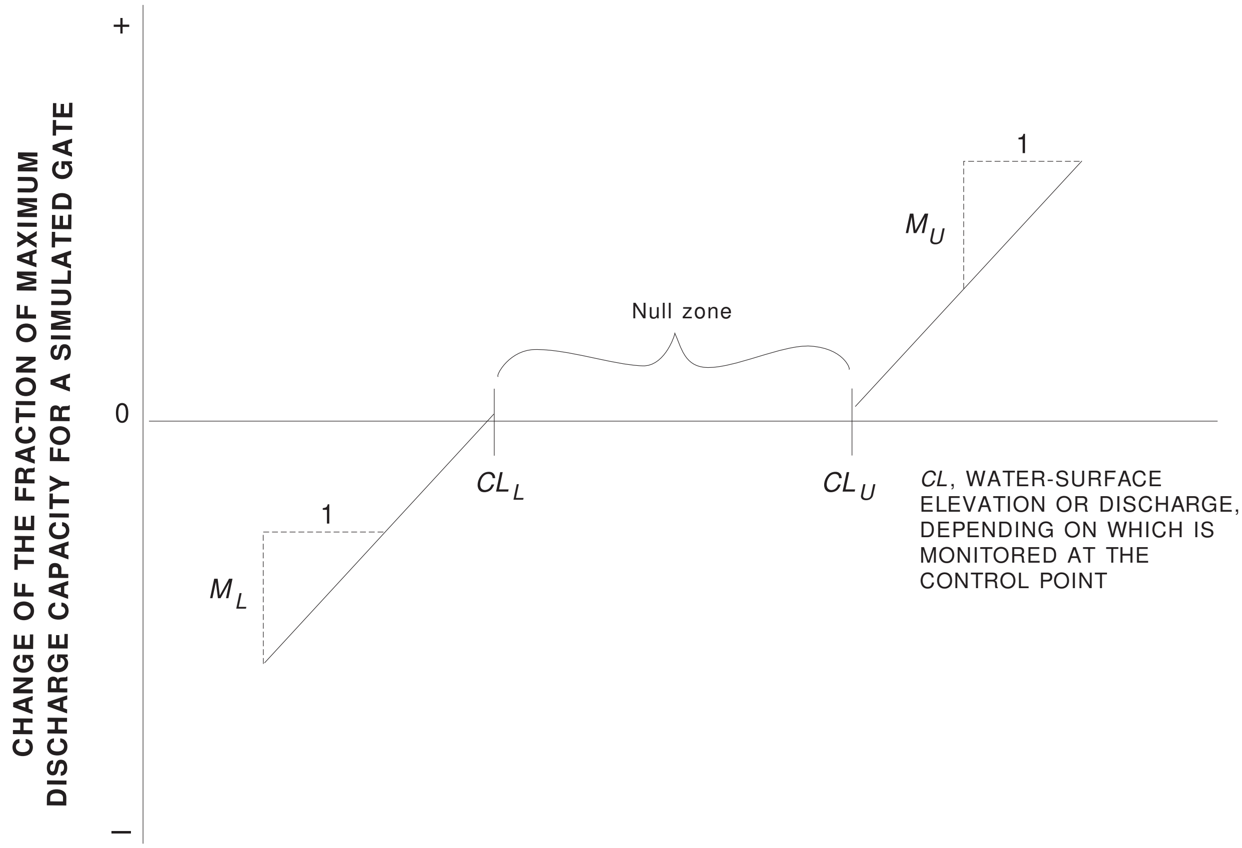
The control-point information must be transformed into a change in the setting for the control structure. The nature of this transformation depends on the type of control structure. If gates are operated, then the setting of the gates has a continuous range, and small changes in gate setting are appropriate. If pump operations are simulated, however, small changes in setting may not be appropriate. The approach taken for gates is to specify the rate of change of the gate-position fraction directly from information at the control point. For pumps, the setting is specified directly with user-defined functions.

8.1.2.2.3.1 Gate Control

The approach applied in FEQ for gates is that a rate of change in the gate-opening fraction, p, is specified directly from information at the control point or points. This rate of change, $\dot{p}$, is then multiplied by the length of the time step to estimate the change in p to apply at the start of that time step. The form of the control function, illustrated in figure 16, consists of three regions: the region below the null zone, the null zone, and the region above the null zone. The lower limit of the null zone is denoted by CL_L, and the upper limit of the null zone is denoted by CL_U. The slope of the line in the region below the null zone is given by M_L, and the slope for the line in the region above the null zone is given by M_U. The values of CL_L, CL_U, M_L, and M_U are specified by the user (section 13.12). The value of p is unchanged if the stream level is in the null zone. Outside the null zone, the rate of change of p, $\dot{p}$, is proportional to the deviation of the discharge or water-surface elevation (depending on which is monitored at the control point) from the closest null-zone boundary.

The null zone is needed to prevent numerous adjustments of gate settings even though the water level at the control point is reasonably stable. If the null zone is too small and the conditions are changing rapidly, adjustments of the gate setting will still be numerous. Other constraints must be applied to the rate at which the setting is changed to make the operation sensible and stable.

If the gate setting is always changed when CL is outside the null zone, the setting may oscillate between fully open and fully closed in model simulation. Oscillations result when a long time is required for the effect of the gate change to be detected at the control point. Therefore, the user also must specify the minimum acceptable rate of movement toward the null zone to avoid frequent changes to the setting. Let $\dot{CL}_{min}$ denote the minimum rate of water-surface elevation or discharge movement (depending on which is monitored) at the control point toward the null zone for which the gate setting remains the same, and $\dot{CL} = (CL_t - CL_{t-1})/\Delta t$ where CL_t is the



EXPLANATION

CL_L LOWER LIMIT OF NULL ZONE DENOTING UNCHANGED GATE PLACEMENT

CL_U UPPER LIMIT OF NULL ZONE DENOTING UNCHANGED GATE PLACEMENT

M_L RATE OF CHANGE OF THE FRACTION OF MAXIMUM DISCHARGE CAPACITY FOR A SIMULATED GATE WITH THE WATER-SURFACE ELEVATION OR DISCHARGE AT THE CONTROL POINT BELOW THE NULL ZONE

M_U RATE OF CHANGE OF THE FRACTION OF MAXIMUM DISCHARGE CAPACITY FOR A SIMULATED GATE WITH THE WATER-SURFACE ELEVATION OR DISCHARGE OF THE CONTROL POINT ABOVE THE NULL ZONE

Figure 16. Example of a typical gate-control function at a control point, as simulated in the Full EQuations model.

level (water-surface elevation or discharge) at the current time t and CL_{t-1} is the level at a time one time step (Δt) before the current time.

Limiting the rate of change for the setting also is useful to help prevent erratic setting changes when conditions at the control points change rapidly. The absolute value of the maximum rate of change is given by $\dot{p}_{max}$.

Finally, the user must establish the relative priority of the action determined on the basis of the level at each control point when the level is in each of the three regions of the control function. When the level is below the null zone, the priority is PR_L; when in the null zone, PR_N; and when above the null zone, PR_H. The priorities are in a simple ordinal relation; that is, priority level 1 is higher than 2, but how much higher is of no concern. The action finally simulated is the action with the highest priority across all control points connected to the control structure.

For a given level at time t, CL_t, at a control point, the processing takes the following general series of steps:

If $CL_t < CL_L$ (the level at the control point is below the null zone), then:

1. Check for rate of movement in the correct direction. If $CL_t < CL_{t-1} + \Delta t \dot{C}L_{min}$, then $\dot{p} = \frac{M_L(CL_t - CL_L)}{\Delta t}$; otherwise, $\dot{p} = 0$.
2. Check for the rate of change of the setting. If $|\dot{p}| > \dot{p}_{max}$, then $\dot{p} = sign(\dot{p})\dot{p}_{max}$; otherwise, $\dot{p} = \dot{p}$.
3. Set the priority, PR_L, for this control point.

If $CL_t > CL_U$ (the level at the control point is above the null zone), then:

1. Check for rate of movement in the correct direction. If $CL_t < CL_{t-1} - \Delta t\ \dot{C}L_{min}$, then $\dot{p} = \frac{M_U(CL - CL_U)}{\Delta t}$; otherwise, $\dot{p} = 0$.
2. Check for the rate of change of the setting. If $|\dot{p}| > \dot{p}_{max}$, then $\dot{p} = sign(\dot{p})\dot{p}_{max}$; otherwise, $\dot{p} = \dot{p}$.
3. Set the priority, PR_H, for this control point.

If $CL_L \le CL_t \le CL_U$ (the level at the control point is in the null zone), then:

1. Specify no change to the setting, $\dot{p} = 0$.
2. Set the priority, PR_N, for this control point.

The action is determined from the control point with the top priority. The equation for the setting change is

$$p_t = p_{t-1} + \dot{p}\Delta t, \tag{120}$$

where

p_t is the new value of setting,
p_{t-1} is the old value of setting, and
Δt is the time step, in hours.

The new setting must always satisfy $0 \le p_t \le 1$.

The determination of the null zones and the priorities depends on the nature of the application. It is common to have levels at one control point indicate an increasing gate opening and levels at another indicate a decreasing gate opening. The assignment of priorities resolves this discrepancy. In general, gate settings should be changed slowly. Rapid changes can result in oscillations in the solution scheme and inefficient gate operation. Furthermore, because the gate setting could potentially be changed every time step, the time step used should be representative of the gates and the objective of the operation and simulation. In some cases, controlling the time step becomes difficult. In certain applications, the complexity of the operational rules may have to be increased so that the gate setting is changed only after sufficient time has elapsed since the previous change.

8.1.2.2.3.2 Pump Control

The speed of pumps is set with user-defined control functions in model simulations. The same code can be used to represent constant-speed, multispeed, or variable-speed pumps. Options are supplied for designing the controls of the pump. For example, the direction of movement of the level at the control point under the gate-control option (section 8.1.2.2.3.1) does not change the control function, only its application. For pumps, however, the direction of movement of the level may be used to specify a different control function for the pump.

A control function for a pump, f_{cp}, is given explicitly in a 1-D table; it is not defined by the slopes above and below a null zone as for gate control. The pump-control function follows several rules:

1. The argument of the control function, CL_{cp}, must be the level that is controlled, either flow or water-surface elevation.

2. The value determined from the control function, n_s, must be a speed in the same units as the speed for the pump so that n_s is equal to $f_{cp}\,(CL_{cp})$.
3. If $f_{cp}\,(CL_{cp}) < 0$, where CL_{cp} is the level (flow or water-surface elevation) monitored at the control point, then the pump is turned off (if it is on; otherwise, it remains off).
4. If $f_{cp}\,(CL_{cp}) = 0$, then the level is in the null zone and the pump state is not changed. If it is off, then it remains off; and if it is on, it remains on and at the current speed.
5. If $f_{cp}\,(CL_{cp}) > 0$, then the pump is turned on if it is off and the speed is set to the value of $f_{cp}\,(CL_{cp})$. If the pump is on, the speed is set to the value of $f_{cp}\,(CL_{cp})$.

This function, $f_{cp}\,(CL_{cp})$, is similar to the control function for a gate but has a more flexible form. The null zone is that range of the function argument for which the function is zero. The zone where the pump is off will have negative function values. The magnitude of the function values in this zone is not important. The zone where the pump is on will define the pump speed at each level in the pump-operation zone.

The example in figure 17 illustrates the case where the level of the water-surface elevation at the control point indicates turning a pump on when water is present and when the destination node is able to accept the water. The water must reach the level where the function becomes positive before the pump is turned on. Once on, the pump is not turned off until the water-surface elevation falls below the level at which the control function becomes negative. The null zone between the levels of turning on and turning off the pump must be used to avoid simulating a continuous sequence of pump cycling. The null zone must be large enough so that the action of turning the pump

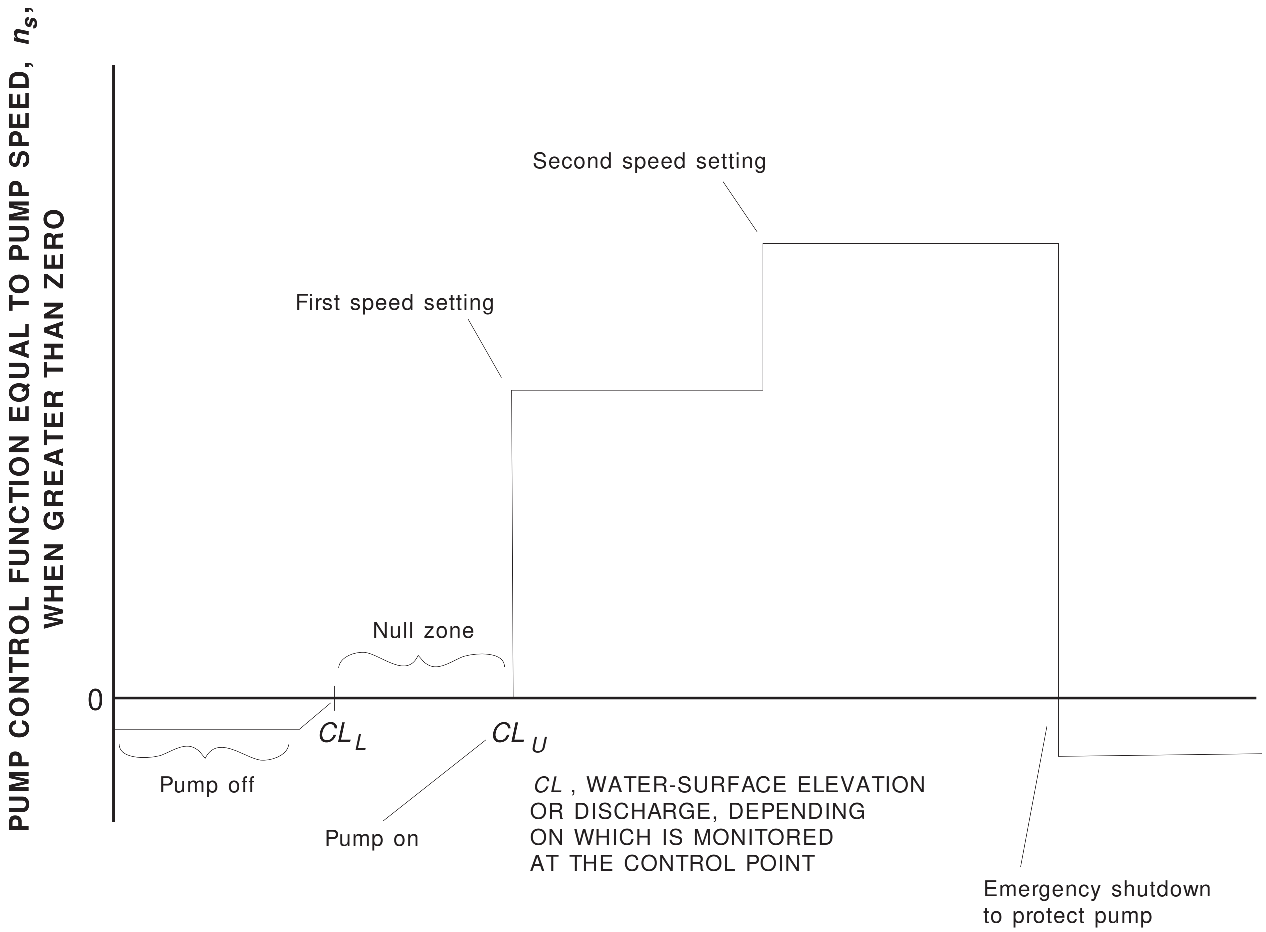


Figure 17. Example of a typical pump-control function at a control point, as simulated in the Full EQuations model.

on does not drop the water-surface elevation to the level for turning the pump off at the end of the next time step. Therefore, the size of the null zone must be selected with consideration of the characteristics of the pump and the source node.

The user has the option of assigning various control functions, depending on the direction of movement of the level sensed. However, a null zone with regard to changes in direction of movement also must be present. If a directional null zone is not present, then endless cycling between off and on could easily result. The directional null zone is given by $\dot{CL}_{min}$ and is defined as for operation of a gate. If δt_s is the time elapsed since the setting was changed and $|CL_t - CL_{t-1}| \le \delta t_s \dot{CL}_{min}$, then the direction of motion will remain unchanged for the selection of the control function to use. If the directional null zone is too small, the action of changing the pump setting may result in a change back to the other control function at the earliest opportunity. Such changes could result in unrealistic variation in pump speed and computational failure.

The priority scheme for gate operation also applies to pumps. Conflicting specifications for pump operation must be resolved because the destination node may not be able to accept the flow at the time when water-surface elevation at the source node indicates that the pump should be turned on. The rules for turning the pump on or off may be set on the basis of the purpose of the pump in the priority scheme. In some cases, turning the pump off may have top priority. For example, in an off-channel flood-control reservoir, the pump should be off either if there is no water to pump or if the stream is still at or near flood stage. Thus, the off regions for the two control points should have a priority rank greater than the on or null regions. Various rules can be devised by a careful selection of the priority rank.

8.2 External Boundary Conditions

External boundary conditions must be given at every flow-path end node not connected to a special feature. Three external boundary conditions are possible: flow as a function of water-surface elevation, flow as a function of time, and water-surface elevation as a function of time.

8.2.1 Flow as Function of Water-Surface Elevation: Code 4 and Code 8

Any one-node control structure can serve as an external boundary condition. These structures express the flow as some function of the water-surface elevation, as has already been discussed in section 8.1. The only difference at an external boundary is that the node providing the water-surface elevation (the head node) and the node providing the flow (the discharge node) must be the same. At an internal boundary, they can be different or the same, depending on the situation. The previous equations for internal boundaries apply with this difference considered.

Cunge and others (1980, p. 36) note that flow as a function of water-surface elevation cannot be used as an external upstream boundary, because the flow would increase without bound. This results because an increase in flow at an external upstream boundary will cause an increase in stage, and this increase in stage will cause an increase in flow from the one-node control-structure relation. Therefore, this boundary condition can only be used as an external downstream boundary.

8.2.2 Flow as Function of Time: Code 6, Type 1

Flow as a function of time at an external node is denoted by $f_{qb}(t)$, where subscript qb denotes the external node. The governing equation for flow as a function of time is

$$Q_{qb} - D_D f_{qb}(t) = 0, \tag{121}$$

where $D_D = 1$ if positive values from f_{qb} enter the stream system at this external boundary node and $D_D = -1$ if positive values from f_{qb} leave the stream system at this external boundary node. This specification of flow at the node will be met for all conditions in the modeled stream system.

Improper use of a boundary of this type can result in gross errors. Typically, flow as a function of time is given at the upstream (inflow) node of a flow path as an external boundary. This implies that downstream conditions do not affect flow at that node. If an effect from downstream is evident, then the boundary point must be moved farther upstream to a region outside the effect. If the boundary point is not moved, a value of flow will be forced in simulation at the external upstream boundary that could not occur at that point.

As outlined by Cunge and others (1980, p. 36), this boundary condition at the downstream end of a stream system must be applied with care. First, the imposed flow may exceed the capacity of the channel to deliver water to that node. Second, if flow is specified as a function of time at both the external upstream and downstream boundaries for a model simulation, then any errors in the values of these flows will be reflected in the water levels. In some cases, the stream system may be partially or completely dewatered because of these errors.

8.2.3 Water-Surface Elevation as Function of Time: Code 6, Type 2

If the flow at the external boundary node is subcritical, then the governing equation for water-surface elevation as a function of time is

$$z_{w_b} - f_z(t) = 0, \tag{122}$$

where z_{w_b} is the elevation of the water surface at the external boundary node and $f_z(t)$ is the water-surface elevation imposed at the external boundary node. The imposed water-surface elevation may result in a depth of flow that is too small for the flow to remain subcritical at the external boundary. This condition is checked for in FEQ, and critical flow is forced in the computations to prevent supercritical flow at the external boundary. The state of the flow at the external boundary is maintained in simulation as for code 5, type 1 (expansion), discussed previously. If flow at the external boundary is critical, then the control is drowned whenever the imposed water-surface elevation exceeds the water-surface elevation at the node at critical flow. In the computation of a critical flow, it is assumed that $\alpha = \beta = 1$; however, if the cross-section table at the external boundary node contains tabulated values of critical flow (table types 22 and 25, see section 11.1.5), then the tabulated value of critical flow is used.

8.3 Initial Conditions

An initial value of flow and water-surface elevation must be known at every node in the stream model before the unsteady-flow computations can begin. These initial values are provided through a steady-flow water-surface profile computation. A steady-flow analog of the unsteady-flow governing equations for branches is used in the steady-flow water-surface profile computations. Most control structures are not represented in these computations. An estimate of the initial conditions is obtained from the steady-flow computations so that the unsteady-flow computations can start. Consequently, an option is provided for holding the boundary conditions and the simulation time constant while initial values of water-surface height are computed. This option is called frozen time. If the frozen-time option is selected by the user, the changes in water-surface height and flow that would take place over a time step are computed, but then the simulation time is reset to the starting time. Normally, only a few frozen time steps must be computed to dissipate transient conditions resulting from the change from steady flow to unsteady flow. If the frozen-time option is selected, a maximum of nine frozen time steps will be computed. In most cases, nine frozen time steps will suffice to reduce computational transitions; however, for simulations of tidally affected flows or streams with many control structures, the computational transients can be particularly strong. In these cases, boundary conditions may have to be held constant (using a hypothetical period of constant conditions) during the start of the unsteady-flow computations (after frozen time) for a period long enough to dissipate the transients induced by the approximate initial condition.

The equations for the steady-flow analysis are not presented here because they are a special case of the governing equations for unsteady flow. A subcritical solution to the governing equations is sought, but (as outlined in section 2.2) a subcritical solution may not exist. If a subcritical solution cannot be computed, then the iterative solution procedure will fail and computations will stop in FEQ. The user must then determine why supercritical flow results. Is the distance step too long or is the slope too steep for subcritical flow? Details of this determination are given in section 13.

9. MATRIX SOLUTION AND NUMERICAL PROPERTIES OF THE FINITE-DIFFERENCE SCHEME

The equations previously developed in sections 6, 7, and 8 describe the flow and water-surface height in a network of open channels. The nature of the stream system and the choices made by the user determine which of the equations are included in a model stream system. Once selected, these equations must be solved simultaneously. The initial conditions define the flows and water-surface heights at some starting time. The unknowns are the flows and water-surface heights at the end of the next time step. At the end of the computations for this time step, the unknowns become the initial values for the following time step. Thus, the equations of motion are solved many times. To be useful, a computer program for simulation of unsteady flow in a network of channels must include an efficient means for solving a system of nonlinear equations.

9.1 Newton's Iteration Method for Solution of Nonlinear Equations

In general, even a single nonlinear equation cannot be solved without some numerical method to approximate the solution to the equation. The example of a single equation illustrates some of the problems that are considered in FEQ simulation. Thus, Newton's iteration method for solution of nonlinear equations is initially described and illustrated for the case of a single nonlinear equation. The discussion of Newton's method is then expanded to the simultaneous solution of many equations.

9.1.1 Application to a Single Equation

If a direct solution to the nonlinear equation is not possible, then some indirect approach must be applied. For a single nonlinear equation, this is done by approximating the residual function over an interval by another simpler function that can be solved directly and easily. The residual function is simply the equation describing the process of interest, organized such that the sum of the relevant factors equals zero. Thus, the residual function of the continuity equation is given by equation 68, and the residual function of the conservation of momentum equation is given by equation 77. If the approximation of the residual function is close, then the solution to the simpler equation will be a close approximation to the solution of the nonlinear equation. In Newton's method, the approximating function is the line tangent to the residual function, F, at some point, $\hat{u}_o$, where $\hat{u}_o$ is close to the location of a root. Let $F_N'(u)$ denote the derivative of the function F_N at any point u. Expanding F_N in a Taylor series about $\hat{u}_o$ and discarding nonlinear terms yields

$$\hat{F}_N(\hat{u}_o) = F_N(\hat{u}_o) + F_N'(\hat{u}_o)(u - \hat{u}_o) , \tag{123}$$

where $\hat{F}_N(\hat{u}_o)$ is the value on the line tangent to F_N, the point of tangency being $\hat{u}_o$. Solving equation 124 for u where $\hat{F}_N(u) = 0$ yields the root for the tangent line,

$$\hat{u}_1 = \hat{u}_0 - \frac{F_N(\hat{u}_0)}{F_N'(\hat{u}_0)} , \tag{124}$$

where $\hat{u}_1$ is the root of the tangent line, which becomes the next approximation to a root of the equation. This process can be repeated until the approximations to the root approach some limit or until the value of the function F_N becomes acceptably small. Equation 124, rewritten to show this process, becomes

$$\hat{u}_{i+1} = \hat{u}_i - \frac{F_N(\hat{u}_i)}{F_N'(\hat{u}_i)}, \tag{125}$$

where i=0, 1, 2, . . . Various applications of Newton's method are shown in figure 18.

Cases that can result in problems in the convergence of Newton's method are shown in figure 18b–d. In the first case, the root is near a point of inflection so that the iterations oscillate and convergence is impossible. In the second case no root is possible: Newton's method will not converge because there is no root to converge to. In the third, the derivative is zero at a root, so the method may not converge or will converge slowly. Hamming (1973, p. 70–72) and Dennis and Schnabel (1983, p. 21–23) discuss means for detecting these problems for a single equation.

9.1.2 Application to Simultaneous Equations

The single-equation Newton's method forms the basis for the method extended to simultaneous nonlinear equations. Instead of a curve with a tangent line, Newton's method for simultaneous equations is based on an n_e-dimensional plane tangent to an n_e-dimensional surface, where n_e is the number of equations. At each iteration, instead of solving a single equation in a single unknown, a linear system of n_e equations in n_e unknowns must be solved. Let $F_k(\boldsymbol{u})$ be the residual function of the kth equation in the system, where $\boldsymbol{u}$ is the vector of unknown values. For a stream system simulated with FEQ, the residual function for the continuity equation is given by equation 68, conservation of momentum equation is given by equation 77, and external boundary conditions are given by equations 91, 121, and 122. To keep the notation compact, let F_{kj}' denote the partial derivative of F_k with respect to the jth variable in the vector of unknowns, $\boldsymbol{u}$. The steps followed are analogous to those for a single variable. The equation for the approximation to the functions is determined about some initial point, and the root of the approximation as an estimate of the root of the equation is then determined.

As an example of the application of Newton's method, consider two equations and two unknowns. To distinguish the iteration number from the number used to denote the variable, let a superscript denote the iteration number, and not an exponent. If there are exponents, they must be set off with parentheses. For example, u_1^0 denotes the initial value for the first unknown in the vector $\boldsymbol{u}$, u_1^2 denotes the second iteration as it affects the first unknown, and $\left(u_2^2\right)^2$ denotes the square of the value of the second unknown at the second iteration. Discarding all nonlinear terms from a Taylor series expansion about the point $\boldsymbol{u}^0$ yields two equations:

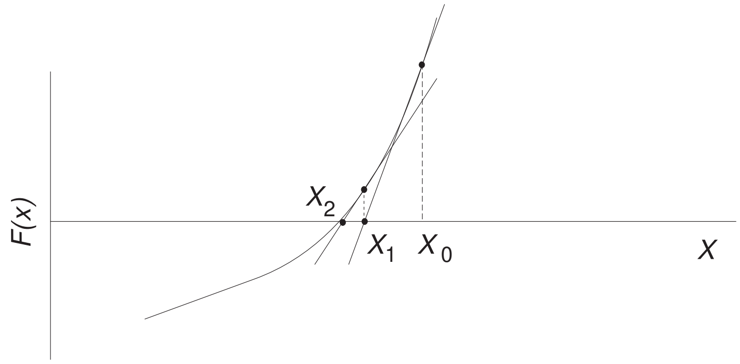
$$\hat{F}_1(u_1, u_2) = F_1\left(u_1^0, u_2^0\right) + \left(u_1 - u_1^0\right)F'_{11} + \left(u_2 - u_2^0\right)F'_{12}\left(u_1^0, u_2^0\right) \tag{126}$$

and

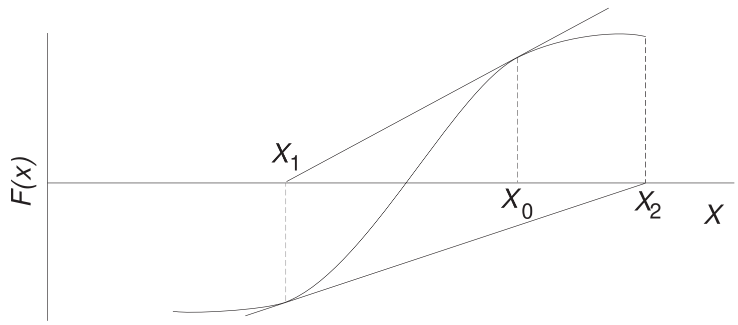
$$\hat{F}_2(u_1, u_2) = F_2\left(u_1^0, u_2^0\right) + \left(u_1 - u_1^0\right)F'_{21}\left(u_1^0, u_2^0\right) + \left(u_2 - u_2^0\right)F'_{22}\left(u_1^0, u_2^0\right). \tag{127}$$

Setting each of these equations to zero and solving for the unknowns gives the first-iteration results for the roots of F_1 and F_2 as for a single equation. Two linear equations in two unknowns result. However, the notation becomes unmanageable as the number of equations increases. FEQ simulation can involve several thousand equations and

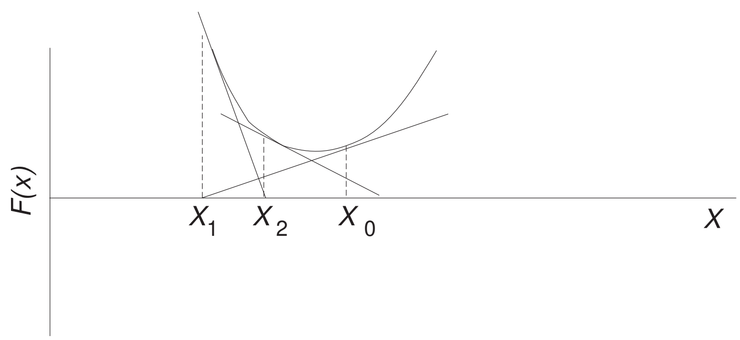
A.



B.



C.



D.

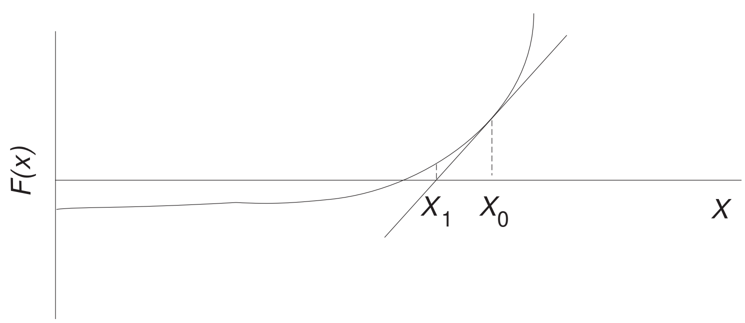


Figure 18. Examples of Newton's method of iterative solution for the value of root, *X*, that results in a function, *F*(*X*), value equal to zero (A) under favorable conditions for convergence to the desired root, (B) near point of inflection, (C) with no root, and (D) with root near zero slope.

as many unknowns. Matrix notation makes for a more compact and more general display of the equations and the solution process. Equation 123 in matrix notation is

$$\hat{\boldsymbol{F}}(\boldsymbol{u}) = \boldsymbol{F}\left(\boldsymbol{u}^0\right) + \boldsymbol{J}\left(\boldsymbol{u}^0\right)\left(\boldsymbol{u} - \boldsymbol{u}^0\right), \tag{128}$$

where $\boldsymbol{u}$ is the vector of values on the tangent planes, $\boldsymbol{F}(\boldsymbol{u})$ is the vector of values of the residual functions for the equations, and $\boldsymbol{J}(\boldsymbol{u})$ is the matrix of partial derivatives of the residual functions, called the Jacobian matrix. If $n_e = 2$, then

$$\hat{\boldsymbol{F}} = \begin{bmatrix} \hat{F}_1 \\ \hat{F}_2 \end{bmatrix}, \tag{129}$$

$$\boldsymbol{F} = \begin{bmatrix} F_1 \\ F_2 \end{bmatrix}, \tag{130}$$

and

$$\boldsymbol{J} = \begin{bmatrix} F'_{11} & F'_{12} \\ F'_{21} & F'_{22} \end{bmatrix}. \tag{131}$$

In equation 128, the argument list following a vector or matrix symbol is applied to each element of the vector or matrix.

Letting $\hat{\boldsymbol{F}}(\boldsymbol{u}) = 0$ and solving for the approximation to the root results in

$$\Delta\boldsymbol{u}^i = \boldsymbol{u}^i - \boldsymbol{u}^{i-1} = -\boldsymbol{J}^{-1}\left(\boldsymbol{u}^{i-1}\right)\boldsymbol{F}\left(\boldsymbol{u}^{i-1}\right), \tag{132}$$

as the final form of the iteration for a simultaneous system of nonlinear equations by means of Newton's method. In equation 132, $\boldsymbol{J}^{-1}$ denotes the inverse matrix for the Jacobian of the system of equations. Dennis and Schnabel (1983, p. 21–23) discuss the conditions for convergence of Newton's method for a system of nonlinear equations. The convergence is quadratic if the first derivatives are sufficiently smooth and the initial point is not too far from one of the roots of the equations.

Obviously, the solution of a system of nonlinear equations is much more complex than for a single equation. Similar convergence problems may result, but now no simple geometric visualization is possible. Furthermore, Dennis and Schnabel (1983, p. 9) point out that problems involving 50 or more equations are difficult to solve unless a good estimate is available before iteration. However, this is only true for systems in which the Jacobian matrix is filled, or nearly so, with nonzero elements. In one-dimensional, unsteady-flow analysis, the Jacobian has a special structure and contains many zeros. For large stream systems, less than 1 percent of the elements in the Jacobian will be nonzero; all other elements are known in advance to be zero. This structure is used in FEQ simulation to reduce the complexity and number of computations.

9.2 Solution of a Sparse, Banded Matrix

Repeated solution of the linear equation system by Newton's method results in a sequence of corrections that decrease to an acceptable value. Criteria are needed to determine when the corrections are small enough to end the iterations. A variety of problems may arise in the search for a solution and techniques for improving the convergence to a solution, so an evaluation of the solutions to these problems is needed.

The coefficient matrix for Newton's method for the system of nonlinear equations describing a stream network is banded, but the bandwidth is not constant. The equations for a branch have a constant bandwidth, but the algebraic relations for the special features add variability to the bandwidth at every junction (see section 10.3). The computation time for the solution of the linear equation system produced with Newton's method is reduced to a feasible value in FEQ by taking advantage of the band structure. A direct method based on the Crout variant of Gaussian elimination is used in FEQ simulation. The method as outlined here is derived from Zienkiewicz and Taylor (1989, p. 479–483).

The system of equations to be solved is

$$\boldsymbol{Ju} = \boldsymbol{b}\,, \tag{133}$$

where

- $\boldsymbol{J}$ is a Jacobian matrix of coefficients for the linear system,
- $\boldsymbol{u}$ is the vector of unknowns, and
- $\boldsymbol{b}$ is the right-hand-side vector of residuals.

To solve this system efficiently, let $\boldsymbol{J} = \boldsymbol{LU}$, where $\boldsymbol{L}$ is a lower triangular matrix and $\boldsymbol{U}$ is an upper triangular matrix. A triangular matrix is a square matrix in which all elements above or below the main diagonal are zero. Requiring the values along the main diagonal of $\boldsymbol{L}$ to be unity makes the two triangular matrices unique. To solve the system of linear equations, let $\boldsymbol{Uu} = \boldsymbol{w}$. The unknowns are then found by first computing $\boldsymbol{w}$ by use of the process of forward elimination. Once $\boldsymbol{w}$ is known, the next step, back substitution, solves for $\boldsymbol{u}$ as follows:

$$w_1 = b_1$$

$$\left(w_i = b_i - \sum_{j=1}^{i-1} L_{ij} w_j, \qquad i = 2, 3, \ldots, n_e \right) \tag{134}$$

$$u_{n_e} = w_{n_e} / U_{n_e n_e}$$

$$u_i = \left(w_i - \sum_{j=i+1}^{n_e} U_{ij} u_j \right) / U_{ii}, \quad i = n_e - 1, n_e - 2, \ldots, 1\,. \tag{135}$$

The factoring of the matrix can be done in place; that is, the elements of the coefficient matrix are replaced by the factor of $\boldsymbol{L}$ or $\boldsymbol{U}$ as computed because the diagonal elements of $\boldsymbol{L}$ and the parts of $\boldsymbol{L}$ and $\boldsymbol{U}$ known to be zero are not stored. In-place factoring is assumed in the following equations. Thus, the same subscripts on any of $\boldsymbol{J}$, $\boldsymbol{L}$, or $\boldsymbol{U}$ refer to the same physical location in the stream system. The value that is needed will be at the location when reached. A row of $\boldsymbol{L}$ to the left of the diagonal (the diagonal element is 1.0 by definition) is calculated in the computations, followed by the column of $\boldsymbol{U}$ at and above the main diagonal. This is done for each row of $\boldsymbol{L}$ and column of $\boldsymbol{U}$ until all are computed; that is, for $j = 2, 3, \ldots, n_e$,

$$L_{ji} = \left(J_{ji} - \sum_{m=1}^{i-1} L_{jm} U_{mi} \right) / U_{ii}, \quad i = 1, 2, \ldots, j-1$$

$$U_{ij} = J_{ij} - \sum_{m=1}^{i-1} L_{im} J_{mj} . \tag{136}$$

The best way to visualize these operations is to partition the coefficient matrix into three parts: the reduced zone where **L** and **U** are already computed, the active zone that is currently computed, and the unreduced zone of the coefficient matrix. These zones are shown in figure 19.

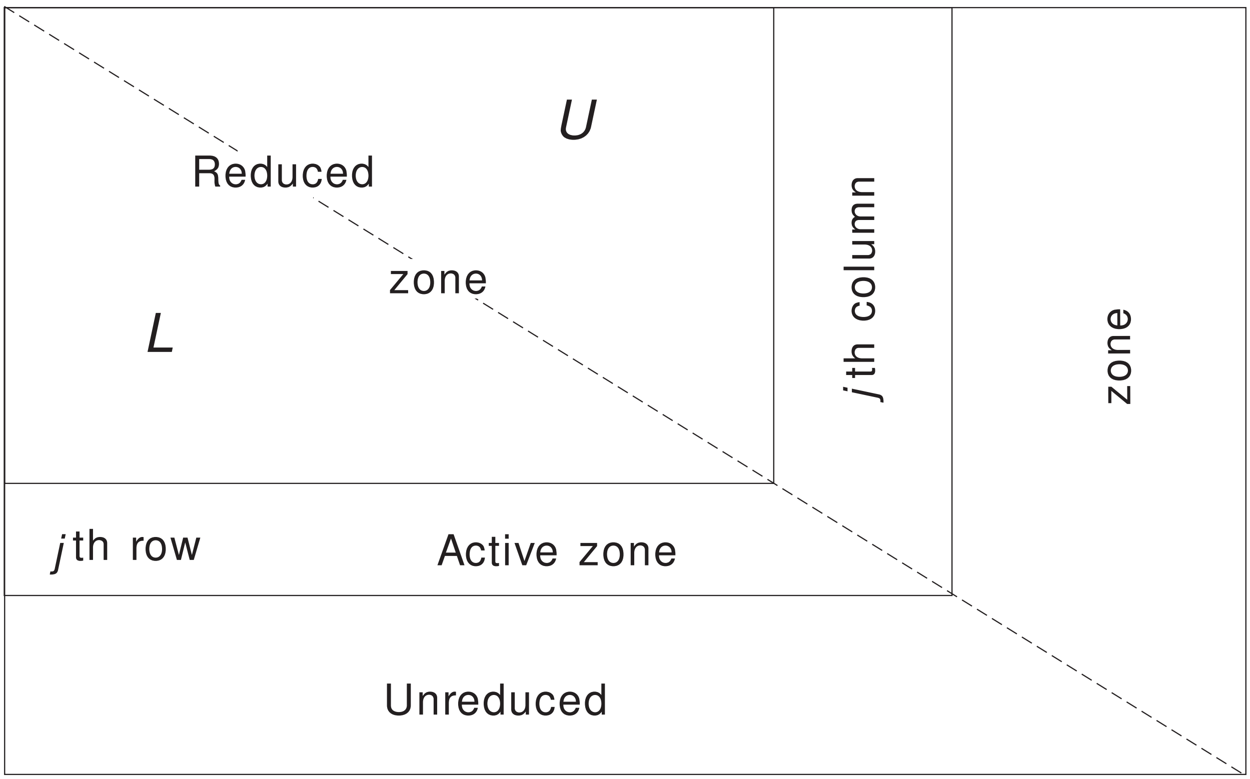


Figure 19. Working zones for lower (*L*) and upper (*U*) factoring of the coefficient matrix in Full EQuations model simulation.

These equations are written with the assumption that all the equations are stored with all the zero and nonzero coefficients. However, the vector inner product for the element of **L** in the *j*th row and the *i*th column involves only elements to the left of the *i*th column in the *j*th row and in the *i*th column at and above the main diagonal, as shown in figure 20. The pattern for the vector inner product for the element of **U** in the *i*th row of the *j*th column includes only elements to the right of the main diagonal in the *i*th row and above the *i*th row in the *j*th column, as shown in figure 21. The banded structure of the Jacobian matrix as developed previously is preserved with this pattern. This preservation of the banded structure results only if no row or column interchanges are made to prevent a zero divide in equation 136. The zeros within the band limits may become nonzero, but the zeros outside the band limits must remain as zeros. Therefore, both zero and nonzero coefficients, within the band as shown previously, are stored.

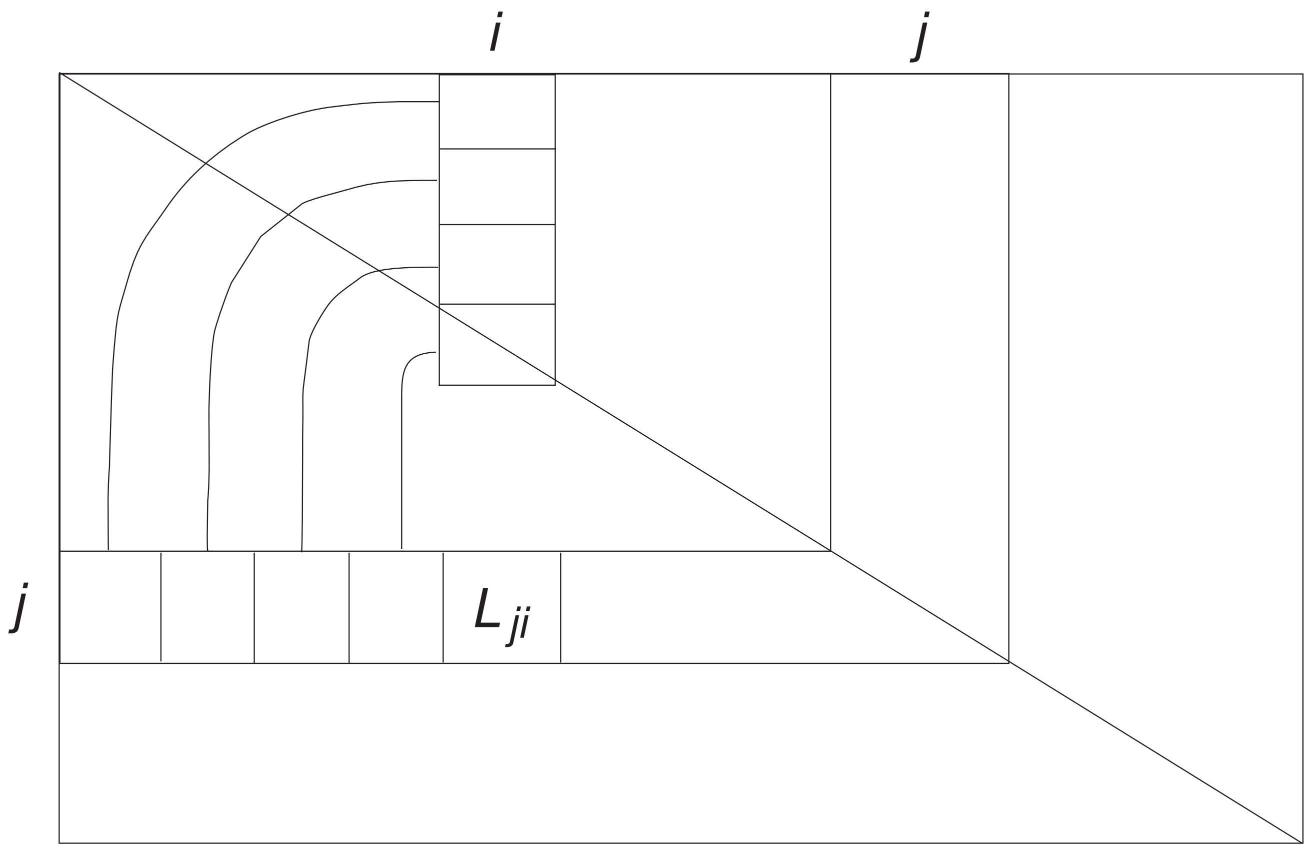


Figure 20. Typical vector inner product for row of the lower diagonal matrix in Full EQuations model simulation.

The band of the Jacobian matrix has constant width and pattern within branches. The junctions cause the band of the Jacobian matrix to be of variable width (section 10.3). Therefore, if enough nodes are in a branch, explicit computation of the vector inner products without iteration is worthwhile; that is, the pattern of the summation is constant for each element, so the summation is written explicitly. This technique is known as loop unrolling in algorithm design. Two block types are available in FEQ computations: a variable-band block and a fixed-band block for the branches. Use of loop unrolling greatly reduces solution time for factoring the matrix so that a fixed-band matrix can be factored with the variable-band algorithm at nearly the same speed as for a fixed-band algorithm.

9.3 Stopping Criteria for Newton's Method

The Jacobian matrix is solved by use of a direct method based on a variant of Gaussian elimination, as described in the previous section, to compute the successive corrections to the estimated unknowns. Criteria are needed to determine when to stop the iteration. Selection of stopping criteria is difficult, as indicated by Hamming (1973, p. 68–70) and Dennis and Schnabel (1983, p. 159–161), because determining an acceptable difference between iterations is complicated. Under good conditions, the corrections from Newton's method should decrease rapidly. This usually results when the conditions for convergence of the method are met. In other cases, however, the corrections may not decrease or may decrease slowly, so alternative actions must be taken in applying FEQ to successfully complete the computations.

Two forms of convergence criteria, relative and absolute, can be used. A relative criterion involves the size of the correction or the size of the residual function relative to some other quantity. An absolute criterion directly

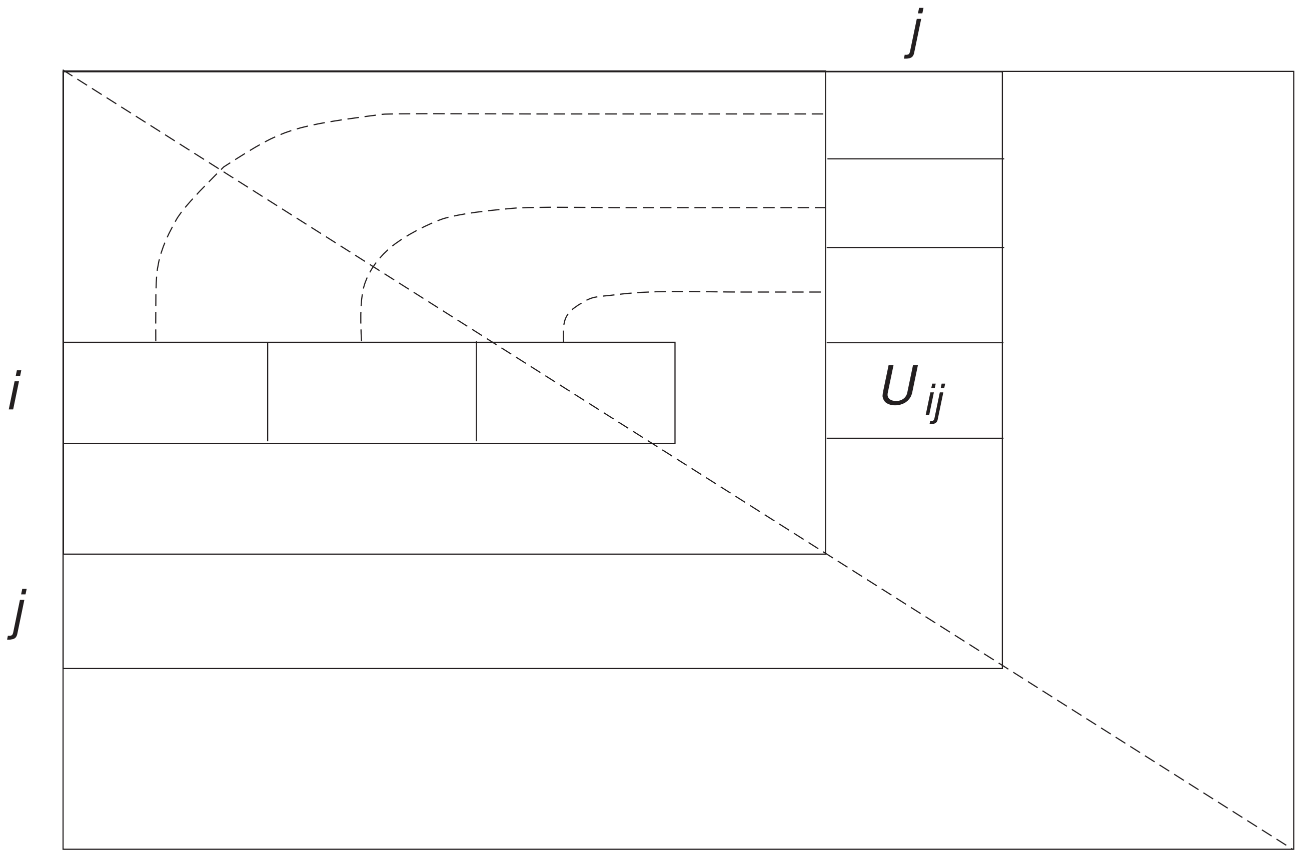


Figure 21. Typical vector inner product for column of the upper diagonal matrix in Full EQuations model simulation.

involves the size of the correction or the residual function. No one criterion will work for all flow conditions. A combination of relative and absolute criteria is applied in FEQ simulation.

A relative criterion for closeness works well in FEQ simulation if the quantity tested is not small. If the quantity tested becomes too small, then the relative convergence criterion is supplanted by absolute criteria in FEQ simulation so that computational convergence is obtained. For example, the relative change in flow rate is meaningful so long as the flow does not become too small. Obviously, a flow of zero cannot be used to define a relative correction. The user must define what flow is too small because FEQ could be applied to simulate the lower Mississippi River or it could be applied to simulate a 5-ft wide brook. Thus, the user specifies a value called QSMALL that is added to the absolute value of the flow to yield the quantity defining the relative change in flow given the correction to the flow rate from Newton's method. If QSMALL is too small, and the user-specified relative change criterion, EPSSYS, is too small, then the criteria for stopping will not be met, and the time step will be reduced in FEQ simulation as described later in this section. QSMALL is usually some small fraction of the flow range of interest. For example, if the flows of interest are greater than 100 ft^3/s, then a value of QSMALL of 0.1 ft^3/s would not be appropriate because the flows are only known at best to within 5 percent. Use of a QSMALL value of 5 to 10 percent of the maximum flows of interest has worked well in typical applications of FEQ. This means that the relative change accepted for the low flows might be considerably larger than EPSSYS.

The stopping criterion for water-surface-height values at locations having a cross-sectional area is defined in terms of the relative change in area. This criterion works well if the depth and area are not too small. The stopping criteria for small flow areas are considered differently than for small flows. If the correction to water-surface height is less than an input value, ABSTOL, then the computations have converged at that point and the relative change is taken as zero. Otherwise, the relative change in the area is computed and must be less than the value of EPSSYS before convergence is achieved. Again, this means that the relative change in area accepted for small areas is larger than for large areas. The elevations at the flow-path end nodes on dummy branches are subject to the relative criterion of change in elevation divided by the current depth at the node.

Relative and absolute criteria also are used for level-pool reservoirs. The relative criterion is $3|dS_R|/S_R$, where S_R is the storage volume in the level-pool reservoir. The absolute criterion is fixed internally at 0.001 ft. The change in storage is given by the product of the correction in water-surface elevation and the current surface area of the reservoir. The storage volume in the reservoir can become quite small. Thus, a lower limit on the value of storage used in calculating the relative convergence is set in FEQ. This lower limit is the larger of 1,000 ft^3 or the volume of 1 ft on the current surface area of the reservoir.

The criteria are applied to each unknown in the equation system. The rule for convergence for the equation system is that all unknowns must satisfy convergence criteria simultaneously. Thus, the maximum value of all relative changes must be less than EPSSYS before convergence is achieved. Experience has shown that, in some cases, only one or two variables will prevent convergence, whereas relative changes in all other variables are only a small fraction of EPSSYS. This is often the case when convergence is slow and the time step is being reduced to solve some computational problem. To increase the robustness of the solution scheme, another convergence criterion is added. Convergence may be declared with this criterion if no more than a user-input number, NUMGT, of unknowns does not meet the current convergence criteria but do meet a less restrictive, secondary convergence criterion. Thus, EPSSYS and ABSTOL together become the primary convergence criteria. NUMGT and the secondary relative change criterion become the secondary criteria for convergence. The number of unknowns not meeting the primary convergence criteria is output at each iteration in model simulation.

In most cases, all unknowns meet the convergence criteria, but disregarding local convergence problems greatly increases the robustness of the unsteady-flow analysis when flows and depths are small. Because the affected flows and depths are small relative to the flows and depths of primary interest, the effect of these additional convergence criteria on the final results is small. Results from several tests have shown that the maximum flows and stages obtained are usually the same or are within the uncertainty implied by the convergence criteria.

If the convergence criteria are not met within a user-supplied maximum number of attempts, the time step is reduced with a user-supplied factor, the integration weight, W_T, is incremented, again with a user-supplied factor, and a solution is computed again. This process continues until the time step becomes too small to continue or until the convergence criteria are met. The minimum time step is specified by the user. A weighted average of the number of iterations to convergence is maintained, and the average is used to increase or decrease the time step. A maximum time step, a minimum time step, and several weights to control the time step are specified by the user. Thus, the time step is under model control during the computations and will vary as the rate of convergence to a solution varies throughout a simulation.

The value used for EPSSYS must reflect the estimation of the stopping criteria based on the results of the most recent correction. All computed corrections are used. Because Newton's method converges rapidly when the root is approached, a 5-percent relative correction made to an unknown value often implies a relative correction of one-tenth or less of that amount in the next correction. Experience with FEQ simulation has supported this conclusion. A reduction of EPSSYS by a factor of 5, from 0.05 to 0.01, generally makes little difference in the results because most of the time steps have converged to a relative correction of 0.01 or less. Run time increases of about 30 percent have been typical with reduction in the system relative tolerance by a factor of 5 or more. Moreover, the stopping criteria are applied to the maximum value anywhere in the model. Commonly, most of the unknowns have relative corrections of a factor of 10 or more smaller than the maximum relative correction in the system.

9.4 Stability, Convergence, and Accuracy of the Solution Scheme

The results of any computation in steady or unsteady flow is an approximation of flow in a stream system. The accuracy of the approximation and the required accuracy for application in decision making, design, or planning are complex questions. Several concepts that relate to these questions are discussed in this section.

The acceptability of the results must be considered in an implicit comparison; that is, comparison of the results with some other set of results either real or, if hypothetical, at least obtainable if sufficient resources are available. The performance of the mathematical model of the stream system developed and computed with FEQ may be evaluated in the same way as for measurements that have been made or could be made on the prototype system. The key question relates to accuracy. How close are the results to some standard? A related but distinct concept is that of precision, the degree of refinement in the result. A value can be precise without being accurate, and an accurate result can be imprecise by certain standards. A typical example relevant to open-channel flow analysis is the reporting of elevations in surveys of stream cross sections. Modern semiautomatic surveying systems commonly report elevations to the nearest one-thousandth of a foot. At first glance, the surveyed channel-bottom elevations appear accurate; yet, in many cases, the elevations are only known within about one-half foot because the leveling rod was placed on mud on a channel bottom. On the flood plain, moving the point of measurement a few feet will change the elevation by much more than the precision of the measurement indicates.

The accuracy of the results is the primary interest, and precision is usually not a limiting factor. However, accuracy can be viewed as absolute or relative. Absolute accuracy is of interest when the model is expected to reproduce the result as measured in the prototype. For example, if measurements for a major flood on a stream are available and are simulated in the model, absolute accuracy is sought. Relative accuracy, on the other hand, is applied for comparing differences between two outcomes. An example would be evaluating the change in water level resulting from the construction of one or more reservoirs in a watershed. Computing the change in water level resulting from different structures on the stream can usually be done more accurately than computing the water level. Typical applications of unsteady flow often involve questions about the changes in flows and water-surface elevations that result from planned changes to watersheds or stream channels (for an example, see Knapp and Ortel, 1992).

9.4.1 Truncation Errors

Every time a differential or integral equation is replaced with an algebraic equation, a truncation error results. This is called a truncation error because the infinite series has been truncated; this also could be called the error of approximation that is introduced because a finite number of terms is used to represent a value requiring an infinite series solution. Finite steps are taken in space and in time, and these errors relate to the size of these steps and to the nature of the approximations used to convert the continuous equations to discrete equations. The truncation error is one of several attributes associated with a numerical method.

A numerical method (also called a scheme) is consistent if the continuous governing equations are obtained as both the time and space increments approach zero. A numerical method is convergent if the results obtained as both the time and space increments are reduced approach a limiting value and that value matches the true solution of the governing equations. Consistency relates to the equations and convergence relates to the solutions of these equations. A numerical method is computationally stable if roundoff and truncation errors do not accumulate such that the solution diverges.

A relation results among these concepts when the governing equations are linear. In this special case, a stable, consistent numerical method also is convergent. This is an important result because it is often easier to demonstrate stability and consistency than it is to demonstrate convergence. In unsteady-flow analysis, however, the governing equations are not linear, so this result does not rigorously hold. Nevertheless, the numerical method used in FEQ simulation is consistent and stable. Comparison of results as the time and distance step are reduced can empirically verify if the numerical method is convergent and that the time and distance steps are small enough for the application.

For each branch in the model, one or more nodes within each computational element defined in the branch description input can be added with the **ADDNOD** option. If one node is added, each computational element length is reduced by half. Thus, the maximum time step should also be reduced by one-half. Empirical tests for convergence involve computing a series of solutions, denoted by $\hat{\gamma}(\lambda)$, where λ is a measure of the size of the time and distance steps. If λ denotes the base level at which the process of reducing the time and distance steps begins, then the next level would be $\lambda/2$ where the time and space increments are one-half the base values. This process can in principle continue to $\lambda/4$, $\lambda/8$, and so on. Values can be compared at node locations common to all solutions to determine a trend toward a common value. A trend provides empirical evidence that the scheme is convergent. This process also can be applied to adjust the time and space increments to reduce the truncation errors if values are too high.

In terms of storage requirements, model size grows in inverse proportion to the size of the time and distance steps. Thus, if the step size is one-fourth the base value, then the storage requirement is increased about four times. The run time for a simulation increases in inverse proportion to the square of the step size. If the step size is again one-fourth of the base value, then the run time will be about 16 times longer than that for the base step. Therefore, limitations on computational time and storage capacity are quickly encountered. A way is needed to use only two such runs to obtain some insight into the effect of varying the time and space steps.

The Richardson extrapolation can be used with results from just two model simulations for which time and distance steps one-half the size of the first simulation are applied in the second simulation. The approximations applied in FEQ computations result in a truncation error being approximately proportional to the square of the time and distance steps. A reduction by half of both the time and distance steps should reduce the errors by about one-fourth. This is true for the time increment only if the value of the temporal integration weight, W_T, is close to 0.5. Let γ be the true solution at a given location; then,

$$\hat{\gamma}(\lambda) - \gamma \approx \kappa\lambda^2 \tag{137}$$

gives an estimate of the error, where κ is a constant of proportionality. In the Richardson extrapolation, this constant is assumed to be independent of the step size. Use of this assumption results in

$$\hat{\gamma}(\lambda/2) - \gamma \approx \kappa(\lambda/2)^2 \tag{138}$$

for the estimated error for the solution with the reduced step size. This error is one-fourth the first error. Eliminating the unknown constant of proportionality in equation 138 results in

$$4(\hat{\gamma}(\lambda/2) - \gamma) \approx \hat{\gamma}(\lambda) - \gamma. \tag{139}$$

Solving this equation results in

$$\gamma \approx \hat{\gamma}(\lambda/2) + \frac{1}{3}(\hat{\gamma}(\lambda/2) - \hat{\gamma}(\lambda)) . \tag{140}$$

Therefore, a better approximation than either $\hat{\gamma}(\lambda/2)$ or $\hat{\gamma}(\lambda)$ can be determined by correcting $\hat{\gamma}(\lambda/2)$.

Application of equation 140 results in an error in the solution with a corrected $\hat{\gamma}(\lambda/2)$ that is about one-third the difference between the two solutions. The error in the solution for the larger time and distance steps also can be estimated because, as shown in equations 137 and 138, it is about four times the error in the solution for the reduced time and distance steps. This cannot be used as a correction as in equation 140, but it may demonstrate that the error in $\hat{\gamma}(\lambda)$ is small enough so that the model can be applied for the intended purpose. This error is approximately 4/3 of the absolute value of the difference between the two solutions. If this error is acceptable, then the time and distance steps as related to the truncation error also are acceptable.

In the empirical convergence analysis applying the Richardson extrapolation, the proportionality constant is assumed to be virtually independent of the step size and neglected higher-order terms are assumed to be small. This applies if the initial step sizes are not too large. General guidelines for selecting the initial time and distance steps are described in the next few paragraphs.

Linear variation of values over each time and distance step is assumed in FEQ computations. Appropriate step sizes should be selected for this linear variation to be reasonable. The matrix-solution method applied in FEQ computations is unconditionally stable with respect to time step but not unconditionally accurate. Small variations in the values of flow at a boundary are often omitted when large time steps are used. For example, if an inflow hydrograph has a 1-hour duration of substantial flow, values for most of the hydrograph will be missed if a 1-hour time step is applied. Therefore, the maximum time step must be selected on the basis of knowledge of expected flow variations.

The distance steps must be related to the water-surface height of interest. A distance step appropriate for a small stream with a maximum water-surface height of 3 ft will be much too small for the main stem of a large river. On the lower Mississippi River, distance steps of several miles may be suitable because the variations in time and space are gradual and the stream is more than 100 ft deep in many places. The spacing of the nodes should be smaller where the water-surface height is expected to vary rapidly; for example, near points of critical or near-critical flow. Node spacing near critical flow may be in the tens of feet or less. The spacing of nodes also should be smaller where the channel changes shape or roughness rapidly. The CHKGEO option in FEQ can be used to identify possible regions of large variation in the channel geometry. Initial steady-flow computations will fail where the distance step is too long. If these requirements for cross-section spacing are met and the frequency of culverts and other controls in urban streams is considered, then the distance steps for models of urban streams will likely be in the appropriate range for accurate simulation.

Generally, the distance steps should be in the range of 100 to 300 times the water-surface heights of concern. Other previously discussed constraints take priority over this range. Choice of a time step such that 10 or more points are within every flow event of concern also is appropriate. Once an initial selection of time and distance steps has been made, the empirical convergence testing can be applied to a typical part of the stream to determine if the truncation errors are acceptable. If model simulation is within the computational capacity of the computer, full convergence testing of a typical event also could prove useful.

9.4.2 Verification of the Accuracy and Convergence of the Full Equations Model

Typically, the accuracy and convergence of numerical models are verified by use of simplified problems for which analytical solutions are available. For unsteady flow, such analytical solutions are not available. In the following paragraphs, the accuracy and convergence of FEQ are demonstrated for unsteady, free-surface flow in a sewer pipe by comparison with unsteady-flow data collected in carefully controlled laboratory experiments done at the Wallingford Hydraulics Research Station in England (Ackers and Harrison, 1964). Two sets of simulations and experiments are considered, one with raw data (experiment 115) and the other with scaled data (data collected in the 0.25-ft diameter pipe and scaled up to a 1-ft diameter pipe by Froude number equivalence principles). The scaled data were derived by Ackers and Harrison (1964) to verify their method of characteristics model of unsteady flow in open channels.

Hydraulic characteristics of the experiment and scaled experiment used to verify FEQ are listed in table 4. For fully turbulent flow, Manning's n is virtually constant for a wide range of flow depths. The experiments done by Ackers and Harrison (1964), however, include flow in the transition region between laminar and fully turbulent flow. Thus, Manning's n varies for the peak-flow and base-flow conditions, as indicated in table 4. At present, variations of Manning's n with depth are not considered from a rigorous fluid-mechanics viewpoint in FEQ computations because, for general applications, variations have negligible effects on the accuracy of the overall routing of a flood wave. The results of FEQ simulations discussed subsequently use the Manning's n corresponding to the base flow. For the scaled experiment, the difference in the n values is negligible, whereas for experiment 115 the wave is rapidly attenuated and flow is simulated acceptably for all downstream locations by use of the higher n.

Table 4. Hydraulic characteristics of the unsteady-flow experiments in sewer pipes done by the Wallingford Hydraulics Research Station, England

[ft, feet; ft^3/s, cubic feet per second; s, second]

Sewer characteristics and units	Experiment 115	Scaled experiment
Diameter, ft	0.25	1
Length, ft	300	1,000
Slope	.001	.001
Roughness (*k*), ft	.00004	.002
Manning's *n* (peak flow)	.0091	.0116
Manning's *n* (baseflow)	.0095	.0115
Base flow, ft^3/s	.0189	.176
Peak inflow, ft^3/s	.0532	.66
Inflow duration, s	36.7	132
Shape of inflow hydrograph	Symmetric trapezoid	Symmetric trapezoid
Duration of peak inflow, s	3.3	12

The results of the FEQ simulations are shown in a series of graphs. The available data confine consideration to the wave-peak depth and timing as the key test parameters. The wave-peak depth plotted with distance is shown for experiment 115 in figures 22–24. The convergence of FEQ calculations as a function of the computation time step, Δt, is shown in figures 22 and 23. The magnitude of the wave-peak depth is accurately simulated for Δt values of 6 seconds or less; thus, the model converged to the measured depth with a Δt as large as one-sixth of the duration of the input hydrograph. The convergence of the FEQ calculations as a function of the computational distance step, Δx, is shown in figure 24. The magnitude of the wave-peak depth is accurately simulated for Δx values of 50 ft or less. Thus, the model computations converged to the measured depth with a Δx as large as one-sixth of the entire length of the test section. The convergence of the FEQ calculations in terms of the timing of the wave peak as a function of the computation distance step, Δx, is shown in figure 25. Even though the Δx of 50 ft yielded accurate results for the wave-peak magnitude, the results are somewhat inaccurate in the simulation of the timing of the wave peak.

The instantaneous depth and time for five data-collection locations in the scaled experiment are shown in figures 26 and 27. The agreement between the measured and the simulated depth at distances of 28.4 and 255.7 ft from the inlet is similar to that achieved by Ackers and Harrison (1964) using their method of characteristics model. Ackers and Harrison's model performs somewhat better in depth simulation than FEQ for the more downstream locations, because of a combination of the depth-variable roughness and possible scaling problems.

The results presented here demonstrate that the numerical scheme used in FEQ can converge to an accurate result with relatively large space and time increments for unsteady flow in a single branch. These computational results for single-pipe laboratory experiments are encouraging with regard to applications of FEQ to large, multiple-branch, real-world stream networks.

9.4.3 Verification of the Full Equations model on the Fox River, Illinois

The accuracy of FEQ simulation also has been tested and verified with field measurements for several stream systems. A particularly thorough evaluation of the simulation accuracy involved an application of FEQ to a 30.7-mi reach of the Fox River in northeastern Illinois (fig. 28), reported by Ishii and Turner (1997). FEQ was tested and verified with stage, discharge, and dye-transport data collected during a 12-day period of unsteady flow induced by operations at the upstream boundary, a sluice-gate control at Stratton Dam. The reach includes 19 bridges and 4 low-head overflow dams, including the dam at the downstream boundary. The channel-bottom profile is shown in figure 29. The reach is effectively split into two subreaches on the basis of slope. The upstream subreach, from Stratton Dam to Algonquin Dam, has a bed slope of only 0.0034 percent and was subject to considerable backwater effects and hysteresis in the stage-discharge relations; the downstream subreach, from Algonquin Dam to South Elgin Dam, has a bed slope of 0.039 percent and was subject to little or no backwater effects. The channel is virtually prismatic, with no trend in width from upstream to downstream.

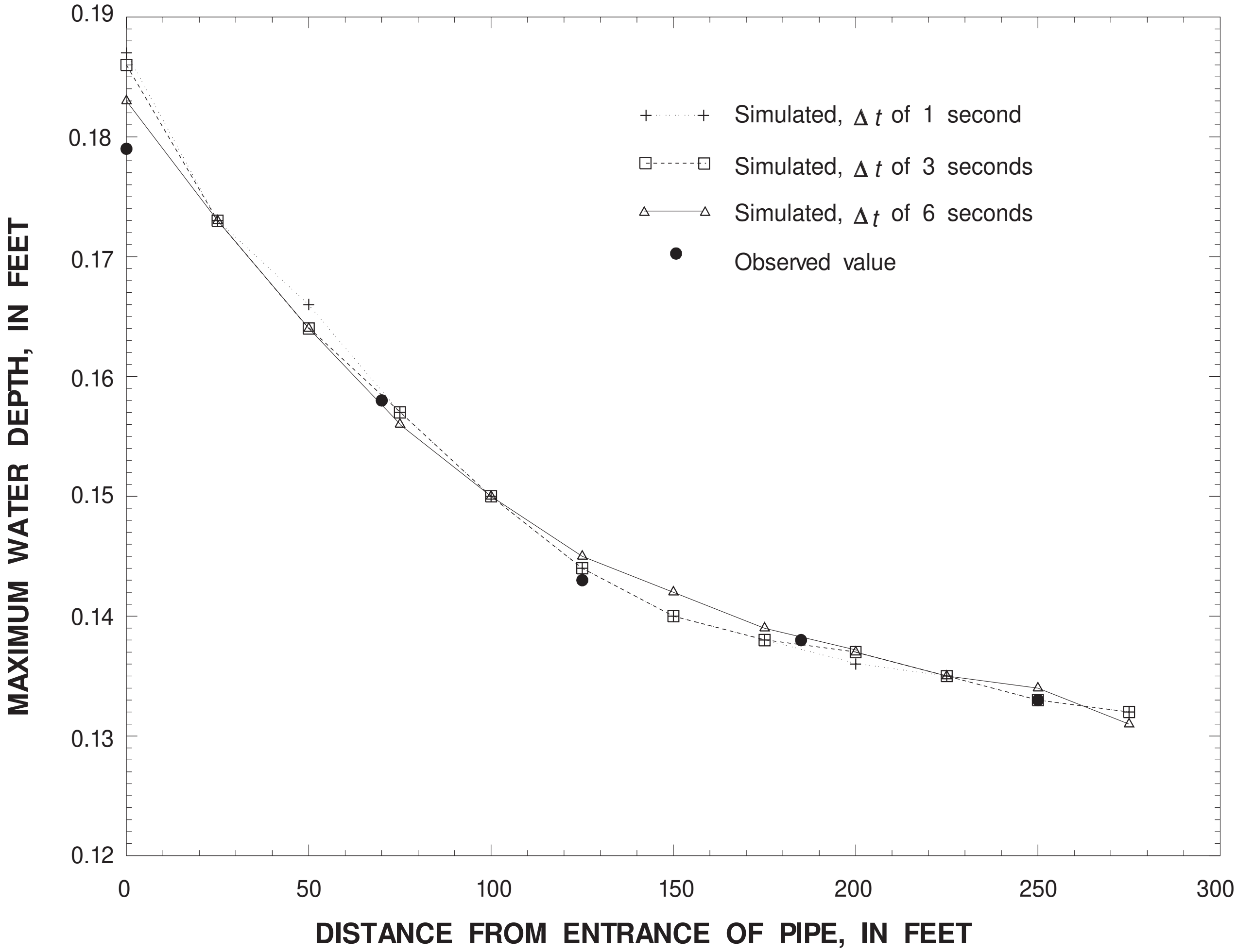


Figure 22. Effect of varying time step, Δt, on maximum water depth simulated in the Full EQuations model for sewer-pipe experiment 115 (Wallingford Hydraulics Research Station, England) with a fixed distance step, Δx, of 25 feet.

The study reach was represented in the FEQ model as a 45-branch network. The number of branches was determined by the number of bridges, dams, and tributary branch inflows simulated. FEQUTL (Franz and Melching, in press) routines were used to compute the 2-D tables for flow through the bridges and over the dams. The model of the Fox River was developed by the Illinois Department of Natural Resources, Office of Water Resources (IDNR-OWR) from a HEC-2 model developed earlier by IDNR-OWR and the U.S. Amy Corps of Engineers. The hydrology for the unmeasured tributary inflows was done by proportioning a percentage of the measured flow of the nearest of the two gaged tributaries among the smaller tributaries by area. Only 26 percent of the tributary-area flow was not measured at least once during the verification field study.

The upper subreach of the model was calibrated by the Illinois State Water Survey (Knapp and Ortel, 1992), and the lower subreach was subsequently calibrated by IDNR-OWR. The upstream boundary condition for the model calibration was discharge at Wilmot Dam, 18.8 mi upstream from the study reach. Stage and discharge data from two flood periods were used in the calibration data set, and an additional six flood periods were used to verify the calibration for the upstream reach. Seven of the eight flood periods were used in the calibration of the downstream reach.

The fully calibrated model was then passed to the USGS for comparison to the measured stage and discharge data collected during the unsteady-flow verification field study. No further adjustment was made to the calibrated model during the verification. All numerical results cited are from the calibrated model with no adjustment. Adjustments in Manning's n of –0.006 for the upstream reach and +0.005 to the downstream reach were made and

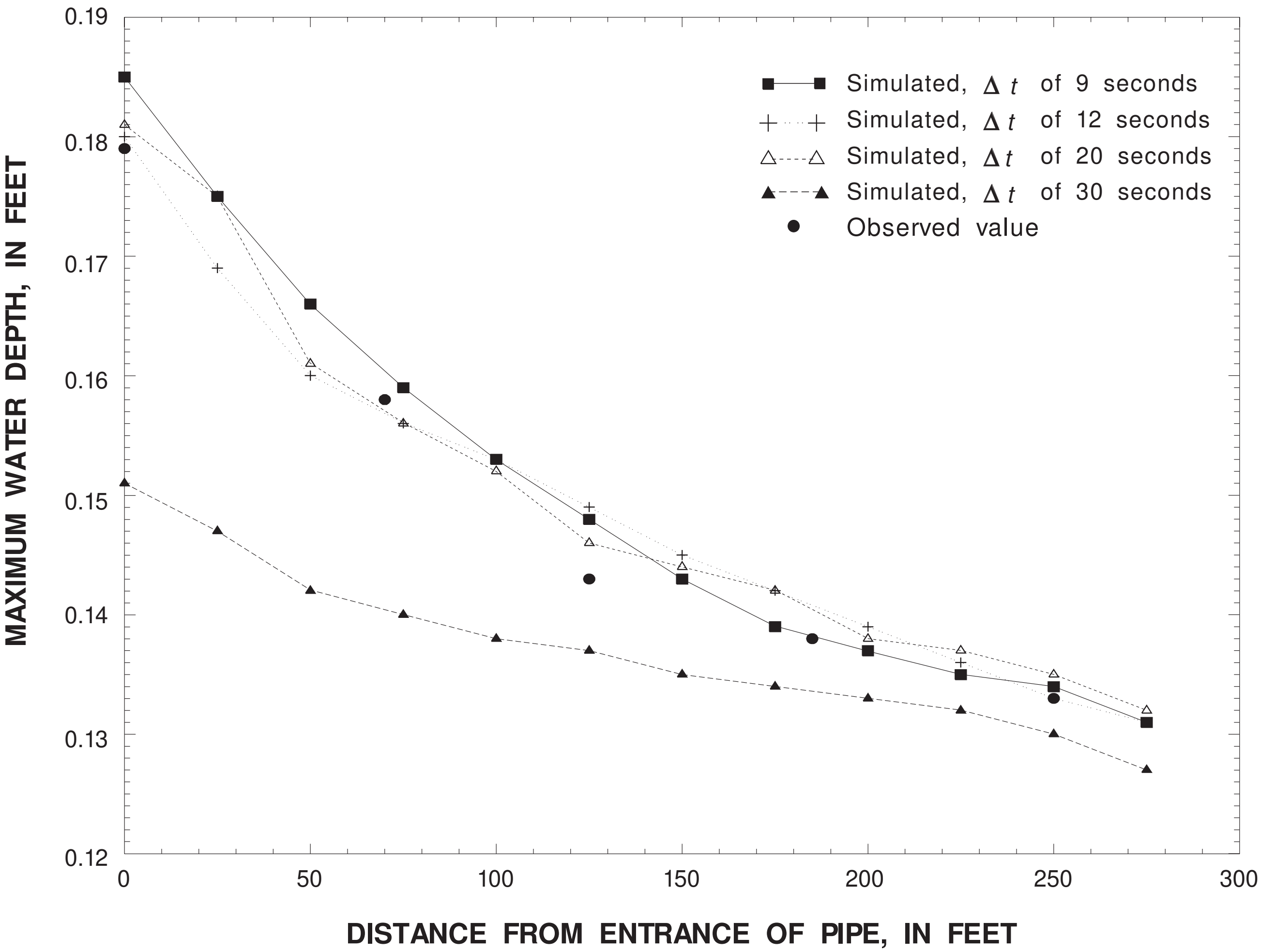


Figure 23. Effect of varying time step, Δt, on maximum water depth simulated in the Full EQuations model for sewer-pipe experiment 115 (Wallingford Hydraulics Research Station, England) with a fixed distance step, Δx, of 25 feet.

are clearly indicated on the figures showing the calibrated results (fig. 30), in order to demonstrate the effect of such an adjustment. The verification periods differed from the calibration periods in that flow was lower and no overbank flooding resulted. The routed wave was a trough rather than a peak. At locations where water-surface height was low, the calibrated values for the roughness coefficient, Manning's *n*, appeared to be too low compared to the effective field values, resulting in simulated stages that were less than measured stages by as much as 0.8 ft during the wave trough at 3 of the 15 locations where stage data were collected. Simulated and field-measured stages were matched closely at the other 12 sites, differences between the stages ranging from 0 to 0.3 ft. The routed discharge results were especially close to field-measured values. The timing of the discharge hydrograph and the stage hydrograph almost exactly matched the observed hydrographs. The discharge and stage results at one site in the upstream subreach and one site in the downstream subreach are shown in figure 30. The results at all sites are presented in Ishii and Turner (1997).

At Huntley Road Bridge, 21.4 mi downstream from Stratton Dam and 5 mi downstream from Algonquin Dam, a difference in the hydrograph near the end of the wave trough is the result of local rainfall that was not simulated. Discharge is otherwise simulated accurately in both timing and amount. The site is one of the three sites where Manning's *n* appears to have been underestimated for low flow. The stage-discharge relations at the same two sites for the calibrated and adjusted simulations and the measured values are shown in figure 31. The hysteresis induced by the variable-momentum slope in the upstream reach is clearly evident. In contrast to this, the stage-discharge relation at Huntley Road (in the downstream reach) shows little evidence of hysteresis.

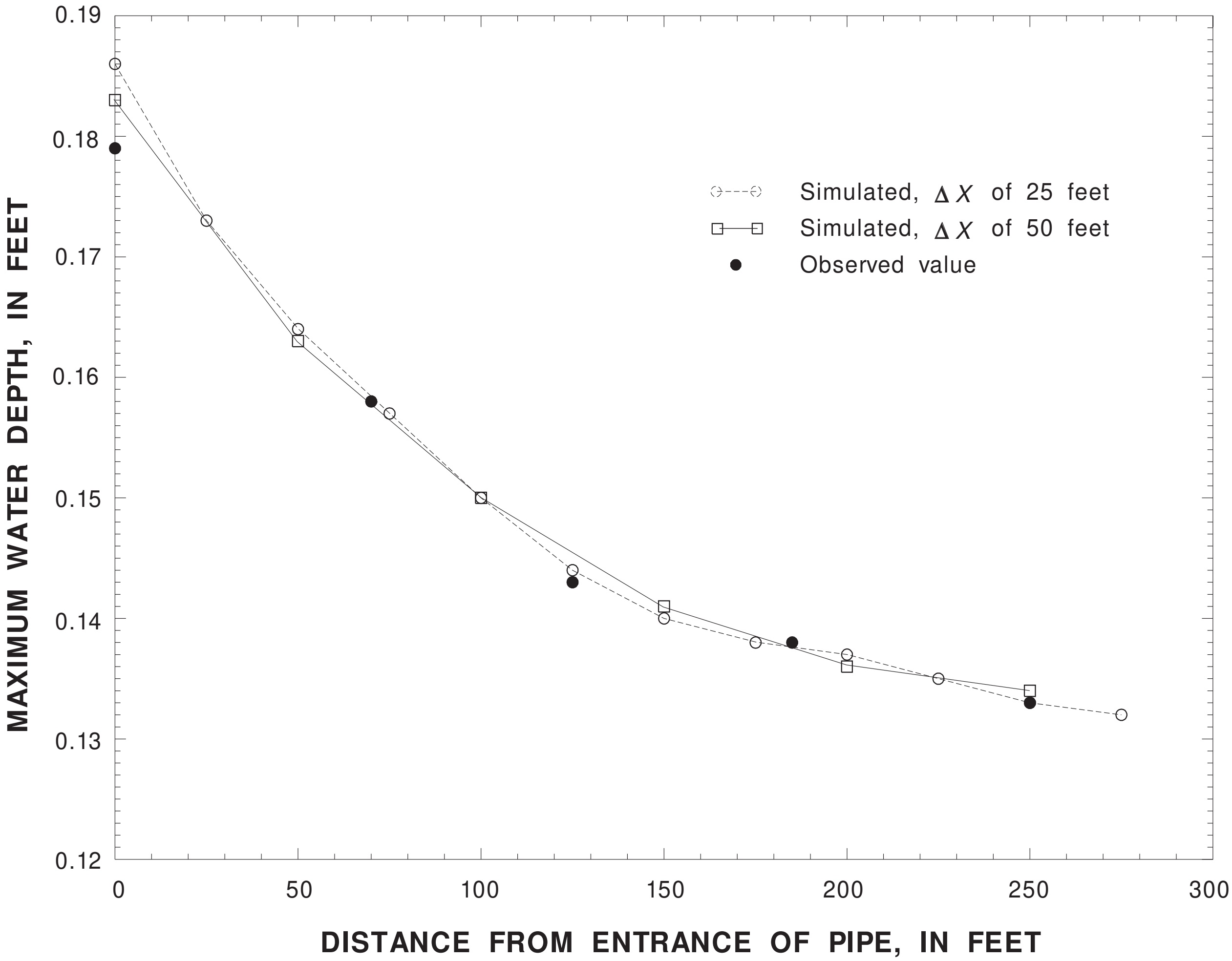


Figure 24. Effect of varying distance step, Δx, on maximum water depth simulated in the Full EQuations model for sewer-pipe experiment 115 (Wallingford Hydraulics Research Station, England) with a fixed time step, Δt, of 3 seconds.

The study also included a semicontinuous dye injection with manual and automatic collection of samples at 18 locations during the unsteady-flow period. The velocity simulated in FEQ was verified by inputting the flow-field and dye-injection data to the Branched Lagrangian Transport Model (BLTM) (Jobson and Schoelhammer, 1987) and comparing the simulated and measured dye concentrations. The match was acceptable given the limit of the temporal resolution imposed by the relative infrequency of dye-sample collection at most sites. The results for Fox River Valley Gardens and Huntley Road Bridge are shown in figure 32 together with the measured and simulated discharges. Although the dye-injection rate was almost constant, a spike in concentration resulted from the higher concentration during the low-flow period. An attenuated peak followed during the high-flow period. The decline in concentration to near zero between the two peaks was because of pump failure that resulted in a zero injection rate for 15 hours between the two flow conditions.

Other results reported in Ishii and Turner (1997) include the sensitivity of the model to computational and physical parameters. Ishii and Wilder (1993) report on the results of using different pairs of exterior boundary conditions. Turner and others (1996) report on a study to verify the culvert and overbank routines for a small stream reach. FEQ was determined to be robust and accurate for these applications.

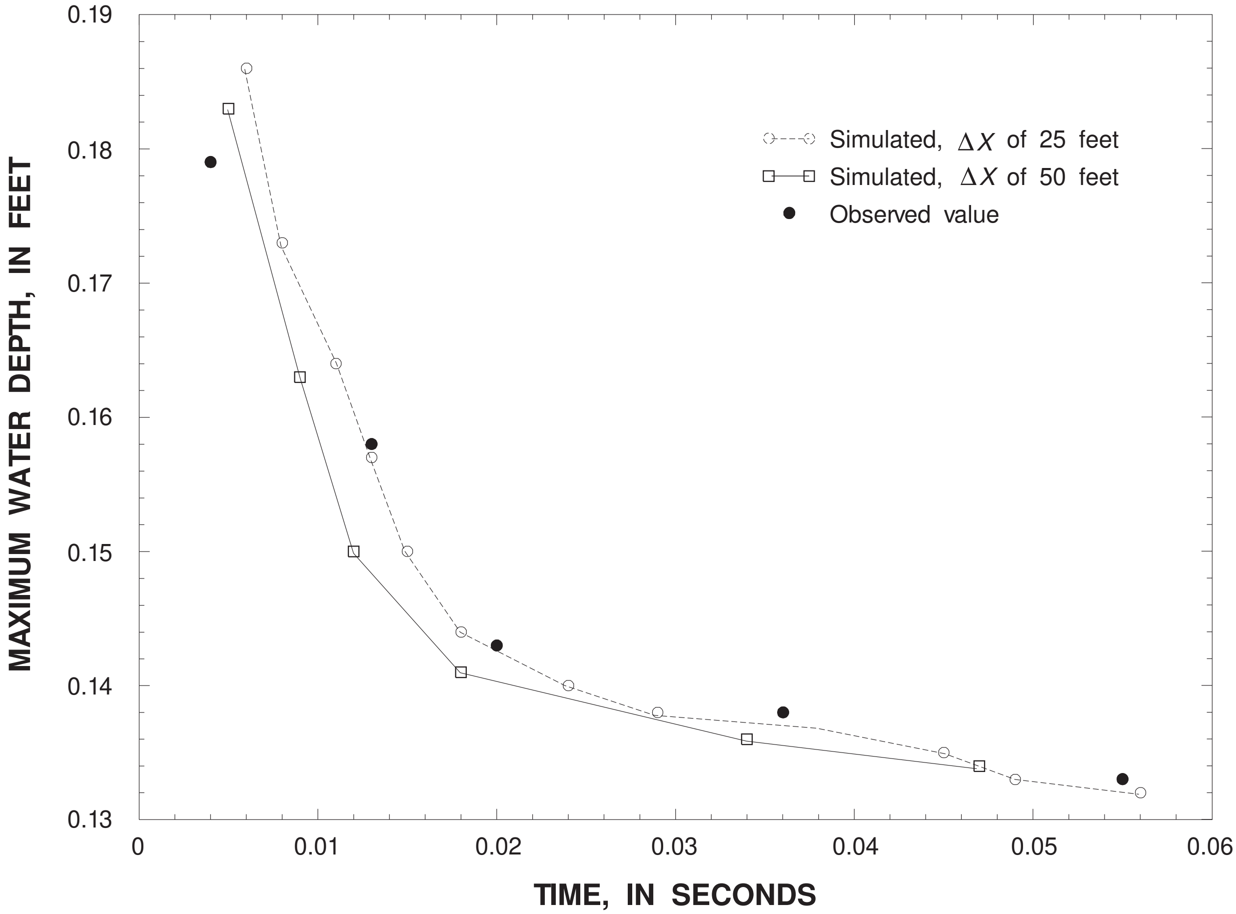


Figure 25. Simulated and observed maximum water depth and time as a function of distance step, Δx, for sewer-pipe experiment 115 (Wallingford Hydraulics Research Station, England) with a fixed time step, Δt, of 3 seconds.

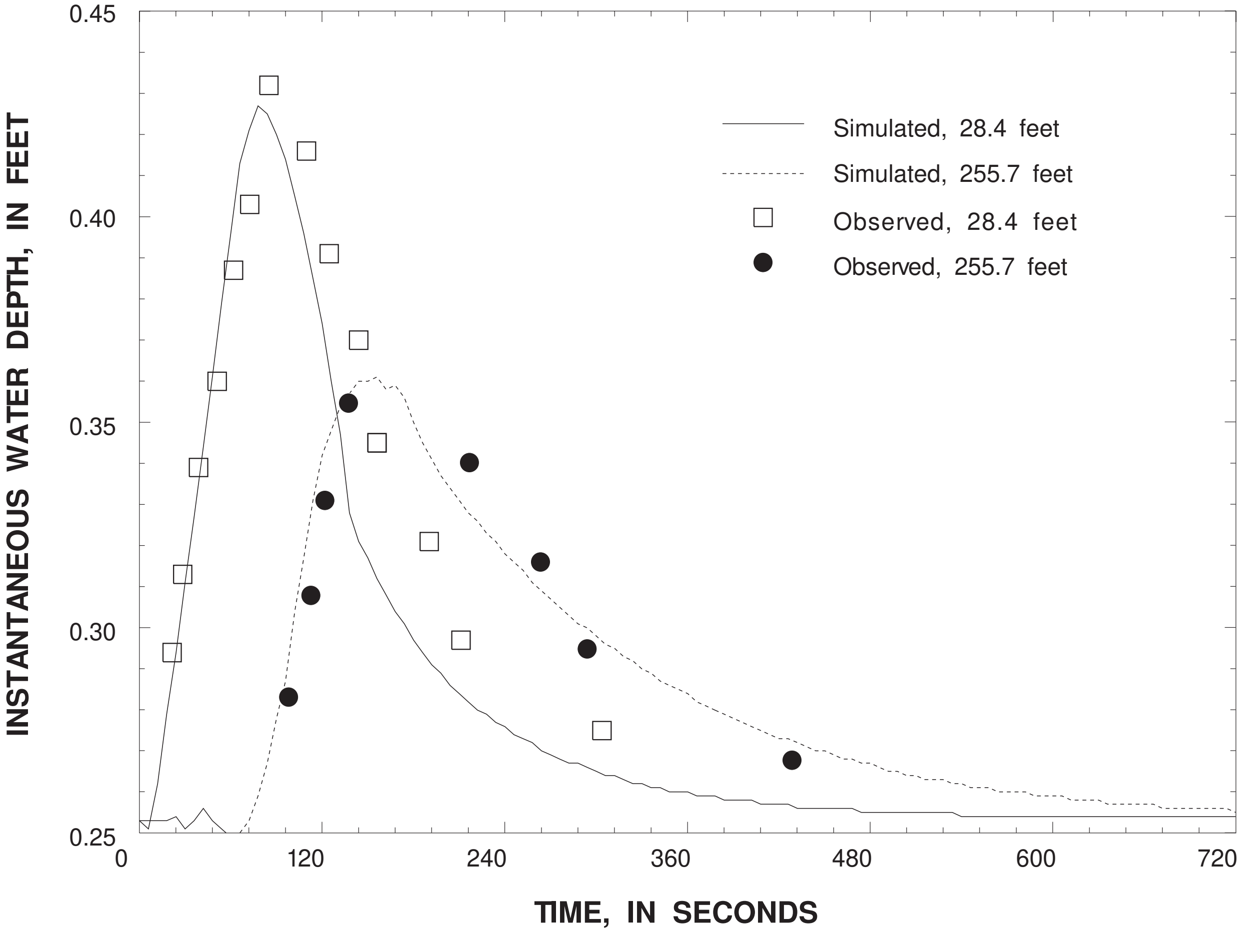


Figure 26. Simulated and observed water depths for locations 28.4 and 255.7 feet downstream from pipe inlet for scaled sewer-pipe experiment (Wallingford Hydraulics Research Station, England).

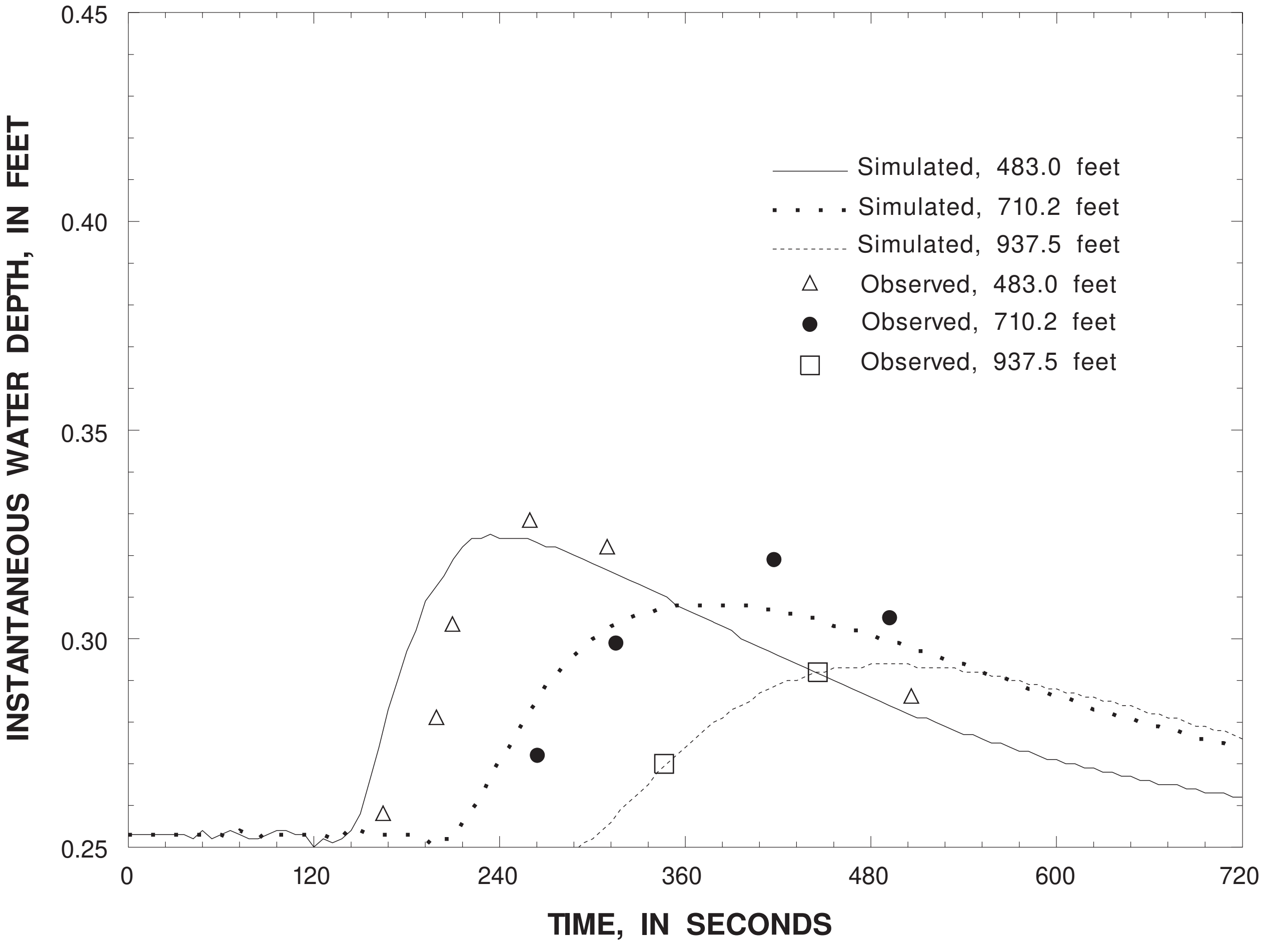


Figure 27. Simulated and observed water depths for locations 483.0, 710.2, and 937.5 feet downstream from pipe inlet for scaled sewer-pipe experiment (Wallingford Hydraulics Research Station, England)

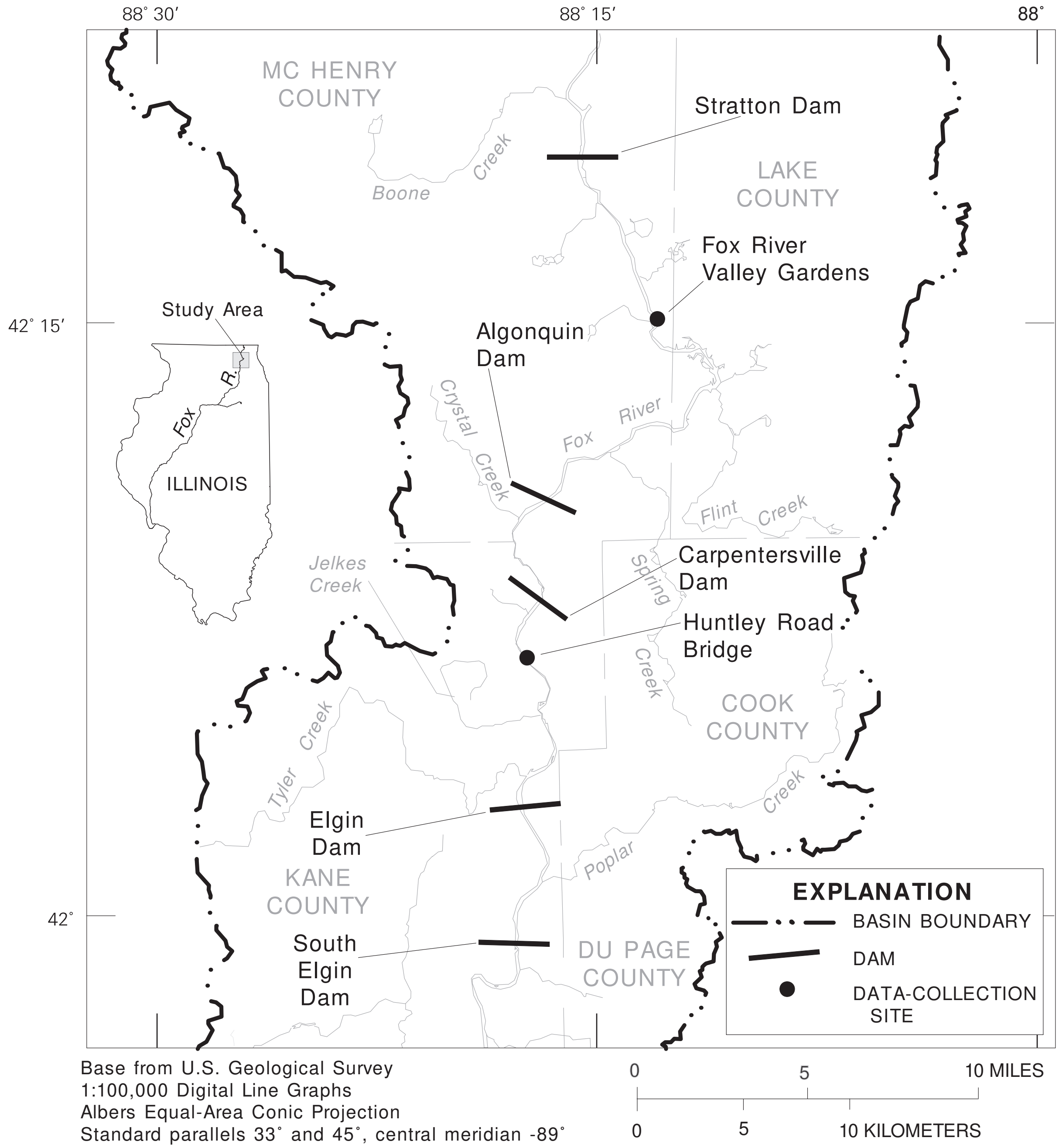


Figure 28. Fox River, Illinois, study reach and data-collection sites considered in this report.

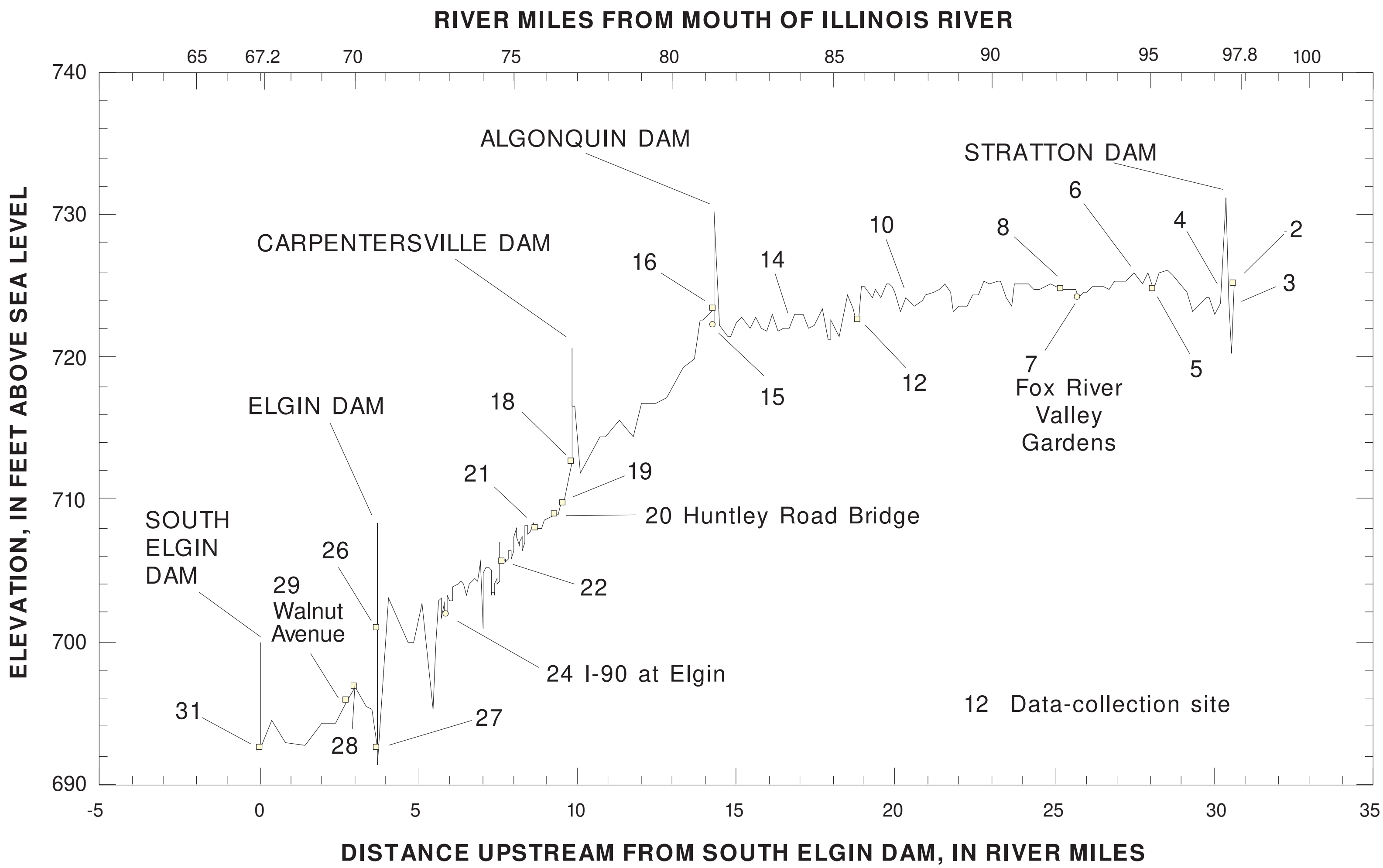


Figure 29. Bottom elevation and data-collection sites on the Fox River, Illinois, study reach..

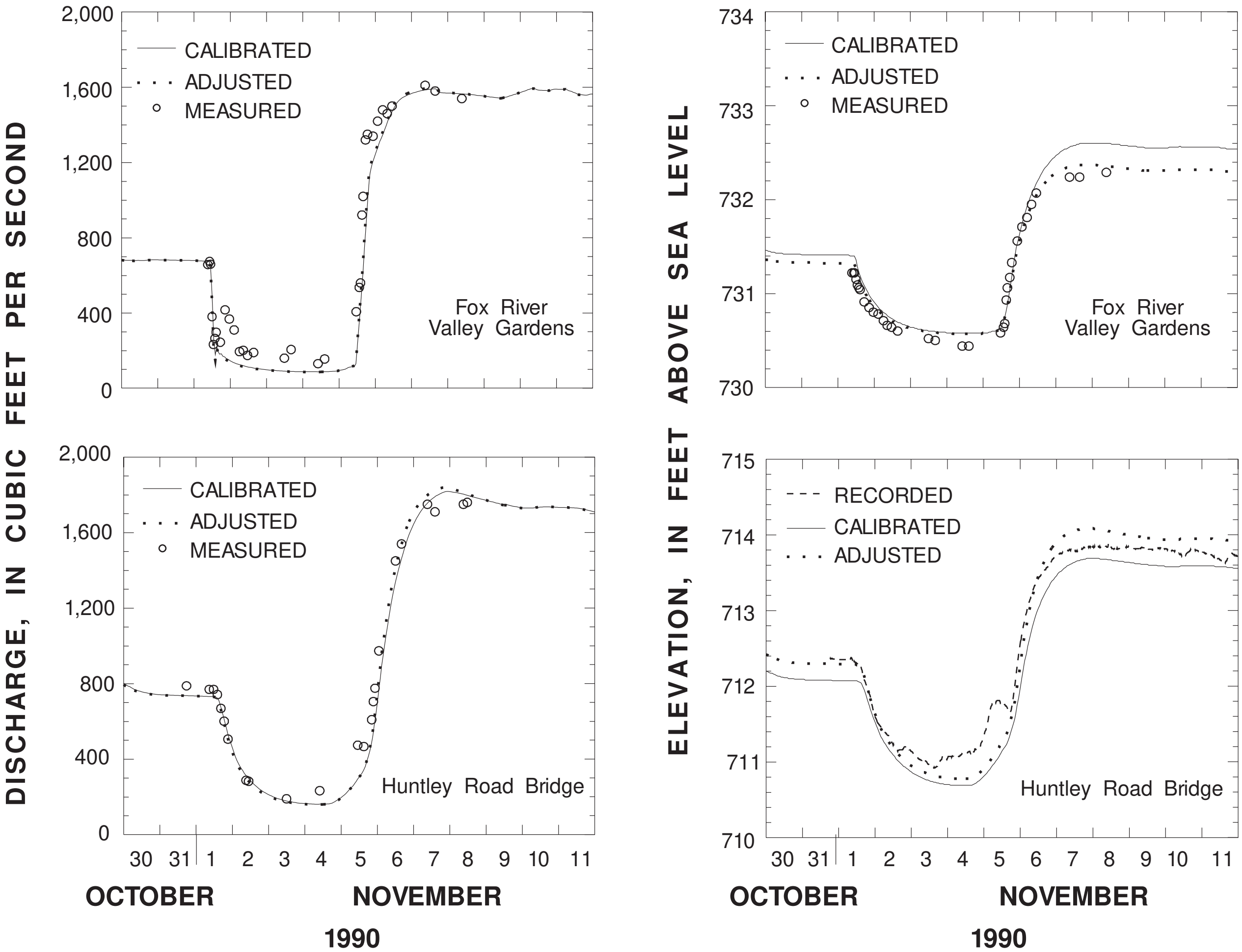


Figure 30. Measured or rated and simulated discharge and stage at data-collection sites on the Fox River in Illinois.

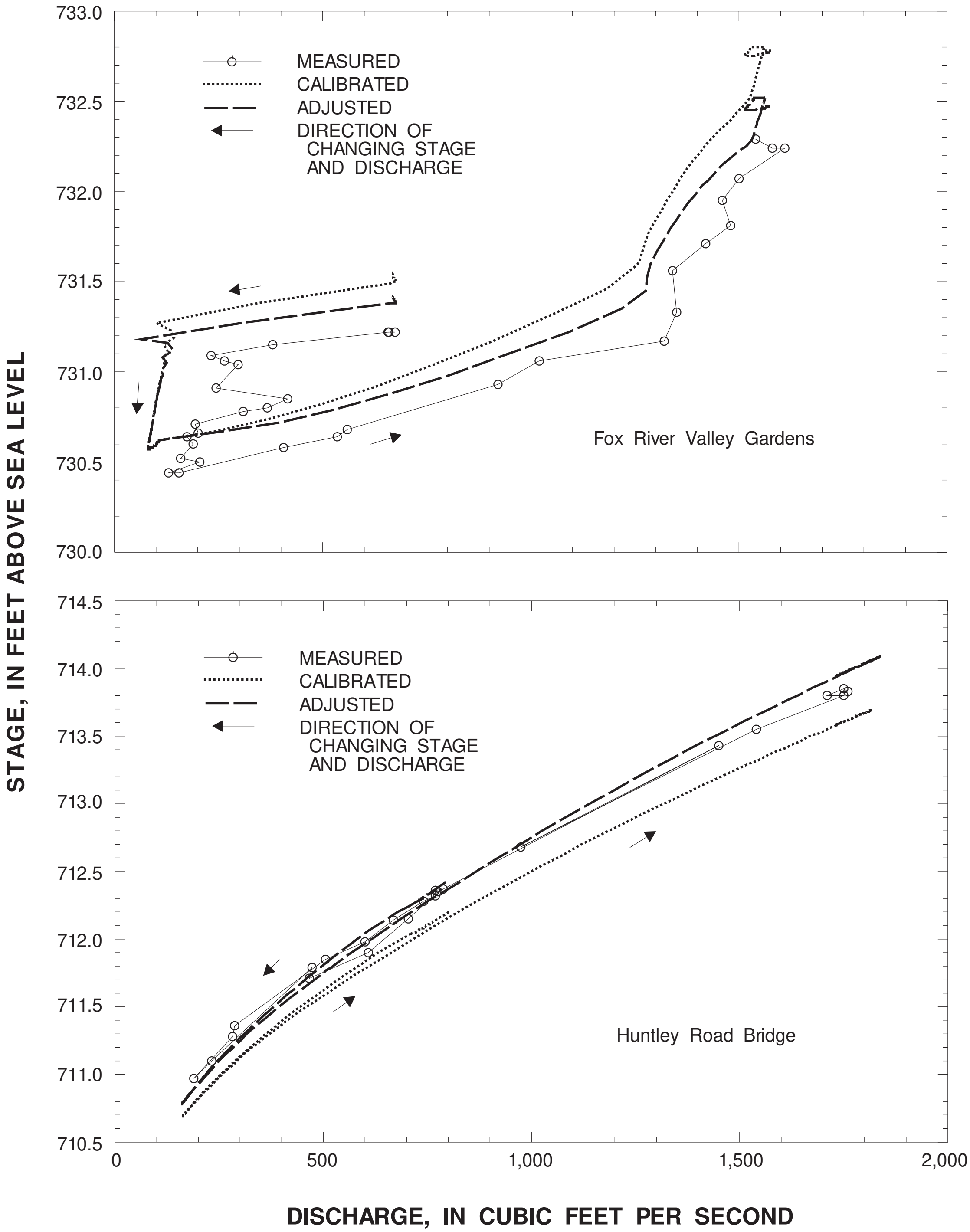


Figure 31. Measured or rated and simulated stage-discharge relations at data-collection sites on the Fox River in Illinois for the October 30–November 11, 1990, study period.

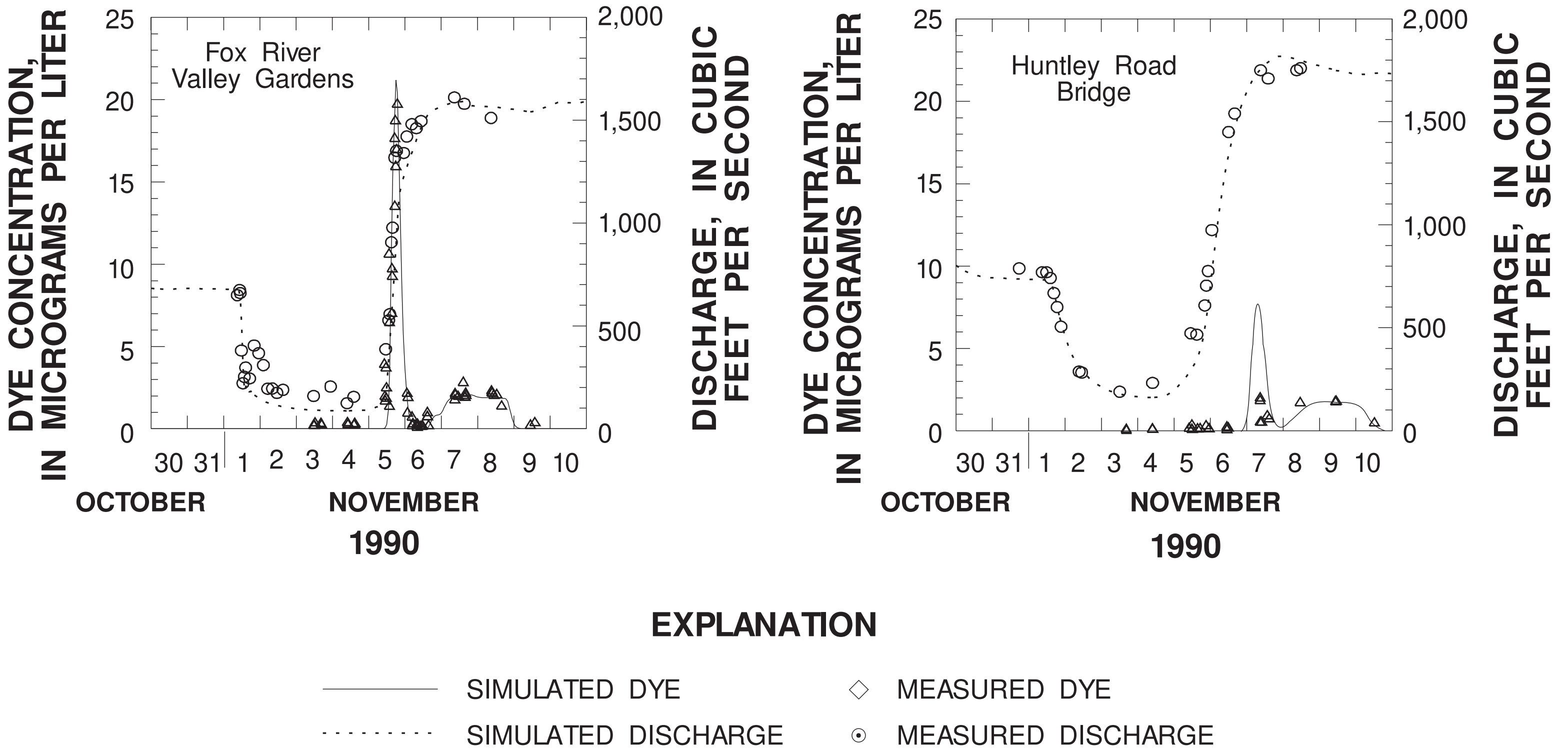


Figure 32. Measured and simulated dye concentrations and discharge at data-collection sites on the Fox River in Illinois.

10. MATRIX FORM OF THE SYSTEM OF EQUATIONS DESCRIBING THE NETWORK OF CHANNELS

Some simple examples are given here to illustrate the matrix form of the system of equations describing the network of channels. These examples increase in complexity to provide insight into the development of the Jacobian matrix for a complex stream system.

10.1 Example: Single Branch

The simplest example is a single branch with an upstream boundary condition of flow as a function of time (an imposed hydrograph) and a downstream boundary condition of flow as a function of water-surface elevation (a rating curve). The example branch with a total of four nodes, two end nodes and two interior nodes is shown in figure 33. The equations describing flow paths, internal boundary conditions, and external boundary conditions placed in the matrix yielding flow and water-surface elevation throughout the stream system are input to FEQ with the Network-Matrix-Control Input (NMCI) Block (section 13.6). The description of this stream system in the NMCI Block for FEQ is the following:

[. . . , additional input values that are not needed for the discussion of this example; blanks indicate no input in these columns]

	Input columns					
Line	Code	N_1	N_2	N_3	N_4	N_5
1	6	1	U1	...		
2	1	1				
3	4	2	D1	-1	D1	...

Only the parts of the NMCI necessary for discussing how the Jacobian matrix is prepared (a NMCI fragment) are given above and explained below; complete details are included in "Input Description for the Full Equations Model" (section 13.6). Code 6 designates the upstream external boundary condition, and code 4 designates the downstream external boundary condition in the form of a stage-discharge relation. Code 1 denotes a branch. The important items in this discussion are the code numbers, the branch numbers, and the flow-path end node labels (section 3.2.2). The upstream boundary is node U1, and the downstream boundary is node D1. This example is the simplest possible in FEQ—at least one branch is required in any stream system. In line 3 of the NMCI fragment, the first of the two nodes given for code 4, D1, is the node defining the head (water-surface elevation) and the second node, D1, is the node defining the flow through the structure; that is, the discharge node discussed above. In this example, and in all cases at a node requiring an external boundary condition, the node for head and the node for discharge will be the same. In some important cases, however, the node for head and the node for discharge must differ at an internal boundary condition.

The nonlinear-equation system in the example consists of eight equations and eight unknowns, two unknowns for each node. The unknowns will be paired flows and water-surface heights. The water-surface elevation becomes known at a node once the water-surface height is known, so water-surface height is used in most FEQ computations on branches. The order of placing the equations in the solution matrix is important because it affects the structure of the resulting Jacobian matrix. At each time step, this matrix is computed, and the linear system of equations discussed in section 9.1 is solved. If the Jacobian matrix is arranged properly, the solution is efficient.

The equations in this example are ordered in the solution matrix with the upstream boundary equation first (eq. 121), followed by the six equations for the three computational elements in the branch; that is, the equations for conservation of mass (eq. 68) and momentum (eq. 77) appear in pairwise order. The final equation (eq. 91) is the downstream boundary condition. This order results in a banded Jacobian matrix, a matrix in which all nonzero

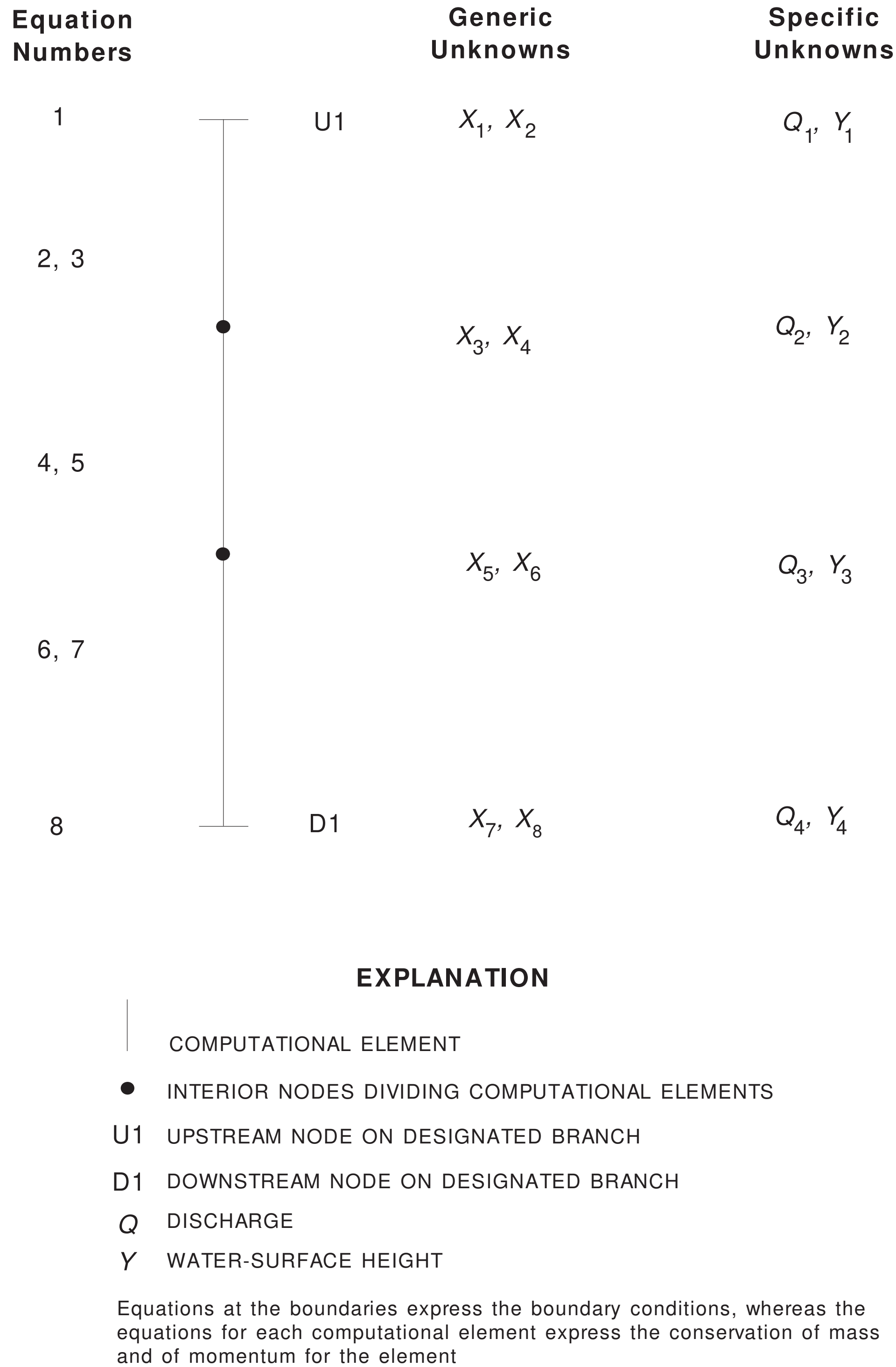


Figure 33. Single-branch example of the development of the Jacobian matrix of flow and water-surface height for a stream in the Full EQuations model.

elements are close to the main diagonal of the matrix. The equation describing the upstream boundary only includes one of the two unknowns, Q_1 and y_1, where the nodes in this example are numbered in downstream order. The Jacobian includes the partial derivatives, F', of the residual function of each motion equation (eqs. 68 and 77), and each external boundary condition equation (eqs. 121 and 91) with respect to each of the unknown variables.

Therefore, the other seven entries in the first row of the Jacobian matrix are zero because the first equation does not include any of these other seven variables. Only four unknowns are included in each pair of equations for the computational elements, the flow and water-surface height at each node. Therefore, at most, four of the eight values in a row of the Jacobian can be nonzero. Finally, the last equation includes only the flow and water-surface height at the downstream node; thus, at most, two values can be nonzero in the last row of the Jacobian. In matrix form, the typical linear system will be similar to

$$\begin{bmatrix} F'_{11} & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ F'_{21} & F'_{22} & F'_{23} & F'_{24} & 0 & 0 & 0 & 0 \\ F'_{31} & F'_{32} & F'_{33} & F'_{34} & 0 & 0 & 0 & 0 \\ 0 & 0 & F'_{43} & F'_{44} & F'_{45} & F'_{46} & 0 & 0 \\ 0 & 0 & F'_{53} & F'_{54} & F'_{55} & F'_{56} & 0 & 0 \\ 0 & 0 & 0 & 0 & F'_{65} & F'_{66} & F'_{67} & F'_{68} \\ 0 & 0 & 0 & 0 & F'_{75} & F'_{76} & F'_{77} & F'_{78} \\ 0 & 0 & 0 & 0 & 0 & 0 & F'_{87} & F'_{88} \end{bmatrix} \begin{bmatrix} \Delta Q_1 \\ \Delta y_1 \\ \Delta Q_2 \\ \Delta y_2 \\ \Delta Q_3 \\ \Delta y_3 \\ \Delta Q_4 \\ \Delta y_4 \end{bmatrix} = \begin{bmatrix} -F_1 \\ -F_2 \\ -F_3 \\ -F_4 \\ -F_5 \\ -F_6 \\ -F_7 \\ -F_8 \end{bmatrix}. \qquad (141)$$

The numbering for subscripts is shown in figure 33. The subscripts for the partial derivatives follow the numbers for the generic unknowns; that is, the second subscript refers to the variable involved and the first subscript to the residual-function number. The rule is simple. An odd-numbered generic unknown (variable) is always a flow, and an even-numbered variable is always a water-surface height. In the equations for a branch, the conservation of mass equation always precedes the conservation of momentum equation. One equation for each boundary condition (internal or external) will always precede the conservation equations on the branch, and one will always come after. Thus, other than the boundary equations, the equations for a branch are ordered such that an even-numbered equation is for conservation of mass and an odd-numbered equation is for conservation of momentum. For example, F'_{53} is the partial derivative of the residual function of the conservation of momentum equation in the second element taken with respect to the third variable, which is the flow at the second node.

The Jacobian matrix has a bandwidth of 5 in the example. All branches where the governing algebraic equations as given above are used will have this bandwidth. In this short branch, the Jacobian matrix has 64 elements, but only 27 (less than one-half) are nonzero. However, if 50 nodes are on the branch, then the Jacobian matrix has 10,000 elements, but only 395 are nonzero. The number of computations needed to solve a linear system with no special structure is proportional to n_e^3. If the system is banded, the number of computations are proportional to $n_e m^2$, where m is the bandwidth. The approximate ratio of the number of computations is then proportional to $(m/n_e)^2$. In the current case, $m = 5;$ with 100 equations for a 50-node branch, the ratio is 2.5×10^{-3}. Thus, the computational effort if the band structure is used is less than 1 percent of the effort if the band structure is not used.

10.2 Example: Two Branches With Overflow Dam

This example introduces a special feature, a low overflow dam, between two branches. As shown in equation 136, concern is concentrated on the structure of the Jacobian matrix and not the vector of changes to the estimated unknowns or the residual vector. Thus, this and the following examples only show the Jacobian matrix pattern. The schematic of the example system is shown in figure 34. Branch 1 has two nodes, and branch 2 has three. This gives a total of 5 nodes and 10 unknowns. The NMCI (section 13.6) fragment for this example is the following:

[..., additional input values that are not needed for the discussion of this example; blanks indicate no input in these columns]

	Input columns					
Line	Code	N_1	N_2	N_3	N_4	N_5
1	6	1	U1	...		
2	1	1				
3	5	6	D1	U2	D1	...
4	2	2	D1	U2		
5	1	2				
6	4	2	D2	-1	D2	...

In line 3 for code 5, the first node, D1, defines the upstream node for head; the second node, U2, defines the downstream node for head; and the third node, D1, defines the discharge node. The discharge node must be either of the two nodes for head. In this case, either choice will be valid. In other cases, only one choice will be valid.

The upstream and downstream external boundary conditions are the same as in the single-branch example, and they provide the first and the last equations as above. The internal boundary conditions at the dam are the conservation of mass at a junction, code 2 (eq. 89), and a two-node control-structure, code 5, for the flow over the dam (eq. 102). The internal boundary condition includes only two flow-path end nodes in this case. Therefore, only 2 out of the 10 elements in the row of the Jacobian for the internal boundary condition will be nonzero. The two-node control structure, in the most general state, includes the flow at one flow-path end node and water-surface heights at both flow-path end nodes. Therefore, at most, this row of the Jacobian matrix will include three nonzero elements. As mentioned earlier, the conservation equations for the elements on each branch are preceded and followed by one equation from the boundary conditions that are attached to the branches. This pattern maintains the same bandwidth for the equations on a branch. At upstream and downstream boundaries the assignment is fixed, and the boundary condition is applied to provide the equation at that end of the branch. At junctions, however, the order of assignment is not fixed and is determined by rules internal to FEQ.

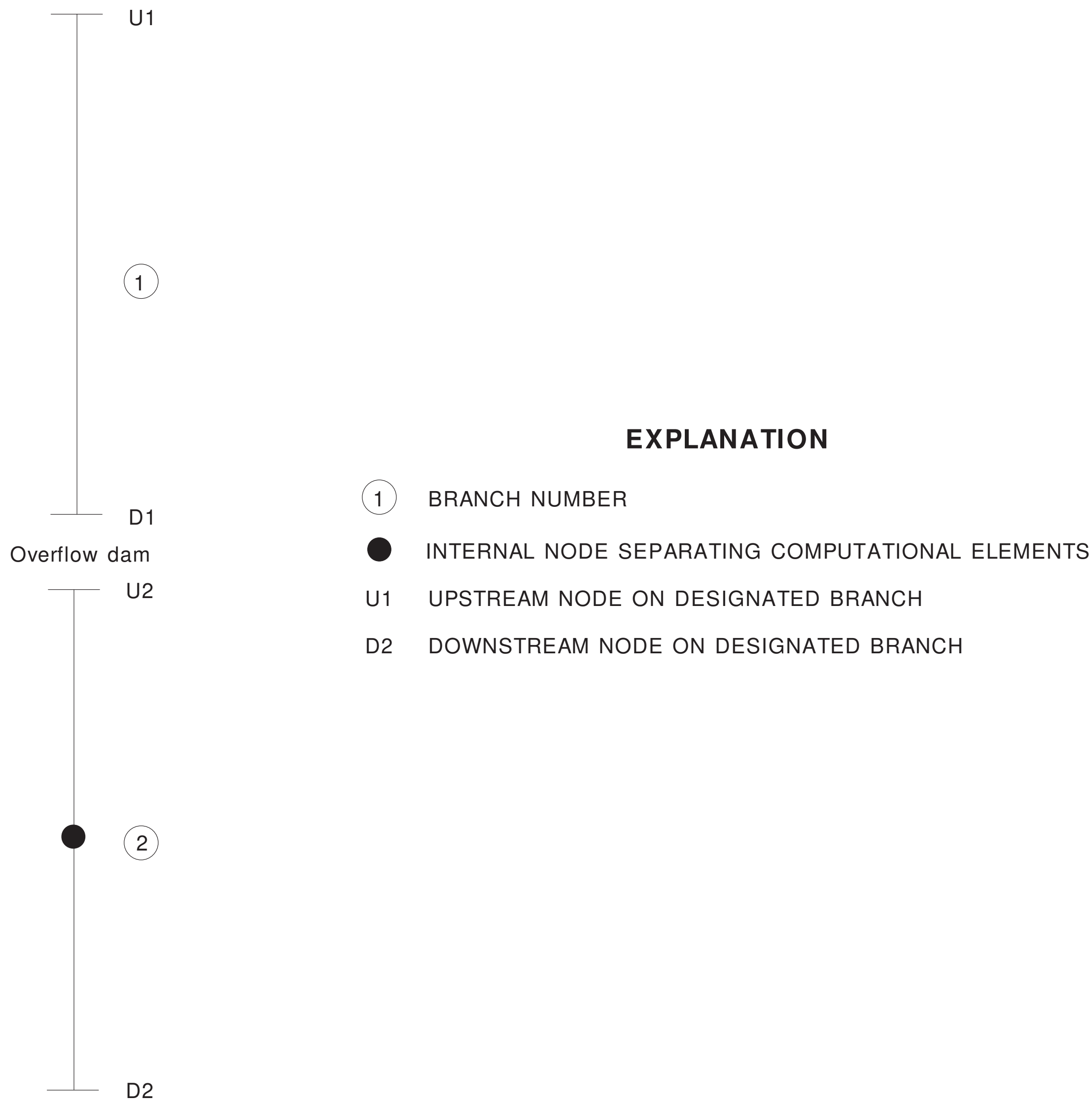


Figure 34. Two-branch example of the development of the Jacobian matrix of flow and water-surface height in a stream in the Full EQuations model.

The Jacobian matrix for this example has the following pattern:

Rows	Q_1	y_1	Q_2	y_2	Q_3	y_3	Q_4	y_4	Q_5	y_5
1	X									
2	X	X	X	X						
3	X	X	X	X						
4			X	X		X				
5			X	0	X	0				
6					X	X	X	X		
7					X	X	X	X		
8							X	X	X	X
9							X	X	X	X
10									X	X

An *X* denotes an element that is nonzero, whereas those elements that are known to be zero are left blank in the matrix. The zeros shown in the matrix are initially zero but may become nonzero. The columns are labeled with the variable involved in that column. In this simple example, the bandwidth is unchanged from that in the single-branch example. Therefore, internal boundary conditions between two branches do not change the bandwidth.

10.3 Example: Three-Branch Junction

If more than two branches are included at a junction, the bandwidth no longer remains constant. The simplest case is that of a three-branch stream system such as shown in figure 20. The NMCI (section 13.6) fragment for FEQ for this three-branch network is the following:

[..., additional input values that are not needed for the discussion of this example; blanks indicate no input in these columns]

	Input columns					
Line	Code	N_1	N_2	N_3	N_4	N_5
1	6	1	U1	...		
2	1	1				
3	3	D1	D2			
4	3	U3	D2			
5	2	3	D1	D2	U3	
6	6	1	U2	...		
7	1	2				
8	1	3				
9	4	2	D3	-1	D3	. . .

The junction is a simple junction in which the water-surface elevation is the same at all three nodes.

The user may specify the boundary node used to initiate the development of the Jacobian matrix. If node U1 is specified as the initial node, then the pattern of the Jacobian matrix is the following:

Columns

Rows	Q_1	y_1	Q_2	y_2	Q_3	y_3	Q_4	y_4	Q_5	y_5	Q_6	y_6	Q_7	y_7	Q_8	y_8	Q_9	y_9
1	X																	
2	X	X	X	X														
3	X	X	X	X														
4			X	X	X	X												
5			X	X	X	X												
6					X	X	X	X										
7					X	X	X	X										
8								X		X								
9							X	0	X	0					X			
10									X	X	X	X			0			
11									X	X	X	X			0			
12											X	X	X	X	0			
13											X	X	X	X	0			
14													X	0	0			
15										X	0	0	0	0	0	X		
16															X	X	X	X
17															X	X	X	X
18																	X	X

The subscripts on Q and y for the columns above correspond to the locations shown in figure 35. An *X* denotes an element that is nonzero, whereas those elements that are known to be zero are left blank in the matrix. The zeros shown in the matrix are initially zero but may become nonzero. The columns are labeled with the variable involved in that column. This pattern is almost banded. In this simple three-branch network, the equation in row 15 and the column under Q_8 modifies the banded pattern of the matrix. If branch 2 had more computational elements, the effect would have been more dramatic. With more branches in the network, the pattern of the matrix would have been a central band with vertical and horizontal spikes projecting out of the band at intervals. Such a matrix has been called a profile matrix or, more aptly, a skyline matrix because the pattern above the diagonal resembles the appearance of a city skyline. The pattern of explicit zeros has been placed in the matrix as an indication of how the matrix will be represented and solved. The discussion of the solution methods in section 9 indicates that only the nonzero and potentially nonzero parts of the matrix must be stored. The solution process is such that the known zero parts of the matrix will remain zero.

10.4 Example: Four Branches with Loop

A four-branch network of channels with a loop is shown in figure 36. In this example, each branch has a single computational element to limit the number of equations that must be presented. The junction conditions are

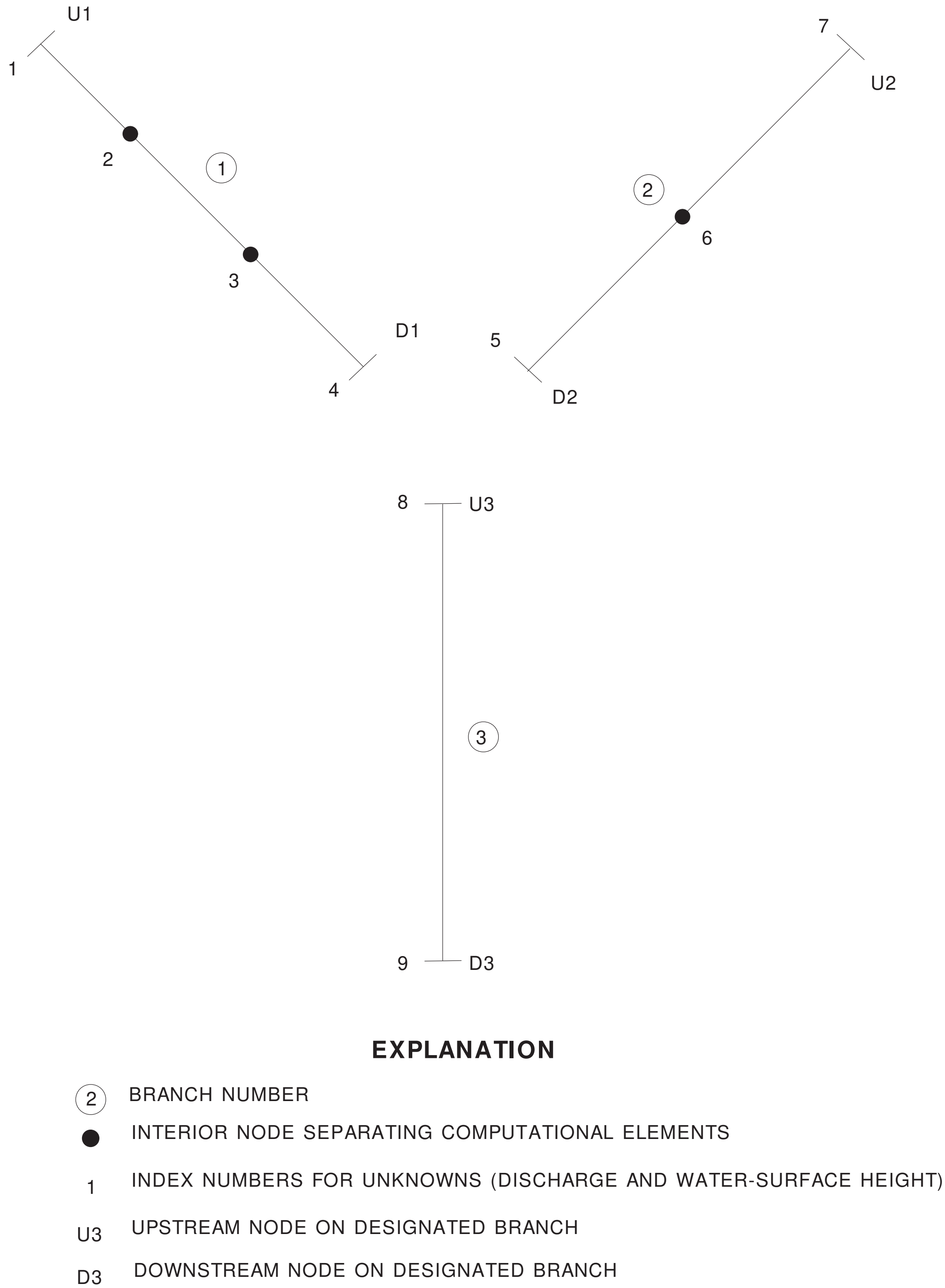


Figure 35. Three-branch example of the development of the Jacobian matrix of flow and water-surface height in a stream system in the Full EQuations model.

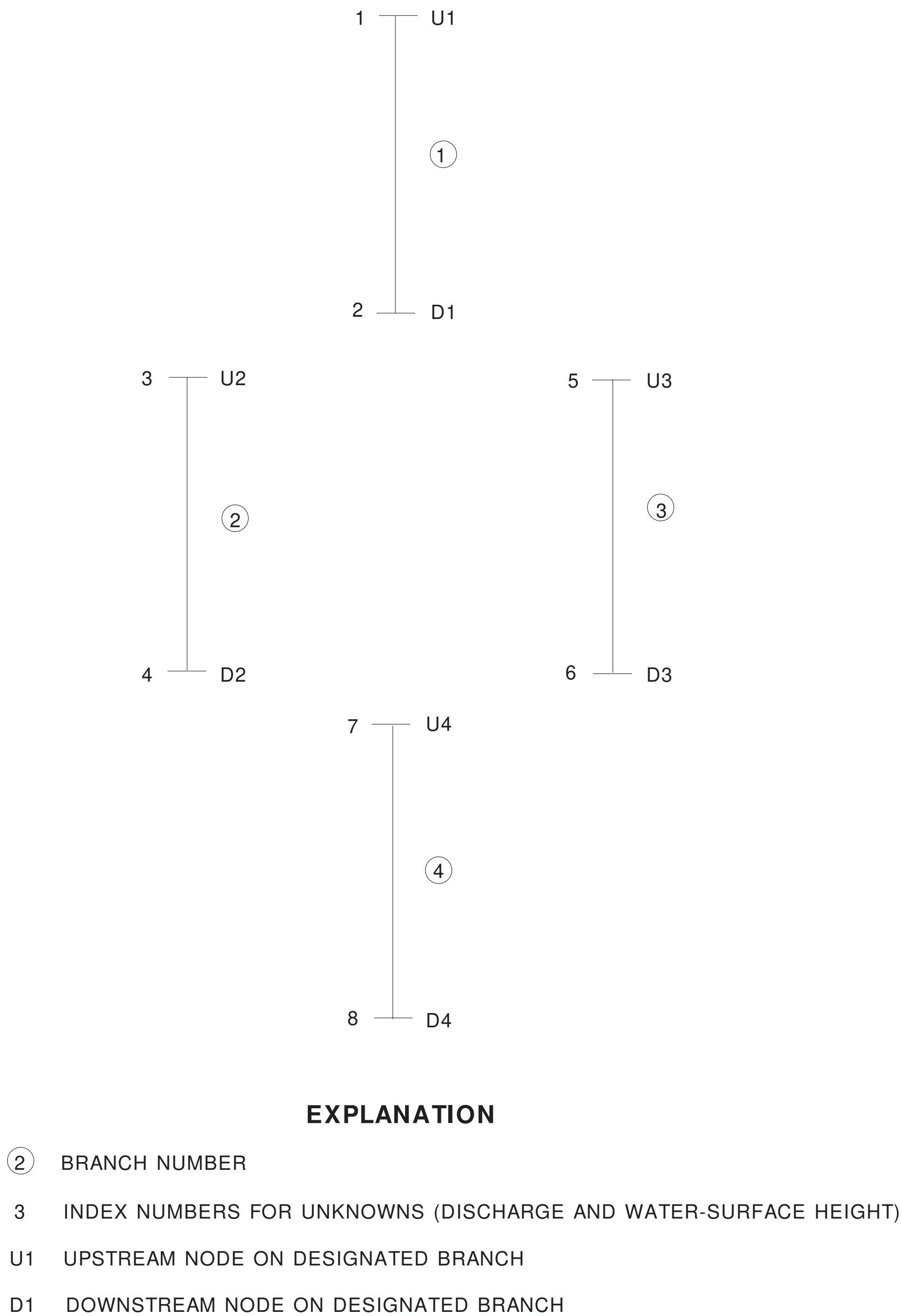


Figure 36. Four-branch (with loop) example of the development of the Jacobian matrix of flow and water-surface height in a stream system in the Full EQuations model.

again the simple requirement that the water-surface elevations in all the branches at a junction should be the same. This is the simplest condition, but yet the most demanding, in establishing nonzero diagonal elements in the Jacobian matrix for the network. The NMCI (section 13.6) fragment in FEQ describing stream-system connections is the following:

[..., additional input values that are not needed for the discussion of this example; blanks indicate no input in these columns]

	Input columns					
Line	Code	N_1	N_2	N_3	N_4	N_5
1	6	1	U1	...		
2	1	1				
3	3	D1	U2			
4	3	D1	U3			
5	2	2	D1	U2	U3	
6	1	2				
7	1	3				
8	3	D2	U4			
9	3	D2	D3			
10	2	3	D2	D3	U4	
11	1	4				
12	4	2	D4	-1	D4	...

If node U1 is specified as the initial boundary node, the pattern of the Jacobian matrix is the following:

Columns

Rows	Q_1	y_1	Q_2	y_2	Q_3	y_3	Q_4	y_4	Q_5	y_5	Q_6	y_6	Q_7	y_7	Q_8	y_8
1	X															
2	X	X	X	X												
3	X	X	X	X												
4				X		X										
5			X	0	X	0			X							
6					X	X	X	X	0							
7					X	X	X	X	0							
8								X	0					X		
9				X	0	0	0	0	0	X				0		
10									X	X	X	X		0		
11									X	X	X	X		0		
12								X	0	0	0	X		0		
13							X	0	0	0	X	0	X	0		
14													X	X	X	X
15													X	X	X	X
16															X	X

There are 71 nonzero or potentially nonzero elements in the Jacobian matrix. This number is called the length of the profile of the matrix. Had the boundary node been D4 instead of U1, the profile length would have been 66. If the user does not request an explicit choice for the initial boundary node, then all possible boundary nodes are attempted in FEQ simulation and the node that minimizes the length of the profile of the matrix is selected. A valid boundary node has the boundary condition of flow as a function of time or flow as a function of elevation. The matrix cannot be formed in FEQ simulation from a boundary where water-surface elevation is specified as a function of time. To study a case where water-surface elevation is a function of time at all upstream boundaries, the user can add a dummy branch with zero flow to the system so that the matrix may be formed.

11. FUNCTION TABLE DESCRIPTION

Several function tables are used in FEQ to represent the various functions in unsteady-flow analyses. Some of these tables are computed with the utility program FEQUTL (Franz and Melching, in press), and others are entered manually from other sources. The origin of the tables is not important as long as they are in the expected structure and format. The concepts and mathematical relations of the tables are described below. A detailed discussion of the formats and examples of some of the tables are given in "Input Description to the Full Equations Model" (section 13). Examples of all of the function tables may be obtained by electronic retrieval from the World Wide Web (WWW) at http://water.usgs.gov/software/feq.html and by anonymous File Transfer Protocol (FTP) from water.usgs.gov in the pub/software/surface_water/feq directory.

Three broad classes of function tables are applied in FEQ: 1-D (one argument), 2-D (two arguments), and 3-D (three arguments). Several options are available within each of the first two classes. The 1-D tables are the simplest and are presented first.

11.1 One-Dimensional Function Tables

Thirteen types of 1-D function tables are available in FEQ to describe physical features of the stream system to be modeled. Many of these table types contain overlapping information, so it is unlikely a given stream system would be simulated by use of all the table types. The following sections describe each of the 1-D table types and the relations among them.

11.1.1 Types 2 and 7

Table types 2 and 7 are similar. Both table types have one argument and one function value, and linear interpolation is applied to determine values not tabulated. A single number is used as an argument in type 2, whereas an external argument with a date/time specification for presenting the time series is used in type 7. This date/time specification is converted internally to a single variable for convenience in interpolation.

By proper choice of the tabulation interval and the choice of the function values, many functions can be represented with these table types. These include rating curves, hydrographs, variation of spillway coefficients with head, and any other function for which a single variable is a function of a single argument.

11.1.2 Types 3 and 8

The relation between table types 3 and 8 is the same as for types 2 and 7, except that the table requires one argument and two function values. The additional function is the derivative of the tabulated function, f_T, with respect to the table argument, f_T'. Linear interpolation in the derivative and integration of the derivative is used in types 3 and 8 to define the function values; that is, the derivative value is defined by

$$\hat{f}_T'(y) = f_{T_i}' + \frac{y - y_i}{y_i - y_{i+1}} \left(f_{T_{i+1}}' - f_{T_i}' \right), \qquad 142)$$

and the function value is defined by

$$\hat{f}_T(y) = f_{T_i} + 0.5\,(y - y_i) \left(f_{T_i} + f_{T_{i+1}} \right), \qquad (143)$$

where the argument y is contained within the interval (y_i, y_{i+1}). The trapezoidal rule is used for integration in equation 143 because it is exact for linear functions. For this table type, values must be computed such that equation 143 is correct. This means that the function value is computed by integration of the piecewise linear derivative.

No formal name has been applied for the interpolation defined by these equations. It will be called integrated linear interpolation for subsequent reference. The function, f_T, is, at most, a piecewise quadratic function of its argument. If the integrated function is quadratic, then the interpolation will be called integrated quadratic interpolation.

Function table types 3 and 8 are ideal for representing the area and storage capacity of a level-pool reservoir because the integral of the surface area is the storage. The storage must be computed by trapezoidal-rule integration of the surface area.

11.1.3 Types 4 and 9

The relation between table types 4 and 9 is the same as for types 2 and 7. In this case, the table includes one argument and two function values, the additional function being f_T'. These tabulated functions are the same as for types 3 and 8, but the interpolation rule differs. For types 4 and 9, a cubic polynomial is defined over each interval by use of the function value and its derivative at each endpoint. The derivative of this cubic polynomial then gives the interpolating polynomial for the derivative of the function. If $\Delta y = y_{i+1} - y_i$ is the tabular interval and $p_I = (y - y_i) / \Delta y$ is the proportion of the tabular interval represented by the point of interpolation, y, then the interpolated function value is

$$\begin{aligned} f_T(y) = {} & f_{T_i} + p_I \Delta y f_{T_i}' + p_I^2 \left[\Delta y \left(2 f_{T_i}' + f_{T_{i+1}}' \right) + 3 \left(f_{T_i} - f_{T_{i+1}} \right) \right] \\ & + p_I^3 \left[\Delta y \left(f_{T_i}' + f_{T_{i+1}}' \right) + 2 \left(f_{T_i} - f_{T_{i+1}} \right) \right], \end{aligned} \tag{144}$$

and the interpolated derivative is

$$\begin{aligned} f_T'(y) = {} & f_{T_i}' - 2 p_I \left[2 f'_{T_i} + f'_{T_{i+1}} + 3 \left(f_{T_i} - f_{T_{i+1}} \right) / \Delta y \right] \\ & + 3 p_I^2 \left[f_{T_i}' + f_{T_{i+1}}' + 2 \left(f_{T_i} - f_{T_{i+1}} \right) / \Delta y \right]. \end{aligned} \tag{145}$$

Function tables of types 4 and 9 are useful whenever a function must be represented smoothly. However, the user must be aware that no checking on the validity of the relation between the function and its derivative is done in FEQ computations. Improper selection of the function and its derivative for tabulation when the function is close to zero can result in an interpolated value that is negative when such a function value is invalid. Many functions of interest in unsteady-flow analysis are not only positive but are always increasing. Linear interpolation of functional results will yield values that are always increasing. Linear integrated interpolation will yield values that are always increasing if the derivative, as tabulated, is always positive. To avoid nonincreasing interpolated values when piecewise cubic interpolation is applied, all tabulated derivatives must be nonnegative and the absolute value of the derivative at either end of the piecewise cubic part of the function must never exceed three times the absolute value of the straight line slope between the function values; that is, $f_{T_i}' \geq 0$ for all i and $\left| f_{T_{i+j}}' \right| \leq 3 \left| f_{T_{i+1}} - f_{T_i} \right| / \Delta y$ for $j = 0,1$ for each interpolation interval.

11.1.4 Type 11

The speed and direction of the wind velocity used to compute the shear stress on the stream surface is tabulated in table type 11. The direction, given in terms of azimuth from north, follows the normal convention for wind; that is, the direction from which the wind is coming, not the direction that the wind is going. Thus, one argument and two function values are in the table, both function values are interpolated linearly as in table types 2 and 7.

11.1.5 Cross-Section Function Tables (Types 20-25)

Six cross-section function tables are supported in FEQ. The type numbers assigned are from 20 to 25. Cross-sectional hydraulic characteristics as a function of the water-surface height in the cross section are tabulated in all table types. Therefore, these function tables are 1-D because only one argument is included. However, all tables contain more than one function value for each argument value.

Depth, top width, area, square root of conveyance, and the momentum-flux correction coefficient are tabulated in all six table types. The first moment of area about the water surface is added in type 21. The first moment of area about the water surface, the energy-flux correction coefficient, and critical flow rate are added in type 22. Types 23 through 25 are similar to types 20 through 22 with the addition of the correction factors for channel curvilinearity. Table contents are summarized symbolically as follows:

Type 20: $y, T, A, \sqrt{K}, \beta$
Type 21: $y, T, A, \sqrt{K}, \beta, J$
Type 22: $y, T, A, \sqrt{K}, \beta, J, \alpha, Q_c$
Type 23: $y, T, A, \sqrt{K}, \beta, M_A, M_Q$
Type 24: $y, T, A, \sqrt{K}, \beta, J, M_A, M_Q$
Type 25: $y, T, A, \sqrt{K}, \beta, J, \alpha, Q_c, M_A, M_Q$

The interpolation mode for each of the cross-sectional characteristics is listed in table 5.

Table 5. Cross-sectional characteristics and interpolation method in the Full EQuations model

Element	Definition	Interpolation method
T	Top width	Linear
A	Cross-sectional area	Integrated linear
$\sqrt{K}$	Square root of conveyance	Linear
β	Momentum-flux correction coefficient	Linear
J	First moment of area about the water surface	Integrated quadratic
α	Energy-flux correction coefficient	Linear
Q_c	Critical flow rate	Linear in logarithms
M_A	Area correction for curvilinear channels	Linear
M_Q	Discharge correction for curvilinear channels	Linear

Integrated quadratic interpolation follows the concept of integrated linear interpolation. The first moment of area about the water surface is given by the integral of the area. This results in

$$\hat{J}(y) = J_i + p_I \Delta y \left[A_i + \hat{A}(y)\right]/2 - (p_I \Delta y)^2 \left[\hat{T}(y) - T_i\right]/12 . \tag{146}$$

Equation 146 is the trapezoidal rule with end corrections and is exact for polynomials of third order (cubic) or less. Therefore, the area is integrated exactly because the area as defined by integrated linear interpolation is a quadratic polynomial.

Linear interpolation of the critical flow in logarithms of flow and depth implies that Q_c is a piecewise power function of water-surface height; that is, for each tabulation interval (y_i, y_{i+1}),

$$\hat{Q}_c(y) = ca_i y^{cb_i} , \tag{147}$$

where ca_i and cb_i are the coefficient and power, respectively, of the power function for interval $_i$. The power and the coefficient will vary between intervals. This interpolation for critical flow is exact for rectangular, triangular, and parabolic cross sections. It also is exact for cross sections where the top width varies as a power function of the water-surface height in the section. For other shapes of cross section, it is a good approximation over moderate ranges of water-surface height, which is acceptable for the purposes of table look up.

Cross-section function tables are input in program unit XSECIN. Table lookup is done in the subroutines XLKT*nn,* where *nn* gives the type number for the table. A function table can be applied in any context for which the table contains the required information; for example, tables of type 25 can be used even though all the characteristics are not needed. If sufficient information for a given context is not available in the selected cross-section table type, an error message is issued and processing stops.

Some obsolete table types are supported in FEQ so that streams simulated with earlier versions of FEQ can be easily adapted to the latest version. These types are types 1 and 12 for cross-section tables. They are exactly the same as the current types 21 and 22. These table types for cross-section function tables are no longer computed in FEQUTL, but they can be used in FEQ simulation. The type designation is converted on input to FEQ. A table of type 5 is computed in FEQ for internal use in abrupt expansions to extract the relation between water-surface height and critical flow from a cross-section table.

11.2 Two-Dimensional Function Tables

One-dimensional tables are limited to one argument. Therefore, only a limited range of functions can be represented in 1-D tables. Two approaches to representing more complex functions by use of 2-D function tables are provided in FEQ. The concepts underlying the two tables are illustrated by a hypothetical case of a hydraulic structure that is subject to tail-water effects.

11.2.1 Type 13

The culvert shown in figure 37 is subject to tail-water effects. Tables of type 13 are developed as follows: Initially, the headwater and tail-water surface elevations are identical, and no flow results; then, the tail water is lowered at the same time that flow rate is increased from the initial zero flow so that the headwater level is maintained. The possible variation in flow for this case is shown in figure 38. As the tail water continues to lower, the flow increases to maintain the headwater level. There is a limit, however, to the continued increase of flow as the tail water is lowered. At some point, called the free-flow limit, a critical-flow or equivalent condition will be reached so that further reductions in tail water do not result in increases in flow rate to maintain a constant headwater level. The difference between the headwater level and the tail-water level at the limit of the effect of the tail water is called the drop to free flow or the free drop. Free flow denotes flow free of tail-water effects. If the relation between flow and tail water is assessed for several initial water levels, a family of lines of constant headwater level is obtained, as shown in figure 39. The figure is divided into two regions, one-node control and two-node control. In the one-node control region, the water level at only one flow-path end node is needed to define the flow. In the two-node control region, the water levels at two flow-path end nodes are needed to define the flow.

In the two-node region, a partial free drop is used to facilitate interpolation. The dashed line that defines the boundary between the one-node and two-node regions is the line of complete or total free drop (the free-flow limit). The horizontal axis where the flow is zero is the line of zero free drop (that is, no difference between headwater and tail water). Other lines of partial free drop are shown in figure 39. For tables of type 13, headwater head and partial free drop are linearly interpolated in the two-node flow region. The flow as a function of headwater head and the partial free drop is tabulated in tables of type 13, the partial free drop varying from 0.0 to 1.0. The free drop for each headwater level also is tabulated in table type 13.

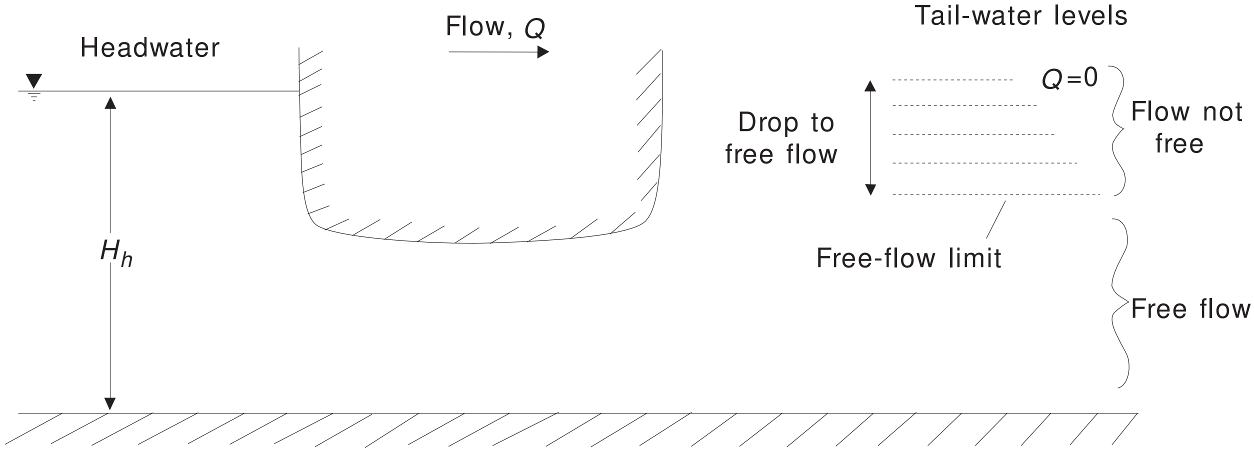


Figure 37. Defining control structure for table type 13 in the Full EQuations model.

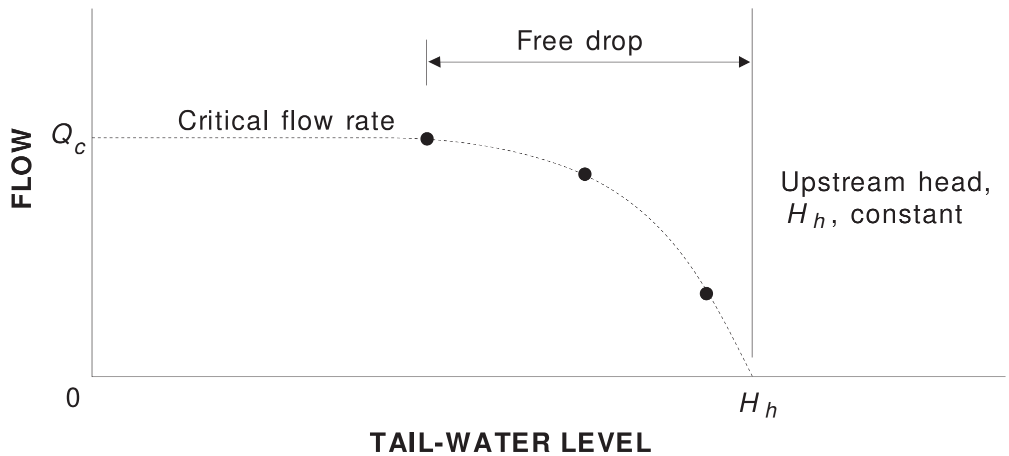


Figure 38. Flow variation for a control structure described by table type 13 in the Full EQuations model.

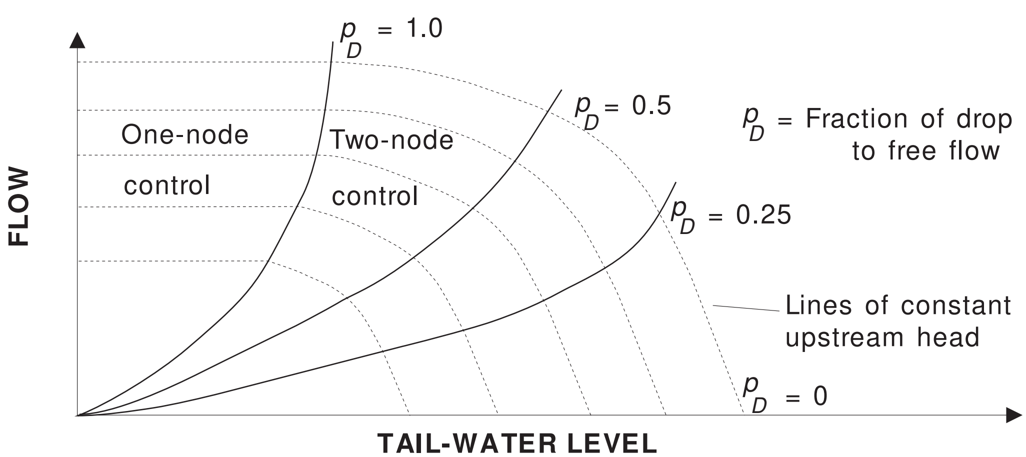


Figure 39. Flow as a function of tail water for a variable two-node control structure as depicted by table type 13 in the Full EQuations model.

Given a headwater head, $H_h > 0$ and a tail water head, H_t, the current flow rate is calculated as follows:

1. The free drop for the headwater head is computed by linear interpolation from the tabulated values of free drop and headwater head by use of the principles and concepts previously outlined. The free drop at H_h is defined as d_f.
2. The actual drop, $d_a = H_h - H_t$, is computed. This drop must be greater than or equal to zero in tables of type 13. If $d_a = 0$, then the flow is zero. If $d_a \geq d_f$, the drop is greater than the drop to free flow and the flow is under one-node control. The free flow for the given headwater head is determined. If $d_a < d_f$, then the flow is under two-node control.
3. To determine the flow, the partial free drop, $p_D = d_a/d_f$, is computed. Using the headwater head and the partial free drop, the flow is computed by bivariate linear interpolation in the tabulated values.

The requirement of a nonnegative drop from upstream to downstream for table type 13 limits its application somewhat. At some structures, notably expansions in flow, a tail-water head that is higher than the headwater head may result because velocity head recovers somewhat (kinetic energy converts to potential energy).

11.2.2 Type 14

A structure similar to that described in type 13 tables is shown in figure 40, but in this example, the downstream water level is held fixed and the upstream water level is estimated. This is the defining basis for tables of type 14. Potential variations of the headwater head as the flow is varied are shown in figure 41. Again, at some point as the flow is increased, critical or some equivalent flow results. At that point, the curve for a given downstream head must end because the relation among the three variables does not apply. For each downstream head, a maximum flow is assumed in type 14 tables above which the relation is not valid. The flow is again called the free flow, and the headwater head for this condition is the head at free flow. The relation defined by the free flow and the headwater head at free flow determines the one-node control relation for the structure. These results are shown graphically in figure 42. In the two-node control region, heads are interpolated in the stream with the downstream head and partial free flow.

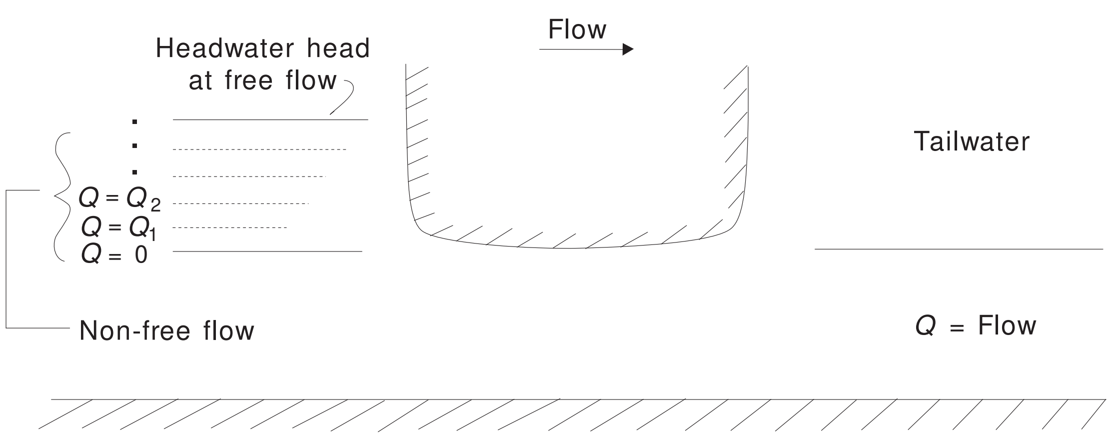


Figure 40. Defining control structure for table type 14 in the Full EQuations model.

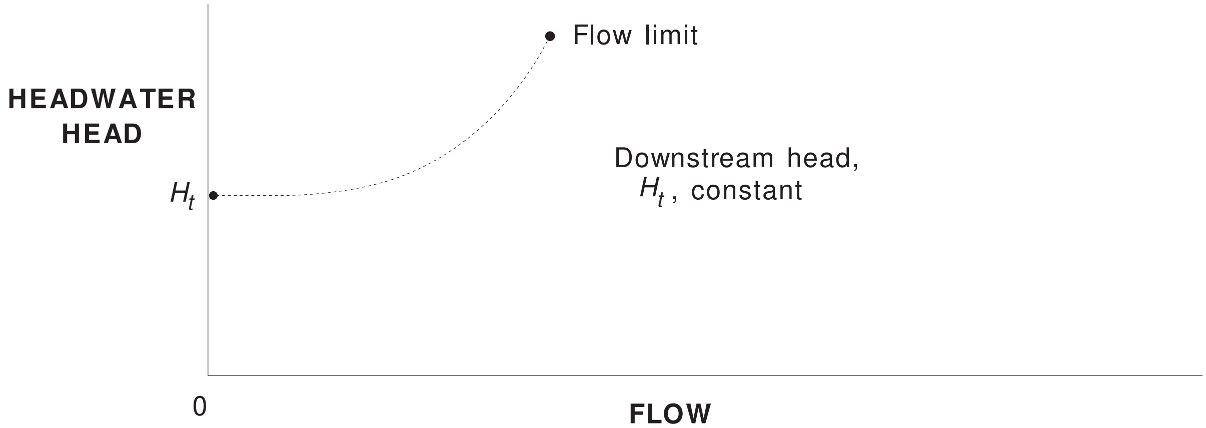


Figure 41. Flow variation for a control structure described by table type 14 in the Full EQuations model.

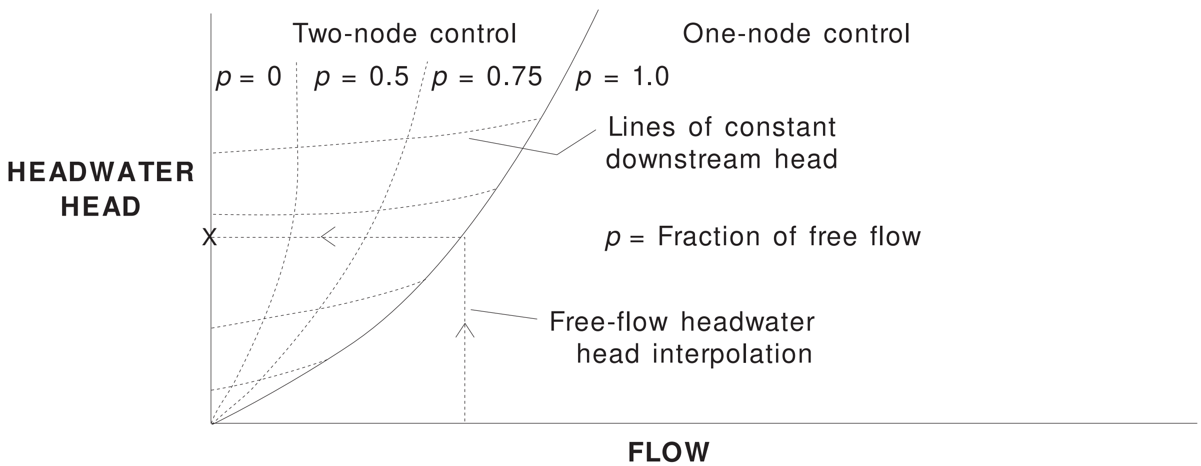


Figure 42. Flow as a function of headwater for a variable two-node control structure, as depicted by table type 14 in the Full EQuations model.

Table lookup for type 14 is more complex than for type 13. The direction of flow for type 13 was always from the higher piezometric head to the lower piezometric head. This does not apply for type 14 tables. The current flow rate determines the direction of flow. However, if the flow rate is zero, as is possible for some water levels, the direction of flow is undefined. This situation could result in problems when flow is initiated through or over a structure. In this case, the flow is zero because the water has not yet overtopped the minimum point of the structure. However, if the flow was zero in the previous time step and one or both of the heads are greater than zero in the current time step, free flow is assumed and is computed in the direction of decreasing piezometric head. This computed direction may be in error, but the error does not matter: The flow has been estimated as nonzero, and subsequent iterations in the solution for the current time step will determine the correct flow direction and magnitude.

The following steps illustrate the procedure for table lookup in a table of type 14 when the special cases involving zero flow have been excluded. If Q is the current flow with $Q > 0$ and H_t is the tail-water head, then the following steps define the headwater head:

1. The free-flow limit, Q_f, is found for the given tail-water head, H_t.
2. If $Q \geq Q_f$, then the flow is under one-node control. The headwater head at the free-flow limit is determined for Q by linear interpolation in the tabulated values of limiting free flow and the headwater head at the limiting free flow. Otherwise, step 3 is done.
3. The partial free flow, $p_F = Q/Q_f$, is computed. The headwater head is interpolated by use of bivariate linear interpolation on partial free flow and tail-water head.

Tables of type 13 and 14 for some control structures are computed in the utility program, FEQUTL (Franz and Melching, in press). An outdated 2-D table, type 6, equivalent to table type 13 with a different format, is supported in FEQ simulation. Table type 13 is preferred because it uses less space and is easier to read.

If structures described by tables of type 13 are in parallel alignment in the stream system, they may be combined into a single table of type 13 because flow rates can be added. Tables of type 14 cannot be combined to describe parallel structures because values of head cannot be added. Therefore, multiple flow paths between two nodes must be represented explicitly if tables of type 14 are used.

Two-dimensional tables of type 10 are used for the CULVERT and UFGATE command parameters in FEQUTL (Franz and Melching, in press), but tables of this type are not used in FEQ simulation. Bivariate linear interpolation for a function of two variables is applied in tables of type 10. Tables of type 10 are mentioned here for completeness. Detailed discussion and examples of tables of type 10 are given in Franz and Melching (in press).

11.3 Three-Dimensional Function Tables

Three arguments are required to represent flow through a variable gate where flow may submerge the gate opening: the water levels upstream and downstream from the gate and the gate setting. Relations are approximated in FEQ simulation with a series of 2-D tables of type 13. Each of these tables is computed for a fixed gate setting. A table of type 15, with the gate setting as the argument, lists the table number for the two-dimensional table for each gate setting and key values of head conditions used to define the state of flow through the gate. Linear interpolation is used for those key values. Together with the pair of table numbers that bracket the gate setting, the key values are then used to determine the flow from the pair of 2-D tables. Interpolation between these tables is discussed in Franz and Melching (in press).

11.4 Summary of Function Tables

Sixteen function-table types are supported in FEQ to assist the FEQ user in choosing the best means to represent the functions needed to describe complex stream systems. Additional table types can be added, if needed, with little difficulty.

A summary of the table types currently (1997) supported in FEQ is given in table 6. The arguments for the 2-D tables are given in the form consistent with the basic program units. The concepts of partial free drop and partial free flow are known only at the function-table level of model calculations.

Table 6. Summary of function tables in the Full EQuations model

[--, none]

Table dimension: 1-D, one dimensional; 2-D, two dimensional; 3-D, three dimensional

First argument: AMV, any meaningful value (for example, stage, head, elevation); YMDH, any meaningful variable with a specific year, month, day and hour designation

Values yielded: T, top width; A, cross-sectional area; $\sqrt{K}$, square root of conveyance; β, momentum-flux correction coefficient; J, first moment of area about the water surface; α, energy-flux correction coefficient; Q_c, critical flow rate; M_A, area-correction coefficient for curvilinear channels; M_Q, discharge-correction coefficient for curvilinear channels;

Table type	Table dimension	First argument	Second argument	Values yielded
2	1-D	AMV	--	Function
3	1-D	AMV	--	Function and derivative
4	1-D	AMV	--	Function and derivative
6	2-D	Head	Head	Flow
7	1-D	YMDH	--	Function
8	1-D	YMDH	--	Function and derivative
9	1-D	YMDH	--	Function and derivative
11	1-D	YMDH	--	Wind speed and direction
13	2-D	Head	Head	Flow
14	2-D	Head	Flow	Head
15	3-D	Gate setting	--	2-D table numbers
20	1-D	Depth	--	$T,A,\sqrt{K},\beta$
21	1-D	Depth	--	$T,A,\sqrt{K},\beta,J$
22	1-D	Depth	--	$T,A,\sqrt{K},\beta,J,\alpha,Q_c$
23	1-D	Depth	--	$T,A,\sqrt{K},\beta,M_A,M_Q$
24	1-D	Depth	--	$T,A,\sqrt{K},\beta,J,M_A,M_Q$
25	1-D	Depth	--	$T,A,\sqrt{K},\beta,J,\alpha,Q_c,M_A,M_Q$

12. PROGRAM OUTLINE

A brief outline of the structure of the FEQ computer program is given here. The program is written in Fortran 77 with extensions limited to those supported by most compilers. FEQ is a complex computer program consisting of about 200 program units: main program, subroutines, and functions. The structure of FEQ is based on the branch and special-feature distinction used to describe the stream system. Because the structure follows that of the simulated unsteady-flow problem, addition of support for a new set of governing equations for a branch or the addition of a new special feature is straightforward.

The program structure is presented here in outline form. This outline is presented visually in figure 43 to illustrate the general computational flow in FEQ. The indentation level and the number of levels in written form give the hierarchy of the relations among program units. The names of the programs units (given in square brackets) are given with the general unit of function. Many details are suppressed here in the interest of clarity and brevity. Many program units are not listed because they are accessed at lower levels or are accessed from various parts of the program. (This is especially true of the program units used for table lookup in the various function tables.) The hierarchy of the program units and subroutines is given below:

1. Get command-line arguments, check file names, and open the requested files. [FEQ]
2. Read user input. [INFO, INFO1, INFO2]
   - 2.1 Read Run Control Block (section 13.1). [INFO1]
   - 2.2 Read Branch Description Block (section 13.2). [BRIN]
   - 2.3 Read Tributary Area Block (section 13.3), if needed. [TRIBIN]
   - 2.4 Read Near-Zero Depth Block (section 13.5), if needed. [NZDCON]
   - 2.5 Read Network-Matrix Control Block (section 13.6). [EXIN]
   - 2.6 Develop internal description of the stream network. [SETCON, MAKJUN]
   - 2.7 Check network for proper connectivity. [DMPJUN]
   - 2.8 Develop pattern for the Jacobian matrix and allocate matrix blocks. [MAKEMC, MAKMAT]
   - 2.9 Read Point-Flows Block (section 13.7), if needed. [PFIN]
   - 2.10 Read Wind Block (section 13.8), if needed. [INFO2]
   - 2.11 Read Special-Output Block (section 13.9). [OUTIN]
   - 2.12 Read Inflow Files Block (section 13.10). [RDFIN]
   - 2.13 Read Outflow Files Block (section 13.11). [WRFIN]
   - 2.14 Read Operation of Control Structures Block (section 13.12). [OPIN]
   - 2.15 Read Function Tables Block (section 13.13). [FTABIN, ATABIN, XSECIN]
   - 2.16 Read Free-Node Initial Conditions Block (section 13.14), if needed. [EXINIT]
   - 2.17 Resolve internal references and do further validity checking. [CHKBR, CHKEX, PMPCTB, CHKPT, CHKWIN]
   - 2.18 Do cross-section table interpolation on branches. [SCAN, MERGE, INTERP]
3. Check geometry, if requested. [CHKGEO]
4. Do static (once per simulation) initializations. [STINT]
   - 4.1 Initialize buffers for inflow files, if any. [RDINIT]
   - 4.2 Read optional initial conditions file. [GETIC]
   - 4.3 Compute initial conditions if not read from file using data input in the Backwater-Analysis Block (section 13.15). [BCKWTR]
   - 4.4 Set flow states in stream-system description. [SETSTA]
   - 4.5 Initialize Operation Control Blocks. [OPINIT]
   - 4.6 Initialize time-series files, if any. [READOC, BFINIT]
   - 4.7 Initialize output files, if any. [WROPEN]
   - 4.8 Write system state to file, if needed. [BWPUT]
5. Make final check of stream system and initial conditions. [FINCHK]
6. Do dynamic (potentially more than one per simulation) initializations. [FEQ, BWGET, SETSTA, SYSVOL, FMXMN]

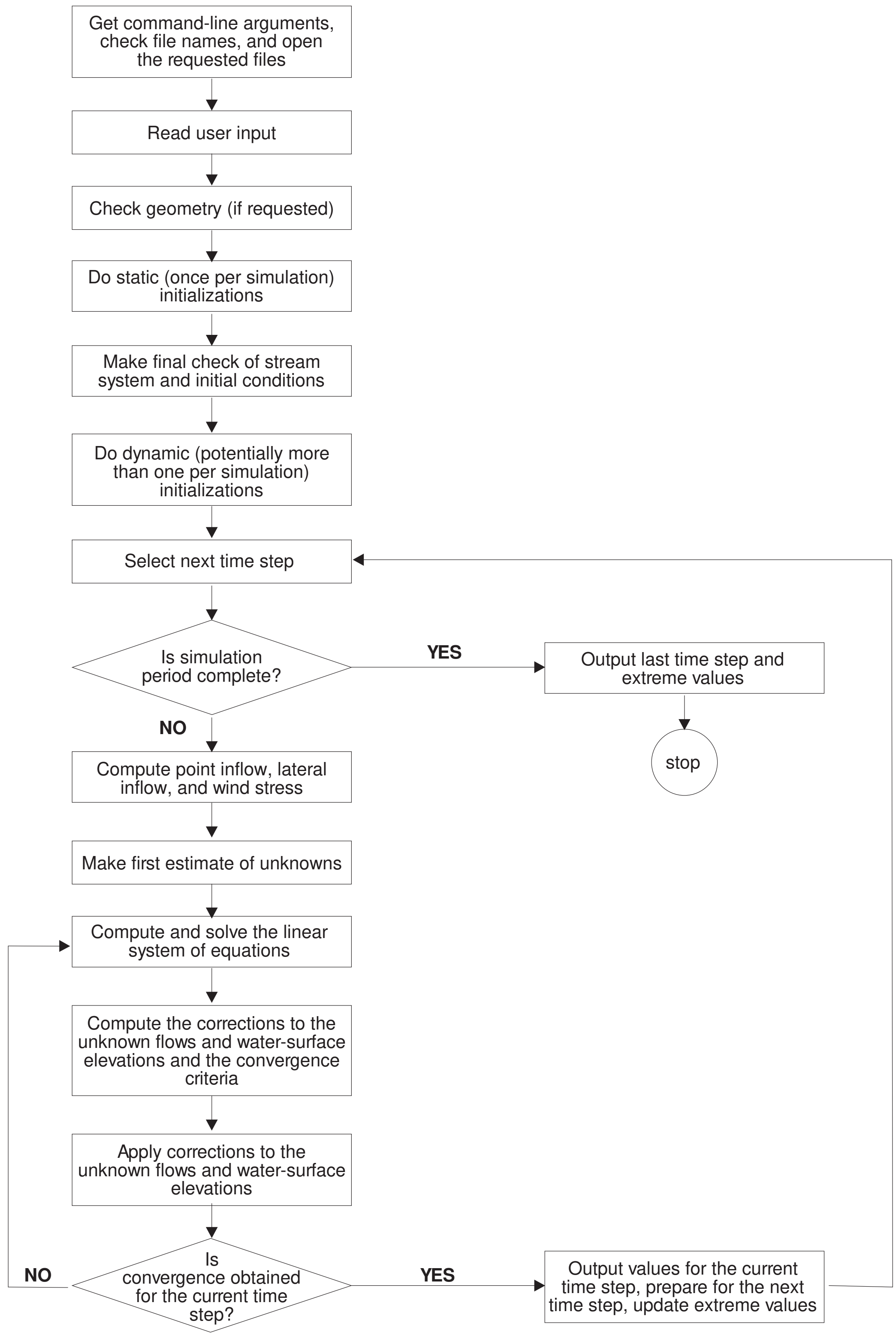


Figure 43. Major computational steps in the simulation of unsteady flow in the Full EQuations model.

7. Select next time step. If simulation period is complete, output last time step and extreme values, and stop. [MANTIM, NWDTWT, TIMINC, RESOUT, ZSUMRY]
8. Compute point inflow, lateral inflow, and wind stress. [LOAD, LKTSF, LKTAB, WSHEAR]
9. Make first estimate of unknowns. [ESTBN, ESTEN]
10. Compute and solve the linear system of equations. [CSMAT]
    10.1 Compute the Jacobian matrix and right-hand side vector. [SETEXT, SETINX, SETINW, SETICX, SETICW, CONTRL, EXCON, BDFTAB, PUMP, CBRID, ABREXP, TWOD6, TWOD14, BDFWR, MCHGAT, SIDEWR]
    10.2 Factor the Jacobian matrix. [PROFAC]
    10.3 Compute the solution. [PROSLV]
11. Compute the corrections to the unknown flows and water-surface elevations and the convergence criteria. [CMPCOR]
12. Apply the corrections to the unknown flows and water-surface elevations. [MAKCOR]
13. If convergence is obtained, output values for current time step, prepare for next time step, and update extreme values. Go to step 7 for next time step. [PRTLOG, BDYFLW, CRSET2, WROUT, OPINIT, RPLSET, RSTSTA, FMXMN, RESOUT, ZSUMRY, SYSVOL, OUTSP]
14. If convergence is not obtained, go to step 10 and repeat subsequent steps. [CSMAT]

13. INPUT DESCRIPTION FOR THE FULL EQUATIONS MODEL: VERSION 8.0

The input to FEQ is broken into blocks. A block consists of one or more tables. A table is a series of lines grouped on some basis. Each block begins with one or more lines of alphanumeric information (called a heading or headings) that delimits the block from the previous input. This is followed with the table or tables of information relevant to that block. The order of the blocks must be maintained. Furthermore, a value input in one block may require that a block of related information appear later in the input sequence to FEQ. For example, the Wind Information Block (section 13.8) is only given if WIND=YES in the Run Control Block (section 13.1).

The input blocks used in simulation of unsteady flow in FEQ are listed in table 7. The order of presentation in table 7 and in this section is the order in which the blocks must be given in the input sequence with the exception of the Function Tables Block (section 13.13). This block may be present in two locations: the location given here or before the Branch Description Block (section 13.2).

Table 7. Summary of input blocks for the Full EQuations model, in order of input

Block name	Summary of block
Run control	Run-control information
Branch description	Description of nodes on each branch
Tributary area	Areas for each land use that are tributary to each computational element
Branch-exterior node	Relation between the branches and the flow-path end nodes
Near-zero depth	Controlling value for depth and interpolation rules applied to increase computational robustness at shallow depths
Network-matrix control	Definition of the equations relating flow and depth values at all flow-path end nodes in the stream system
Point flows	Point inflows and (or) outflows to or from a branch
Wind information	Wind speed and direction and air properties
Special output locations	Specification of the nodes where output to a designated file is desired
Input files	Information describing files used to input flow or water-surface elevation at a flow-path end node
Output files	Information describing files used to output flow or water-surface elevation at any node in the stream system
Operation of control structures	Operational rules for any dynamically operated control structure
Function tables	Specification of all function tables in a user-selected order
Free-node intial conditions	Sign, elevation, and initial values of any free node
Backwater analysis	Initial flows and starting depth at the control points applied to estimate initial conditions at all nodes

Comments can be added at any line in the input if the first column of the line contains either an asterisk (*) or a semicolon (;). If the first character is an asterisk, then the comment is read and written to the output file. If the first character is a semicolon, then the comment is read but not written to the output file. All heading and label lines as described in the input must appear in the proper order. However, the addition of comments allows the input to be labeled or described in as much detail as desired.

Format Descriptions. In the line descriptions that follow the input format codes used are the same as for the Fortran language. An "X" format code indicates skipping the number of spaces given by the number preceding the X. The "A" format code indicates that a character string with no more characters given than the number following the A is to be entered. A character string is any sequence of characters that can be printed. Numbers, letters, and most special characters can be used in character strings. Character strings should be left justified in the specified field on the input line unless other instructions are given. The "I" format indicates an integer number with the maximum number of digits given by the number following the I. No decimal point should appear in an integer number. The number can appear anywhere within the assigned field. The "F" format code indicates a floating point number that can include a decimal point; in FEQ input, this number should always include a decimal point unless the number completely fills the field. The field width is given by the number following the F. A number preceding the A, I, or F format codes denotes repetition of that field width and type the given number of times. Thus, 8F5.0 indicates 8 fields each 5 columns wide and containing a number with a decimal point.

Reference to Files. Many files are used in FEQ simulation. Twenty or more files can be referenced in a complex stream system. The file described by this input description is often called the input file. However, other files will supply information to FEQ and could be considered input files. Function tables can be placed in one or more files and these files can then be referenced in the input to FEQ for access to part of the input. These files are called function table files or auxiliary files. Function tables also can be placed directly in the main input file for FEQ simulation.

Files are available that transfer flow or water-surface-elevation data between segments of the stream system simulated in FEQ. A stream system may be so large that a single model may be unwieldy. Therefore, under appropriate conditions, the user can develop two models; for example, one upstream and the other downstream from a control point. The upstream boundary for the downstream model must be supplied by the results at the downstream boundary on the upstream model. This is done in a special file developed from the results of the upstream model, sometimes referred to as a connection file or also a point-time-series file. The file name is given in the Input Files Block (section 13.10) in the input to FEQ. Sometimes these files contain water-surface elevation data. When these files contain flow data, they could be referred to as inflow files.

One or more files are developed in FEQ in the process of simulating a stream. The primary output file, commonly just called the output file, includes information on how the input has been processed. The input, as read and interpreted in FEQ, is placed in the output file together with error and warning messages. A complete record of the results also can be placed in the output file, as specified by the user. This complete record includes results at every node in the system for every time step. This file can be many megabytes in length. Usually, the output file is restricted to include summary information about the results, and detailed information is included in what is called a special-output file. An output connection file also can be developed; details are given in the Output Files Block (section 13.11).

Files are available to record the hydraulic conditions in the model calculations so that computations can begin at the exact point of where the previous simulation stopped. This feature is useful for forecasting operations. A diffuse time-series file also may be used. A point-time-series file represents the flow at a point in the stream, such as a downstream or upstream boundary. A diffuse-time-series file represents the flows that originate from an area, where no particular point can be identified as the location where the flow enters the stream system. Descriptions of point-time series files and diffuse-time series files are given in appendix 2.

13.1 Run Control Block—Run Control Table

Purpose: Parameters that are used to control the time span to be simulated, including specification of the convergence tolerances, the number of iterations allowed in the computations, and other run control information are supplied with this block.

Headings: Three lines of user-selected information to identify the input sequence.

LINE 1

Variable: NBRA

Format: 5X, I5

Example: NBRA=00005

Explanation: NBRA is the number of branches in the stream system. The maximum number of branches allowed in FEQ is specified in parameter MNBRA in the INCLUDE file ARSIZE.PRM (appendix 3). This number may be increased when necessary and FEQ recompiled.

LINE 2

Variable: NEX

Format: 4X, I5

Example: NEX=00010

Explanation: NEX gives the number of flow-path end nodes in the stream system. This number must be even because flow-path end nodes are always in pairs. The maximum number of flow-path end nodes allowed in FEQ is specified in the parameter MNEX in the INCLUDE file ARSIZE.PRM (appendix 3). This number may be increased when necessary and FEQ recompiled.

LINE 3

Variable: SOPER

Format: 6X, A4

Example: SOPER=YES

Explanation: SOPER=YES means that control structures are present, that they will be operated dynamically in FEQ simulation and, therefore, that the Operation of Control Structures Block (section 13.12) must be in the input sequence. SOPER=NO means that the Operation of Control Structures Block is not needed.

LINE 4

Variable: POINT

Format: 6X, A4

Example: POINT=NO

Explanation: POINT=NO means that no point flows are present and that the Point Flows Block (section 13.7) will not be used. POINT=YES means that point flows are present and the Point Flows Block will be used.

LINE 5

Variables: DIFFUS, MINPRT, LAGTSF, DMYEAR, DMMN

Format: 7X, A4, 2I1, 2I5

Example: DIFFUS=YES 10 1902 1

Explanation:

DIFFUS=YES means that diffuse inflows from the land area tributary to the stream system are used in the simulation. A Tributary Area Block (section 13.3) must be present in the primary input, and a DTSF in the format required in FEQ (appendix 2) must be present in the disk space for computations. DIFFUS=NO means that diffuse inflows will not be included in the unsteady-flow simulations, and no Tributary Area Block should be included in the primary input file.

MINPRT is a variable used to control the output of the normal summary information given at the completion of each time step when DIFFUS=YES. This output is not controlled by the variable PRTINT specified on Line 18. If MINPRT=0, then the summaries are printed; the summary output is suppressed if MINPRT=1. Error and warning messages are never suppressed.

LAGTSF is a time-lag variable applied to data in a time-series file. If LAGTSF is nonzero, the data in the time-series file are lagged; the result is a greater delay in water reaching the stream channels. This option is a simple means for approximating the effect of less abrupt variation of the lateral inflows to the stream. The lagging is always by one time interval of the time-series file. Instead of each runoff value for each time interval being treated as a constant, the runoff is lagged so that, for an isolated interval with runoff, the runoff increases from zero to the peak value over the interval. The runoff then decreases to zero from the peak over the subsequent time interval. Thus, the volume is not changed but the average delay time (that is, the time measured from centroid of runoff before and after lagging), is one-half the time interval. Adjacent time intervals are treated in the same manner, and superposition is used in combining the overlapping flows that result. This adjustment may not be realistic, but it can applied as a quick test to see whether greater refinement of the distribution of lateral inflows will increase the robustness of the computations.

DMYEAR is the year for the dummy event. If omitted, the year for the dummy event defaults to 1925.

DMMN is the month for the dummy event. If omitted, the month for the dummy event defaults to 1.

The dummy event—the first event in the DTSF (appendix 2)—contains a standard sequence of unit-area runoff intensities to establish a standard initial condition in the stream system. This standard initial condition is then saved in the file specified in Line 32 by the variables BWFDSN and BWFNAM and is used to reinitialize the stream system before each of the subsequent events in the DTSF. The dummy event should have either zero runoff or constant small runoff so that the simulated flows in the stream system attain a steady state at levels that will serve as the initial condition for all subsequent events. The specified year must be 1859 or later to be valid for the date computations in FEQ. The year of the dummy event should not include any other events.

The interpretation of the starting and ending times (given on Lines 9 and 10) is affected by the DTSF. The DTSF contains one or more disjoint time segments (called events) containing the unit-area runoff intensities for one or more land-cover types or land uses. These unit-area runoff intensities are multiplied by areas specified in the Tributary Area Block (section 13. 3) to derive the inflow from these land uses into the stream system. The events in the DTSF are given in ascending order of time. The first event must always be simulated to establish the initial conditions to be used for all subsequent events. The subsequent events are independent of each other and need not be simulated in any particular sequence. Thus, one can skip events and simulate only the event of interest. To do this, a starting time equal to or less than the starting time of the event of interest must be specified. The first event that has a starting time equal to or greater than the specified start time will be simulated. The ending time also is checked and computations will end at the ending time specified even if that time is within one of the events stored in the DTSF. The computations stop at the end of an event in the DTSF or at the end time specified by the user. Thus, if the entire DTSF is to be simulated, an end time known to be beyond the end time of the last event stored in the DTSF should be specified.

The information in the DTSF can come from any source, including continuous rainfall-runoff models such as the many variations of the Stanford Watershed Model, HEC-1, or one of the procedures from the U.S. Soil Conservation Service (now known as the Natural Resources Conservation Service). Data must be in the proper format in FEQ. The details of the format are presented in appendix 2, “Unformatted Data-File Structures.”

LINE 6

Variable: WIND

Format: 5X, A4

Example: WIND=NO

Explanation: WIND=NO means that the effect of wind-shear stress on the water surfaces of the streams will not be simulated. WIND=YES means that the wind-shear stress on the water surfaces of the streams will be simulated, and a Wind Information Block (section 13.8) must be included in the primary input file in this case. Wind-shear stress is computed for branches only and cannot be computed for level-pool reservoirs.

LINE 7

Variable: UNDERFLOW

Format: 10X, A4

Example: UNDERFLOW=NO

Explanation: This variable is used to suppress underflow messages and should always be UNDERFLOW=NO. This variable is not currently used in FEQ, but it was previously needed for some compilers and operating systems.

Some compilers and operating systems, by default, issue messages for each computational underflow resulting during the execution of a program. Many thousands of underflow messages may be generated in FEQ simulation. Issuance of these messages only slows the computations and clutters the output file. These messages can be suppressed by calling a compiler- and operating-system-specific routine. Therefore, this line is retained for possible future use.

LINE 8

Variable: ZL

Format: 3X, F10.0

Example: ZL=1.200

Explanation: ZL is the cutoff depth for applying the zero-inertia approximation in a computational element to simplify computation of supercritical flows. Inertia terms in the governing equations are suppressed whenever the depth at either end of the computational element is less than ZL. Unless special problems arise, ZL should be set to zero.

LINE 9

Variables: SYR, SMN, SDY, SFRAC

Format: 6X, I4, 1X, I2, 1X, I2, 1X, F12.0

Example: STIME=1980/01/20:12.0

Explanation: The starting time for the current simulation is specified with this line.

SYR is year (all four digits),

SMN is number of the month (two digits),

SDY is day of month (two digits), and

SFRAC is hour of the day.

The hour of the day can range from 0.0 to 24.0. This convention permits two different time designations for the boundary between consecutive days. For example: 1982/06/21:0.0 and 1982/06/20:24.0 refer to the same time point. In another example, the year can differ if the day boundary also is a year boundary: 1982/12/31/24.0 and 1983/01/01/0.0 refer to the same time point. The time designation to apply in a given context is user dependent. Starting times of zero and ending times of 24.0 are probably the most appropriate in many cases.

The example gives noon on the 20th day of January 1980 as the starting time. The year should be 1859 or later for internal date/time computations to function properly.

LINE 10

Variables: EYR, EMN, EDY, EFRAC

Format: 6X, I4, 1X, I2, 1X, I2, 1X, F12.0

Example: ETIME=1981/12/31:14.56

Explanation: The ending time for the current simulation is specified with this line. Values are defined analogously to the starting times on Line 9.

LINE 11

Variable: GRAV

Format: 5X, F10.0

Example: GRAV=32.2

Explanation: The acceleration due to gravity is specified in the proper system of units. Two systems of units are supported in FEQ. If GRAV=32.2, then English units are used. In this case, the internal units are lengths in feet, areas in square feet, volumes in cubic feet, and times in seconds. Volume units in the English system are in acre-feet when printed. SFAC (specified on Line 24) is used to convert cross-section stations from the unit used in the input to the internal model unit. The square of SFAC is used to convert tributary area from the input unit to the internal unit. Tables prepared and input by the user may be in any units the user prefers. A multiplying factor specified by the user is applied to convert the values input to the internal units for FEQ simulation. For example, storage values for level-pool reservoirs may be input in acre-feet with an accompanying multiplying factor of 43,560 to convert acre feet to cubic feet. The unit of the unit-area runoff intensity from a diffuse time-series file must be consistent with the internal units; that is, feet per second. Example tables may be obtained by electronic retrieval from the World Wide Web (WWW) at http://water.usgs.gov/software/feq.html and by anonymous File Transfer Protocol (FTP) from water.usgs.gov in the pub/software/surface_water/feq directory.

If GRAV= 9.8 the internal units are meters for lengths, square meters for areas, cubic meters for volumes, and time in seconds. The volume unit for printing volumes is 1,000 cubic meters. Again, multiplying factors are available for making conversions between the units in the input and the internal units.

LINE 12

Variable: NODEID

Format: 7X, A4

Example: NODEID=YES

Explanation: NODEID=YES means that an identification string of as many as eight characters may be given in the Branch Tables for each node on a branch. Free nodes described in the Free-Node Initial Conditions Block (section 13.14) also may have an identification string of the same length. The string may be left blank if no printed node identification is desired. The identification string is printed with the node when results are reported. NODEID=NO means that no identification string can be given with the nodes. NODEID=YES should be used because clearer output is produced if node names are carefully selected.

LINE 13

Variable: SSEPS

Format: 6X, F10.0

Example: SSEPS=0.10

Explanation: SSEPS is the convergence tolerance for the volume of water in ponding storage when sewers are simulated. The maximum relative change in surcharge storage volume allowed anywhere in the system during the iterative computation of the surcharge storage is specified with SSEPS. If SSEPS is made too small, convergence problems will result. If SSEPS is too large, inaccurate ponding volumes may result.

LINE 14

Variables: NAME, PAGESP

Format: A4, 1X, I5

Example: PAGE=00022

Explanation:

NAME is the identifying character string.

PAGESP is the number of lines per page for the special output file. The maximum number of lines per page in the special outfile allowed in FEQ is specified in parameter MSPKNT in the INCLUDE file ARSIZE.PRM (appendix 3). This value should probably not be changed.

Additional information is presented in the input description for the Special Output Locations Block (section 13.9).

LINE 15

Variable: EPSSYS, ABSTOL, FAC

Format: 7X, 3F5.0

Example: EPSSYS=0.0500.010

Explanation: The convergence tolerances for the matrix solution are specified in this line.

EPSSYS is the primary relative tolerance specifying the maximum relative change in the value of an unknown (flow or water-surface height depending on context) between iterations. The maximum relative change as computed is based on the last correction from Newton's method. The last correction is made so that the relative error in the unknowns is probably much less than the stated tolerance. Therefore, the tolerance should not be made too small. Some simulation experiments have shown that a change from 0.05 to 0.005 for EPSSYS changed the computed maximum water-surface elevations by about 0.02 ft or less. The flows differed by a few percent at most. The execution time with EPSSYS=0.05 was about 30 percent less than the execution time with EPSSYS=0.005.

ABSTOL is a tolerance on the change in water-surface height at nodes on branches in the stream system. If the absolute value of the change is less than ABSTOL, then the relative change for that variable is treated as if it were zero. ABSTOL must be nonzero if small depths are to be simulated. ABSTOL provides a criterion for small depths similar to that of QSMALL for low flow rates. The analogous tolerance for level-pool reservoirs, RESEPS, is set to 0.001 ft.

FAC is a factor used to establish secondary relative tolerance. If input for FAC is omitted, then FAC is set at 2.0. The secondary tolerance is computed by multiplying the primary relative tolerance (EPSSYS) by FAC. This secondary relative tolerance is used when NUMGT (given in next input line) is positive. Convergence is obtained if all but NUMGT variables satisfy the primary relative tolerance and the NUMGT variables that exceed the primary relative tolerance are all less than the secondary relative tolerance. The secondary relative tolerance can sometimes be used to increase the robustness of the computations in FEQ.

LINE 16

Variables: MKNT, NUMGT

Format: 5X, 2I5

Example: MKNT=00005 1

Explanation:

MKNT is the maximum number of iterations for the matrix system solution. If MKNT is exceeded, then the time step is reduced by multiplication with the LFAC value given on Line 28 and convergence is attempted again. This process continues until either convergence is obtained or the time step becomes less than the minimum time step given on Line 28. The maximum number of iterations allowed in FEQ is specified in the parameter MNITER in the INCLUDE file ARSIZE.PRM (appendix 3). The value of MNITER should be changed only as a last resort to obtain computational convergence. If MNITER is changed, then FEQ must be recompiled.

NUMGT is the number of nodes at which the secondary relative convergence criterion may be applied. Use of NUMGT solves a frequent problem of robustness. Convergence of the computations will frequently be prevented by problems at only one or two nodes; all other nodal variables may meet the convergence criterion for relative change by a large margin, yet convergence cannot be declared for the system until all nodal variables meet the criterion. Test simulations in which statistics on the relative change for the nodal variables were calculated and printed indicated that, in many cases, problems in only one or a few nodes can greatly delay or prevent convergence. Convergence can be obtained if the number of variables not meeting the current primary convergence criteria does not exceed NUMGT and if these variables satisfy the secondary relative tolerance defined in Line 15. The number of variables not meeting the current convergence criteria is printed in the iteration log. The value NUMGT should be small. A value of 1 generally results in greatly increased robustness and only minor effects on the final computed flows and depths. In some cases, the only way to obtain convergence without greatly modifying the stream-system representation is to apply NUMGT. The iteration log should be checked to determine when, where, and how often NUMGT is used in simulation. In most cases, relative changes will not exceed the convergence criterion. Often, cases in which the criterion is exceeded at one node result during low-flow periods and will have little effect on the higher flows and depths of interest.

LINE 17

Variables: OUTPUT, PROUT

Format: 7X, I5, 1X, I5

Example: OUTPUT=00001

Explanation:

OUTPUT controls the level of detail in the output. Setting OUTPUT equal to zero results in the minimum output detail, and setting OUTPUT equal to 5 results in the maximum output detail for debugging of simulation errors. A value of zero is used in normal operation. A value of 1 results in a complete echo of the cross-section tables in addition to the headings as for OUTPUT=00000. A dump of some internal table values for checking purposes also results for OUTPUT=00001. For values of OUTPUT greater than 1, the actual computations are detailed in the output from FEQ. OUTPUT=00005 should be applied sparingly because large amounts of output are produced. If OUTPUT is greater than 1, then Lines 19 and 20 must be in the input sequence.

PROUT specifies the output level applied after the simulation time has passed the optional date/time given after PRTINT on Line 18.

LINE 18

Variable: PRTINT, YR, MN, DY, HR

Format: 7X, I5, 1X, I4, 1X, I2, 1X, I2, 1X, F10.0

Example: PRTINT=00005

Explanation:

PRTINT controls the frequency of output of the stream-system results. PRTINT=00001 means that output is printed every time step, PRTINT=00002 means that output is printed every second time step, PRTINT=00003 means that output is printed every third time step, and so forth. The value of PRTINT is set to 1 so that the complete results of every time step are output. (This setting should be used only for debugging of simulation errors.)

The date/time value after which output is printed with the value of PROUT instead of the value of OUTPUT (Line 17) is specified as follows:

YR is the four-digit year,

MN is the month number,

DY is the day number, and

HR is the hour of the day with possible fractional hours.

LINE 19

Variable: STPRNT

Format: 7X, I5

Example: STPRNT=00010

Explanation: STPRNT is the number of the time step at which detailed output is to start when OUTPUT is greater than 1. A more useful option for debugging is application of the options on Lines 17 and 18.

LINE 20

Variable: EDPRNT

Format: 7X, I5

Example: EDPRNT=00100

Explanation: This variable specifies the number of the time step at which detailed output is to stop when OUTPUT (Line 17) is greater than 1. A more useful option for debugging is application of the options on Lines 17 and 18.

LINE 21

Variables: NAME7, NAMEGE

Format: A7, A5

Example: GEQOPT=STDX

Explanation:

NAME7 is the identifying character string “GEQOPT=.” The string GEQOPT= must be used because both parts of the input are read; if the first six characters are not GEQOPT, then a warning message is issued and the option STDX is selected.

NAMEGE is the default Governing EQuation OPTion for the branches in the stream system. The options for the governing equations are: STDX, STDW, STDCX, and STDCW. These options are described in detail in section 6.2. If STDW or STDCW are selected, then the Near-Zero-Depth Block (section 13.5) must appear in the input immediately before the Network-Matrix Control Input.

LINE 22

Variable: EPSB

Format: 5X, F10.0

Example: EPSB=0.00001

Explanation: EPSB is the convergence tolerance for the steady-state flow analysis used to define the initial conditions in the stream system. This value should be small to produce small residuals in the steady-flow governing algebraic equations because this residual is, in effect, multiplied by the time step in the unsteady-flow solution. A large residual may require extra computational effort during the frozen-time stage of the FEQ computations.

LINE 23

Variable: MAXIT

Format: 6X, I5

Example: MAXIT=00020

Explanation: This variable specifies the maximum number of iterations permitted in the steady-state flow analysis to define the initial conditions in the stream system.

LINE 24

Variable: SFAC

Format: 5X, F10.0

Example: SFAC=5280.0

Explanation: SFAC is the multiplying factor for converting the stations for the nodes along the stream system given in the Branch Description Block (section 13.2) to the internal units. The combination of SFAC and GRAV (Line 11) determine the internal units. If GRAV=32.2, English units are used and internal lengths are in feet, areas are in square feet, and volumes are in cubic feet. Time is in seconds. For convenience, SFAC is used to allow different units for the stations along the stream and for tributary areas. SFAC is applied to multiply whatever values are used for the stations, and SFAC×SFAC is used to adjust the tributary areas. If GRAV=32.2 and SFAC=5,280, then stations along the stream are given in miles and tributary areas are given in square miles.

LINE 25

Variables: QSMALL, QCHOP

Format: 7X, 2F10.0

Example: QSMALL= 50.0 0.01

Explanation:

QSMALL is the value of flow rate to add to a flow before computing a relative change in the flow. Use of QSMALL prevents a division by zero if the flow is zero. QSMALL should be small relative to the flows of interest, but it should not be made too small because unneeded computational effort is required for reducing the relative changes between iterations to a value smaller than needed. A value on the order of a few percent of the maximum flows expected is generally reasonable.

QCHOP is the truncation level for output of flows. Any flow less than or equal to QCHOP is set to zero. This option is used to eliminate the output of small flows that result from roundoff during the computations. The default value for QCHOP is 0.001.

LINE 26

Variable: IFRZ

Format: 5X, I5

Example: IFRZ=00005

Explanation: Losses from flow through most control structures are not currently included in the steady-flow analysis used to determine initial conditions for simulation of unsteady flow. Therefore, simulated time is “frozen” for the number of time steps given by IFRZ to yield initial conditions that do include these losses. Because time is frozen, all the inflows and outflows that are functions of time remain constant. The following line gives the time steps to be used during this process. The concept of frozen time is explained in detail in section 8.3. The maximum number of frozen time steps allowed in FEQ is specified in the parameter MNFRDT in the INCLUDE file ARSIZE.PRM (appendix 3). The number may be changed, but FEQ must then be recompiled.

LINE 27

Variables: DTVEC(*)

Format: 10F8.0

Example: 1800. 1250. 850. 600. 400. 275. 200. 150. 100.

Explanation: The IFRZ time-step values to apply while time is frozen are specified on this line. The first time step is given last, and the last time step is given first. Normally, the first time step in frozen time should be small, and the time step should become progressively larger for each frozen-time period simulated. This line is not present if IFRZ=0.

LINE 28

Variables: MAXDT, MINDT, AUTO, SITER, HIGH, LOW, HFAC, LFAC

Format: 8F5.0

Example: 1800 0.1 0.7 2.9 3.2 2.4 2.0 0.5

Explanation: The parameters used to select time steps are specified on this line.

MAXDT is the maximum time step allowed in FEQ simulation of unsteady flow.

MINDT is the minimum time step allowed in FEQ simulation of unsteady flow.

AUTO is a weighting factor used to compute a weighted running sum of the number of iterations, which is applied to detect convergence problems. AUTO must be in the range of 0 to 1.

SITER is an initial value for the computation of the weighted running sum of the number of iterations.

HIGH is the upper bound on the weighted running sum of the number of iterations. If the running sum is greater than HIGH, then the time step is decreased.

LOW is the lower bound on the weighted running sum of the number of iterations. If the running sum is less than LOW, then the time step is increased.

HFAC is a multiplier used to increase the computational time step to improve computational convergence.

LFAC is a multiplier used to decrease the computational time step to improve computational convergence.

Time steps are selected during model computation on the basis of the computational effort required to reach convergence in the recent iterations. When the computations go smoothly, convergence is attained in one, two, or sometimes three iterations. However, as the flows and stages in the stream change, the current time step that previously resulted in reasonable convergence may prove to be too large or too small. If the current time step is assumed to be too large for the current flow conditions, then the number of iterations required to obtain convergence will increase. If the current maximum number of iterations, MKNT (Line 16) is exceeded, then the current time step is multiplied by LFAC to yield a smaller time step. The computations are then repeated at this smaller time step. In this process, the time step may become less than MINDT; a warning message is written and simulation stops. For smaller changes in the time step, the number of iterations may not be greater than MKNT, but excessive iterations may result. In such cases, computation with a smaller time step is probably better to promote more rapid convergence. The opposite situation (continuing to apply a small time step even when the computations converge with only one iteration) also must be avoided.

In order to detect convergence problems, a weighted running sum, SITER, is computed from the number of iterations to reach convergence or to exceed MKNT iterations. The running sum is updated by use of the relation SITER=AUTO×SITER + (1 − AUTO)×ITER, where ITER is the current iteration count. To keep SITER from growing without bound, it is required in FEQ that $0 \leq \text{AUTO} < 1.0$. As an example, assume that the number of iterations, ITER, remains constant for many time steps. In this case, the value of SITER will approach ITER, and the speed of approach will depend on the value of AUTO. If AUTO is 0.0, then SITER is the number of iterations to convergence for the previous time step. This value of SITER may result in time-step changes that are too rapid. In contrast, if AUTO is nearly 1.0, the value of SITER will change only slowly and convergence will be obtained slowly.

With SITER determined, the time step is unchanged if $\text{LOW} \leq \text{SITER} \leq \text{HIGH}$. If SITER > HIGH, then the time step is multiplied by LFAC to reduce it. If SITER < LOW, then the time step is multiplied by HFAC to increase it. The goal is to keep the number of iterations at about 2. Keeping time steps so large that the number of iterations increases is not useful because convergence of the iterations does not mean that an accurate solution has been obtained. Suggested values are AUTO=0.7, HIGH=3.2, LOW=2.4, HFAC=2.0, and LFAC=0.5. The value given for SITER should be midway between HIGH and LOW. The maximum time step allowed, MAXDT, should be selected on the basis of the expected rate of change of flow during the simulation. If MAXDT is too large, major events may be missed; and if MAXDT is too small, unneeded computations will result. MINDT should be set small so that short-duration computational problems will not result in premature termination of the simulation. Values of MINDT of 1 second or less are suggested from modeling experience.

LINE 29

Variables: MRE, FAC1

Format: 4X, 5F10.0

Example: MRE=0.1 1.0

Explanation:

MRE is the maximum relative change permitted in a variable during extrapolation.

FAC1 is a factor used to regulate the degree of extrapolation from previous values at each node to aid in the solution of the equations. If FAC1=1.0, then full linear extrapolation in time from the two previous values at each node is used to estimate the unknowns at the end of the current time step. If FAC1=0.5, one-half the extrapolation is used and so on. If diffuse flows are included in the simulation and are dominant, then the best results are obtained when FAC1=0. If diffuse flows are not included in the simulation, then use of FAC1=1.0 often reduces the time required to complete the computations.

LINE 30

Variable: DWT

Format: 4X, F10.0

Example: DWT=0.1

Explanation: DWT is the change in the current value of W_T, the weight factor for approximating integrals with respect to time. DWT and BWT (described in Line 31) are used to vary the value of W_T depending on the conditions simulated in the computations. Whenever the time step must be reduced because convergence was not obtained within MKNT iterations, the current value of W_T is incremented by DWT. If the resulting value for $W_T > 1.0$, then $W_T = 1.0$. Whenever the time step is decreased because SITER < LOW, then the current value of W_T is decreased by DWT. If the resulting value of $W_T <$ BWT, then $W_T =$ BWT. Varying W_T during the computations balances the need for accuracy and the need to damp nonlinear oscillations. A fixed value of W_T can still be applied throughout the simulation by setting DWT=0.0.

LINE 31

Variable: BWT

Format: 4X, F10.0

Example: BWT=0.5

Explanation: BWT is the base value of the weight factor for approximating integrals with respect to time. The valid range for this factor is $0.5 \leq$ BWT ≤ 1.0. Values in the range of 0.55 to 0.7 represent a balance between accuracy and damping of nonlinear oscillations.

LINE 32

Variables: BWFDSN, BWFNAM

Format: 7X, I2, A64

Example: BWFDSN=15

Explanation:

BWFDSN is the Fortran unit number (see appendix 3) for the file used to save the initial conditions when DIFFUS=YES. A value is read but not used in FEQ when DIFFUS=NO.

BWFNAM is the name for the file opened in FEQ simulation to store the initial conditions needed for events in the DTSF. If BWFNAM is not blank, then a file will be opened with a name given by the contents of BWFNAM (in most microcomputers). On IBM mainframes, the name given in BWFNAM will be the ddname for the DD statement defining the data set. If BWFNAM is blank, then an implicit open procedure will be done on an IBM mainframe if the proper DD statement defining the unit number given by BWFDSN is present. In some microcomputer systems, the user will be prompted for the file name in this case, whereas execution of FEQ will be aborted on other microcomputer systems. This file is written at the end of the dummy event in the DTSF. The dummy event is always the first event and usually has been set in 1925. If an event prior to 1925 is simulated, then the date for the dummy event must be given on Line 5.

LINE 33

Variable: CHKGEO

Format: 7X, A4

Example: CHKGEO=YES

Explanation: CHKGEO is used to select geometric checking. If geometric checking is selected, then execution of FEQ is terminated at the end of the checking process so that changes to the input can be made.

The unsteady-flow computations in FEQ are often sensitive to the variations in channel geometry. If the geometric variations among cross sections are too large, then a solution may not be obtained or an invalid solution may result, producing very large Froude numbers that have no basis in physical reality. The governing equations are approximated by use of simple rules of numerical integration in each computational element. If the changes in top width, area, or conveyance from the upstream node to the downstream node of the element are too large, then large errors in the approximations are introduced. The list of depths that appear in the cross-section tables at each end of each computational element in the system are checked in CHKGEO. For each of these depths, the ratio of the downstream hydraulic-characteristic value to the upstream hydraulic-characteristic value is computed, where the hydraulic characteristics are top width, area, and conveyance. If this ratio is too small or too large, then a message is printed alerting the user to the value of the ratio.

The absolute change in the cross-sectional hydraulic characteristics from one end of the computational element to the other is important, but the rate of change of these elements with distance at a constant depth also is important. Natural channels commonly include rapidly expanding or contracting reaches. The Manning's roughness values used to define the conveyance of the channels normally relate to the losses resulting in straight reaches. Additional losses must be included in reaches that are rapidly expanding or contracting. This input is considered in FEQ; the rate of change of top width and the rate of change of area, normalized to be in the same units as the rate of change of top width, is computed in CHKGEO as an indicator of the need to compute additional losses. If the rates of change are too large or too small, then a message is printed alerting the user to the value of the ratio.

LINE 34

Variable: ISTYLE

Format: 7X, A4

Example: ISTYLE=OLD

Explanation: The style of labeling flow-path end nodes is selected with ISTYLE. If ISTYLE=OLD, then the flow-path end nodes are numbered and the Branch-Exterior Node Block (section 13.4) must appear in the input. This is applied in versions of FEQ prior to 7.0. If ISTYLE=NEW, then the flow-path end nodes are labeled with a prefix character and the branch number if on a branch. An upstream branch end node is labeled with the prefix letter "U" followed by the branch number. A downstream branch end node is labeled with the prefix letter "D" followed by the branch number. Labels for level-pool reservoir and dummy-branch end nodes are prefixed by the letter "F" followed by a number chosen by the user in the range 1 to 999. As an example, the upstream end node on branch 231 is U231, and the downstream node is D231. A level-pool reservoir node and the associated inflow node could be labeled F500 and F499, respectively. The even number denotes the downstream end of the flow path represented by the level-pool reservoir.

LINE 35

Variable: EXTTOL

Format: 7X, F10.0

Example: EXTTOL= 0.5

Explanation: EXTTOL is the distance above the top of section that can be extrapolated during the unsteady-flow computations before a warning message is issued at the completion of the simulation. The extension of each cross section by the EXT option is checked to determine whether the maximum elevation of water exceeds the unextended maximum level of the cross-section table by more than EXTTOL length units.

LINE 36

Variable: SQREPS

Format: 7X, F10.0

Example: SQREPS=1.E25

Explanation: "SQREPS=" is always checked; therefore, it must be spelled as given here. SQREPS is used to solve special problems with Newton's method. An application of Newton's method will sometimes oscillate about a solution at one or more nodes that are close to a point of inflection on a *n*-dimensional solution surface, where *n* is the number of unknowns. In this case, the correct direction for the correction has been determined in Newton's method, but the magnitude of the correction is too large. A sketch of an example of this problem with one unknown is shown in figure 18B. The sum of the squares of the residuals is computed from all of the residual functions in the nonlinear system of equations describing the flow in the model of the stream system. If the magnitude of the correction computed from Newton's method is small enough, the sum of squares of the residuals should be smaller from iteration to iteration. The tolerance on the change in the sum of squares is given in SQREPS. If SQREPS=00000, then the sum of squares must stay the same or decrease to avoid reducing the Newton correction in simulation to what is called a partial Newton correction. If SQREPS=10000, then the sum of squares must increase by more than 10,000 before a partial Newton correction is applied.

The recommended procedure is to suppress partial Newton corrections until necessary. Therefore, if the value for SQREPS is left blank, a default value of 1.E30 is assigned, effectively suppressing any partial Newton corrections. Partial Newton corrections may be helpful if the maximum relative correction is at the same node and if the solution is close to convergence for successive iterations at that node. The corrected values of the sum of squares also should oscillate, at least approximately, between two values, both of which are close to but still not within the convergence criteria. The time step will continue to be reduced until the computations are terminated because the minimum-time-step limit has been reached. In this case, reducing SQREPS to a smaller value might circumvent the convergence problem. The iteration log, printed for each time step, includes (under the heading SUMSQR) the sum of squares of the residuals for the equation system. Examining these values for several time steps for which the computations did converge can help determine what size of tolerance would likely result in convergence. Too small a tolerance can result in unneeded partial Newton corrections and can lead to failure of convergence. The sum of squares does not always get smaller from iteration to iteration, especially in the earlier iterations for a time step. Therefore, partial Newton corrections should be used only in those cases that clearly indicate oscillation about a root.

LINE 37

Variables: NAME2, GETDSN, GETNAM

Format: A6, I5, A64

Example: GETIC=00009START

Explanation: The file used to define the initial conditions for starting the computations is specified in this line. This file will have been output from FEQ with the PUTFC option in a previous simulation (described in Line 38).

NAME2 is the identifying character string “GETIC=.” The string “GETIC=” must appear and must be spelled as shown.

GETDSN is the Fortran unit number (see appendix 3) for the file containing a previous FEQ simulation to be used as the initial condition for the current FEQ simulation. When the value of GETDSN is greater than zero, the initial conditions will be given in the file named after the number.

GETNAM is the name of the file containing the previous FEQ simulation to be used as the initial condition for the current FEQ simulation. If no file name is given (useful primarily on IBM mainframes), then the file is implicitly opened in FEQ following the IBM Fortran conventions. If the name is given and an IBM mainframe is being used, then the DD name is used to access the file. If this feature is not used, then the input following GETIC= should be left blank.

LINE 38

Variables: NAME3, PUTDSN, PUTNAM

Format: A6, I5, A64

Example: PUTFC=00009START

Explanation: Saving the final conditions computed in FEQ for the simulation in the named file is specified with this line.

NAME3 is the identifying character string “PUTCF=.” The string “PUTFC=” must appear and must be spelled as shown.

PUTDSN is the Fortran unit number (see appendix 3) for a file containing the final conditions computed in FEQ simulation to be used as the initial condition for a subsequent FEQ simulation. When the value of PUTDSN is greater than zero, the final conditions will be output to the file named after the number.

PUTNAM is the name of file in which the final conditions computed in FEQ simulation will be placed to be used as the initial condition for a subsequent FEQ simulation. The interpretations of the file name on various computer systems are listed in the explanation for Line 37.

PUTFC and GETIC are used for elementary operational hydrology. PUTFC is used to compile and update the hydraulic conditions as the model is run with measured inflow data. However, some forecast data may be available for the inflows, so the GETIC options with the PUTFC option left blank can be applied to estimate what the forecast implies without changing the current initial condition. The simulation used in operational hydrology must have IFRZ=0 (Line 26) to start properly. The line of time steps provided when IFRZ $>$ 0 must then be deleted (Line 27).

A further and more common use for the PUTFC and GETIC options is to use them to save an initial condition from which to start FEQ simulation. The computations are often difficult to start. Once the computations are started the boundary conditions are held fixed, by use of dummy input tables or files, until transients from the assumed conditions damp out. Applying the PUTFC option at the end of the run then saves these results for later access by use of GETIC. An initial condition can be shifted by starting from a known initial condition and specifying the input hydrographs needed to attain the final condition of interest. Saving this final condition then makes it available to start subsequent simulations. This applies only if the stream-system model does not change.

Addition or deletion of flow paths invalidates the saved initial conditions, but other changes, such as to an inflow hydrograph, do not.

13.2 Branch Description Block—Branch Tables

Purpose: A description of the nodes on each branch is supplied with this block. The location of each node on the branch is given together with the elevation of the minimum point in the stream channel. Additional data such as special loss coefficients or tables also can be given. The lines in this block must be entered for each branch in the stream system. Line 3 is repeated as needed for each node on the branch. Lines 1 and 2 are given for each branch, followed by three or more Line 3's.

Heading: One line of user-selected information. The suggested string is BRANCH DESCRIPTION; only the first six characters of the heading are read and these must be BRANCH. This block is required so that the Function Tables Block (section 13.13) may be given before the Branch Description Block, thereby allowing table lookup of the station and the elevation at one or more nodes on branches in the Branch Description Block.

LINE 1

Variables: BNUM, INERT, CFRATE, WDFAC, ADDNOD

Format: 5X, I5, 9X, A5, 8X, F5.0, 9X, A5, 8X, A5

Example 1: BNUM=00020

Example 2: BNUM=00020 INERTIA= 0.05 CFRATE= 100.WINDFAC= 1.2 ADDNOD= −1

Explanation:

BNUM specifies the branch number described in the branch table. Each branch number must be numbered but not necessarily consecutively. Thus, the number of branches and the maximum branch number do not need to be the same. For example, numbers for a two-branch system could be 100 and 200. The branch number must be between 1 and 999.

INERTIA refers collectively to the local-acceleration and convective-acceleration terms in the momentum equation. The momentum equation can be computed in FEQ by applying partial inertia terms. A value of INERTIA=1 (the default value in example 1) results in the application of the full inertial terms in the momentum equation. A value of INERTIA=0 results in the application of the noninertia approximation to the momentum equation. Setting INERTIA=0 may result in a division by zero or other computational problems. Thus, if the noninertia approximation is applied, INERTIA should be set to a value close to zero but not exactly zero.

CFRATE specifies the effective inflow area, CF, per unit length of sewer. The unit length of sewer is specified in SFAC in the Run Control Block (section 13.1). Specification of CFRATE permits CF to be assigned to each computational element in a branch in proportion to the computational-element length. If CFRATE is nonzero, then any values of CF given explicitly for the branch on Line 3 will be ignored. The default value in example 1 is 0.

WINDFAC is an adjustment factor for the wind-shear stress on the water surface in the branch. Use of this factor permits adjustment of the effective wind shear to represent the differences in exposure to wind among the branches in the stream system. The default value in example 1 is 1.

ADDNOD requests the addition of nodes to each computational element. ADDNOD=0 is the default in example 1. The added nodes are output in the detailed and summary tables produced in FEQ simulation when a positive value of ADDNOD is used. The added nodes are not output, even though they are shown in the echo of the branch table input and are used in the computations, when a negative value of ADDNOD is used. In Example 2, a node will be placed at the mid-point of each element in the branch table to form two computational elements. This additional node will not be shown in the output from FEQ; however, this node may be cited in error and warning messages. In this example, the node numbers on the branch are output as if there are no additional nodes. This is done to simplify empirical testing for convergence.

To test for convergence, model simulation is done with the selected element lengths and maximum time step. The output is saved, and ADDNOD= −1 is specified for each branch in the model. Specification of ADDNOD= −1 will nearly double the number of nodes and computational elements. To execute FEQ, array sizes must be large enough for this expansion. The maximum time step should be reduced by one-half, and another model simulation should be made. Node numbers and the number of nodes in the output will be the same. However, the second simulation should be an improved approximation to the governing equations. If desired, another simulation with ADDNOD= −3 could be made with the maximum time step being one-fourth of the original time step. Four computational elements will now be within each original computational element. The simulation time and storage required in the program also will increase. With four times the number of elements and one-fourth the time step, this last simulation will take about 16 times longer than the first simulation. The results will converge to a consistent value with succeeding simulations. Obviously, extensive convergence testing can only be done for small stream systems and for short-duration simulations. However, analyzing part of a stream system can provide insight into the distance and time steps required to obtain a reasonable approximation to the governing equations for the entire stream system.

LINE 2

Variable: HEAD

Format: A80

Example: NODE XNUM STATION ELEVATION KA KD HTAB AZM CF YC STD

Explanation: This is a heading to describe the information on subsequent lines.

LINE 3 (One for each node on the branch, at least three lines per branch)

If NODEID=NO (Line 12, Run Control Block (section 13.1)), then

Variables: NODE, XTAB, X, Z, CA, CD, HL, AZM, CF, YC, STD

Format: 2I5, 2F10.0, 2F5.0, I5, 3F5.0, F10.0

If NODEID=YES, then

Variables: NODE, NAME4, XTAB, X, Z, CA, CD, HL, AZM, CF, YC, STD

Format: I5, 1X, A8, 1X, I5, 2F10.0, 2F5.0, I5, 3F5.0, F10.0

Explanation: The values describing each node on a branch are specified.

NODE is the node number on the current branch. Node numbers must be greater than zero. A negative value for the node number indicates the end of a branch table. Numbers for nodes on a branch must be assigned in ascending consecutive order. Only the upstream node on a branch must be given a node number. Subsequent nodes will then be automatically numbered in FEQ to satisfy this requirement. This is the preferred method of input to minimize errors and user confusion. The same node number can be used for nodes on different branches, but such duplication is not advised. A useful system is to assign four-digit numbers for a node. The first two digits are the branch number, and the second two digits are the node number on the branch. To apply this numbering procedure, there must not be more than 99 branches in the system or more than 99 nodes on a branch. These conditions apply for most stream systems. For example, the first node on branch 10 would be given the number 1000 or 1001 in this system, depending on user preference of numbering from 1 or from 0. Application of this method, which provides a self-identifying number for a node on a branch with the branch number and location on the branch, can save considerable time and reduce input errors.

The compile time variable, MNBN (Maximum Number Branch Nodes), in the INCLUDE file ARSIZE.PRM (appendix 3) defines the maximum number of nodes. This number may be set to any desired positive value by the user. The value of MNBN must always include the nodes at each end of each branch and any other nodes not on the ends of the branch (interior nodes). The variable MEXTRA in the INCLUDE file ARSIZE.PRM (appendix 3) gives the count of the interior nodes on a branch. A new number does not take effect until FEQ is recompiled and linked. The range of values in current use for MEXTRA is 400 to 700.

NAME4 is the identification string for the node.

XTAB is the number of the table listing the hydraulic characteristics of the cross section at the node.

X is the station of the node.

Z is the elevation of the lowest point in the stream at the node.

CA is the loss factor to apply to the difference in velocity head in the computational element upstream from the node when the flow is accelerating with respect to distance.

CD is the loss factor to apply to the difference in velocity head in the computational element upstream from the node when the flow is decelerating with respect to distance.

HL is the number of the table listing the factor applied to the mean velocity head in the element as an estimate of point losses.

AZM is azimuth of the downstream flow direction, in degrees clockwise from north, for the computational element upstream of the node. The azimuth need only be given if the effect of wind-shear stress on the water surface is simulated. A value of azimuth propagates into blank fields. If the azimuths for all computational elements in a branch are the same, then only the azimuth for the first element must be given. The azimuth applies to a computational element, not a node. Thus, the azimuth for the first element is given at the second node on the branch. If the azimuth is the same for all elements, however, a value given for the first node on the branch will propagate to all subsequent elements.

CF is equal to A_{si}, the effective area of the inlets to a storm sewer. Guidelines for determining CF are discussed in section 13.2.2.

YC is depth of water (distance of hydraulic-grade line above invert) when ponding begins or elevation of the water surface when ponding begins for a surcharged storm sewer. Guidelines for determining YC are discussed in section 13.2.2.

STD is the standard flood elevation for computation of the valley storage for floodway computations. The valley storage below the standard flood elevation will be computed if the elevation is input, and the storage will be reported for each branch and for the entire stream system. Valley storage is the volume of water contained in the branch if the water-surface elevation is at the standard flood elevation at every node on the branch. If the standard flood elevation for the first node on the first branch is set to zero, then the valley storage will not be computed, and normal computations will continue in FEQ. The minimum input for the standard flood elevation is at the upstream and downstream node of each branch. Missing values will then be linearly interpolated. Intermediate values can be given between the upstream and downstream nodes on the branch, and missing values will be linearly interpolated by means of the given standard flood elevations.

The station and elevation given here are used in FEQ simulation unless the table option for these fields is invoked. If the Function Tables Block (section 13.13) is placed ahead of the Branch Description Block, then the string “TAB” or “tab” in either the elevation or the station column indicates that the corresponding value in the cross-section table specified on the same line of input is used in FEQ simulation. The table must be in the input and must include the desired values of station and elevation. These values are then used as if read directly in simulation.

Not all the input values must be specified for each node. The values that must be defined either explicitly by the user or implicitly in FEQ following user directions are the node number, the cross-section table number, the station, and the elevation of the bottom profile. The other fields on the line can be left blank if these features are not needed in the analysis.

13.2.1 Computational-Element Interpolation

Frequently, the spacing of measured cross sections is too large for computation of the unsteady flows. Thus, cross sections must be added to reduce the length of one or more computational elements. Two methods for adding intermediate cross sections are available in FEQ. The first method is simple propagation of the last known cross-section shape at equal station intervals and with linear interpolation for the bottom-profile elevation. This method is selected by leaving one or more blank lines in the branch-table input. Lines of complete information must be above and below the blank lines. The upstream cross-section table number is assigned to each blank line, and the stations and elevations are distributed uniformly between the two lines of known values. The second method is linear interpolation of cross-section hydraulic characteristics between two known cross sections. This method is selected by giving a negative table number for the cross-section table or, more conveniently, by giving a minus sign in the rightmost column of the field for the table number. An available table number will be assigned and supplied at the proper time in simulation. Use of the minus sign circumvents the problem of the user remembering which table numbers are available for interpolated cross sections. If the station and elevation values are given, then they will be used in the interpolation. Otherwise, the station values will be uniformly distributed, and elevations will be linearly interpolated. The first method should be applied only if the channel is prismatic over the stream reach to which cross sections are added. The second method should be applied if the channel hydraulic characteristics vary over the stream reach between the known cross sections.

Another interpolation feature for cross sections is computation of bottom-profile elevations for specified cross-section stations. The elevation of the bottom profile at the first and last node on the branch and the corresponding stations must always be given. With these first and last values given, the bottom-profile elevations are determined at the nodes for which nonzero station values were given and for which the bottom-profile elevation field was left blank. Then the station and the bottom-profile elevation for the nodes for which both the station and the bottom-profile elevation were left blank are computed at equal station intervals between specified stations. For example, if cross-section geometry is known at locations 2,400 ft apart (stations 0+00 and 24+00) and the user wishes to interpolate five cross sections between these locations, especially one at station 18+00, then the interpolated cross sections will be at stations 6+00, 12+00, 18+00, 20+00, and 22+00.

If the ADDNOD option for the branch is nonzero, then new stations, bottom elevations, and the standard flood elevations are interpolated in simulation. Cross-section tables are assigned on the basis of the specification for the unexpanded element. If the table numbers at the nodes for the element are the same, then this table number will be used for nodes added to this element. If the table numbers differ between the element to be expanded and the unexpanded element, then the cross-section table is interpolated at the added nodes. Any computational element for which a nonzero value of HL is specified cannot include added nodes. This is done because no means is available to divide the head loss represented by the table referenced by HL. All other variable values (KA, KD, YC, and CF) propagate into the new computational elements. It is best to leave the CF column blank and use the CFRATE option for the branch to assign the effective inlet areas for storm sewers when ADDNOD is applied.

13.2.2 Storm Sewers as Branches

Simulation of flow in storm sewers presents several special problems. These problems are solvable and flow can be simulated in storm and combined sewers; however, the user must supply additional information to obtain these solutions. The first problem is that the free surface disappears in a closed conduit when flowing full. The governing equations as written in FEQ are then invalid. The solution to this problem is to add a narrow slot (the so-called Preissmann slot; see Cunge and Wegner, 1964) to the top of each closed-conduit cross section so that the conduit is never closed. The slot need be only a fraction of an inch wide so that the distortion of cross-sectional hydraulic characteristics is negligible. Wave-propagation speed in a closed conduit is large and is infinite where the water is treated as incompressible, as in FEQ. If the narrow slot is applied, an infinite celerity will not result in the closed conduit, but a large celerity will result (this is all that is needed for realistic simulation). The slot must be high enough so that the section is never overtopped. The depth printed in the output will be the distance from the invert of the conduit to the hydraulic-grade line in the conduit. The problem of no free surface because of pipe surcharge is solved by always providing a hypothetical free surface.

The second problem with closed conduits is that no storage (negligible storage in the slot) is available when flowing full. As a convenience in the hydraulic analysis, the inflow from the watershed surfaces into the storm sewers is simulated separately by use of a hydrologic model. The runoff generated by rainfall and snowmelt on the land surface is simulated by this model. This runoff water will reach the channel and flow downstream; however, the state of flow in the channel is not simulated in the hydrologic model, and downstream conditions leading to surcharge of the storm sewer are not simulated. Therefore, provision must be made for some storage mechanism to represent, at least approximately, the ponding of water in streets, parking lots, basements, and other forms of depression and detention storage in the watershed. To do this, two additional parameters must be input for each computational element representing a sewer that receives inflow from the land surface. If storage is not considered, all inflow reaching the conduit inlet in a time interval will be forced into the conduit. The computed hydraulic-grade line elevation will become very large (hundreds of feet) and computational failure may result.

The first parameter, A_{si}, is an estimated effective area of the inlets to a storm sewer for the computational element. This effective area is the product of a discharge coefficient and an approximate area of inlets to the storm sewer in the computational element. This is only an approximation because the equations used to represent the inflow to and outflow from the storm sewer are for orifice flow. The flow through the inlets is orifice flow only when the ponding is deep enough to completely submerge the inlets. Realistic simulation must be maintained to avoid major errors without becoming overly concerned about physical details that are poorly understood and difficult and costly to simulate. The second parameter, YC, gives the distance between the water surface when ponding begins in the element and the invert of the conduit. This value, which does not vary with time, is an approximate value that relates to the depth of the invert below the ground surface and the assumed or actual ponding areas.

The A_{si} and YC parameters are used to determine the rate at which water can enter or leave through the conduit inlets. Thus, the inflow capacity is determined by orifice flow computed from the difference between the value of YC and the current depth in the conduit. When the current depth is the same as the YC, no water flows into the conduit. Any inflow in excess of the inflow capacity is stored in the ponding area. Water is always ponded such that the water surface is YC above the conduit invert. The capacity of the ponding area to store water is unlimited and of undefined extent and location. The ponded water will all be retained until the inflow or depth in the sewer decrease such that inflow is less than or equal to the inlet capacity. Thus, the excess water is retained without loss. This approach for ponded flow is very simple and gives only a rough approximation to the hydraulic details of a storm-sewer system. This approach is adequate, however, to yield a realistic representation of the major features of flow in a storm-sewer system. If this simple model of ponded flow is inadequate, then the surface ponding and alternative flow paths can be simulated explicitly in FEQ if sufficient data are available.

YC is most often estimated as the distance from the ground surface to the invert of the sewer pipe, y_G. The value at the downstream node on each element is used as the average value for the computational element. Therefore, if the YC values are given, the first value should appear for the second node on the branch, and the field for YC for the first node should be left blank. YC values propagate into empty fields. If the same YC value is to be used for the branch, it need be given only for the second node on the branch. More than one YC value can be given. The various YC values will propagate into any subsequent blank fields.

A more convenient option for many situations is to input an approximate land-surface elevation instead of the depth to the invert directly. Therefore, elevations of the ponding level (land-surface elevations) may be input instead of the depth to the invert. The land-surface elevation option is selected by entering a value for the first node on the branch. All other values in the column for YC are then used in FEQ as elevations. The value of y_G is computed as the difference between the ponding elevation and the invert of the conduit. If the elevation option is selected, the minimum data needed are the elevation at the first node on the branch and the elevation at the last node on the branch. Linear interpolation for intermediate points is applied in FEQ computations. Intermediate elevations also can be input, and elevations at points between known elevations will be interpolated.

13.3 Tributary Area Block—Tributary Area Tables

Purpose: The areas for each land use that are tributary to each computational element are supplied with this block. This block only appears if DIFFUS=YES in the Run Control Block (section 13.1). The unit number of the Diffuse Time-Series File (DTSF) (appendix 2) containing the runoff intensity for each land use as a function of time also is given in this block. All branches must appear in the input even if they have no tributary area. The branch mode of input is used to assign a tributary area of zero to these branches. Lines 7, 8, and 9 or 9a are repeated as needed until all branches have been described.

Heading: One line of user-selected information. The suggested string is TRIBUTARY AREA.

LINE 1

Variable: TSFDSN, TSFNAM

Format: 7X, I5, A64

Example: TSFDSN=00012\SALT\TSFLONG

Explanation:

TSFDSN is the Fortran unit number (see appendix 3) for the DTSF containing the unit-area runoff intensities on the tributary areas.

TSFNAM is the name of the DTSF containing the unit-area runoff intensities on the tributary areas. If TSFNAM is entered, then a file will be opened with a name given by the contents of TSFNAM (most microcomputers) as the DTSF. On IBM mainframes, the name given in TSFNAM will be the ddname for the DD statement defining the data set. If the TSFNAM is blank, an implicit open will be done on IBM mainframes if the proper DD statement defining the unit number given in TSFDSN is listed. In some microcomputer systems, the user will be prompted for the file name in this case, whereas program execution will be terminated on other microcomputer systems.

For convenience in conceptualizing the stream system, the tributary area contributing runoff to the branches is divided into subareas on the basis of the source of the rainfall data used to estimate the unit-area runoff intensities stored in the DTSF. These data may come from a single rain gage, from multiple rain gages, or from some other source such as weather radar. The Tributary Area Block is primarily designed to read time series of unit-area runoff computed in the Hydrologic Simulation Program Fortran, HSPF, (Johanson and others, 1984) as tributary inflow to a stream system simulated with FEQ; however, the source of the unit-area runoff data in the DTSF is not important for FEQ application. Any rainfall-runoff simulation model or approximation can be used to compute the time series of unit-area runoff in the DTSF. In FEQ simulation, the tributary-area runoff data must be stored and retrieved with respect to (1) a rain gage, (2) some subarea of the total area, and (3) land uses or land covers within that subarea. At least one rain-gage subarea must be specified in the input to FEQ, and at least two land uses must be stored in the DTSF even if only one is used in FEQ simulation. This requirement is a result of the detailed format used to store the runoff events in the DTSF, as explained in appendix 2. The relations among rain gages, land uses, and time series of unit-area runoff are described in the following discussion.

The user assigns rain-gage numbers to each rain-gage subarea for later reference. These numbers are required to start at 1 and be consecutive. Land-cover types are used in the hydrologic analysis and are associated with the rain-gage number. For example, three rain gages might be used for a given watershed, and five different land-cover types (impervious, flat-slope grassland, medium-slope grassland, steep-slope grassland, and forest) might be present in the watershed. Then, 15 combinations of rain gages and land-cover types are possible. For each time in the hydrologic computations, 15 numbers of unit-area runoff intensities from the watershed are stored in the DTSF in an order defined in the input to the utility program applied to develop the DTSF. Additional details are given in appendix 2. The order of the diffuse time series is under the control of the user, but a logical ordering should be used because the order of input for tributary areas is related to the order of appearance of the runoff values in the DTSF. Giving the runoff values in land-cover-type order for each rain gage is suggested.

The user must specify the number of land-cover types to be associated with each rain gage. This number is often the same for each rain gage, but it could vary. Also, the land-cover types do not have to be the same for each rain gage, but the user must make sure that the order of storing the runoff values in the DTSF is the same as the order in which the tributary areas are given for the land-cover types in the input.

Each computational element is assigned the runoff from only one rain gage. The tributary area for a computational element should be kept small to prevent computational problems. Computational elements can be added if several rain gages are required for one or more reaches in the stream.

The tributary areas for a branch can be input in three modes. In the simplest and least flexible input mode, branch, the user must give the total area for each land-cover type for the entire branch. This area is distributed to each element in proportion to the ratio of the length of the element to the length of the branch. The tributary area is assumed to be uniformly distributed over the length of the branch in this mode of input.

In the second input mode, station interval, the user must specify a series of upstream and downstream stations along the branch and, for each station interval, the total area for each cover type. The area is distributed over that station interval in proportion to element length. In this mode, a nonuniform distribution of tributary area over the length of the branch can be specified. The upstream and downstream stations given must match computational-element boundaries.

In the final mode of input, the node mode, the user must give the tributary area for each element explicitly. This gives complete freedom of area assignment, but the user must input the tributary area for each element on the branch. Furthermore, if elements are added in the branch description table to solve computational problems, then the user must update the tributary area input accordingly.

On completion of tributary-area input, the areas are output as assigned to each branch and element so that the user can verify that the proper result has been obtained. Furthermore, a complete summary of tributary area is given for each branch and for the entire stream system. The total area for each land-cover type, rain gage, branch, and system is given to help in verifying the input.

LINE 2

Variable: FFFDSN, FFFNAM

Format: 7X, I5, A32

Example: FFFDSN=00011\SALT\UPMS\FFF

Explanation:

FFFDSN is the Fortran unit number (see appendix 3) for the file used to store the flows and stages required for a flood-frequency analysis.

FFFNAM is the name of the file used to store the flows and stages required for a flood-frequency analysis. If FFFNAM is not blank, then a file will be opened with a name given in FFFNAM (most microcomputers) to store the values needed for flood-frequency analysis. On IBM mainframes, the name given in FFFNAM will be the ddname for the DD statement defining the data set. If the FFFNAM is blank, an implicit open will be done on a IBM mainframe if the proper DD statement defining the unit number given by FFFDSN is present. In some microcomputer systems, the user will be prompted for the file name in this case, whereas execution will be aborted in other microcomputer systems.

LINE 3

Variable: NLUSE

Format: 6X, I5

Example: NLUSE=00006

Explanation: NLUSE is the number of land-cover type and rain-gage combinations represented by the tributary areas. For example, if three land covers are in each of two rain-gage segments in the hydrologic simulation then NLUSE=00006. The maximum number of diffuse, tributary areas allowed in FEQ is specified in the parameters MNDIFA and MXGLU in the INCLUDE file ARSIZE.PRM (appendix 3). The number may be increased and FEQ recompiled.

LINE 4

Variable: NGAGE

Format: 6X, I5

Example: NGAGE=00003

Explanation: NGAGE is the number of rain gages used to define the runoff intensities on the tributary area for the watershed. The maximum number of rain gages allowed in FEQ is specified in the parameter MXGAGE in the INCLUDE file ARSIZE.PRM (appendix 3). The number may be increased and FEQ recompiled.

LINE 5

Variable: HEAD

Format: A80

Example: GAGE NCOV

Explanation: This is the heading for the land-cover table for a given rain gage.

LINE 6 (one for each rain gage)

Variable: GAGE, NCOV

Format: 2I5

Example: 1 3

Explanation:

GAGE is the rain gage number (in ascending order).

NCOV is the number of land-cover types for the given rain gage. The sum of the number of land-cover types must be the same as the value of NLUSE given on Line 3.

LINE 7 (one for each branch in combination with Lines 8 and 9)

Variable: BRA, FAC2

Format: 7X, I5, 5X, F10.0

Example: BRANCH=00001 FAC= 1.05

Explanation:

BRA is the branch number for the tributary-area table. The branch numbers in the tributary-area input must be in ascending order with no omissions. If the branch number is zero, then the table describes the areas tributary to reservoirs. If the branch number is negative, then the station-interval mode is applied to specify tributary areas. Otherwise, the branch mode of input is used. A branch number of zero may be repeated as many times as needed to apply a different factor to the tributary area for reservoirs. A warning will be issued to the user that more than one branch number of zero is in the input. If the adjustment factor for each tributary area of each reservoir is the same, then only one branch number of zero is needed. All assignments of a branch number of zero must precede nonzero branch numbers.

FAC2 is a multiplier on each of the tributary areas given in the branch. The value of FAC2 is user specified. FAC2 is typically used to check the sensitivity of FEQ output to tributary-area flows without having to repeatedly run the hydrologic model. If omitted, the default is FAC2=1.0.

LINE 8 (one for each branch in combination with Lines 7 and 9)

Variable: HEAD

Modes: Node and branch

Format: A80

Example: NODE GAGE AREA1 AREA2 AREA3 AREA4 AREA5

Mode: Station interval

Format: A80

Example: USTAT DSTAT GAGE AREA1 AREA2 AREA3

Explanation: These are user-supplied headings descriptive of the tributary areas.

LINE 9 (one for each branch in branch mode and one for each node on the branch except the upstream node in node mode in combination with Lines 7 and 8)

Modes: Node and branch

Variables: NODE, GAGE, TRIBA(*)

Format: I5, I5, 10F6.0

Explanation:

NODE delineates the downstream node for the computational element to which the specified tributary area and gage apply. The nodes on a branch in the tributary-area input must be in ascending order.

GAGE denotes the rain gage used to estimate lateral inflow from a tributary area entering a computational element in FEQ simulation. The lateral inflow is identified among the various DTSF's of runoff by the rain gage most representative of rainfall on the tributary area, denoted by GAGE, and the area consisting of a given land-cover type, denoted by TRIBA.

TRIBA specifies the area tributary to a computational element on the branch consisting of a given land-cover type. The values in the TRIBA array must be given in the order of the DTSF's. In node mode, all nodes except the upstream node must be explicitly input even if the tributary area for that branch is zero. If the branch number on Line 7 is zero, then the node input is the flow-path end node label of the simulated reservoir. In this latter case, the node list must be terminated by a −1 because only the reservoirs with tributary area must be entered in the table. As a result, the number of reservoirs included in model simulation is not known in advance. If the value for NODE is zero, then the branch mode of input for tributary area is applied, and the tributary areas given will be the total for the branch in each land-cover type.

The current limit on land-cover types that may contribute to a computational element is 10. Line 9 is repeated as needed to complete the input for the branch. The same mode of input must be used to complete the input for any branch.

LINE 9a (more than one for each branch in station-interval mode in combination with Lines 7 and 8)

Modes: Station interval

Variables: USTAT, DSTAT, GAGE, TRIBA(*)

Format: F10.0, F10.0, I5, 10F6.0

Explanation:

USTAT is the upstream station for the stational interval.

DSTAT is the downstream station for the station interval.

GAGE denotes the rainfall gage used to estimate the lateral inflow from a tributary area entering a computational element in FEQ simulation. The lateral inflow is identified among the various DTSF's of runoff by the rain gage most representative of rainfall on the tributary area, denoted by GAGE, and the area consisting of a given land-cover type, denoted by TRIBA.

TRIBA denotes the area tributary to all computational elements between the specified upstream and downstream stations in the station-interval mode consisting of a given land-cover type. The values in the TRIBA array must be given in the order of the DTSF's. The branch number given on Line 7 must be negative to select the station-interval mode.

The upstream and downstream stations both must be available in the branch-description table for the branch. Any station interval not used will have zero tributary area. Line 9a is repeated as required to describe the tributary area for the branch. The last line must be a null tributary area with the identical upstream and downstream stations. This designation indicates that input for the branch is complete.

13.4 Branch-Exterior Node Block—Branch-Exterior Node Table

Purpose: The relation between the branches and the branch-end nodes is supplied with this block so that the information in the Network-Matrix Control Block (section 13.6) will properly define a system of equations describing the flow in the stream system. This block is required only if ISTYLE=OLD is in the Run Control Block (section 13.1). It is retained for support of the stream systems modeled using FEQ versions prior to 7.0. If ISTYLE=NEW, this block should not appear in the input.

Heading: One line of user-selected information. The suggested string is BRANCH-EXTERIOR NODE.

LINE 1

Variable: HEAD

Format: A80

Example: BRAN UEXN DEXN

Explanation: This is a user-supplied heading for the subsequent lines of information.

LINE 2 (one for each branch)

Variables: BNUM, UEXN, DEXN

Format: 3I5

Explanation:

BNUM is the branch number.

UEXN is the upstream branch-end node number.

DEXN is the downstream branch-end node number.

The branches should be given in ascending order to make input easier to follow, but they need not be in ascending order.

13.5 Near-Zero Depth Block—Near-Zero Depth Table

Purpose: Optional block given only if GEQOPT=STDW or STDCW appears in the Run Control Block (section 13.1) or if these options appear for one or more branches. Only those branches that require the information must be included in the input. The controlling values of depth and the rule for interpolation for the variable weight in the distance integral approximation are given with this block to increase the robustness of computations at shallow depths.

Heading: One line of user-selected information. The suggested string is NEAR-ZERO DEPTH.

LINE 1

Variable: HEAD

Format: A80

Example: BRAN YATONE YATHAF INTERP

Explanation: These are user-supplied headings for subsequent information on Line 2.

LINE 2

Variables: BRAN, YATONE, YATHAF, CHAR8

Format: I5, 2F10.0, A8

Explanation: The parameters to improve computational robustness when simulating near-zero depths are specified on this line. A set of parameters must be input for each branch for which GEQOPT=STDW or STDCW is used so that Line 2 is repeated for each of these branches.

BRAN is the branch number,

YATONE is the water-surface height below which selected distance-integral approximations are fully off-centered,

YATHAF is the water-surface height above which these approximations are again fully centered,

CHAR8 is an option for interpolation between the two limiting values for centering. The interpolation options are LINEAR or linear for linear interpolation, CUBIC or cubic for cubic interpolation, and left blank for linear interpolation. For cubic interpolation, the rate of change of the off-centering weight with respect to water-surface height at the limiting values is always zero.

A computational element is off-centered whenever the minimum water-surface height falls below YATHAF for that branch. The off-centering of the integral approximations reduces the accuracy of the approximations but increases the computational robustness. This approach is applied on the basis of the result that depths near zero are not of primary interest but are often the primary source of computational problems. Reducing the accuracy of the computations at these shallow depths will have little effect on the results at the depths of interest.

The off centering of the integral approximations has been tested only on a few streams simulated in FEQ, but simulation of flow values approaching zero has been successful. The values to choose for YATONE and YATHAF are not yet clear. A preliminary guideline is that YATHAF should be close to the depth that results when flows become of interest. Clearly it should be less than bankfull stage for the branch. The YATONE value should be much less than bankfull stage but yet not so small that the stream is dry before the depth given by YATONE is reached.

13.6 Network-Matrix Control Block—Network-Matrix Table

Purpose: The equations relating the flow and water-surface height values at all the flow-path end nodes in the stream system are defined with this block. The input in this block specifies how the flow-path end nodes are connected with internal boundary conditions (at special features) and external boundary conditions to represent the system. Complete details on stream-network schematization are given in section 3; details on internal boundary conditions are presented in section 8.1.

Heading: One line of user-selected information. The suggested string is NETWORK MATRIX CONTROL.

The code/type combinations applied in FEQ simulation are the following:

Code 1: Branch

Code 2: Number of nodes at a junction

Code 3: Equality of water-surface elevation between nodes

Code 4: One-node head-discharge relation

- Type 1: Flow over a weir
- Type 2: Table relating discharge and head
- Type 3: Channel control of flow
- Type 4: Structure capacity as function of time
- Type 5: Structure capacity varied dynamically
- Type 6: Pump with capacity limited by tail water

Code 5: Two-node head-discharge relations

- Type 1: Expansion or contraction with critical flow
- Type 2: Bidirectional flow in tables, plus pumping
- Type 3: Variable-speed variable-head pump
- Type 4: Bridge with flow over roadway
- Type 5: Abrupt expansion with inflow or outflow
- Type 6: Two-dimensional tables
- Type 7: Variable height weir
- Type 8: Sluice gates at Stratton Dam at McHenry, Ill.
- Type 9: Underflow gates tables

Code 6: Node with forced value of flow or elevation (Boundary conditions)

Code 7: Level-pool reservoir

Code 8: Critical depth

Code 9: Not in use

Code 10: Not in use

Code 11: Conservation of momentum or constant elevation

Code 12: Match average elevation at two nodes

Code 13: Conservation of momentum or energy

Code 14: Side-weir flow

Code 15: Dummy branch

LINE 1

Variable: HEAD

Format: A80

Example: CODE A B C D E F G H I J FA FB FC FD FE

Explanation: This is a user-selected heading for the lines of input to follow.

LINE 2 (repeated as many times as needed to input all the equations describing flow and water-surface elevation in the stream system including external boundary conditions)

Variables: CODE, N(1),. . .,N(10), F(1),. . .,F(5)

Format: I5, 10I4, 5F7.0

Example: See example input available by electronic retrieval from the World Wide Web (WWW) at http://water.usgs.gov/software/feq.html and by anonymous File Transfer Protocol (FTP) from water.usgs.gov in the pub/software/surface_water/feq directory.

Explanation: Line 2 is the information used to define an equation or equations in the matrix relating the flow-path end nodes and branches. The relation and the meaning of the remainder of the line is specified by the entry for CODE. All fields on the line are read whether needed or not. The following sections describe the meaning of each input variable for each code.

If CODE=1 (Branch equations):

N(1) is the branch number. Two equations in the matrix are generated for each branch.

If CODE=2 (number of nodes at a junction, sum of flows=0):

N(1) is the number of flow-path end nodes at the junction. Must be at least 2 and no more than 9.
N(2), . . . ,N(N(1)+1) are the flow-path end nodes at the junction.

If CODE=3 (Equality of water-surface elevation between nodes):

N(1) is the first flow-path end node.
N(2) is the second flow-path end node.

If CODE=4 (One-node head-discharge relation at a node):

N(1) is the type number for the relation, with $1 \leq N(1) \leq 6$.
N(2) is the number of the flow-path end node at which the head is specified.
N(3) is the direction of positive flow; +1 means flow into the system, and −1 means flow out of the system where the system is that part of the stream that includes the discharge node for the hydraulic control structure.
N(4) is the number of the flow-path end node at which the flow is specified.

In most cases, the node at which the head is specified and the node at which the flow is specified will be the same. Separate specification of these nodes is allowed, however, to include exceptional cases. These four values must always appear if CODE=4. The remainder of the line depends on the value of N(1) in which the TYPE is specified.

If TYPE=1 (Flow over a weir):

N(5) is the number of the table that includes the weir coefficient as a function of head.
F(1) is the elevation of the reference point used for defining head.
F(2) is the weir length.

If TYPE=2 (Table relating discharge and head):

N(5) is the number of the table specifying flow as a function of head.
F(1) is the elevation of the reference point used for defining head.

If TYPE=3 (Channel control of flow):

N(5) is the slope source for defining flow. If N(5)=−1, then the slope is given in F(1); if N(5)=0, then the bottom slope of the stream channel is used; and if N(5)=1, then the water-surface slope at the previous time point is used.
F(1) is the value of slope if N(5)=−1.

If TYPE=4 (Structure capacity given as a function of time):

N(5) is the number of the table specifying the maximum flow through the structure as a function of head.
N(6) is the number of the table specifying proportion of maximum flow as a function of time.
F(1) is the elevation of the reference point used for defining head.
It is assumed that time varying flow can be represented by the product of two functions. The first function is the proportion of the maximum flow rate as a function of time. The second function is the maximum flow rate as a function of head at the head node. These two functions are simple to define given records of the operation of the facility.

If TYPE=5 (Structure capacity varied dynamically in FEQ computations):

N(5) is the number of the table specifying maximum flow through the structure as a function of head.
N(6) is the number of the operation block controlling the operation of the structure. All operation blocks must be numbered consecutively, starting at 1. The operation table appears in the Operation of Control Structures Block (section 13.12).
F(1) is the elevation of the reference point used for defining head.

If TYPE=6 (Pump with capacity limited by tail water):

N(5) is the number of the table specifying the pumping rate as a function of upstream head only. Tail-water variations are assumed to have only a small effect on the effective pumping rate.
N(6) is the source for the controlling level in the tail water. If value is greater than zero, it represents a time-series table number. If value is less than zero, it represents the Fortran unit number (see appendix 3) for a point-time-series file. This file must be specified in the Input Files Block (section 13.10). The controlling level can be stage or flow.
N(7) is the number of the table in which the tail-water level is converted into a limiting flow for the pumping rate. This table includes an argument of stage or flow taken from the source in N(6), and the limiting pumping rate is specified in this table. The actual pumping rate is the lesser of the limiting value or the rate from the table in N(5).

F(1) is the elevation of the reference point used for defining head for the table in N(5).

If CODE=5 (Two-node head-discharge relations):

N(1) is the type of relation, where $1 \leq N(1) \leq 9$.
N(2) is the upstream node of the two nodes in the relation.
N(3) is the downstream node of the two nodes in the relation.
N(4) is the node at which the flow through the structure is specified; it must be either N(2) or N(3).

Each structure of CODE=5 includes two nodes and a nominal downstream direction of flow used to establish the designation of the upstream and downstream nodes. The upstream node should be placed where the water approaches the structure when flowing in the nominal downstream direction. The downstream node should be placed at the exit point from the structure when the water is flowing in the nominal downstream direction.

If TYPE=1 (Expansion or contraction with critical flow):

N(5) is the sign of the transition. Sign=+1 if expansion in flow results when flow is from upstream node to downstream node; sign=−1, otherwise.
N(6) is the number of the table giving the hydraulic characteristics of the cross section where critical flow is computed.
F(1) is the loss factor on velocity-head change for flow from upstream node to downstream node.
F(2) is the loss factor on velocity head change for flow from downstream node to upstream node.
F(3) is the elevation for defining depth in the section of critical flow.

Discussion: Code 5, type 6, and an expansion-contraction table computed in FEQUTL (Franz and Melching, in press) with the EXPCON command, is often preferred to applying this option. In code 5, type 1, a contraction is assumed to always be a contraction and an expansion is always an expansion. Furthermore, initiating flow from zero is sometimes difficult in simulation for code 5, type 1. If water is always present and the direction of flow through the transition does not change, then code 5, type 1 can be applied. If, however, convergence problems occur and can be traced to this option, the problem is often eliminated by switching to code 5, type 6 and a pre-computed table.

If TYPE=2 (Bidirectional flow given by tables, plus pumping):

N(5) is the number of the table specifying flow in the positive direction; that is, from upstream node to downstream node. If N(6)=0, then N(5) specifies the number of the table containing square root of conveyance as a function of water-surface height.
N(6) is the number of the table specifying submergence reduction for positive flow. If N(6)=0, the conveyance option is applied. The pumping parameters are ignored if the conveyance option is selected.
N(7) is the number of the table specifying flow in the negative direction.
N(8) is the number of the table specifying submergence reduction for negative flow.

N(9): If N(9)=0, water-surface elevation is detected at the destination node for control of the pump. If N(9)=1, flow rate is detected at the destination node for control of the pump.
N(10) is the node to monitor if pump is to be switched off whenever flow at the flow-path end node given by N(10) is > 0.
F(1) is the elevation for the reference point used for computing head.
F(2) is the flow distance for the conveyance option.
F(3) is the pumping rating. If F(3)=0, then no pump is simulated. If F(3) > 0, then pumping from the upstream node to the downstream node is simulated. If F(3) < 0, then pumping from the downstream node to the upstream node is simulated. The node from which water is pumped is the source node and the node to which water is pumped is the destination node.
F(4) is the inlet elevation for the pump. The water-surface elevation at the source node must exceed the inlet elevation before the pump can be turned on. If the water-surface elevation at the source node is at least 0.1 ft below the inlet elevation, the pump will be turned off.
F(5) is the cutoff value at the destination node. F(5) is either flow or water-surface elevation as defined by the value of N(9). The cutoff value is defined by the user to turn the pump on or off depending on conditions at the destination node. If this value gives a cutoff elevation, then setting F(5) < F(1) prevents convergence problems resulting from the pump cycling in subsequent iterations of the computations for a solution at a time step.

Discussion: This Network-Matrix Control Option is designed to represent various structures. The conveyance option can be used to represent flows into and out of extensive slack-water areas next to a channel for which storage is represented by a level-pool reservoir and the inflow and outflow are controlled primarily by boundary friction and the inertial terms are negligible. The bidirectional-flow option can represent flow over a weir, a road, a spillway, and other similar features. The flow is zero if both the upstream and downstream elevations are below the elevation used for defining head. Otherwise, the flow is given by the product of the flow from the flow table for the node with the larger head and the submergence reduction factor from the corresponding submergence table. The head used does not include the velocity head of approach.

The pump option is used to represent the simple on-off operation of a constant-flow pump. Water always moves in the direction given by the sign of the constant pumping rate defined in F(3). The pump is either on or off. If the pump is off, then it is turned on whenever the inlet elevation is exceeded by the water-surface elevation at the source node and the monitored value at the destination node is in the proper range. If the monitored value at the destination node is water-surface elevation, then the water-surface elevation at the destination node must be less than the cutoff value given in F(5) to be in the proper range for turning the pump on. If the monitored value at the destination node is flow rate, then the flow rate at the destination node must be positive and less than the cutoff value given in F(5) to be in the proper range for turning the pump on.

If the pump is on, some similar set of rules must be used to determine when to turn the pump off. The action of the pump will affect the monitored values at the destination node, and some tolerance region or null region must be provided to prevent endless on-off cycling of the pump. The pump is turned off if the water-surface elevation at the inlet is at least 0.1 ft below the pump-inlet elevation given in F(4). If the monitored value at the destination node is elevation, then the pump is turned off if the water-surface elevation at the destination node is greater than the elevation given in F(1). If the monitored value at the destination node is flow rate, then the pump is turned off if the flow at the destination node is greater than the turn-on value plus twice the absolute value of the pumping rate. Finally, if N(10) > 0, then the pump is turned off if the flow at the node given in N(10) is greater than zero.

If TYPE=3 (Variable-speed variable-head pump):

N(5) is the flow direction: 1 means pumping from upstream node to downstream node; −1 means pumping from downstream node to upstream node. The node from which water is being drawn is called the source node, and the node to which water is being pumped is called the destination node.
N(6) is the number of the table specifying the flow through the pump for each head; that is, a pump-performance curve. The speed of the pump used to define this curve is the base speed to which all speeds are relative. The base speed should be the highest speed for the pump so that all other speeds are smaller. For example, if a pump has a speed range from 0 to 1,800 revolutions per minute (rpm), then 1,800 rpm should be used to define the head across the pump. A relative speed of 0.5 means that the pump is operating at 900 rpm. The relation between flow through the pump and head must be unique. There are pumps for which this is not true; that is, pumps with ranges of flow for which the head increases as the flow increases followed by a decreasing head with increasing flow. Thus, for a given head, two flows are possible. Operation of pumps with nonunique head-discharge relations could be unstable if applied for pumping in an open-channel network. Operation of these pumps also could prove unstable in FEQ simulation. Therefore, to represent the pump performance in FEQ, the pump curve must be modified for pump operation stability when applied in an open-channel network.
N(7) is the number of the table specifying the sum of the entrance losses at the inlet, friction losses in the inlet conduit, and friction losses in the outlet conduit. The sum is expressed in terms of head as a function of flow through the pump. In simulation, losses are assumed to increase with flow. This table is optional and if omitted, the losses are assumed to be included in the pump-performance curve table given in the table specified in N(6).
N(8) is the number of the table specifying the exit-loss coefficient on the velocity-head difference between the end of the outlet conduit and the destination node. This coefficient is given as a function of the depth of submergence of the outlet conduit. The loss coefficient must be 1.0 at zero submergence. If this table is omitted, then an exit-loss coefficient of 1.0 is applied for all levels of submergence.
N(9): The number of the table specifying the pump speed as a function of time is given if N(9) < 0. The operation block number (see section 13.12) controlling the pump is given if N(9) > 0. The pump is kept running at the base speed all the time if N(9)=0. Water will not be pumped if the water-surface elevation at the source node is below the inlet elevation.
N(10) is an optional name for the pump. The name can be four alphanumeric characters at most. The first character should be alphabetic. If a pump is given a name, its state may be printed in the Special-Output File by specifying the name in the field as for a flow-path end node in the Special-Output Locations Block (section 13.9). In this case, the state of the pump includes the relative speed and the nature of flow. NO H2O denotes that the pump is on but that the inlet is above the water surface. FP denotes free flow out of the pump outlet. SP denotes submerged flow out of the pump outlet. OFF denotes that the pump is currently not running.
F(1) is the elevation of the outlet conduit and the point of reference for submergence of the outlet. If the water-surface elevation at the destination node is below this elevation, then a loss equal to the exit velocity head from the outlet conduit is assumed. If the water-surface elevation at the destination node is greater than this elevation, then a loss coefficient is determined from the table given in N(8) if this table is specified; otherwise, the loss coefficient is specified as 1.0. The loss coefficient is used in the computation of the fraction of the velocity-head difference between the conduit exit

and the destination node that is an exit loss. The argument for the table is the depth of submergence computed relative to the elevation of the outlet conduit.
F(2) is the area of the outlet conduit when flowing full. The outlet conduit is assumed to be flowing full at all pump speeds.
F(3) is the elevation of the inlet conduit. Pump flow is zero if the water-surface elevation is below the inlet.
F(4) is the factor on the velocity head at the source node, usually 0 or 1 to exclude or include, respectively, the velocity head at the source node. This factor is specified as 0 if the source node is a free node.
F(5) is the factor on velocity head at the destination node, usually 0 or 1 to exclude or include, respectively, the velocity head at the destination node. This factor is specified as 0 if the destination node is a free node.

Discussion: This network-matrix control option is designed to represent a variable-speed variable-head pump. If the head on the pump exceeds its cutoff head, then the flow through the pump will reverse if a check valve of some kind is not present in the outflow conduit. It is specified in the pump characteristic table whether reverse flow is to be allowed. In most cases, check valves are present, and reverse flows are not allowed. If this is the case, the table given in N(6), must be extended to a head higher, and sometimes much higher than any expected with flow through the pump set to zero for the additional heads.

If TYPE=4 (Bridge with flow over the roadway):

N(5) is the number of the table specifying the bridge-loss coefficient as a function of water-surface height at the bridge opening for positive flow.
N(6) is the number of the table specifying the bridge-loss coefficient as a function of water-surface height at the bridge opening for negative flow.
N(7) is the number of the table specifying the area of bridge opening as a function of water-surface height at the bridge opening.
N(8) is the number of the table specifying flow over the roadway as a function of head for positive flow.
N(9) is the number of the table specifying flow over the roadway as a function of head for negative flow.
N(10) is the number of the table specifying the submergence effect as a function of head ratio for flow over the roadway.
F(1) is the maximum flow area through the bridge.
F(2) is the elevation of the high point of the bridge opening. Two options are available for this elevation. The first is the actual elevation of the high point of the bridge opening, and the second is a value that is at least 1 ft higher. If the true value is given, a submerged-flow equation is applied in simulation if the upstream end of the bridge becomes submerged. Otherwise, the free-flow equation is used with the free-flow loss adjusted to match closely the submerged-flow loss. This later option is applied to avoid flow discontinuities during the transition from free flow to submerged flow. These discontinuities can result in severe computational problems.
F(3) is the submerged-flow discharge coefficient for the bridge.
F(4) is the elevation for computing head on the roadway. This value can be set so large that no water will flow over the roadway. This should be done to simulate bridges for which flow over the roadway is impossible instead of using the TYPE=3 option.

Discussion: The required bridge-loss tables are computed with FEQUTL (Franz and Melching, in press).

If TYPE=5 (Abrupt expansion with inflow or outflow):

N(5) is the number of the table specifying the critical flow as a function of depth at the upstream node for the abrupt expansion. A table of type 2, type 4, type 22, or type 25 may be used. Types 22 or 25 are preferred, but types 2 and 4 are retained for consistency with earlier versions of FEQ.

Discussion: An abrupt expansion can be any feature that results in an abrupt increase in the area available for flow at all water-surface heights. Thus, a hydraulic drop is an abrupt expansion, as is a sudden increase in channel size. Critical depth is possible only at the upstream node. The user must supply a critical-flow table so that this condition can be detected. Inflow-outflow of water may result between the upstream and downstream nodes of the abrupt expansion. The direction of this flow is taken to be at right angles to the direction of flow from the upstream node to the downstream node. These features make it possible to represent structures used to divert water from the side of a channel. Frequently, an abrupt expansion is used to reduce the velocity of the water so that the performance of the side discharge is easier to evaluate and more efficient in diverting water.

The Type=5 option has several restrictions. First, the nodes must be on branches in their natural condition; that is, the upstream node for the abrupt expansion must be the downstream node on a branch upstream, and the downstream node for the abrupt expansion must be the upstream node for the branch downstream. Second, the discharge node for the abrupt expansion must be the upstream node of the abrupt expansion. Third, the flow cannot reverse at the discharge node. (Flow may reverse at the downstream node of the abrupt expansion.) Fourth, gravity and friction forces are ignored in the momentum balance.

If TYPE=6 (Two-dimensional tables):

N(5) is the number of the table specifying flow from the upstream node to the downstream node.
N(6) is the number of the table specifying flow from the downstream node to the upstream node.
N(7) is the number of an optional table specifying a multiplying factor to apply to the values derived from the tables given in N(5) and N(6). If left blank, the multiplying factor is taken to be 1.0.
N(8) is the number of an optional table specifying the elevation for computing heads as a function of time. If left blank, the elevation for computing heads remains fixed at the value given in F(1).
N(10) is the continuation field. If N(10) > 0, then the next line of input gives another set of values N(5) through F(1) so that more than one set of flow relations is available between the two nodes. The end of the input is defined by a value of N(10)=0. The number of sets of tables input is limited only by the memory allocated for the storage of the Network-Matrix Control Input.
F(1) is the elevation for computing heads.

Discussion: This option represents bidirectional flow through a structure. If the tables referenced in N(5) and N(6) are type 6 or 13, then there must be a drop in water-surface elevation in the direction of flow. For these table types, more than one set of tables can be used to describe the flow between the two nodes. For example, there may be several openings through a long highway or railroad fill crossing the flood plain. Each opening could be represented by its own set of tables. The sum of the flows through all the active openings are reported in the output.

With tables of type 14, a drop in water-surface elevation in the direction of flow is not required; however, only one flow path can be defined between a pair of flow-path end nodes. If more than one flow path is required, then additional branches or dummy branches must be added so that only one table exists for each flow path between nodes. This restriction results because type 14 tables include the downstream head and flow as arguments, and the upstream head is determined from these arguments. Thus, the values from these tables are not additive. Conversely, tables of type 6 and 13 include the upstream and downstream head as arguments, and flow values, which are additive, are determined from these arguments.

If TYPE=7 (Variable-height weir):

N(5): If N(5) > 0, it is the operation block number (see section 13.12); if N(5) < 0, it is the number of the table specifying the opening fraction as a function of time.
N(6) is the number of the table specifying the elevation of the weir crest as a function of the opening fraction.
N(7) is the number of the table specifying the weir coefficient for flow from the upstream node to the downstream node (positive flow) as a function of the opening fraction.
N(8) is the number of the table specifying the weir coefficient for flow from the downstream node to the upstream node (negative flow) as a function of the opening fraction.
N(9) is the number of the table specifying the submergence correction for flow over the weir.

N(10) is the optional name for the weir. The name can be four alphanumeric characters at most. The first character should be alphabetic. If the weir is given a name, its state may be printed in the Special-Output File by specifying the name in the field as for a flow-path end node in the Special-Output Locations Block (section 13.9). In this case, the state of the weir includes the crest elevation and the nature of flow. NO FLOW denotes absence of flow over the weir. FW denotes flow over the weir free of downstream effects (free flow). SW denotes submerged flow over the weir.
F(1) is the weir length.
F(2) is the factor to apply to velocity head when computing the total head applicable to the weir equation.

Discussion: This option may be applied to simulate flow for overflow gates. The opening fraction, *p*, is taken to be 0.0 when the gate is fully raised and 1.0 when the gate is fully lowered. The value of *p* is set by the rules given in the operation block (see section 13.12) or table referenced in N(5). The velocity-head factor can be applied to eliminate the velocity head from the weir equation by setting F(2)=0. Conversely, the velocity-head factor can be applied to reduce the effect of weir height on the weir coefficient.

If TYPE=8 (Sluice gates at Stratton Dam at McHenry, Ill.):

N(5): If N(5) > 0, it is the operation block number (see section 13.12); if N(5) < 0, it is the number of the table specifying the gate opening as a function of time.

F(1) is the sill elevation for the sluice gates (731.15 ft).

F(2) is the Maximum gate opening, in feet. A value of 8 ft is about the limit of the fitted relations representing the flow through the gates.

F(3) is the factor to apply to the flows to represent change in width from the standard width at McHenry of 68.75 ft (five gates at 13.75 ft each). The default value is 1.0 if left blank. If the gates simulated differ in width from those at Stratton Dam on the Fox River at McHenry, Ill., then the ratio of widths can be used for this factor. Adjustments for differing approach conditions also may have to be considered.

Discussion: This option is specific to a particular set of gates on the Fox River, Ill. The gate relations are internal to the subprogram units in FEQ, so no parameters or tables describe the gates. Only the three values given above can be used to change the flow characteristics at the gates.

If TYPE=9 (Underflow gates tables):

N(5): If N(5) > 0, it is the operation block number (see section 13.12); if N(5) < 0, it is the number of the table specifying the gate opening as a function of time. This is the opening in the same units used for the maximum gate opening in F(2).
N(6) is the name for the gate. The name can be four alphanumeric characters at most. The first character should be alphabetic. If a name is given for the gate, its state may be printed in the Special-Output File by specifying the name in the field for a flow-path end node in the Special-Output Locations Block (section 13.9). In this case, the state of the gate includes the opening and the nature of flow. FW denotes free-weir flow. SW denotes submerged-weir flow. FO denotes free-orifice flow. SO denotes submerged-orifice flow. OR denotes orifice-flow conditions such that it cannot be determined in the table lookup process whether the flow is free or submerged, only that the flow is orifice flow. NO FLOW denotes that either the gate is closed or the water-surface level is below the gate opening.
N(7) is the number of a type 15 table for flow from the upstream node to the downstream node.
N(8) is the number of a type 15 table for flow from the downstream node to the upstream node.
F(1) is the sill elevation for the sluice gates.
F(2) is the maximum gate opening in feet. This value is multiplied by the fractional gate opening generated in an Operation of Control Structures Block (section 13.12) to convert to an actual gate opening for table lookup in the table given in N(7) or N(8).

Discussion: This option is designed to approximate the hydraulic characteristics of an operable gate subject to submergence. The gate can be any of a number of types. The gate type is defined when the two-dimensional tables are computed. Underflow gates, such as sluice gates and Tainter gates, can be easily simulated with this option. Simulation of combined underflow and overflow gates also should be possible. Much microcomputer memory may be needed if many two-dimensional tables are used to simulate the gates.

If CODE=6 (Node with forced value of flow or elevation):

The maximum number of CODE=6 boundaries in the stream system allowed in FEQ is specified in the parameter MNCO6 in the INCLUDE file ARSIZE.PRM (appendix 3). In addition, the maximum number of input files for CODE=6 boundaries allowed in FEQ is specified in the parameter MNFIN in the INCLUDE file ARSIZE.PRM (appendix 3). These parameters may be increased as needed and FEQ recompiled.

N(1) is TYPE of the forced value. TYPE=1 if flow as a function of time is forced at the node, and TYPE=2 if elevation as a function of time is forced at the node.
N(2) is the number of the flow-path end node where forcing takes place.
N(3) is the direction of positive flow at the node. Direction is given as 1 if a positive value of flow from the flow source is into the stream system and −1 if out of the system.
N(4) is the source for the forcing values. If N(4) > 0, then the source is the table given by the number in N(4). If N(4)=0, then the source is the constant value given in F(1). If N(4) < 0, then the source is the file referenced by the Fortran unit number (see appendix 3) specified by the absolute value of N(4).
F(1) is the value of constant flow if N(4)=0 and TYPE=1. This value is the minimum value if the source for the forcing values (N(4)) is nonzero. This minimum value applies to both flow and elevation and to values from a table and from a file. If applied with tidal data including values possibly less than zero, F(1) should be less than the minimum tide expected. Otherwise, tide levels below this value will be truncated.
F(2) is the multiplier on the value given by the source for flows and elevation. The value of F(2) is specified by the user. F(2) is typically used to check the sensitivity of FEQ output to source flows and elevations without making major changes in model input. If left blank, a default value of 1.0 is applied.

If CODE=7 (Level-pool reservoir):

N(1) is the node number of the reservoir.
N(2) is the number of the table specifying storage as a function of elevation for the reservoir.
N(3), the number of inflow nodes, must be 1. Every level-pool reservoir must have one inflow node.
N(4) is the inflow node number.
F(1) is the slope factor for making the elevation at the reservoir node and the inflow node slightly different. F(1) defaults to a value of 1×10^{-8} if left blank.

Discussion: Two equations for flow through the level-pool reservoir are generated and placed in the network matrix with this code. To obtain reliable solution of the equations with the current network matrix, level-pool reservoirs must be not quite level. Complete details are given in the section 7.2. The default value for the slope factor results in a relatively small elevation difference between the entrance and exit of the level-pool reservoir for all but the largest flows.

If CODE=8 (Critical depth):

N(1) is the number of the branch-end node where cross-sectional hydraulic characteristics are defined.

If CODE=11 (Conservation of momentum or constant elevation):

N(1) is the upstream node.
N(2) is the downstream node.

Discussion: The option is used to represent the effect of an inflow or outflow at right angles to the channel. If there is an inflow of water between the two nodes, conservation of momentum is applied to allow for a difference in water-surface elevation. If there is an outflow of water between the two nodes, the two nodes must have the same elevation.

Several restrictions apply to the option. The cross sections at both nodes must be identical and must be given by the same table number. Furthermore, the bottom elevations must be identical. There also should be a change in flow rate; otherwise, CODE=3 is a better option for this situation.

If CODE=12 (Match average elevation at two nodes):

N(1) is the upstream source node for averaging.
N(2) is the downstream source node for averaging.
N(3) is the node at which average elevation of nodes N(1) and N(2) will be forced.
F(1) is the weight to be used in computing the average. The weight is applied to N(1), and the complement of the weight is applied to N(2). The weight, W, must satisfy $0 \leq W \leq 1$.

Discussion: This option is applied to attach a branch or free node to a junction defined by code 11 or code 13 that can have a difference in elevation at the nodes. An average of the two elevations is then defined with Code 12.

If CODE=13 (Conservation of momentum or energy):

N(1) is the upstream node.
N(2) is the downstream node.
F(1) is the loss coefficient to apply to the energy equation. The coefficient is a factor applied to the change in velocity head that results when water is taken from the channel. The loss coefficient must be nonnegative and less than 1. The energy losses when water is discharged from a channel are generally small. Care must be applied in setting this value because computational failure can result if it is too large.

Discussion: This option is preferred for representing an inflow or outflow at right angles to the stream. If there is inflow, then conservation of momentum is applied; whereas if there is outflow, conservation of specific energy is applied. Several restrictions apply to this option. The upstream node must be the downstream node on the branch upstream from the junction, and the downstream node for this option must be the upstream node for the branch downstream from the junction. Moreover, the bottom elevations must match, and the cross-section table numbers must be identical.

If CODE=14 (Side-weir flow):

N(1) is the upstream node on the source channel.
N(2) is the downstream node on the source channel.
N(3) is the node used to represent the outflow or inflow to the source channel.
N(4) is the flow table of type 13 defining the outflow over the side weir. This flow is computed as if the weir were a normal weir. Adjustments are made, on the basis of flow conditions, for the side-weir flow.
N(5) is a flow table of type 13 representing inflow over the side weir into the source channel. This flow is taken to be for a normal weir; no adjustment is made.
F(1) is the weight coefficient used in computing the average water-surface height, elevation, and flow rate in the source channel. The weight is applied to node N(1), and the complement of the weight is applied to node N(2). The value of the weight must satisfy $0 \leq W \leq 1$.
F(2) is the elevation from which the heads included in the flow tables are measured.

Discussion: Flow over a side weir is difficult to compute even under steady-flow conditions. A detailed integration of flow along the side weir cannot be simulated in FEQ. However, one approach to approximating this integration is described in section 8.1.2.1.3.2.

Side weirs are a potential source of computational problems in FEQ simulation. If too much flow is diverted, then the water upstream from the weir in the source channel may approach critical depth or even become slightly supercritical. This may result in computational failure; most side weirs do not function well with a hydraulic jump at some point along the weir. Careful thought must be given to sizing the source channel and the weir so that supercritical flow will not result.

If CODE=15 (Dummy branch):

N(1) is the upstream node for the dummy branch.
N(2) is the downstream node for the dummy branch.
F(1) is the slope factor for assigning the elevation at the downstream and upstream nodes slightly different values. F(1) defaults to $1{\times}10^{-8}$ if left blank.
F(2) is the surface area to use in routing flows through the dummy branch. F(2) defaults to 2.0 if left blank.

Discussion: Two equations are generated and placed in the network matrix with this code. A dummy branch is a flow path for which only the flow rate is of interest. Area, roughness, and (or) other properties are not of interest in the segment. A small surface area and a small difference in elevation between the two nodes are applied in FEQ simulation. An example of a dummy-branch application is to represent the flow over an emergency spillway on a dam. All that is of interest is the flow of water over the spillway. Water-surface height and velocity may be of interest in designing the spillway chute and stilling basin, but these involve computations not done in FEQ. Another dummy-branch application is to represent the flow over a levee when it is overtopped. Again, primary interest is the amount of water following that path. The depth and velocity are of secondary interest.

If CODE = −1 (Input for network-matrix table is complete).

LINE 3

Variable: NAME1, BNODE

Format: A6, A5

Example: BNODE= U25

Explanation:

NAME1 is the identifying character string “BNODE=.” This string must appear even if no boundary node is specified to initialize the matrix development. The name must be in upper case and must be given exactly as shown, beginning in column 1. Otherwise, a message will be issued that the boundary node for starting the network-matrix construction is missing.

BNODE is the boundary node to use for initiating the development of the network matrix. In order to develop the network matrix, a beginning flow-path end node on the boundary of the stream system is required in simulation. This must be a boundary node but must not be a boundary node at which elevation as a function of time is given. If elevation as a function of time is specified at all boundary nodes, then a dummy branch must be added to the model so that one of the other two boundary types can be specified. If the node field is left blank, then a matrix will be developed from each valid boundary node in the model and the node resulting in the smallest matrix length will be selected. It is best to leave the field blank and let the optimum starting boundary node be selected in simulation. Specifying this node will save only a fraction of a second of microcomputer time even when simulating a complex stream system. As the model is changed, the optimum boundary node also may change.

A new pattern for the network matrix for each valid boundary node as the starting node is considered during simulation to determine the optimum starting node. This means that the pattern for the network matrix must be recomputed once the optimum boundary node is selected. Occasionally, a slightly different length for the optimum boundary node will be found than in the original search. This results because of the internal order of the Network-Matrix Control Input changes as each pattern is computed. Certain decisions in the development of the network-matrix pattern are based on path lengths. Sometimes the available path lengths are all equal, and an arbitrary choice must be made. As the internal order of the Network-Matrix Control Input varies, the arbitrary choice may select a different path. Path differences are usually small.

13.7 Point Flows Block—Point Flow Tables

Purpose: The inflows and (or) outflows to or from within a branch of the stream system that can be localized to a point in, or at least a small section of, the stream are specified with this block. These flows might be water intakes for a city or a discharge point for a treatment plant. This block is present only when POINT=YES in the Run Control Block (section 13.1). In most cases, for robustness in model simulation, flows such as these should be input into a junction between branches instead of within a branch.

Heading: One line of user-selected information. The suggested string is POINT FLOWS.

LINE 1

Variable: HEAD

Format: A80

Example: BRAN NODE TYPE TABN MFAC

Explanation: These are user-supplied headings for subsequent information.

LINE 2 (one for each point flow)

Variables: BRA, NODE, TYPE, TABLE, MFAC

Format: 4I5, F10.0

Explanation: Information for a point flow is specified on this line

BRA is the branch number.

NODE is the number of node on the branch at the downstream end of the element included with the point flow.

TYPE=1 and 2 for inflow and outflow, respectively.

TABLE is the number of the table giving the flow as a function of time.

MFAC is the multiplying factor to apply to the flow to approximate the downstream component of momentum flux.

Line 2 is given for each point flow. The same branch and node number may appear as many times as needed for multiple point flows in a given element. The end of the table is indicated by assigning a negative value for the branch number.

Specifying point flows is primarily done when the DTSF (appendix 2) is not used. The point flows referenced here can only be in a function table and not a file and, therefore, their duration is limited. The argument for function tables representing a time series is the elapsed time in seconds from the start of the analysis. A DTSF may contain several distinct nonoverlapping time periods to be simulated. The elapsed time is set to zero at the start of each period in the DTSF. Thus, any time series given in a function table will be reused for each time segment. Input and output files are a better means of specifying point flows for an extended period of time or with a DTSF.

13.8 Wind Information Block—Wind Table

Purpose: The information on wind (such as wind-shear coefficient, air density, wind velocity table number, and other information) is supplied with this block. This block is present only when WIND=YES in the Run Control Block (section 13.1). If wind shear is simulated, the azimuth information in the Branch Description Block (section 13.2) should be specified. If the azimuth information is not specified, then all elements will be assigned an azimuth of zero, meaning that downstream direction points north for the computational elements.

Heading: One line of user-selected information. The suggested string is WIND INFORMATION.

LINE 1

Variable: WINTAB

Format: 7X, I5

Example: WINTAB=00007

Explanation: WINTAB is the number of the table specifying wind velocity and direction as a function of time. Function table type 11 is reserved for this purpose. An inconsistency often results in common practice for the designation of wind direction. The direction for wind has traditionally been the point from which the wind is *coming* and not the direction the wind is going. This traditional designation is retained and applied in FEQ. Thus, the direction of the wind is in degrees clockwise from north for the direction from which the wind is coming. For example, an east wind is given an azimuth of 90 degrees, a south wind an azimuth of 180 degrees, a west wind an azimuth of 270 degrees, and a north wind an azimuth of 0 degrees. The azimuth for the orientation of an element in a branch is based on the downstream direction that water flows, not the direction from which it is *coming*. Therefore, an element with downstream flow going east also is given an azimuth of 90 degrees. In this case an east wind will be in the opposite direction even though the wind and the downstream flow direction are given the same value for azimuth. One-hundred eighty degrees are added internally in FEQ to convert the wind direction to the same basis as the flow direction.

Estimates of the drag coefficient for wind-shear stress on the water surface are computed with results determined by Wilson (1960). Wilson indicates that the drag coefficient depends on the wind velocity, with the coefficient increasing to a limit as the wind velocity increases. He also stated that there is evidence that the drag coefficient decreases slightly to a minimum at about 13 to 16.5 ft/s for low wind speeds. The variation between low wind and high wind speeds is nonlinear but not well established. This variation is supported in FEQ by requiring the following values: the wind speed at minimum drag coefficient, VAMIN; the minimum drag coefficient, CDMIN; the wind speed at maximum drag coefficient, VAMAX; and the maximum drag coefficient, CDMAX. The minimum drag coefficient is applied to all wind speeds less than VAMIN and the maximum drag coefficient to all wind speeds greater than VAMAX. The drag coefficient for intermediate wind speeds is interpolated by use of a cubic polynomial between the minimum and maximum points, assuming that the slope of the function is zero at both extremes.

The drag-coefficient values given below are computed under the assumption that the point of wind measurement is 33 ft above the water surface. The wind speed can be adjusted to this standard height by assuming an appropriate variation for wind velocity in the vertical (Linsley and others, 1982, p. 37–41).

LINE 2

Variable: AIRSPW

Format: 7X, F10.0

Example: AIRSPW= 0.075

Explanation: AIRSPW is the specific weight of air, in pounds per cubic foot. The specific weight of air varies slightly with temperature, moisture content, and atmospheric pressure. These variations over the course of the simulation are considered negligible. The specific weight of air can be estimated from

$$\gamma_a = 0.0807 \frac{491.6}{T + 491.6} \frac{P_a - 0.3783e}{1013.3}, \tag{148}$$

where

γ_a is the specific weight of air, in pounds per cubic foot;
T is the air temperature, in degrees Fahrenheit;
P_a is the atmospheric pressure, in millibars; and
e is the vapor pressure of water in air, in millibars.

The specific weight of air at standard atmospheric pressure varies from 0.075 lb/ft^3 in dry air at 40°F to 0.0654 lb/ft^3 in fully saturated air at 100°F. The variation is small but might be detectable in a carefully controlled experiment in a laboratory setting. It is unlikely that a field test would require any more than a possible adjustment to average conditions of atmospheric pressure, temperature, and humidity.

LINE 3

Variable: WATSPW

Format: 7X, F10.0

Example: WATSPW= 62.4

Explanation: WATSPW is the specific weight of water, in pounds per cubic foot. The ratio of the specific weight of air to the specific weight of water is used in FEQ computations. The respective specific weights can be supplied in any convenient units so long as the ratio is correct. The specific weight of water varies slightly with temperature but the variation from 62.4 lb/ft^3 over the range of possible temperatures is less than 1 percent.

LINE 4

Variable: VAMIN

Format: 6X,F10.0

Example: VAMIN= 15

Explanation: VAMIN is the velocity of the wind where the drag coefficient reaches an approximate minimum value. The recommended value from Wilson (1960) is about 15 ft/s.

LINE 5

Variable: CDMIN

Format: 6X,F10.0

Example: CDMIN= 0.0015

Explanation: CDMIN is the minimum value of the drag coefficient. The recommended value from Wilson (1960) is 0.0015.

LINE 6

Variable: VAMAX

Format: 6X,F10.0

Example: VAMAX= 75

Explanation: VAMAX is the wind velocity where the drag coefficient reaches an approximate maximum. The recommended value from Wilson (1960) is 75 ft/s (about 50 mi/hr).

LINE 7

Variable: CDMAX

Format: 6X,F10.0

Example: CDMAX= 0.0025

Explanation: CDMAX is the maximum value of the drag coefficient. The recommended value from Wilson (1960) is 0.0025.

13.9 Special-Output Locations Block—Special-Output Table

Purpose: The nodes where output to a designated file is desired are specified with this block. This block is required even if no special output is needed. No nodes are specified in this case, but a null block is still required in the input. The maximum number of special outputs in the special-output file allowed in FEQ is specified in the parameter MNSOUT in the INCLUDE file ARSIZE.PRM (appendix 3). The maximum number of gates for output of special values to the special-output file allowed in FEQ is specified in the parameter MNGATE in the INCLUDE file ARSIZE.PRM (appendix 3). These parameters may be changed as needed and FEQ recompiled.

Heading: One line of user-selected information. The suggested string is SPECIAL OUTPUT LOCATIONS.

LINE 1

Variable: UNIT, OUTNAM

Format: 5X, I5, A32

Example: UNIT=00009SPOUT

Explanation:

UNIT is the Fortran unit number (see appendix 3) of the file used for storing the special output.

OUTNAM is the name for the file used for storing the special output. If OUTNAM is nonblank, then a file will be opened with the name given by the contents of OUTNAM (most microcomputers). On IBM mainframes, the name given in OUTNAM will be the ddname for the DD statement defining the data set. If the OUTNAM is blank, then an implicit open will be done in an IBM mainframe if the proper DD statement defining the unit number given by UNIT is present. In some microcomputer systems, the user will be prompted for the file name in this case, whereas program execution will be aborted on other microcomputer systems.

LINE 2

Variable: HEAD

Format: A80

Example: BRAN NODE IDENTIFICATION

Explanation: This is a user-supplied heading for subsequent information.

LINE 3 (one for each node at which special output is desired)

Variables: BRA, NODE, HEAD1, HEAD2

Format: I5, A5, 1X, 2A7

Example:
```
    1  100   NOD 100
```

Explanation: The output node locations are specified on this line.

BRA is the branch number. If BRA=0, then the node column contains an exterior node label. The table is terminated if the branch number is given as negative.

NODE is the node number. If the node number is negative, the diffuse inflows to either the reservoir or the computational element specified by the node with the elevation values set to zero will be output. Diffuse inflows for reservoirs not having tributary area should not be requested.

Output of the gate opening (in feet) for the sluice gates at Stratton Dam on the Fox River at McHenry, Ill., and the flow type (FO, free orifice; SO, submerged orifice; FW, free weir; SW, submerged weir) is requested by placing the string “MCHN” right justified under the NODE heading. The branch number is zero. If code 5, type 8 is not present in the Network Matrix Control Input, the values output are undefined.

Output of the gate opening and flow type for the underflow gates supported by code 5, type 9 is requested by placing the user-selected four-character string for the gate under the NODE heading. The flow types are the same as for the gates at the Stratton Dam on the Fox River at McHenry, Ill.

HEAD1 and HEAD2: Up to 14 characters of identification information can be given for each output location. This identification is broken into the two strings HEAD1 and HEAD2, each seven characters long, and serves as a two-line heading for each column of hydraulic data in the special-output file. These headings are repeated every PAGESP lines, where PAGESP is input as PAGE in the Run Control Block (section 13.1). The number of lines per page including the two heading lines is specified with PAGESP. Thus, if at least one heading is to be visible while the user is reviewing the special output on a computer screen, then PAGESP should be set to 22. If only one heading at the top of the file is desired, then PAGESP should be set to a number larger than the maximum number of lines expected in the special-output file. Then, only three extra lines will appear in the special-output file: The first line will give the value of PAGESP, and the following two lines will give the identification information. PAGESP is output to the special-output file so that the special-output file can be read by other computer programs modified to read the number of lines per page and then skip over the headings as they read the file. This is a simple modification and should be made on all programs applied to read the special-output file. When DIFFUS=YES, the headings are given at the beginning of each period simulated even though PAGESP lines have not been used.

Two lines are output for each time point in the simulation. The year, month, and day followed by the water-surface elevations at the requested output locations are listed in the first line. The hour of the day (in fractional form) and then the flows at the requested locations are listed in the second line.

This table must always be present even if no output is needed. The headings must be given and the input terminated as usual, resulting in a null table (that is, a table with no values).

13.10 Input Files Block—Input File Table

Purpose: Information describing files used to input flow or water-surface elevation at a flow-path end node is specified with this block. These files are referred to as Point Time Series Files (PTSF) (appendix 2) to distinguish them from the single DTSF used to specify the diffuse flows into the stream system. This block is required at all times and, if no files are input, a null block is specified.

Heading: One line of user-selected information. The suggested string is INPUT FILES.

LINE 1

Variable: HEAD

Format: A80

Example: UNIT NAME

Explanation: This is a user-supplied heading for subsequent information.

LINE 2 (one for each PTSF required)

Variables: UNIT, NAME

Format: I5, 1X, A64

Explanation:

UNIT is the Fortran unit number (see appendix 3) of the PTSF to be read in FEQ. The block is terminated if a negative value for UNIT is given.

NAME is the left-justified file name for the PTSF file to be read in FEQ. The table must be present even if there are no input files. The block is then null consisting of only the heading and a −1 in the

UNIT column. Each input file is associated with a single flow-path end node by specification the negative of the unit number in the source field for a forced boundary condition (CODE=6) in the Network-Matrix Table (section 13.6).

Specification of point inflows in a PTSF can be applied whether or not a DTSF is present. If a DTSF is used, the user must ensure that all required time segments are included in the file. The input/output file system was designed to permit subdivision of a large system at points of known hydraulic control such that a series of smaller systems could be analyzed in sequence from upstream to downstream to represent the larger system. In a case where a large stream system is subdivided in smaller subsystems, the output from an upstream subsystem is stored in a file for later use as input to the downstream system. For this approach to work properly several conditions must be satisfied. First, the same DTSF must be used for all subsystems to ensure that the same length and sequence of time segments appears in all input/output files. Second, the points of subdivision must be at points of known relation between flow and stage in the stream (control points). If this is not the case, then the subdivision of the system into smaller parts will distort the results. If no control points can be identified, then the system should not be subdivided.

13.11 Output Files Block—Output File Table

Purpose: The information describing files to output flow or water-surface elevation at any node in the stream system is specified with this block. These files can later be used for input to another stream system with the Input Files Block (section 13.10). This block is always required and, if no files are output, a null block must be specified. The maximum number of point time series files output allowed in FEQ is specified in the parameter MNFOUT in the INCLUDE file ARSIZE.PRM (appendix 3). This number may be increased as necessary and FEQ recompiled.

Heading: One line of user-selected information. The suggested string is OUTPUT FILES.

LINE 1

Variables: HEAD

Format: A80

Example: UNIT BRAN NODE ITEM TYPE NAME

Explanation: This is a user-supplied heading for subsequent information.

LINE 2 (one for each Point Time-Series File (PTSF) required)

Variables: UNIT, BRA, NODE, ITEM, TYPE, NAME

Format: 3I5, 1X, A4, 1X, A4, 1X, A64

Explanation: The location and type of information output to create a PTSF are specified on this line.

UNIT is the Fortran unit number (see appendix 3) for the file. The block is terminated if the UNIT given is negative.

BRA is the branch number.

NODE is the node number.

ITEM=FLOW for output of flow and ITEM=ELEV for output of water-surface elevation.

TYPE=PNT or TYPE=STAR: Specifies that instantaneous values are to be output (the only ones allowed).

NAME is the left-justified name of the file.

If an Output-File Table is not used in a FEQ simulation, a null table must be present (as in the Input-File Table).

13.12 Operation of Control Structures Block—Operation Tables

Purpose: The operation rules for any dynamically operated control structures are given with this block. This block is only required if SOPER=YES in the Run Control Block (section 13.1). The maximum number of Operation-Control Blocks allowed in FEQ is specified in the parameter MNBLK in the INCLUDE file ARSIZE.PRM (appendix 3). This number may be increased as necessary and FEQ recompiled.

Heading: One line of user-selected information. The suggested string is OPERATION OF CONTROL STRUCTURES.

LINE 1

Variable: BLK

Format: 4X, I5

Example: BLK=00001

Explanation: BLK is the number of the Operation-Control Block table. The input of operation blocks is terminated if BLK=−1. Operation-Control Block tables must be numbered consecutively, beginning with 1.

LINE 2

Variable: BLKTYP

Format: 8X, A4

Example: BLKTYPE=PUMP

Explanation: BLKTYP is the type of the Operation-Control Block. Two types of operation blocks are used: BLKTYPE=PUMP denotes that a pump is controlled, and BLKTYPE=GATE denotes that a gate is controlled.

LINE 3

Variable: MINDT

Format: 6X, F10.0

Example: MINDT=300.

Explanation: MINDT is the minimum time, in seconds, between changes to the gate or pump setting. Changes in the setting can be made only at the boundary between time steps, but at least MINDT seconds must have elapsed since the last change before the next change can be made.

LINE 4

Variable: PINIT

Format: 6X, F5.0

Example: PINIT=0.62

Explanation: PINIT is the initial value for the opening fraction. The opening fraction varies between 0 and 1. When PINIT=0, the gate is completely closed (underflow gates: sluice gates, tainter gates, and others) or fully raised (overflow gates: drum gates and related devices). The gate is so positioned to either force the flow to zero or reduce it to a minimum value. When PINIT=1.0, the gate is positioned such that the flow is maximized for the given water-surface elevation.

For a pump, opening-fraction values specify the speed of the pump relative to the standard speed chosen for the table defining the relation between flow and head for the pump. The pump is off when PINIT=0.0, and the pump is on at the relative speed given if PINIT > 0.

If BLKTYPE=GATE (Gate Operation)

LINE 5

Variable: HEAD

Format: A80

Example:
BRAN NODE KEY MODE MNRATE LRATE LOWLIM HGHLIM HRATE LPRI NPRI HPRI DPDT

Explanation: These are user-supplied headings for subsequent information.

LINE 6 (one for each control point)

Variables: BRA, NODE, KEY, MODE, MNRATE, ML, LL, LU, MU, LPRI, NPRI, HPRI, DPDT

Format: I5, 2A5, I5, 5F7.0, 3I5, F5.0

Explanation: The parameters defining a control point used in the operation of a structure are specified on this line.

BRA is the branch number for the control point. BRA=0 if the node is a flow-path end node. BRA < 0 denotes the end of input for the current block.

NODE is the node label for the control point.

KEY=ELEV if water-surface elevation is monitored at the control point, KEY=QCON if flow rate is monitored at the control point and the null zone limits are constant values, and KEY=QVAR if flow rate is monitored at the control point and the null zone limits are variable. The null zone is that range of the monitored value (flow or elevation) in which the setting of the structure will not be changed.

In the QVAR option, the gate will be operated so that the outflow will follow the variable null zone closely if the outflow changes slowly enough. If the null zone changes rapidly, the simulation may not be able to follow the null-zone changes because of inadequacies in the rules for opening the gate or in the head available. The QVAR rule may be overridden by other rules on the basis of the priority of the various control points.

MODE=0 means that the structure opening is changed whenever the monitored value at the control point is outside the null zone. MODE=1 means that the structure opening is changed only if the monitored value at the control point is outside the null zone and is not moving in the right direction with sufficient speed.

MNRATE is the minimum change per hour under MODE=1 required to avoid changing the structure opening. For example, if KEY=ELEV, indicating that water-surface elevation is monitored, and if MNRATE=0.01, then the elevation at the monitored point must be moving toward the null region at a rate exceeding 0.01 ft/hr to avoid changing the structure opening.

The null region is needed to prevent or minimize searching for the structure’s opening fraction. Thus, the action indicated in the null region is no change in the opening fraction. Experience indicates that MODE=1 is the best choice because the response at the control point to the structure-opening change can be delayed so that conditions at the control point are still out of the null zone but are moving toward the null zone. If MODE=0 is used, the structure-opening fraction will increase to the maximum value, resulting in a large overshoot of the desired result. This will then be followed by a decrease in the structure opening to its minimum value and another overshoot of the desired result. Use of MODE=1 and a properly sized null zone can reduce the search for the opening fraction.

ML is the rate factor for the rate of change of opening when conditions at the control point are below the null zone. This rate factor is a multiplier on the distance of the monitored value from the closest boundary of the null region. The resulting rate is the change per hour of the structure opening as measured by an opening fraction, p, which is taken as 0.0 when the flow is restricted by the maximum amount and as 1.0 when the flow is restricted by the minimum possible amount. The actual rate of opening used is limited by DPDT, given later in the input. The rate-factor sign is determined by the location and design of the control structure relative to the control point.

LL is the lower limit for the null zone.

LU is the upper limit for the null zone.

MU is the rate factor for the rate of change of opening when conditions at the control point are above the null zone.

LPRI is the numerical priority of the action for this control point when the monitored value is below the null zone. Numerical priority 1 is the highest priority, 2 is next highest, and so forth. The numbers are ordinal only; that is, used only for relative ranking. There is no degree of priority difference, so the only relation used is the quality of equal to, greater than, or less than.

NPRI is the priority of the action for this control point when the monitored value is in the null zone.

HPRI is the priority of the action for this control point when the monitored value is above the null zone.

DPDT is the absolute value of the maximum permitted rate of change in the opening fraction for the structure.

LINE 6a (one for each case where KEY=QVAR on Line 6)

Variable: HEAD

Format: A80

Example: NUMB NZHW NODE NDWT NODE NDWT NODE NDWT

Explanation: These are user-supplied headings for subsequent information.

LINE 6b (one for each case where KEY=QVAR on Line 6)

Variables: NUMB, NZHW, (NDVEC(J), NDWT(J), J=1,NUMB)

Format: I5, F5.0, 7(I5,F5.0)

Explanation: The values required to define the variable null-zone limits are specified on this line.

NUMB is the number of flow-path end nodes used in computing the flow to define the midpoint of the null zone.

NZHW is the null-zone half width. The null-zone half width is added to the midpoint flow for the null zone to specify the upper limit of the null zone. It is subtracted from the midpoint flow to give the lower limit.

NDVEC specifies the numbers of the NUMB flow-path end nodes that are used to compute the midpoint of the null zone.

NDWT is the weight used for the flow at each flow-path end node for computing the midpoint of the null zone. This weight can be positive or negative. The weight on the flows at the flow-path end nodes can be used to deduct flow leaving the stream system by some path other than the structure being operated in the control block. In addition, the weight can be used to account for other sources of inflow, such as diffuse inflow not considered at any flow-path end node. As the heading indicates, the flow-path end node and the associated weight are given pairwise in the input.

If BLKTYPE=PUMP (Pump operation)

LINE 5

Variable: HEAD

Format: A80

Example: BRAN NODE KEY MNRATE RISE FALL ONPR OFPR

Explanation: These are user-supplied headings for subsequent information.

LINE 6 (one for each control point)

Variables: BRA, NODE, KEY, MNRATE, RISE, FALL, ONPR, OFPR

Format: I5, 2A5, F7.0, 4I5

Explanation: The parameters defining a control point used in the operation of a variable-speed pump are specified on this line.

BRA is the branch number for the control point. If the node is a flow-path end node, BRA=0 because the node label will provide complete information on the node location. BRA < 0 denotes the end of input for the current block.

NODE is the label for the node at control point.

KEY=ELEV if water-surface elevation is monitored at the control point, and KEY=QCON if flow rate is monitored at the control point. Pump control does not permit a variable-width null zone as for gates when KEY=QVAR is specified. The null zone is that range of the monitored value (flow or water-surface elevation) in which the setting of the pump will not be changed.

MNRATE is the minimum change per hour in the monitored variable (flow or water-surface elevation) required to change the table defining the pump operation. Two pump-operation tables are considered: one for a rising level at the control point and one for a falling level. The level must have been moving at least as fast on the average as the rate given in MNRATE before the computed level is considered to be really rising or falling. This tolerance is included to prevent rapid switching between the tables.

RISE is the number of the table specifying the pump speed as a function of the monitored variable when the level is increasing in magnitude. This function includes the definition of the null zone for the pump. A discussion of the pump-speed function is presented in section 8.1.2.2.3.2.

FALL is the number of the table specifying the pump speed as a function of the monitored variable when the level is decreasing.

ONPR is the priority assigned to the control-point action when the pump-speed function indicates that the pump be turned on.

OFPR is the priority assigned to the control-point action when the pump-speed function indicates that the pump be turned off.

A dynamically operated structure can include more then one control point, and different actions may be indicated by levels at the various control points. The method used in FEQ simulation to determine which action should be taken is to attach a priority to each of the three possible actions indicated by levels at the control point. The highest priority action for all the control points is the action taken.

Application of the simple priority method for selecting the proper action ignores the direction of motion of the monitored values. Sometimes the action taken when the monitored value is increasing differs from that when the monitored value is decreasing. This complication has been included for pump control but not for gate control. The number of rules used for the operation of control structures is virtually unlimited. Thus, changes will be necessary for specific examples not fitting the current generalized scheme.

13.13 Function Tables Block—Function Tables

Purpose: All function tables are given with this block in a user-selected order. Each table has a unique identifying number, so order is not important. Some or all tables can be stored in one or more files, and the table information from these files, can be obtained during simulation. A negative TYPE value for TYPEs 2, 3, 4, 6, 7, 8, 9, 11, 13, 14, 15, or 20-25 suppresses echoing of the table values. Only the header information will be output by FEQ in this case.

Heading: One line of user-selected information. The suggested string is FUNCTION TABLES; only the first eight characters of the heading are read and must be FUNCTION if the Function Tables Block is moved ahead of the Branch Description Block.

LINE 1 (one for each function table or reference to a table of auxiliary files)

Variable: TABLE, FNAME

Format: 7X, I5, A64

Example: TABLE#=00029

Explanation:

TABLE is the identification number of the function table. If TABLE=−1, then input of function tables is terminated. If TABLE=−n where n is an integer > 1, function-table input is continued from a file specified by Fortran unit number *n* (see appendix 3). The maximum function table number allowed in FEQ is specified in the parameter MFTNUM in the INCLUDE file ARSIZE.PRM (appendix 3). This number may be increased as necessary and FEQ recompiled.

FNAME is the name of the file to be read in FEQ. If FNAME is nonblank and TABLE=−*n*, a file (called an auxiliary file) will be opened with the name given by the contents of FNAME (most microcomputers) to continue input of the function tables. On IBM mainframes, the name given by FNAME will be the ddname for the DD statement defining the data set. If the FNAME is blank, an implicit open will be done on an IBM mainframe if the proper DD statement defining the unit number given by *n* is present. In some microcomputer systems, the user will be prompted for the file name, whereas program execution will be aborted on other microcomputer systems.

Each specified file in this option is opened, read, and then closed so that the function tables can be divided into many groups for convenience in development and adjustment. The relation between the files depends on how each of the auxiliary files is ended. If the last line of the file is a negative table number other than −1, then the current auxiliary file is closed, the file specified in this last line is opened, and function tables are read from this new auxiliary file. The number of separate files that can be used is unlimited because each file points to the next file. The last file in the sequence is terminated with TABLE=−1.

If the auxiliary file ends without a negative table number, then the auxiliary file is closed and the next table number is read from the input file. Thus, one or more auxiliary files can be given in sequence in the input file to complete the input of the function tables. This technique is preferred because all the auxiliary file names appear in one place. The first method is inconvenient in that the next auxiliary file name appears at the end of the current auxiliary file, and this requires that all of the auxiliary files be scanned in order to establish the files being used. To avoid confusion and to simplify programming, only table numbers describing additional auxiliary files or with the −1 terminator can appear after an auxiliary file has been given. This means that function-table input from the input file must be completed before auxiliary files are referenced to provide the remainder of the function tables.

LINE 2

Variable: TYPE

Format: 5X, I5

Example: TYPE=00002

Explanation: TYPE is the type of function table. The type defines the format and role of the table. A summary of the table types used in FEQ is given in section 11 and table 6.

If TYPE=2 (Single function linear):

LINE 3 (one for each table of type 2 in sequence with Lines 4 and 5)

Variable: REFLEV, FAC3

Format: 5X, F10.0, 5X, F10.0

Example: REFL= 0.0 FAC= 43560.0

Explanation:

REFLEV is the reference level read by FEQ but not now used.

FAC3 is a multiplier on the function values in the table. If the FAC3 entry is omitted, then the multiplier defaults to 1.0.

LINE 4 (one for each table of type 2 in sequence with Lines 3 and 5)

Variable: HEAD

Format: A80

Explanation: This is a user-selected heading for subsequent information.

LINE 5 (one for each argument value for each table of type 2 in sequence with Lines 3 and 4)

Variables: ARG, F(1)

Format: 2F10.0

Explanation:

ARG is the argument value for the tabulated function.

F(1) is the value of the function at the given argument value.

The table is terminated by an argument value less than the argument value on the previous line. Negative values can be used for an argument. Intermediate function values are determined by linear interpolation.

If TYPE=3 (Dual function: linear and integrated linear):

LINE 3 (one for each table of type 3 in sequence with Lines 4 and 5)

Variable: REFLEV, FAC4

Format: 5X, F10.0, 5X, F10.0

Example: REFL=0.0 FAC= 43560.0

Explanation:

REFLEV is the reference level read in FEQ but not now used.

FAC4 is a multiplier on the function values in the table. If the FAC4 entry is omitted, the multiplier defaults to 1.0.

LINE 4 (one for each table of type 3 in sequence with Lines 3 and 5)

Variable: HEAD

Format: A80

Explanation: This is a user-selected heading for subsequent information.

LINE 5 (one for each argument value for each table of type 3 in sequence with Lines 3 and 4)

Variables: ARG, F(1), F(2)

Format: 3F10.0

Explanation:

ARG is the argument value for the functions tabulated.

F(1) is the value of the first function at the given argument value.

F(2) is the value of the second function at the given argument value. F(2) is the derivative of the function tabulated in F(1).

For a table of type 3, F(1) and F(2) must be related with the trapezoidal rule with F(1) being given by F(2). The table is terminated if the argument value given is smaller than the argument value on the previous line.

If TYPE=4 (Dual function: cubic Hermite):

Same as for TYPE=3 except that F(2) is given by the derivative of a piecewise cubic Hermite polynomial describing F(1).

If TYPE=6 (Two dimensional):

LINE 3 (repeated as needed to represent the range of flows for the table of type 6, paired with Line 4 and in sequence with Lines 5 and 6)

Variable: HEAD

Format: A80

Example: HEADUP FREEDROP

Explanation: This is the heading for the part of the table detailed by Line 4.

LINE 4 (repeated as needed to represent the range of flows for the table of type 6, paired with Line 3 and in sequence with Lines 5 and 6).

Variable: HU, FDROP

Format: 2F10.0

Explanation:

HU is the value of upstream head. A value of HU < 0 terminates input to this part of the table.

FDROP is the corresponding drop in water-surface elevation required from the upstream node to the downstream node for the maximum physically possible flow. The maximum possible flow is called the free flow, and the drop in elevation is called the free drop.

The values of upstream head and free drop should be entered in ascending order of upstream head.

LINE 5 (repeated as needed to represent the range of flows for the table of type 6, paired with Line 6 and in sequence with Lines 3 and 4)

Variables: HEAD

Format: A80

Example: PROPORTION OF FREE DROP AND FLOW FOR CONSTANT UPSTREAM HEAD

Explanation: This is the heading for the part of the table detailed in Line 6.

LINE 6 (repeated as needed to represent the range of flows for the table of type 6, in sequence with Lines 3, 4, and 5)

Variable: P, Q

Format: 2F10.0

Explanation:

P is the proportion of free drop.

Q is the corresponding flow for constant upstream head.

Values must be specified for each of the upstream heads in the first part of the table specified on Line 4. The first line for each upstream head must be $P = 0$ and $Q = 0$. Subsequent lines for each upstream head must give *P* and *Q* in ascending order of P until the final value with $P = 1.0$ and $Q =$maximum physically possible flow for the given upstream head. Thus, each upstream head in the first part of the table has a minimum of two lines in the second part of the table. The sets of lines for the upstream heads appear in the same order as the upstream heads. Each of the sets must end with a value of $P = 1.0$.

The control structure is between an approach section and a departure section, as shown for a sluice gate in figure 44. The length of channel between the approach section and the entrance to the control structure is called the approach reach, and the length of channel between the exit of the control structure and the departure section is called the departure reach. The approach and departure sections will be input to FEQ. Thus, the 2-D table represents the relation among the water-surface elevations at these sections and the flow through the control structure. The lengths of these reaches must be selected with consideration of possible flow directions. If the flow through the control structure is unidirectional, then the approach section should be beyond the drawdown local to the structure. In the case of culvert or bridge openings, the approach reach should be at least as long as the total opening width. The length of the departure reach should be such that the velocity distribution in the channel is almost fully developed. One approach for selecting a length is to use the rate of expansion of a free jet in the range of 1:4 to 1:5, where this is the ratio of opening width to jet width. This is the rate of expansion on one side of the jet, so the expansion in width is really about 1:2 to 1:2.5. With this rate of expansion and the width of the structure opening, the length of the departure reach required such that the jet will completely fill the departure section can be computed. It is assumed that the velocity distribution in the departure section will be virtually the same as the velocity distribution in that section if the control structure were not present.

The method suggested above often results in departure-reach lengths much longer than the approach-reach length. Furthermore, bidirectional flow through the control structure results in approach and departure reaches functioning conversely as the flow reverses direction. It is not practical to vary the length of the corresponding branch lengths with changes in direction of flow. To simulate bidirectional flow, it is assumed that the 2-D table can represent losses in the departure reach greater than those resulting from the normal frictional resistance if the structure were not present. Thus, the departure reach can be compressed to the same length as the approach reach, and only local losses resulting from the structure are of concern. The frictional losses are then approximated in the branch, which now extends into the area of disturbed flow downstream of the structure. In this process, the local losses resulting from the expansion of the flow are assumed to be much larger than those resulting from boundary friction, as is most often the case.

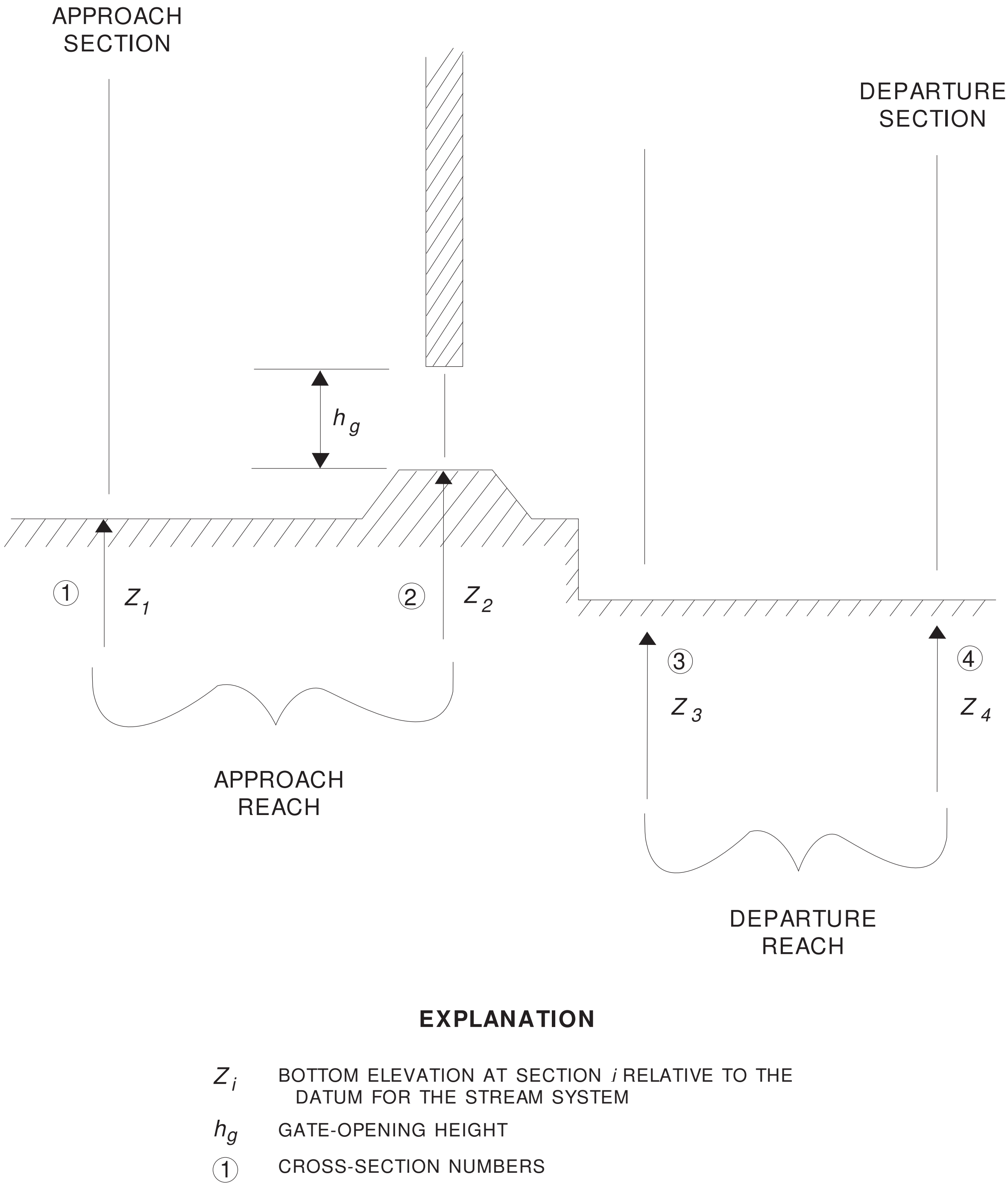


Figure 44. Location of cross sections for sluice-gate analysis, as simulated in the Full EQuations model.

A 2-D table represents all possible flows for a range of water-surface elevations in the approach and departure sections. It is convenient to define an upstream head (in the approach section) and a downstream head (in the departure section) where upstream and downstream denote the actual flow direction and not just the nominal location of a node. The flow is always from approach section to control-structure entrance to control-structure exit to departure section. The upstream and downstream head are referenced to the same datum, and equality of upstream and downstream head results in zero flow.

If TYPE=7, 8, or 9 (Alternative time argument for table types 2, 3, or 4):

These three table types include a time argument that differs from the argument given in table types 2, 3, and 4. The time argument is given in the form of year, month, day, and hour. Line 3 is exactly the same as for types 2, 3, and 4 but Lines 4 and 5 differ.

LINE 4

Variable: HEAD

Format: A80

Example: YEAR MN DY HOUR F(1) F(2)

Explanation: This is a user-selected heading for subsequent information.

LINE 5 (repeated as needed to input the entire time series of input data)

Variables: YR, MN, DY, HR, F(1), [F(2)]

Format: I4, 1X, I2, 1X, I2, 3F10.0

Explanation:

YR is the year for which the function values apply. All four digits of the year must be given.

MN is the month (1-12) for which the function values apply.

DY is the day of the month (1-31) for which the function values apply.

HR is the hour of the day, including any fractional hour for which the function values apply.

F(1) is the value of the first function at the given time argument.

F(2) is the value of the second function at the given time argument, if required (table types 8 and 9).

The values for year, month, and day need be given only for the first argument in the table and whenever they change. Blank fields for these three values are replaced with the most recent nonblank values; that is, the values of year, month, and day propagate into blank fields. The hour of the day must be given for each entry because this value will always differ. The year, month, and day must be valid calendar dates (an error message is issued if they are not). Further, the year must be 1859 or later. An error message is issued if the year falls outside the valid range. The end of input for a table with the year, month, and day arguments is indicated by a specified year, month, day that is earlier in time than the value on the preceding line.

If TYPE=11 (Dual-function wind speed and direction):

LINE 3

Variable: REFLEV, FAC

Format: 5X, F10.0, 5X, F10.0

Example: REFL= 0.0 FAC= 1.0

Explanation:

REFLEV is the reference level read in FEQ but not now used.

FAC is a multiplier on the wind-speed values in the table but not the direction. Thus, FAC can be used to adjust measured wind speeds to the correct units and measurement elevation.

LINE 4

Variable: HEAD

Format: A80

Example: YEAR MN DY HOUR SPEED AZIMUTH

Explanation: This is a user-selected heading for subsequent information.

LINE 5 (repeated as needed to input the entire time series of input data).

Variables: YR, MN, DY, HR, F(1), F(2)

Format: I4, 1X, I2, 1X, I2, 3F10.0

Explanation:

YR is the year for which the wind values apply. All four digits must be in the year given.

MN is the month (1–12) for which the wind values apply.

DY is the day of the month (1–31) for which the wind values apply.

HR is the hour of the day, including any fractional hour, for which the wind values apply.

F(1) is the speed of the wind at the given time argument.

F(2) is azimuth of the wind direction, measured clockwise from north in degrees. This value corresponds to the tradition of wind direction being that from which the wind is coming. For example, an east wind has an azimuth of 90 degrees.

The values for year, month, and day need be given only for the first argument in the table and whenever they change. Blank fields for these three values are replaced with the most recent nonblank values; that is, the values of year, month, and day propagate into blank fields. The hour of the day must be given for each entry because this value should always differ. The year, month, and day must be valid calendar dates (an error message is issued if they are not). Further, the year must be 1859 or later. An error message is issued if the year falls outside the valid range. The end of input for a table with the year, month, and day arguments is indicated by specifying a year, month, day that is earlier in time than the value on the preceding line.

If TYPE=13 (Two dimensional):

Type 13 tables contain the same information as type 6 tables except that the proportions of free drop must be the same for all upstream heads. Tables of this type are computed in FEQUTL (Franz and Melching, in press) by applying the same values of proportion of free drop for each upstream head. Therefore, the preferred output table type is type 13 because it is more compact in output, more easily read by a user, and takes less space in FEQ storage. A small part of a large table of type 13 follows:

```
TABLE#=  300
TYPE=  -13
LABEL=A-CRK1
NHUP=    7
NPFE=   10
   HUP   1000-4  1500-4  2000-4  4000-4  6000-4  8000-4  1000-3
 FDROP   2482-4  2687-4  2916-4  3787-4  4598-4  5293-4  5958-4
   PFD   Flows for HUP and Proportion of FDROP
 1000-5  3582-5  9038-5  1674-4  8882-4  2471-3  5298-3  9747-3
 4000-5  6462-5  1701-4  3182-4  1706-3  4757-3  1023-2  1884-2
 9000-5  8740-5  2332-4  4452-4  2428-3  6796-3  1469-2  2712-2
 1600-4  9930-5  2809-4  5423-4  3014-3  8480-3  1849-2  3416-2
 2500-4  1054-4  3080-4  6080-4  3447-3  9758-3  2150-2  3982-2
 3600-4  1054-4  3211-4  6459-4  3734-3  1063-2  2399-2  4325-2
 4900-4  1054-4  3275-4  6722-4  3925-3  1120-2  2519-2  4674-2
 6400-4  1054-4  3275-4  6722-4  3925-3  1155-2  2600-2  4837-2
 8100-4  1054-4  3275-4  6722-4  3999-3  1170-2  2635-2  4914-2
 1000-3  1054-4  3275-4  6722-4  4005-3  1174-2  2643-2  4934-2
```

NHUP is the number of upstream heads in the table, and NPFD is the number of partial free drops tabulated in the table. The values following HUP are the upstream heads in compressed floating-point form. The first four digits of the number give the value to be multiplied by the power of 10 given by the last two characters of the number. The first character of the last two characters is always a plus or a minus sign. Thus, the first upstream head value given in the example table on the line starting with HUP is 0.1 and the last upstream head given is 1.0. The next line lists the drop to free flow for the upstream head immediately above it. The last drop to free flow in the table is 0.5958. The column headed by PFD lists the proportion of free drop (partial free drop). A partial free drop of zero is absent because the flow at zero partial free drop is zero. The numbers under each of the upstream-head and free-drop pairs are the flow for the partial free drop at the left of the line. Thus, the flow from the table for an upstream head of 0.6 and a partial free drop of 0.25 is 9.758. For this same upstream head the flow at free drop (partial free drop of 1.00) is 11.74.

If TYPE=14 (Two dimensional):

Tables of type 14 tabulate the upstream head for a structure as a function of the downstream head and the flow rate. An example of part of such a table follows:

```
TABLE#= 960
TYPE= -14
LABEL=CONTRACTION INTO LEFT GAP
NHDN=  9
NPFQ= 20
 QFREE   4177-1  5076-1  5928-1  6999-1  8404-1  1015+0  1216+0  1444+0  1702+0
   HDN   9000-3  9200-3  9400-3  9600-3  9800-3  1000-2  1020-2  1040-2  1060-2
   PFQ   Ups heads for HDN and Proportion of QFREE
 6746-5  9023-3  9225-3  9425-3  9626-3  9827-3  1003-2  1023-2  1043-2  1064-2
 1259-4  9078-3  9284-3  9485-3  9686-3  9890-3  1010-2  1030-2  1051-2  1072-2
 1813-4  9152-3  9363-3  9566-3  9769-3  9977-3  1019-2  1040-2  1062-2  1084-2
 2349-4  9239-3  9456-3  9662-3  9867-3  1008-2  1030-2  1052-2  1075-2  1097-2
 2872-4  9332-3  9557-3  9766-3  9974-3  1019-2  1042-2  1065-2  1089-2  1113-2
 3384-4  9430-3  9663-3  9875-3  1009-2  1031-2  1055-2  1079-2  1104-2  1129-2
 3887-4  9530-3  9770-3  9986-3  1020-2  1044-2  1068-2  1094-2  1119-2  1145-2
 4384-4  9630-3  9879-3  1010-2  1032-2  1056-2  1082-2  1108-2  1135-2  1162-2
 4874-4  9731-3  9989-3  1021-2  1044-2  1069-2  1095-2  1123-2  1150-2  1179-2
 5359-4  9832-3  1010-2  1033-2  1056-2  1081-2  1109-2  1137-2  1166-2  1195-2
 5839-4  9933-3  1021-2  1044-2  1068-2  1094-2  1122-2  1152-2  1182-2  1212-2
 6314-4  1003-2  1032-2  1055-2  1079-2  1106-2  1136-2  1166-2  1198-2  1229-2
 6786-4  1013-2  1042-2  1067-2  1091-2  1119-2  1150-2  1181-2  1213-2  1246-2
 7254-4  1023-2  1053-2  1078-2  1103-2  1131-2  1163-2  1195-2  1229-2  1262-2
 7719-4  1033-2  1064-2  1089-2  1114-2  1144-2  1176-2  1210-2  1244-2  1279-2
 8181-4  1043-2  1075-2  1100-2  1126-2  1156-2  1190-2  1224-2  1260-2  1295-2
 8639-4  1053-2  1086-2  1111-2  1138-2  1168-2  1203-2  1238-2  1275-2  1312-2
 9095-4  1063-2  1096-2  1122-2  1149-2  1181-2  1216-2  1253-2  1290-2  1328-2
 9549-4  1073-2  1107-2  1133-2  1161-2  1193-2  1229-2  1267-2  1305-2  1344-2
 1000-3  1083-2  1118-2  1145-2  1172-2  1205-2  1243-2  1281-2  1321-2  1361-2
```

NHDN is the number of downstream heads, and NPFQ is the number of partial free flows tabulated. The values in this table are in compressed floating-point form. The first four digits of the number give the value to be multiplied by the power of 10 given by the last two characters of the number. The first character of the last two characters is always a plus or a minus sign. The numbers in the line following QFREE are the flow rate required to yield free flow at the downstream head given immediately below in the line beginning with HDN. For a given downstream head, a flow greater than the QFREE value indicates free-flow conditions; thus, the relation between flow and upstream head must be applied as the internal boundary condition. The upstream head for each of the values of QFREE appears in the last line of the table at a partial free flow (PFQ of 1.00). For example, at a downstream head of 9.6, the upstream head at free flow is 11.72 and the free flow is 699.9. As an example, assume that the table is accessed by use of a downstream head of 9.60 and a flow of 840.4. Free-flow conditions would be simulated because the value of the flow argument, 840.4, is larger than the flow at the free-flow limit for a downstream head of 9.6. The upstream head sequence at free flow and the free flow would be used to determine the upstream head for the given flow. In this special case, the tabulated upstream head is 12.05. Keeping the downstream head at 9.60 and assuming that the flow argument is 236.85 results in a partial free flow of 0.3384. For these flow and downstream-head values, the upstream head is 10.09.

If TYPE=15 (Table of table numbers):

Tables of type 15 are 1-D with a single argument and four function values used to develop a 3-D table from a series of 2-D tables. The single argument is the vertical extent of the rectangular gate opening. The first function value is the table number of a 2-D table listing the flows for a range of upstream and downstream heads for a gate setting held constant at the value given for the argument. The three additional function values listed in the table for each gate opening describe boundaries between flow conditions. H1FWULR is the ratio between the upstream head at section 1 (the approach section) at the upper limit of free-weir flow and the gate opening. H4FWULR is the ratio between the head at section 4 (departure section) at the upper limit of free-weir flow and the gate opening. H4SWSOMDR is the ratio between (1) the head at section 4 at the boundary between submerged-weir flow and submerged-orifice flow when the head at section 1 is midway between a head equal to the gate opening and a head equal to the upper limit of free-weir flow and (2) the gate opening. Complete details on the interpolation method are given in the FEQUTL documentation report (Franz and Melching, in press).

LINE 3

Variables: REFLEV, LABEL

Format: 5X, F10.0, 1X, A50

Example: REFL=   0.0 LABEL= Tainter gates at Lock and Dam 21.

Explanation:

REFLEV is a currently unused reference level.

LABEL is an identifying label for the table. This label is appended to the gate opening and is used to compute the 2-D tables applied in the tables of type 15.

LINE 4

Variable: HEAD

Format: A80

Example: OPENING TAB#   H1FWULR   H4FWULR H4SWSOMDR

Explanation: The heading shown is the one generated in the UFGATE command in FEQUTL (Franz and Melching, in press).

LINE 5 (repeated as necessary to input all the arguments needed to describe gate operation)

Variables: ARG, TAB2D, F(1), F(2), F(3)

Format: F10.0, I5, 3F10.0

Explanation:

ARG is the argument for the function tabulated. This is the vertical gate opening.

TAB2D is the table number for the 2-D table of type 13 that represents the flows through the gate for a range of upstream and downstream heads with the gate setting fixed at the argument value. The gate openings are all positive. A gate opening of zero must not appear in the table. Guidelines for selecting the gate-opening sequence are given in the discussion of the UFGATE command in Franz and Melching (in press).

F(1:3) is the three values defining boundaries between flow regions computed in the UFGATE command. These values were described previously.

The table is terminated by an argument value less than the argument value on the preceding line.

If TYPE=20 (Cross-sectional hydraulic characteristics):

LINE 3

Variable: STAT

Format: 8X, F10.0

Example: STATION=1.456

Explanation: STAT is the station of the cross section, read and used if the “tab” or “TAB” option is invoked in the Branch Description Tables Block (section 13.2). Otherwise, the value is ignored.

LINE 4

Variables: ELEV, CUTOFF, EXT, FAC5

Format: 10X, F12.0, 7X, F10.0, 5X, F10.0, 5X, F5.0

Explanation:

ELEV is the elevation of the minimum point in the cross section represented, read and used if the “tab” or “TAB” option is invoked in the Branch Description Tables Block (section 13.2). Otherwise, the value is ignored.

CUTOFF is the elevation below which information in the cross-section table is input but discarded.

EXT is the maximum extent of extrapolation allowed from the last two entries in the table. If EXT < 0, then the top width is held constant at the maximum tabulated value during the extrapolation. If EXT > 0, then the top width is extrapolated linearly.

FAC5 is the value used to adjust all values of conveyance in the table. FAC5 is a divisor on the conveyance (not the square root of conveyance). If the cross section is not subdivided and Manning’s *n* is constant, then FAC5 is a multiplier on Manning’s *n*. If FAC5 is not given, then the default is 1.0.

LINE 5

Variable: HEAD

Format: A80

Explanation: These are user-selected headings for subsequent information. Tables of TYPE=20 are computed in FEQUTL (Franz and Melching, in press), which also produces the headings.

LINE 6 (repeated as necessary to input data for all depth values needed to represent the cross section)

Variables: Y, T, A, KH, BETA

Format: 2F10.0, 2F13.0, F10.0

Explanation:

Y specifies the values of water-surface height used in the tabulation of the hydraulic characteristics. Water-surface height is defined relative to the point of minimum elevation in the cross section. A negative value of water-surface height terminates the table.

T is the top width of the water surface at water-surface height Y.

A is the area of the cross section at water-surface height Y.

KH is the square root of the conveyance of the cross section at water-surface height Y.

BETA is the momentum-flux correction factor for flow in the cross section at water-surface height Y.

If TYPE=21-25 (Cross-sectional hydraulic characteristics):

These additional table types for cross-sectional hydraulic characteristics follow a format similar to that for TYPE=20 with the addition of more cross-sectional hydraulic characteristics. The hydraulic characteristics added for each table and the methods used for interpolating in the tables are given in section 11.1.5. The detailed format for the tables can be determined by computing tables of the appropriate type by use of the utility program, FEQUTL (Franz and Melching, in press), or the examples obtained by electronic retrieval from the World Wide Web (WWW) at http://water.usgs.gov/software/feq.html and by anonymous File Transfer Protocol (FTP) from water.usgs.gov in the pub/software/surface_water/feq directory.

13.14 Free-Node Initial Conditions Block—Free-Node Table

Purpose: The sign, elevation, and initial values of any free nodes are assigned with this block. This block is required only if the system has free nodes. Free nodes are present whenever twice the number of branches is less than the number of flow-path end nodes. The maximum number of free nodes allowed in FEQ is specified in the parameter MNFREE in the INCLUDE file ARSIZE.PRM (appendix 3). This number may be increased as necessary and FEQ recompiled.

Heading: One line of user-selected information. The suggested string is FREE NODE INITIAL CONDITIONS.

LINE 1

Variable: HEAD

Format: A80

Example: NODE DEPTH DISCHARGE DEP DATUM SIGN

Explanation: This is a user-selected heading for subsequent information.

LINE 2 (one for each free node)

If NODEID=NO then

Variables: NODE, DEPTH, DISCH, ELV, SIGN

Format: I5, 3F10.0, I5

If NODEID=YES then

Variables: NODE, NAME, DEPTH, DISCH, ELV, SIGN

Format: I5, 1X, A8, 1X, 3F10.0, I5

Explanation:

NODE is the node label of the free node.

NAME is an optional name for the free node.

DEPTH is the initial water-surface height at the free node.

DISCH is the initial discharge at the free node.

ELV is the elevation of the reference point defining the datum for water-surface height at the free node. If this datum is zero, then water-surface height and water-surface elevation are the same. Assigning a zero value is often a good choice for level-pool reservoirs and generally is the most convenient choice for other free nodes.

SIGN is the sign of the free node (+1 or −1). The sign for each of the free nodes is read in FEQ simulation to be consistent with input data from older versions of FEQ. In Version 7.0 and later, however, the sign of each free node is determined from the Network-Matrix Control Input. A warning message is issued for incorrect signs, and the correct sign from context in the Network-Matrix Control Input is applied. A downstream and an upstream node should have positive and negative signs, respectively.

No terminator is needed for this block because the number of free nodes is known at this point in the analysis of the input.

13.15 Backwater Analysis Block—Backwater Table

Purpose: The initial flows and the starting depth at control points are specified with this block to estimate the initial conditions at all nodes in steady-flow analysis.

Heading: One line of user-selected information. The suggested string is BACKWATER ANALYSIS.

LINE 1 (one for each branch; repeated in combination with Line 2)

Variable: NBR

Format: 14X, I5

Example: BRANCH NUMBER=00001

Explanation: NBR is the branch-number identification for the flow-specification table for a branch. Branch numbers must be given in ascending order.

If NBR < 0 on Line 1, then

LINE 2 (one for each branch; repeated in combination with Line 1)

Variable: DISCH

Format: 10X, F10.0

Example: DISCHARGE=250.0

Explanation: This is the flow in the branch if the flow is the same for all nodes on the branch. If the flows change along the branch, then a more detailed input is required.

If NBR > 0 on Line 1, then

LINE 2

Variable: HEAD

Format: A80

Example: NOD1 NOD2 DISCHARGE

Explanation: This is a user-selected heading for subsequent information.

LINE 3 (repeated as needed to define a flow for each node on the branch)

Variables: ND1, ND2, DISCH

Format: 2I5, F10.0

Explanation:

ND1 is the first node in range of node numbers.

ND2 is the second node in range of node numbers.

DISCH is the discharge to use for ND1 through ND2.

The first and last node in the range must be given even if they are the same number.

LINE 4

Variable: HEAD

Format: A80

Example: BRAN CODE ELEVATION   EXN

Explanation: These are user-selected headings for subsequent information.

LINE 5 (one for each branch)

Variables: BRA, CODE, ELEV, EXN

Format: 2I5, F10.0, I5

Explanation:

BRA is the number of the simulated branch. If BRA=0 and if CODE > 0, then the flow-path end node given in CODE is assigned the elevation of the flow-path end node given in EXN.

CODE is the code defining the operation to be done:

CODE=0 indicates that the starting elevation for the branch given by BRA is the sum of the elevation of the water surface at the flow-path end node specified in EXN and the value specified for ELEV. The value specified for ELEV in this case may not be an elevation. It will likely be an increment in water-surface elevation.

CODE≠0 indicates that the starting elevation is taken from ELEV. The starting elevation may be a first guess to be refined in FEQ simulation. The following options are available:

CODE=1 indicates that the starting elevation is used as the final elevation. This is the method for setting a known fixed elevation.

CODE=2 indicates that the critical depth is computed at the downstream end of the branch by use of the elevation given in ELEV as a first estimate. The resulting elevation at critical depth is then used as the starting elevation for the branch.

CODE=3 indicates that the water-surface height is computed from a stage-discharge relation as specified in the Network-Matrix Control Block (section 13.6) for CODE=4, TYPE=1, 2, or 3, by use of the elevation in ELEV as a first estimate. The resulting elevation is then used as the starting elevation for the branch.

CODE=–3 indicates that the water-surface height is computed from a stage-discharge relation specified at the free node given in EXN in the same manner as for CODE=3, BRA=0 in this case. With this value, the initial condition for this process is taken from the Free Node Initial Conditions Block (section 13.14).

CODE=5 indicates that critical depth is computed at the downstream end of the branch by use of the elevation in ELEV as a first estimate. The resulting water-surface elevation is compared with the elevation at the node specified in EXN. The elevation is then used at critical depth if it is greater than the elevation specified in EXN. Otherwise, the elevation at the node specified in EXN is used.

CODE=6 indicates that the upstream water-surface elevation is computed from the flow and the downstream water-surface elevation for a two-node control structure is defined by CODE=5, TYPE=6 in the Network Matrix Control Block (section 13.6). The source for the downstream water-surface elevation is given in EXN. This must be the downstream node for head for the two-node control structure. The ELEV value gives an estimate of the initial water-surface elevation at the upstream node of the control structure. If this upstream node is on a branch, the branch number is given in BRA and the water-surface profile for the branch is computed starting at the elevation found for the upstream node of the control structure. If the upstream node is a new input-style free node, the node label is given in BRA and the elevation found is assigned to the free node. If the upstream node is an old input-style free node number, the node number prefixed by a minus sign is given in BRA, and the elevation found is assigned to the free node. In this option, it is assumed that the flow at the flow node of the control structure is non-zero. If the flow is zero, the user will know the elevation and should supply it directly.

Line 5 is repeated for each branch in the stream system but not necessarily in branch number order. The order is defined by the requirement that the water-surface elevation be known at the downstream end of the branch. Thus, the first branch appearing in the input will be the most downstream branch, where a boundary condition will define the water-surface elevation directly or indirectly. The branches are then given in an order such that the result of the analysis of one branch will provide an elevation for one or more upstream branches. A value of −1 in the BRA column terminates input for the backwater computations and indicates that all flow depths have been defined.

REFERENCES CITED

Abbott, M.B., 1974, Continuous flows, discontinuous flows and numerical analysis: Journal of Hydraulic Research, v. 12, no. 4, p. 417–467.

Abbott, M.B., and Basco, D.R., 1989, Computational fluid dynamics: New York, Longman Scientific and Technical with John Wiley and Sons, 425 p.

Ackers, P., and Harrison, A.J.M., 1964, Attentuation of flood waves in partfull pipes: Proceedings of the Institution of Civil Engineers, London, v. 28, p. 361–382.

Ahlberg, J.H., Nilson, E.N., and Walsh, J.L., 1967, The theory of splines and their applications: New York, Academic Press, 284 p.

Blalock, M.A., and Sturm, T.W., 1981, Minimum specific energy in compound open channel: Journal of the Hydraulics Division, American Society of Civil Engineers, v. HY107, no. 6, p. 699–717.

———, 1983, Closure for "Minimum specific energy in compound open channel": Journal of Hydraulic Engineering, American Society of Civil Engineers, v. 109, no. HY3, p. 483–486.

Bradley, J.N., 1970, Hydraulics of bridge waterways: U.S. Department of Transportation, Federal Highway Administration, Bureau of Public Roads, Hydraulic Design Series, no. 1, 111 p.

Chaudhry, Y.M., and Contractor, D.N., 1973, Application of the implicit method to surges in open channels: Water Resources Research, v. 9, no. 6, p. 1605–1612.

Chow, V.T., 1959, Open-channel hydraulics: New York, McGraw-Hill, 680 p.

Cunge, J.A., Holly, F.M., Jr., and Verwey, A., 1980, Practical aspects of computational river hydraulics: London, Pitman Publishing Limited (reprinted 1986, Iowa City, by the Iowa Institute of Hydraulic Research), 420 p.

Cunge, J.A., and Wegner, M., 1964, Integration numerique des equations d'ecoulement de Barre de St. Venant par un schema implicite de differences finies. Application au cas d'une galerie tantot en charge tantot a surface libre: La Houille Blanche, no. 1, p. 33–39.

Daugherty, R.L., and Franzini, J.B., 1977, Fluid mechanics with engineering applications: New York, McGraw-Hill, 564 p.

DeLong, L.L., 1989, Mass conservation—1-D open-channel flow equations: Journal of Hydraulic Engineering, American Society of Civil Engineers, v. 115, no. 2, p. 263–269.

———, 1991, Closure for "Mass conservation—1-D open-channel flow equations": Journal of Hydraulic Engineering, American Society of Civil Engineers, v. 117, no. 8, p. 1080–1082.

Dennis, J.E., Jr. and Schnabel, R.B., 1983, Numerical methods for unconstrained optimization and nonlinear equations: Englewood Cliffs, N.J., Prentice-Hall, 378 p.

El-Khashab, A., and Smith, K.V.H., 1976, Experimental investigations of flow over side weirs: Journal of the Hydraulics Division, American Society of Civil Engineers, v. 102, no. HY9, p. 1255–1268.

Fisk, G.G., 1988, Discharge ratings for control structures at McHenry Dam on the Fox River, Illinois: U.S. Geological Survey Water-Resources Investigations Report 87–4226, 24 p.

Franz, D.D., and Melching, C.S., in press, Approximating hydraulic properties of open channels and control structures, using the Full EQuations UTiLity (FEQUTL) program: U.S. Geological Survey Water-Resources Investigations Report 97–4037.

Fread, D.L., 1974, Numerical properties of implicit four-point finite difference equations of unsteady flow: Silver Spring, MD., National Weather Service, NOAA Technical Memorandum NWS HYDRO-18, 38 p.

Froehlich, D.C., 1991, Discussion of "Mass conservation—1-D open-channel flow equations": Journal of Hydraulic Engineering, American Society of Civil Engineers, v. 117, no. 8, p. 1078–1080.

Hager, W., 1987, Lateral outflow over side weirs: Journal of Hydraulic Engineering, American Society of Civil Engineers, v. 113, no. 4, p. 491–504.

Hamming, R.W., 1973, Numerical methods for scientists and engineers (2d ed.): New York, McGraw-Hill, 721 p.

Henderson, F.M., 1966, Open channel flow: New York, MacMillan, 522 p.

Hulsing, H., Smith, W., and Cobb, E.D., 1966, Velocity-head coefficients in open channels: U.S. Geological Survey Water-Supply Paper 1869–C, 45 p.

Ishii, A.L.and Turner, M.J., in press, Verification of a one-dimensional, unsteady-flow model for the Fox River in Illinois: U.S. Geological Survey Water-Supply Paper 2477, 66 p.

Ishii, A.L., and Wilder, J.E., 1993, Effect of boundary condition selection on unsteady-flow model calibration; *in* Proceedings, XXV Congress of the International Association for Hydraulic Research, Tokyo, Japan: p. 193–200.

Jaeger, C., 1956, Engineering fluid mechanics: London, Blackie and Sons, 529 p. [Translation from German to English by P. O. Wolf]

Jobson, H.E., and Schoellhamer, D., 1987, User's manual for a branched Lagrangian transport model: U.S. Geological Survey Water-Resources Investigations Report 87–4163, 73 p.
Johanson, R.C., Imhoff, J.C., Kittle, J.L., Jr., and Donigian, A.S., Jr., 1984, Hydrological Simulation Program - FORTRAN (HSPF); user's manual for release 9.0: U.S. Environmental Protection Agency, EPA–600/3–84–066, 767 p.
Knapp, H.V., and Ortel, T.W., 1992, Effect of Stratton Dam operation on flood control along the Fox River and Fox Chain of Lakes: Champaign, Ill., Illinois State Water Survey Contract Report 533, 79 p.
Lai, C., 1987, Numerical modeling of unsteady open-channel flow; *in* Yen, B.C., ed., Advances in Hydroscience, v. 14, New York, Academic Press, p. 161–333.
Liggett, J.A., 1975, Basic equations of unsteady flow; *in* Mahmood, K., and Yevjevich, V., eds., Unsteady Flow in Open Channels: Littleton, Colo., Water Resources Publications, p. 29–62.
Linsley, R.K., Kohler, M.A., and Paulhus, J.L.H., 1982, Hydrology for engineers (3rd ed.): New York, McGraw-Hill, 508 p.
Miller, S., and Chaudhry, M.H., 1989, Dam-break flows in curved channel: Journal of Hydraulic Engineering, American Society of Civil Engineers, v. 115, no. 11, p. 1465–1478.
Petryk, S., and Grant, E.U., 1978, Critical flow in rivers with flood plains: Journal of the Hydraulics Division, American Society of Civil Engineers, v. 104, no. HY5, p. 583–594.
Preissmann, A., 1961, Propagation des intumescences dans les canaux et rivieres: First Congress of the French Association for Computation, Grenoble, France.
Schaffranek, R.W., Baltzer, R.A., and Goldberg, D.E., 1981, A Model for Simulation of Flow in Singular and Interconnected Channels: U.S. Geological Survey Techniques of Water-Resources Investigations book 7, chapter C3, 110 p.
Shearman, J.O., 1990, Users manual for WSPRO—A model for Water-Surface PROfile computations: Washington, D.C., U.S. Department of Transportation, Federal Highway Administration, FWHA-IP-89-027, 187 p.
Streeter, V.L., and Wylie, E.B., 1985, Fluid mechanics (8th ed.): New York, McGraw-Hill, 586 p.
Strelkoff, T., 1969, One-dimensional equations of open-channel flow: Journal of the Hydraulics Division, American Society of Civil Engineers, v. 95, no. HY3, p. 861–876.
Snohomish County Public Works, 1989, Snohomish River unsteady flow model (FEQ)—Results of model calibration and reevaluation of the five-year + one foot level of protection: Surface Water Management Division, 52 p.
Turner, M.J., Pulokas, A.P., and Ishii, A.L., 1996, Implementation and verification of a one-dimensional, unsteady-flow model for Spring Brook near Warrenville, Illinois: U.S. Geological Survey Water-Supply Paper 2455, 35 p.
Tynes, K.A., 1989, Hydraulics of side-channel weirs for regional detention basins: University of Texas at Austin, Master's Thesis, 128 p.
Whitaker, S., 1968, Introduction to fluid mechanics: Englewood Cliffs, N.J., Prentice-Hall, 457 p.
Wilson, B.W., 1960, Note on surface wind stress over water at low and high wind speeds: Journal of Geophysical Research, v. 65, no. 10, p. 3377–3382.
Xia, R., and Yen, B.C., 1994, Significance of averaging coefficients in open-channel flow equations: Journal of Hydraulic Engineering, American Society of Civil Engineers, v. 120, no. 2, p. 169–190.
Yen, B.C., 1973, Open-channel flow equations revisited: Journal of the Engineering Mechanics Division, American Society of Civil Engineers, v. 99, no. EM5, p. 979–1009.
———, 1992, Hydraulic resistance in open channels; *in* Yen, B.C. ed., Channel flow resistance—Centennial of Manning's formula: Littleton, Colo., Water Resources Publications, p. 1–135.
Yen, B.C., Wenzel, H.G., Jr., and Yoon, Y.N., 1972, Resistance coefficients for steady spatially varied flow: Journal of the Hydraulics Division, American Society of Civil Engineers, v. 98, no. HY8, p. 1395–1410.
Zienkiewicz, O.C., and Taylor, R.L., 1989, The finite element method, Volume 1: New York, McGraw-Hill, 648 p.

APPENDIXES

APPENDIX 1: LIST OF NOTATION

The following symbols are used in this report.

a	Lower bound of the function region integrated in the weighted four-point scheme of numerical analysis
A	Total cross-sectional area
$A[x,y(x)]$	Total cross-sectional area at the location along the distance axis given by x
A_C	Total cross-sectional area at critical flow
A_O	Exit area of the outlet conduit for a variable-speed pump
A_p	The area of a submerged object projected on a cross section orthogonal to the approach velocity
A_s	Constant surface area for the reservoir assumed to be present at a dummy branch
A_{si}	Effective area of a storm-sewer inlet
A_x^y	Derivative of the cross-sectional area with respect to distance x if water-surface height y is held constant
ΔA	A small incremental area in the channel cross section
dA	The differential area taken as a vector normal to the control surface of the control volume
b	Upper bound of the function region integrated in the weighted four-point scheme of numerical analysis
$\boldsymbol{b}$	Vector of residuals in the solution of a system of nonlinear equations with Newton's method
c	Wave celerity
ca_i	Coefficient for the power equation used to interpolate tabulated critical flows in tabulation interval i
cb_i	Power for the power equation used to interpolate tabulated critical flows in tabulation interval i
$C+$	Upstream wave trajectory
$C-$	Downstream wave trajectory
C_D	Dimensionless drag coefficient representing the drag resulting from a submerged object or obstruction in an open channel
$C_D^{(w)}$	Dimensionless drag coefficient for wind-shear stress
$C_{du}(p_G)$	The weir coefficient when flow is from the downstream node to the upstream node at a variable-height weir
$C_{SW}(l)$	Side-weir coefficient
$C_{ud}(p_G)$	The weir coefficient when flow is from the upstream node to the downstream node at a variable-height weir
CL	Level of discharge or water-surface elevation monitored at a control point for gate operation
$\dot{C}L$	Rate of movement of water-surface elevation or discharge (depending on which is monitored) at the control point
CL_{cp}	Level of discharge or water-surface elevation monitored at a control point for pump operation
CL_L	The lower limit of the null zone for simulation of gate or pump operation
$\dot{C}L_{min}$	The minimum rate of movement of water-surface elevation or discharge (depending on which is monitored) at the control point toward the null zone for which the gate setting remains the same
CL_U	The upper limit of the null zone for simulation of gate or pump operation

CS	Control surface of the control volume
CV	Control volume in open-channel flow
d	The piezometric head of the tail water on the crest of a weir
d_a	The actual head drop across a hydraulic structure represented by a table of type 13
d_f	The free drop at headwater head, H_h, for a hydraulic structure represented by a table of type 13
d_L	Nominal upstream piezometric head on a variable-height weir
d_R	Nominal downstream piezometric head on a variable-height weir
D_C	The control structure sign
D_D	Direction of flow specified by the user
D_F	The sign the flow at the discharge node of a control structure must have
D_p	Pumping direction
D_q	The discharge node sign
D_S	The system sign for a two-node control structure
D_T	The transition sign for a flow expansion
$E(Q,Z_w)$	Elevation of the total energy line for water-surface elevation, Z_w, and flow, Q
$E_s(Q,y)$	Specific energy for flow rate, Q, and water-surface height, y.
$f_C(z_w)$	Function specifying critical flow at the upstream node of an abrupt expansion with a diversion channel
$f_{cp}(.)$	Control function for a pump
$f_f(Q_q)$	The head losses resulting from separation at the entrance to the pump intake, flow resistance in the intake conduit, and flow resistance in the outlet conduit
$f_{gc}(p_G)$	Gate-crest-elevation function for a variable-height weir
$f_i(.)$	Function specifying flow through structure i that conveys flow between the upstream and downstream nodes at a special feature in the stream system
$f_I\left(\bar{h}, h_m\right)$	Function defining inflow to a channel over a side weir
$f_K(z_w)$	The conveyance function for the flow path between two nodes in the simple-conveyance option
$f_m(z_w)$	Maximum-capacity function for a variable geometry, one-node control structure
$f_q\left(z_{w_h}\right)$	Flow function for a one-node control structure
$f_{qb}(t)$	Boundary condition of flow as a function of time
$f_{qs}(h_L,h_R,w_g)$	Function defining the flow through the sluice-gate opening
$f_Q(\Delta h_1)$	The flow rate delivered by a variable-speed pump operating at the standard operating speed against a head difference of Δh_1
$f_{Qdu}(h_w)$	Weir-flow function for flow from downstream node to upstream node
$f_{Qud}(h_w)$	Weir-flow function for flow from upstream node to downstream node
$f_s(r_h)$	Submergence factor for flow over a variable-height weir
$f_{Sdu}(.)$	Function denoting the table of weir-flow submergence factors for flow from downstream node to upstream node

$f_{Sud}(.)$ Function denoting the table of weir-flow submergence factors for flow from upstream node to downstream node

$f_{tw}(t)$ The tail-water condition for a simple pump at any time, t

$f_{twc}(x_{tc})$ Tail-water conversion function for a simple pump, where x_{tc} equals the tail-water condition obtained from $f_{tw}(t)$

$f_T(y)$ Tabulated function value for argument value y

$\hat{f}_T(y)$ Interpolated function value for argument value y

$f_T'(y)$ Tabulated derivative of tabulated function value with respect to argument y

$\hat{f}_T'(y)$ Interpolated derivative of tabulated function value with respect to argument y

$f_w\left(\bar{h}, h_M\right)$ Function defining the flow over a weir

$f_z(t)$ Boundary condition of water-surface elevation as a function of time

$f_{14}(.)$ A function that yields the head at the upstream node given the piezometric head at the downstream node and the flow at the flow node

$\tilde{F}$ The drag rescaled by the upstream hydrostatic-pressure force at a submerged object or obstruction in the channel

$\boldsymbol{F}_a$ Froude number for the flow approaching a submerged object or obstruction in the channel

$\boldsymbol{F}_W$ Froude number of the channel flow relative to the head on the side weir

F_D Drag from a submerged object or obstruction determined from measured drag coefficients

F_D' Term included in the equation of motion representing drag from a submerged object or obstruction determined from measured drag coefficients

F_{DE} Drag from a submerged object or obstruction approximated from an equivalent energy slope determined from head-loss relations

F_{DEC} The equivalent drag combining drag resulting from submerged objects of obstructions and eddy losses resulting from channel expansions or contractions

F_p Hydrostatic pressure force on the cross section

F_x Forces acting on a control volume in the x direction

$F(.)$ The residual function

$\boldsymbol{F(u)}$ Vector of values of the residual function for the equations of motion describing the stream system

$F_{ij}'(.)$ Partial derivative of the residual function of equation of motion i with respect to variable j

$F_N(u)$ Nonlinear function solved with Newton's method

$F_N'(u)$ Derivative with respect to u of nonlinear function solved with Newton's method

$\hat{F}_N(\hat{u}_o)$ The value on the line tangent to F_N, the point of tangency being $\hat{u}_0$

g Acceleration of gravity

G First point on the x-t plane affected by disturbances at X_L and X_R at time t_0; lower boundary in time of the region of influence for X_L and X_R

$\bar{h}$ The average piezometric head in the source channel for a side weir

$h(s)$ Local flow depth at offset s in a cross section

h_{L_i} Upstream piezometric head relative to a head reference point for structure i at a special feature in the stream system represented with an explicit two-dimensional table

h_M	Head at the middle node in the representation of flow over a side weir
h_{R_i}	Downstream piezometric head relative to a head reference point for structure *i* at a special feature in the stream system represented with an explicit two-dimensional table
$h_{SW}(l)$	Head on the side weir
h_w	The piezometric head on a weir
h_0	Base stage for the numerical experiments of Xia and Yen (1994) of the effect of β on the momentum equation
Δh	The head difference between water ponded at the entrance to a storm sewer and the pressure head in the storm sewer
Δh^*	A small value of the head difference used to linearize the computation of the maximum current rate of inflow to a storm sewer to avoid computational failure when $\Delta h < \Delta h^*$
Δh_{ad}	Head loss for accelerating or decelerating flow in channel contractions or expansions, respectively
Δh_p	Head loss in a control volume in open-channel flow resulting from a submerged object or obstruction
$H_{SW}(l)$	Height of the side-weir crest above the channel bottom
H_h	Headwater head for a hydraulic structure represented by a table of type 13
H_t	Tail-water head for a hydraulic structure represented by a table of types 13 and 14
$I(t)$	Inflow of water that enters a control volume over or through the sides of the channel
$J[x,y(x,t)]$	First moment of area with respect to the water surface at location *x* for water-surface height *y* at time, *t*
$\boldsymbol{J}(u)$	Matrix of partial derivatives (Jacobian matrix) of the residual functions for the equations describing the stream system
J_x^y	Derivative of the first moment of area with respect to the water surface with distance *x* if water-surface height *y* is held constant
$k(s)$	The conveyance per unit width in a cross section at offset *s*
k, k'	Correction coefficient for a nonhydrostatic pressure distribution in the nearly exact momentum equations
k_a	Head-loss coefficient for accelerating flow in a channel contraction
k_d	Head-loss coefficient for decelerating flow in a channel expansion
k_p	Head-loss coefficient for the submerged object or obstruction at location x_p
K	Total channel conveyance
K_E	Head-loss coefficient for flow through an expansion corrected for flow direction
K_F	Friction-loss coefficient applied at a dummy branch
K_M	Mean conveyance in a computational element
K_o	Exit-loss coefficient for a variable-speed pump
K_{vh}	A correction factor applied to the approach-channel velocity head to account for the increase in energy head because of converging flow in the immediate vicinity of a variable-height weir
K_+	The head-loss coefficient when the flow is from the upstream node to the downstream node in a flow expansion

K_-	The head-loss coefficient when the flow is from the downstream node to the upstream node in a flow expansion
l	Distance measured along a side weir
L	Length of the side weir
$\boldsymbol{L}$	Lower triangular matrix of the Jacobian matrix, $\boldsymbol{J}$
L_G	Length of an overflow gate
m	Bandwidth of a Jacobian matrix
m_c	The number of control structures conveying water between the upstream and downstream nodes at a special feature represented with an explicit two-dimensional table
m_D	The number of submerged objects or obstructions in a computational element for which drag may be estimated from measured drag coefficients
m_p	The number of submerged objects or obstructions in a computational element for which drag must be estimated from an equivalent energy slope determined from a head-loss relation
$M(Q,y)$	Specific force for flow rate, Q, and water-surface height, y
$M_A(x,y_0)$	The weight coefficient that will result in a valid volume per unit length when multiplied with the cross-sectional area at location x for water-surface height y_0
M_f	The weight coefficient to correct for the effects of channel curvilinearity on the friction term in the momentum equation
M_F	Momentum flux through the cross section.
M_L	Rate of change of the fraction of maximum discharge capacity for a simulated gate with the monitored level at the control point below the null zone
$M_Q(x,y_0)$	The weight coefficient that will result in a valid momentum content per unit length when multiplied with the total flow rate through the cross section at location x for water-surface height y_0
M_T	The weight coefficient that will result in a valid water-surface area per unit length when multiplied by the top width
M_U	Rate of change of the fraction of maximum discharge capacity for a simulated gate with the monitored level at the control point above the null zone
n	Manning's roughness coefficient
n_e	Number of nonlinear equations to be solved simultaneously with Newton's method
n_j	Number of flow-path end nodes at a junction
n_r	Relative operating speed of a variable-speed pump
n_s	Pump operation speed
$q(x,t)$	Lateral inflow per unit length along the channel
$q_{SW}(l)$	Rate of outflow per unit length of side weir
$q_w(s)$	The flow per unit width in a cross section at offset s
$Q(x,t)$	The total flow rate through the cross section at location x at time, t
$Q(x,y_0)$	The total flow rate through the cross section at location x for water-surface height, y_0
Q_C	Critical flow rate determined for steady flow in a compact channel $(\alpha \approx \beta \approx 1)$
Q_{cs}	The flow through a variable-geometry, one-node control structure

Q_E	Critical flow rate determined from minimization of specific energy in a channel where $\alpha \neq 1$
Q_{EXi}	The flow at the ith flow-path end node at a junction
Q_f	Free-flow limit for a given tail-water head for a hydraulic structure represented by a table of type 14
Q_M	Critical flow rate determined from minimization of specific force in a channel where $\beta \neq 1$
Q_{MAX}	The maximum current rate of inflow to a storm sewer
Q_P	Pumping rate
Q_q	Discharge at the discharge node for a control structure
Q_{qb}	Discharge at an external boundary of the stream system
Q_{SW}	Flow over a side weir
Q_{SWM}	Flow over a side weir at the midpoint of a computational interval of the side weir length
Q_w	Flow over a weir
$p(_t)$	The opening-fraction function for a variable-geometry control structure
$\dot{p}$	The rate of change of the gate-opening fraction
p_D	Partial free drop for a hydraulic structure represented by a table of type 13
p_F	Partial free flow for a hydraulic structure represented by a table of type 14
$\dot{p}_{max}$	Absolute value of the maximum rate of change in the gate-opening fraction
$p_G(t)$	Gate-position fraction for a variable-height weir
p_I	The proportion of the function-table interval represented by the point of interpolation
$P[x,y(x,t)]$	The wetted perimeter of the channel at location x for water-surface height y at time, t
P_{GF}	The numerical approximation to the integral of the pressure, gravity, and friction terms in the momentum equation
PR_H	Priority assigned for gate-setting changes when the level at the control point is above the null zone
PR_L	Priority assigned for gate-setting changes when the level at the control point is below the null zone
PR_N	Priority assigned for gate-setting changes when the level at the control point is in the null zone
r	The ratio of downstream water-surface height to upstream water-surface height at a submerged object or obstruction in the channel
r_h	The ratio upstream piezometric head to downstream piezometric head for a variable-height weir
$R(x,y)$	Hydraulic radius for location x and water-surface height y
s	Offset distance across a cross section
s_B	Offset at the beginning of the wetted top width for the cross section
s_E	Offset at the end of the wetted top width for the cross section
S	The volume of water between adjacent cross sections in the stream channel
$S_h(x_1,x_2)$	The correct volume of water between cross sections at locations x_1 and x_2 for a given water-surface height
$S_q(x_1,x_2)$	The correct momentum content of the flow between cross sections at locations x_1 and x_2 for a given water-surface height
S_R	Storage volume of a level-pool reservoir

S_0	Bottom slope of the channel, positive when the bottom elevation decreases in the downstream direction
S_{ad}	Equivalent slope corresponding to the eddy loss distributed over the length of the control volume for accelerating or decelerating flow in channel contractions or expansions, respectively
S_f	Friction (momentum) slope for flow in a channel
S_p	Slope introduced to represent the loss of head (mechanical energy) in a control volume in open-channel flow resulting from a submerged object or obstruction
S_w	Water-surface slope between two nodes applied in the simple-conveyance option, positive when the water-surface elevation decreases in the downstream direction
$ST(z)$	Storage capacity of a level-pool reservoir at water-surface elevation z
t	Time
t_D	The time point that is down relative to the center of the rectangular box in the x-t plane used to define the four-point numerical solution scheme
t_U	The time point that is up relative to the center of the rectangular box in the x-t plane used to define the four-point numerical solution scheme
t_0	Initial time for unsteady-flow computations
δt_s	Time elapsed since the pump speed was changed
Δt	Time step in FEQ computations
T	Top width of the channel
T_C	Top width at critical flow
T_x^y	Derivative of the top width with respect to distance x if water-surface height y is held constant
u	Independent variable for a nonlinear function solved with Newton's method
$\boldsymbol{u}$	Vector of unknown discharge and water-surface elevation values for the stream system
$\hat{u}_0$	Point near a root in the solution of a nonlinear equation solved with Newton's method
$\hat{u}_{i+1}$	The root of the tangent line, which becomes the next approximation to a root of the nonlinear equation solved with Newton's method
U	Wind velocity
$\boldsymbol{U}$	Upper triangular matrix of the Jacobian matrix, $\boldsymbol{J}$
v	Local velocity for a small incremental area, ΔA, in a cross section
v_x	Local velocity in the x direction
V	Cross-sectional average velocity
$\boldsymbol{V}$	The velocity vector
V_a	Cross-sectional average velocity in the approach reach upstream from a submerged object
δv	The deviation of the point velocity from the average velocity
$d\forall$	The volume differential
w	Vector product of the upper triangular matrix of the Jacobian matrix, $\boldsymbol{J}$, and the vector of unknown discharges and water-surface elevations for the stream system, $\boldsymbol{u}$
w_g	Sluice-gate opening distance
W	Weight applied in the weighted four-point scheme of numerical analysis

W_A	Weight applied to integrals of area with respect to distance in the weighted four-point scheme of numerical analysis
W_E	Weight applied to average water-surface elevation if an unmodeled side channel joins the channel
W_{SM}	Time-averaged wind-stress term in the numerical approximation of the momentum equation
W_T	Weight applied to integrals with respect to time in the weighted four-point scheme of numerical analysis
W_X	Weight applied to integrals of conveyance with respect to distance in the weighted four-point scheme of numerical analysis
x	Distance along the distance axis for the channel
x_D	Location of a drag-producing submerged object or obstruction in an open channel
x_L	Location of upstream face of a control volume in open-channel flow
x_R	Location of downstream face of a control volume in open-channel flow
x_p	Location in an open channel of a drag-producing submerged object or obstruction for which drag is determined from an equivalent energy slope
Δx	Distance between cross sections in FEQ computations; also, length of computational element in steady- and unsteady-flow analysis
Δx_c	Distance between the nodes in the simple-conveyance option
Δx_{CV}	Length of a control volume in open-channel flow
Δx_s	Small increment along the distance axis of the channel, applied to determine the weight coefficients that correct for the effects of channel curvilinearity
X_L	Upstream point along a channel disturbed at time t_0
X_R	Downstream point along a channel disturbed at time t_0
$y(x)$	The height of the water surface above the minimum point in the cross section at the location given by x
y_G	The height of the ground surface above the storm-sewer inlet
y_M	Mean value for the water-surface height in the control volume for a computational time step
y_R	Maximum depth of water in a level-pool reservoir
y_0	Constant water-surface height applied to determine the weight coefficients that correct for the effects of channel curvilinearity
Δy	The tabular interval of a function table
z	Height above the thalweg
$\bar{z}$	The average water-surface elevation in the source channel for a side weir
z_{hSW}	The elevation of the datum for defining heads for flow over a side weir
z_{h_i}	Head-reference point for special feature i represented with an explicit two-dimensional table
z_{lhs}	The total energy head at the upstream node of a flow expansion
z_m	The minimum elevation of a weir
z_{rhs}	The total energy head at the downstream node of a flow expansion
z_w	Water-surface elevation
$\bar{z}_w$	Average water-surface elevation between two nodes in the simple-conveyance option

z_{w_b}	Water-surface elevation at a boundary node
z_{w_c}	Water-surface elevation at a critical control
z_{w_h}	Water-surface elevation at the head node for a control structure
Δz	Incremental height
Δz_w	The change in water-surface elevation across a computational element
α	Kinetic-energy-flux correction coefficient
β	Momentum-flux correction coefficient
$\hat{\gamma}(\lambda)$	Series of solutions used in empirical tests of method convergence
κ	A constant of proportionality applied in the Richardson extrapolation
λ	A measure of the size of the time and distance steps in empirical tests of method convergence
θ	The angle between the direction of the water velocity and the direction of a line normal to the small incremental area, ΔA
ρ	Density of water
ρ_a	Density of air
$\sigma(s)$	The rate of change of distance along the flow line at offset s to the rate of change of distance along the main-channel axis (sinuosity at offset s)
$\bar{\tau}$	The average shear stress exerted on the water by the channel boundary
ϕ	The angle between the direction of the velocity and the direction of the distance axis at the location of the cross section
ψ	Angle between the downstream flow direction in the channel and the velocity of the wind
ω	A correction factor applied to the normal weir coefficient to approximate the discharge coefficient for each location along a side weir
$\bar{\omega}$	A correction factor applied to the normal weir coefficient to approximate the discharge coefficient for each location along a side weir computed at the average values at the upstream and downstream nodes

APPENDIX 2: UNFORMATTED DATA-FILE STRUCTURES: FULL EQUATIONS MODEL VERSION 8.0

Various files are used in FEQ simulation for input and output of data. Many of these files are formatted in an easily understood manner or as described in the Input Description for the Full EQuations Model (section 13). However, some of the input files are unformatted. The data in unformatted files are stored in the internal binary form that is directly accessible only by a computer. Thus, the file structure cannot be read and easily understood by the user. These files are used because computer processing is more efficient and storage space is less than that for formatted files. Thus, unformatted files are the best means for recording and retrieving large amounts of data in this application.

Two classes of unformatted data files are supported in FEQ. The diffuse time-series file (DTSF) is used to store the unit-area runoff intensities resulting from precipitation measured at one or more rain gages, each representing one or more land uses. These values represent the runoff from the area tributary to the branches in an unsteady-flow model. A single file is used to store all the values for a model. In contrast, more than one point time-series file (PTSF) may be used in an unsteady-flow model. These files, called point because they represent the flow or water-surface elevation at a point in the model of the stream system, record or provide the flow or water-surface elevation from or to an unsteady-flow model. They are used to connect submodels in a watershed and are called connection files.

Diffuse Time-Series File

The structure of the DTSF is designed to enable FEQ to simulate many disjoint runoff events stored in a single file. In most applications, the major runoff events in each water year are selected and the unit-area runoff intensities for each of these events are computed with a rainfall-runoff model. Furthermore, more than one rain gage and (or) land use in the watershed normally is simulated. Therefore, the structure of the DTSF must allow for the storage of multiple disjoint events, each containing runoff time series for each of one or more rain gages or land uses.

In the structure of the DTSF, the following assumptions are made:

1. The runoff intensities are average values over a time step.
2. The time step at any given time is the same for all rain gages and land uses simulated in the watershed.
3. The number of rain gages and land uses is a constant across runoff events.

The DTSF must always contain at least two time series, each containing the sequence of unit-area runoff intensities for one land use and one rain gage. If only one time series is needed, the second time series must be stored with values of zero runoff. The runoff intensities are in terms of velocity so that multiplication by the tributary area will yield a flow rate. For inch-pound units, the velocity is in feet per second; and for metric units, in meters per second.

File Header Records

A DTSF contains three file-header records that describe various aspects of the file. Each of these records is of the same fixed length as all subsequent records in the file. The record length is $(7 + N_{ts})$ four-byte words or $(28 + 4N_{ts})$ characters, where N_{ts} is the number of time series being stored. Let $N(i)$ denote the ith four-byte word in a record of the DTSF. The first record in the file contains

$N(1)$, the year in which the DTSF was prepared,
$N(2)$, the month in which the DTSF was prepared,
$N(3)$, the day in which the DTSF was prepared,
$N(4)$, the time of day (as an integer) when the DTSF was prepared,
$N(5)$, the number of header records in the file $(= N_{ts} + 3)$,
$N(6)$, the number of time series $(= N_{ts})$, and
$N(7)$, the flag for DTSF versions (must always be 1).

The remaining words of the first record should all be set to zero. Items $N(1)$ through $N(4)$ are designed to help the user keep track of all DTSF's used in modeling. The second file-header record contains the following data for the record from which the events were extracted:

$N(1)$, the starting year,
$N(2)$, the starting month,
$N(3)$, the starting day,
$N(4)$, the ending year,
$N(5)$, the ending month, and
$N(6)$, the ending day.

The remaining words of the second record should be set to zero.

The third file-header record contains any alphanumeric description of the DTSF. It must have $(28 + 4N_{ts})$ characters with the space (blank) being counted as a character. This record is written to the output file to document the DTSF used in a model.

Time-Series Descriptions

The three file-header records are followed by N_{ts} records, each with an alphanumeric description of the time series. Each of these records must have a length of $(28 + 4N_{ts})$ characters to match the length of the file-header records. The time-series descriptions are written to the output file to document the time series used in the unsteady-flow model.

Data Records for a Runoff Event

The data records for each runoff event follow after the $(3 + N_{ts})$ header record. An event is defined as a consecutive sequence of time intervals extracted from a longer time series. The beginning time and ending time of the event must be specified before simulation of the flows resulting from the runoff intensities begins. All events in the DTSF must be in chronological order. Each data record contains the following data:

JTIME is a double-precision (two four-byte words) floating-point value giving the modified Julian time for the end of the time interval defining the average runoff values being stored. This is the elapsed time in days from the start of November 17, 1858. The modified Julian time at the start of January 1, 1993, is 48988.0. The modified Julian time at 6 a.m. on the same day is 48988.25.

DYFRAC is a double-precision (two four-byte words) floating-point value giving the fraction of the day for the date given in JTIME.

YEAR is an integer (one four-byte word) giving the year at the end of the time interval.

MONTH is an integer (one four-byte word) giving the month at the end of the time interval.

DAY is an integer (one four-byte word) giving the day at the end of the time interval.

RUNOFF $(1 : N_{ts})$ is a single-precision (one four-byte word) floating-point runoff values for each of the time series for the time interval ending at JTIME. The order of the time series is the same as the order of the tributary areas in the Tributary Area Block (section 13.3) in FEQ. A logical order is to give the land uses for each gage in a standard sequence. It is recommended that the time-series description records be in the same order.

The first record of each event is different in terms of the time specification. All other records in the event give the time at the end of the time interval defining the average runoff. The modified Julian time for the first record is the time at the start of the first time interval of the event. All but the first two runoff values in the first record are set to zero. The first two runoff values are used to store the modified Julian time of the end of the event. This is done by use of the EQUIVALENCE statement in Fortran to overlay a double-precision floating-point value onto two single-precision floating-point values. (This is the source of the requirement that there be at least two time series in the DTSF.) This ending time is extracted in simulation, and the two runoff values are set to zero to match the other values when the first record of each event is encountered. The end of the DTSF is marked by writing a data record in which all the fields are set to zero.

The structure of the DTSF is specific to computer systems with commensurable sizes for integers, single-precision floating-point numbers, and double-precision floating-point numbers. Most computer systems developed in the United States follow the pattern assumed by the DTSF. However, some systems have a different word size for integers and single-precision floating-point numbers. Furthermore, some systems have a double-precision floating-point value that does not take twice the space of a single-precision value. When the storage sizes are not commensurable as assumed above, a new version of the DTSF will be required.

Point Time-Series File

A point time-series file (PTSF) has a much simpler structure than a DTSF. It includes a single header record with an integer code (one four-byte word) describing the contents. If CODE = 2, then the values stored are water-surface elevations, and if CODE = 4, the values stored are flow rates. The subsequent records in the file contain the modified Julian time and the flow or water-surface elevation at that time. The first record contains four bytes of information and all subsequent records contain 12: eight bytes for the modified Julian time and four bytes for the flow rate or elevation. Linear interpolation is applied between consecutive values in a PTSF. No restriction is placed on the time interval between values. The user must ensure that linear interpolation is reasonable.

Warning—Unformatted Files are System Specific

Unformatted input and output statements used in the DTSF and PTSF are supported by all standard Fortran languages. These files are computer-system specific. They can sometimes be compiler specific, but most compiler makers include flags to create a standard unformatted file. Computer systems differ when processor chips are from different manufacturers. The files may be compatible, but compatability must always be checked. For example, an unformatted file from an IBM mainframe computer cannot be transferred to an IBM personal computer or compatible. Files created on a specific workstation may not be compatible with a workstation from another vendor and will not be compatible with any computer with INTEL microprocessors.

APPENDIX 3: DIMENSIONS OF ARRAYS AND SPECIFICATION OF FORTRAN UNIT NUMBERS IN THE FULL EQUATIONS MODEL

Proper application of FEQ for simulation of unsteady flow in a network of open channels requires the use of many arrays and the specification of Fortran unit numbers for input files, output files, and other operations. Many of the arrays have identical dimensions that have increased throughout the years as Fortran compilers and operating systems have become more efficient. To facilitate changes in the size of the arrays, the dimensions of the arrays are specified in an INCLUDE file named ARSIZE.PRM, which is used when the program is compiled. The contents of ARSIZE.PRM are described below. Typically, the sizes of these arrays will not need to be changed from the values in ARSIZE.PRM that may be obtained by electronic retrieval from the World Wide Web (WWW) at http://water.usgs.gov/software/feq.html and by anonymous File Transfer Protocol (FTP) from water.usgs.gov in the pub/software/surface_water/feq directory. However, for very large stream systems, some of the dimensions may need to be changed as described below. Because FEQ has been developed over a period of more than 20 years, many aspects of the code use Fortran unit numbers to identify input files, output files, and other program features. Selection and application of these unit numbers also are discussed below.

Dimensions of Arrays in the Full Equations Model

A stream system larger than that permitted by the size declared for a variety of vectors and arrays cannot be simulated in FEQ. (A vector is an array with only a single dimension.) The sizes of these arrays and vectors have been declared by use of Fortran parameters. A file, ARSIZE.PRM, contains these dimension parameters in a PARAMETER statement. An INCLUDE compiler directive appears with ARSIZE.PRM for the arguments in any program unit in which one or more of the parameters in ARSIZE.PRM is required to establish the dimension of some array or vector.

Many of the parameters giving the dimensions begin with the letter “M” and may be followed with the letter “R” to denote the number of rows for a vector or an array. All vectors are viewed as being column vectors. Therefore, each row for a vector contains only one element. The second letter may be “C” to denote the number of columns for arrays; that is, the number of elements in each row of the array. The remaining four characters available in a Fortran name are the same as or are related to the variable name being dimensioned. Some parameters that are used for many different vectors or arrays do not follow these guidelines. For example, MNBN gives the maximum number of branch nodes and is used to dimension vectors that contain information relevant to nodes on a branch.

The following list gives a brief description of each parameter.

CD5TY1 Used in storing and processing code 5, type 1 entries in the Network Matrix Control Block (section 13.6).

CD5TY2 Used in storing and processing code 5, type 2 entries in the Network Matrix Control Block (section 13.6).

CD5TY3 Used in storing and processing code 5, type 3 entries in the Network Matrix Control Block (section 13.6).

CD5TY4 Used in storing and processing code 5, type 4 entries in the Network Matrix Control Block (section 13.6).

CD5TY5 Used in storing and processing code 5, type 5 entries in the Network Matrix Control Block (section 13.6).

CD5TY6 Used in storing and processing code 5, type 6 entries in the Network Matrix Control Block (section 13.6).

CD5TY7 Used in storing and processing code 5, type 7 entries in the Network Matrix Control Block (section 13.6).

CD5TY8 Used in storing and processing code 5, type 8 entries in the Network Matrix Control Block (section 13.6).

CD5TY9 Used in storing and processing code 5, type 9 entries in the Network Matrix Control Block (section 13.6).

MCPFPT Maximum number of columns in point-flow-pointer vector PFPNT.

MCTD10 Maximum number of columns in a table of type 10. Set to 1 because tables of type 10 are not currently used in FEQ; however, the input of these tables is supported in FEQUTL, and the code that processes these tables is included in FEQ. Therefore, a value for MCTD10 is required even though tables of type 10 are not currently used in FEQ.

MEXTRA Number of branch nodes in excess of the nodes on each end of a branch.

MFTNUM Maximum function table number. Gives length of function-table pointer vector FTPNT. One value in this vector is used by every function table in the model of the stream system. The value of MFTNUM is the maximum value for a function table number. For example, if MFTNUM is 10,000, then no table number can be larger than 10,000.

MLPDA Maximum length of the partial derivative array PDAVEC. This array is stored as a vector with rows and columns accessed with pointer vectors C and R.

MNBLK Maximum number of Operation-Control Blocks (section 13.12).

MNBN Maximum number of branch nodes.

MNBRA Maximum number of branches.

MNCD6 Maximum number of Code=6 forced boundaries in the stream system.

MNDBUF Maximum number of diffuse flow buffers. Should be in the range of 20 to 30.

MNDEP Maximum number of depth values in an interpolated cross-section table.

MNDIFA Maximum number of diffuse areas. This is the sum across all rain gages of the land uses in each rain-gage area.

MNETRY Maximum number of entries in the Network-Matrix Control Block (section 13.6). Each CODE value in the input defines an entry.

MNEX Maximum number of flow-path end nodes.

MNFIN Maximum number of input files allowed for Code=6.

MNFOUT Maximum number of output files allowed. This refers to the output of point time-series files.

MNFRDT Maximum number of frozen time steps.

MNFREE Maximum number of flow-path end nodes not on branches. Total of level-pool reservoir and dummy-branch end nodes.

MNGATE Maximum number of gates for output of special values to the special-output file.

MNITER Maximum number of possible iterations for system solution.

MNMID Maximum number of contiguous interpolated cross sections in a branch.

MNSOUT Maximum number of special outputs in the special-output file.

MORG Offset parameter for command-line processing. Should be 0 for UNIX systems and 1 if a Lahey compiler is used.

MREMC Maximum number of rows in the Network-Matrix Control vector. This vector is used to store the information in the Network-Matrix Control Block (section 13.6) in an internal coded form. Each item stored takes a row in the vector. Thus, this vector should have several thousand rows.

MRFTAB Maximum number of rows in the function-table vector pair (FTAB, ITAB). Each number in a function table takes a row in this vector. Values for this parameter are usually 250,000 or larger. Large models may require a value of 1 million or more elements.

MRMAT Maximum number of rows in the array of partial derivatives. The number of rows in this array must be twice the number of distinct nodes in the system. MNBN is the number of nodes on branches, and MNFREE is the number of nodes on dummy branches and level-pool reservoirs. Thus, MRMAT is twice the sum of MNFREE and MNBN.

MRMBLK Maximum number of rows in the matrix-block control system. One row is used for each matrix block in the array of partial derivatives.

MRPFPT Maximum number of rows in the point-flow-pointer vector PFPNT. Each point inflow takes one row. These are point flows into an element of a branch. In general, such point inflows should not be used. The use of a dummy branch or level-pool reservoir to represent point inflows is the preferred alternative.

MRRBUF	Maximum number of rows in the read buffer for input files. This space is shared by all input files attached to Code=6 in the Network-Matrix Control Block (section 13.6). Thus, it should be large enough so that each file obtains 100 or more elements.
MSPKNT	Default maximum number of lines per page in the special-output file.
MUNIT	Maximum unit number for input or output. Selection of input and output unit values is described in detail below.
MXGAGE	Maximum number of rain gages.
MXGLU	Maximum number of land-use/rain-gage values in the system. Same value as MNDIFA.
OFF234	Offset of the first argument in a function table of type 2, 3, or 4 from the address of the table.
XTIOFF	Offset of the first argument in cross-section function table from the address of the table.

Setting the System Size

The key parameter values in setting the system size are the maximum number of branches, MNBRA, the maximum number of free nodes, MNFREE, and the maximum number of extra nodes on branches, MEXTRA. The other parameter value that may need to be changed is the maximum number of rows in the function table storage, MRFTAB. Other parameter values can be changed, but the default values are generally large enough to represent most stream systems. If many input or output files are going to be used, then MNFIN, MNFOUT, and MRRBUF should be increased.

Input-Output Unit Number Selection

FEQ has been operational long enough that some components predate recent developments in operating systems. In the past, some versions of Fortran and some operating systems required that an input or output operation in a computer program must be associated with a file on a hard disk using a unit number. Thus, the user had to specify an explicit unit number in order to associate the file on the hard disk with the proper point of input or output in the Fortran program. In addition, it was not possible to reference a file name as such in Fortran programming in the past. Therefore, the intermediary of a unit number had to be applied.

As a consequence, a considerable history of input streams was developed for FEQ that required explicit unit numbers. These numbers continue to be supported in FEQ to maintain consistency with past use even though the use of unit numbers is no longer required in the current Fortran compilers and operating systems. As time permits and demand dictates, the old usage of unit numbers will be replaced with new usage that applies the more flexible facilities available on current operating systems. In the interim, the following guidelines for choosing unit numbers are offered.

1. Each operating system/language combination has a unique upper limit for unit numbers. In some cases, this is 99, but it may be larger or smaller than this value. The purpose of MUNIT in the INCLUDE file ARSIZE.PRM is to set this level so that the input can be checked in FEQ computations to make sure that the specified unit numbers are not too large for the operating system.
2. Fortran programming has defined standard unit numbers since the early beginnings, in the 1960's. Unit 5 was defined as a standard input, unit 6 as standard output to a line printer, and unit 7 was defined as output to a standard card punch. Card punches are not used anymore, and line printers are rarely used at this time. However, unit numbers 6 and 7 may have special significance for a given compiler of the Fortran programming language. Thus, these numbers should not be used for anything other than their original intent. Compilation and operation system difficulties may result if these unit numbers are applied in a nonstandard manner.
3. Command-line arguments are used in FEQ. A command line is the line following the DOS prompt on a DOS machine and the UNIX prompt on a UNIX machine. This is the line where words and strings of symbols are typed to tell the computer what actions should be done (the line where commands are issued to the computer).

The first item included on a command line is the command. Items that follow the command, usually separated from the command and from each other by one or more spaces, are called command-line arguments. Two arguments are required with each a file name in FEQ simulation. The first argument is the name of the file that contains the input to FEQ as defined in the Input Description for the Full Equations model (section 13). This is the standard input file. The second argument is the name of the file that contains the items that are computed in the course of a simulation. This is the standard output file. An example command line for FEQ is FEQ FEQ.IN FEQ.OUT, where FEQ.IN is the name of the standard input file and FEQ.OUT is the name of the standard output file. The names for the standard input and output file are selected by the user and can be any valid name available in the operating system. As many as 64 characters may be used for the specification of the file name in FEQ. Therefore, a path can be included with the name.

4. Two standard unit numbers that relate the standard input and standard output files for FEQ are defined in a file STDUN.PRM, which may be retrieved electronically as described in section 1.1. These are the unit numbers that are equivalent in concept to unit numbers 5 and 6 in traditional Fortran. The Fortran parameter STD5 defines the unit number to use for processing the standard input file. The Fortran parameter STD6 defines the number to use for processing the standard output file. These parameters are 5 and 6, respectively, by default for DOS systems, but they need not have those values. Any valid values that the user selects may be used. Once selected, however, the unit numbers cannot be used for another purpose in the input to FEQ.
5. The Fortran-defined unit designation for writing to the display screen is used in FEQ. The unit field in an output statement in Fortran is given as an asterisk to denote output to the display screen. This means that an output to the standard output unit, STD6, will appear in the standard output file, whereas an output to a unit designated by an asterisk is to the display screen. This is the result on DOS systems. Some UNIX systems, however, define the asterisk unit differently. On these UNIX systems, the asterisk unit is associated with the standard output unit, 6. If this also is the unit used for the standard output file, then no text is written to the display screen and all text appears in the standard output file. Thus, to write to the display screen when running under UNIX, STD6 should not be given the number 6. Also, the standard unit number 5 is often associated with the keyboard in some UNIX systems. Therefore, STD5 and STD6 should not have the values of 5 and 6 under UNIX, but should be given some other numbers before the program is compiled. Values of 3 and 4 have been used in the past and have worked. On UNIX systems, unit numbers 5, 6, and 7 should be avoided because they may have a special meaning.
6. Aside from the standard unit numbers specified for STD5 and STD6 and the traditional standard unit numbers of 5, 6, and 7, the user may select the unit numbers. This choice is limited only by the following constraints:
   (A) the same unit number should not be used more than once in any context other than the input of files containing function tables, and
   (B) a unit number larger than the maximum value set by parameter MUNIT in INCLUDE file ARSIZE.PRM at the time the program was compiled should not be used.
   The units that have been referenced in FEQ simulation are monitored, and any duplicate reference results in an error.
7. The reuse of a unit number when processing files that contain function tables is allowed because these files are all processed in sequence and are processed completely before any other files referenced in the standard input to FEQ are processed. Use of a standard unit number for this purpose is recommended; unit number 15 is commonly applied for the unit number for processing each file containing function tables.
8. Standard numbers are recommended for files that appear in a common context. This would include the special-output file, the diffuse time-series file, and the files used to save and restore the state of the model. These numbers could all be less than some reasonable limit, such as 15. Then, numbers larger than 15 but no more than MUNIT are available for use in accessing point time-series files for input or output. The use of standard ranges will help avoid errors.

APPENDIX 4: FULL EQUATIONS MODEL ERROR MESSAGES, WARNINGS, AND BUGS: VERSION 8.0

Messages issued in FEQ to the user are in three categories. The first category of messages describe some condition in the input for which the intent of the user is not understood. Some simple errors can be identified; standard corrective action can then be taken, and reading and checking the input to search for additional errors can continue in simulation. The run, however, will terminate when the input has been checked. Some errors may be detected only during execution of the unsteady-flow computations. These errors are not corrected and, in most cases, the computations are terminated.

The second category consists of warning messages reporting conditions that may indicate a user error or that highlight the possibility of some later computational problem. Some assumption to circumvent the problem is made, and computations continue. The user should always check the warning messages to make sure that the default action taken in FEQ simulation has not affected the results.

The third category of messages consists of bug messages reporting conditions that should not occur. Such messages, which are usually indicative of some problem in the computer code, should be infrequent. However, an application may result in conditions not tested, and a bug may be revealed. Bug messages are not documented here.

All messages given by FEQ are of the form *ERR:nn*, *WRN:mm*, or *BUG-kk* where *nn*, *mm*, and *kk* are the respective message numbers. Error messages in any other form originate from the computer operating system, not FEQ.

ERR:NN MESSAGES

A fairly extensive set of error messages is available in FEQ to alert the user to errors in specification of the stream system as early as possible. As FEQ evolves, changes to program code will be made so that some of the currently undetected errors will be detected before creating computational problems or incorrect results. However, it is possible for one to specify an improperly defined stream system so that no errors are detected. As a simple example, the stationing used for the node locations on a branch might actually be in river miles but the stationing might be erroneously input as thousands of feet. This error cannot be detected because there will be a proper value in SFAC in either case. However, the results of the run, should it be completed, would be incorrect. Thus, three types of errors are possible: an error that can be detected in FEQ and is detected; an error that could have been detected in FEQ but is not because no code for the detection is available; and an error that, by its nature, could not be detected in FEQ either now or in the future. Over time, the number of errors of the first type will probably increase and the number of errors of the second type will probably decrease. The number of errors of the third type, however, will stay the same.

The error messages are given in numerical order below. There may be gaps in the order as changes are made to improve the code and circumvent errors. The order of numbering is arbitrary.

Messages with numbers of 60 or less are reported in a special subroutine, KILL, and relate to the early processing of the input to FEQ. Each message includes a separate line, not shown here, with a reference number. This number is defined from the context of the following message. For example, message 2, “NODE ON A BRANCH OUTSIDE VALID RANGE,” will show the errant node number in the reference-value line.

In many cases, the message includes numbers and, in a few cases, character strings. These numbers and character strings will have a value that depends on the specific error. The positions of these values are denoted by “nn” for an integer, “ff” for a floating-point number (that is, one containing a decimal point or in floating point format), and “aa” for a character string.

ERR: 1 BRANCH NUMBERS OUT OF RANGE
A branch number < 0 or > 999 has been detected.

ERR: 2 NODE ON A BRANCH OUTSIDE VALID RANGE
A node on a branch has been detected that is ≤ 0.

ERR: 3 NODE ON A BRANCH OUT OF SEQUENCE OR A DUPLICATE
Nodes on a branch must have node numbers assigned in ascending consecutive order. A node has been found that does not follow this sequence or is a duplicate on the branch.

ERR: 4 UPSTREAM EXTERIOR NODE NUMBER OUT OF RANGE
A node number for the upstream node on a branch has been detected, which is either ≤ 0 or > 1998. Issued only when ISTYLE=OLD.

ERR: 5 DOWNSTREAM EXTERIOR NODE NUMBER OUT OF RANGE
A node number for the downstream node on a branch has been detected, which is either ≤ 0 or > 1998. Issued only when ISTYLE=OLD.

ERR: 6 Not in use.

ERR: 7 INVALID DEVICE TYPE FOR A CONTROL STRUCTURE
Each control-structure category, code 4 or code 5, has its valid range of device type numbers listed in table 3. A type number has been detected that is outside the valid range for the control-structure category.

ERR: 8 I/O UNIT NUMBER ALREADY IN USE
The input/output units specified either explicitly or implicitly by the user are monitored in FEQ. A unit number has been detected that is the same as one already given by the user.

ERR: 9 TABLE NUMBER OUT OF VALID RANGE
A table number for a function table to be supplied by the user must be greater than zero and less than the compile time variable, MFTNUM (Maximum Function Table NUMber), given in the INCLUDE file ARSIZE.PRM (appendix 3). The current standard maximum function table number is 10,000. This number may be set to any desired positive value by the user. The new number does not take effect until FEQ is recompiled and linked.

ERR:10 FUNCTION TABLE SPACE EXCEEDED
An area of computer memory is maintained for the storage of all function tables in input to FEQ. The size of this area is given by the compile time variable, MRFTAB (Maximum Row Function TABle), in the INCLUDE file ARSIZE.PRM (appendix 3). This value must be large enough for the stream system being simulated. This number may be set to any desired positive value by the user. The new number does not take effect until FEQ is recompiled and linked. The typical range of values is 40,000 to 300,000 elements. Each element, on byte-oriented computers, takes 4 bytes. Thus, a request for 150,000 elements will use 600,000 bytes in one array.

ERR:11 INVALID CODE FOR NETWORK MATRIX CONTROL
A code value that is not defined by FEQ has been detected in the Network-Matrix Control Block (section 13.6). The current code values are given in table 3.

ERR:12 Not in use.

ERR:13 NETWORK MATRIX CONTROL SPACE EXCEEDED
A predetermined amount of space for storage of the network-matrix control information is allocated in FEQ. This space is given by the compile time variable, MREMC (Maximum Row Network-Matrix Control), in the INCLUDE file ARSIZE.PRM (appendix 3). This number may be set to any desired positive value by the user. The new number does not take effect until FEQ is recompiled and linked. The typical range is 2,000 to 3,000 elements.

ERR:14 NUMBER OF NODES AT A JUNCTION > 9 OR < 2
The number of nodes at a junction (Code = 2) is outside the valid range indicated in the message. There must be at least two nodes at a junction.

ERR:15 UNKNOWN BRANCH NUMBER IN NMC INPUT
A branch number appears in the Network-Matrix Control Block (section 13.6), but that branch number did not appear in the Branch Description Block (section 13.2). Therefore, the branch number has not been assigned.

ERR:16 TYPE MUST BE 1 OR 2 FOR A FORCED BOUNDARY
An invalid type has been found for a forced boundary (Code = 6).

ERR:17 NUMBER OF RESERVOIR INFLOW NODES MUST BE 1
In version 7.0 and later, the number of inflow nodes to a reservoir must be 1. This node becomes the upstream node for the reservoir analogous to the upstream node on a branch. Use code 2 (number of nodes at a junction, sum of flows equals zero) to attach nodes to the upstream end of a reservoir.

ERR:18 TYPE FOR POINT FLOWS > 2 OR < 1
The type for point flows must be either 1 or 2; a different value has been detected.

ERR:19 TOO MANY POINT FLOWS GIVEN
A predefined space is maintained for the point flows, and this space is too small. The size of this space is given by the compile time variable, MRPFPNT (Maximum Row Point Flow PoiNTer), in the INCLUDE file ARSIZE.PRM (appendix 3). This number may be set to any desired positive value by the user. The new number does not take effect until FEQ is recompiled and linked. The current value is 3 because the point-flow option is rarely used.

ERR:20 Not in use.

ERR:21 Not in use.

ERR:22 THE NODES FOR HEAD MUST BE DISTINCT
The two nodes for head in a two-node control structure are the same. This is invalid and meaningless.

ERR:23 THE NODE FOR FLOW MUST EQUAL ONE OF THE HEAD NODES
The flow node for a two-node control structure is different than either of the head nodes. The flow node must match one of the two head nodes.

ERR:24 RESERVOIR NODE DUPLICATES AN EXISTING NODE
A reservoir node number is a duplicate of a number already given to a previously defined flow-path end node.

ERR:25 DUPLICATE FUNCTION TABLE NUMBER
A function table number is a duplicate number already used for a previously defined function table.

ERR:26 Not in use

ERR:27 TOO MANY NODES
The number of nodes on branches has exceeded the maximum predefined in FEQ. The compile time variable, MNBN (Maximum Number Branch Nodes), in the INCLUDE file ARSIZE.PRM (appendix 3) defines the maximum number of nodes. This number may be set to any desired positive value by the user. The value of MNBN must always include the nodes at each end of each branch and any other nodes not on the ends of the branch (interior nodes). The variable MEXTRA in the

INCLUDE file ARSIZE.PRM (appendix 3) gives the count of the interior nodes on a branch. A new number does not take effect until FEQ is recompiled and linked. The range of values in current use for MEXTRA is 400 to 700.

ERR:28 POINT FLOW INVALID FOR UPSTREAM NODE
A point flow must enter a computational element. Computational elements are identified and referenced by the number of the downstream node. The upstream node on a branch cannot, therefore, refer to any computational element, and any such reference is reported as an error.

ERR:29 TOO MANY BRANCHES
A predefined space for branches is allocated in FEQ. The size of this space is given by the compile time variable, MNBRA (Maximum Number BRAnches), in the INCLUDE file ARSIZE.PRM (appendix 3). This number may be set to any desired positive value by the user. The new number does not take effect until FEQ is recompiled and linked.

ERR:30 TOO MANY EXTERIOR NODES
A predefined space for storage of flow-path end node information is allowed in FEQ. The size of this space is defined by the compile time variable, MNEX (Maximum Number EXterior flow-path end nodes), in the INCLUDE file ARSIZE.PRM (appendix 3). This number may be set to any desired positive value by the user. The new number does not take effect until FEQ is recompiled and linked.

ERR:31 INVALID OPERATION BLOCK NUMBER IN CHKEX
Operation block numbers must be positive but not greater than the number of blocks given in the Operation of Control Structures Block (section 13.12). The blocks should be numbered consecutively starting at 1.

ERR:32 LEVEL POOL RESERVOIR INFLOW NODE IS NOT FREE OR IS IN USE ELSEWHERE.
A node assigned as the inflow node to a reservoir either is not a free node, or if it is a free node, has already been used elsewhere for a dummy branch or for an inflow node to another level-pool reservoir.

ERR:33 FREE NODE ON A DUMMY BRANCH IS NOT FREE OR IS IN USE ELSEWHERE.
A free node assigned to a dummy branch either is not free, is used on another dummy branch, or is used as an inflow node to a level-pool reservoir.

ERR:34 Not in use.

ERR:35 INVALID PRINT OUT OPTION
A value for OUTPUT in the Run Control Block (section 13.1) < 0 or > 5 was detected. Values greater than 1 are designed for application in debugging and should be used with care because large amounts of output can be produced.

ERR:36 INVALID OPTION FOR POINT FLOWS
The option for POINT in the Run Control Block (section 13.1) must be YES or NO. Any other response will result in this error message.

ERR:37 INVALID OPTION FOR DIFFUSE FLOWS
The option for DIFFUSE in the Run Control Block (section 13.1) must be YES or NO. Any other response will result in this error message.

ERR:38 INVALID OPTION FOR WIND LOADING
The option for WIND in the Run Control Block (section 13.1) must be YES or NO. Any other response will result in this error message.

ERR:39 Not in use.

ERR:40 I/O UNIT NUMBER OUT OF RANGE
A predefined range for input/output unit numbers is used in FEQ. The minimum unit number is 1, and the maximum number is given by the compile time variable, MUNIT (Maximum UNIT number), in the INCLUDE file ARSIZE.PRM (appendix 3). This number may be set to any desired positive value by the user. The new number does not take effect until FEQ is recompiled and linked.

ERR:41 SPECIAL OPERATION BLOCK NUMBER OUT OF RANGE
Operation block numbers (section 13.12) must be positive but not greater than the maximum number of blocks given by the compile time variable, MNBLK (Maximum Number of BLocKs), in the INCLUDE file ARSIZE.PRM (appendix 3). This number may be set to any desired positive value by the user. The new number does not take effect until FEQ is recompiled and linked.

ERR:42 INVALID OPTION FOR SPECIAL OPERATIONS
The option for SOPER in the Run Control Block (section 13.1) must be YES or NO. Any other response will result in this error message.

ERR:43 EXTERIOR NODE USED AS A FLOW NODE IN A CONTROL STRUCTURE MORE THAN ONCE
A flow-path end node has been used as flow node in more than one control structure. A flow node must be unique; therefore, more than one use is invalid.

ERR:44 EXTERIOR NODE APPEARS ON MORE THAN ONE BRANCH
A flow-path end node has been assigned to more than one branch. The flow-path end node numbers assigned to the upstream and downstream nodes on each branch must be unique.

ERR:45 INVALID SLOPE SOURCE FOR CHANNEL CONTROL:-1,0,1, ARE VALID
A slope source code number other than the possible values given in the error message has been detected.

ERR:46 THIS NODE MATCHES ANOTHER WHEN NODES MUST BE DISTINCT.
Duplicate flow-path end node numbers have appeared when nodes should be distinct. This is possible in several of the Network-Matrix Control Block (section 13.6) options, codes 3, 5, 11, 12, 13, and 14.

ERR:47 Not in use.

ERR:48 TOO MANY FORCED NODES WITH TABLES OR FILES.
Increase MNCD6 in the INCLUDE file ARSIZE.PRM (appendix 3) and recompile and link FEQ.

ERR:49 TOO MANY INPUT FILES.
Increase MNFIN in the INCLUDE file ARSIZE.PRM (appendix 3) and recompile and link FEQ.

ERR:50 INVALID DIRECTION: 1 OR −1 ARE VALID
An invalid direction has been given in the Network-Matrix Control Block (section 13.6).

ERR:51 BRANCH HAS ALREADY APPEARED IN NMC INPUT.
The branch number has already been read. Indicates an error in input of a branch number, in labeling the schematic, or in writing the Network-Matrix Control Block (section 13.6).

ERR:52 Not in use.

ERR:53 Not in use.

ERR:54 Not in use.

ERR:55 Not in use.

ERR:56 Not in use.

ERR:57 Not in use.

ERR:58 Not in use.

ERR:59 Not in use.

ERR:60 Not in use.

ERR:61 ELEMENT LENGTH IS ZERO
The computational element length is given by the absolute value of the difference in the stations at the upstream and downstream ends. This length must be positive, but an element has been detected with zero length, an indication that two consecutive nodes on a branch were given the same station value.

ERR:62 Not in use.

ERR:63 X-SECT TABLE AREA RELATIVE ERROR > .02 TABLE # = nn DEPTH = ff
The area and first moment of area of all cross-section tables are recomputed. If the recomputed area differs from the area in the table by more than 2 percent, an error is issued if the area > 0.1. This indicates some error in the table. Perhaps the table has been distorted in processing or was not computed properly.

ERR:64 TABLE# = nn HAS INVALID VALUES FOR STRUCTURE SETTING.
Values in the given table are for the opening for a variable-geometry control structure. A value for the setting > 1.0 or < 0.0 has been detected. The specified setting value must be between 0.0 and 1.0.

ERR:65 TABLE# = nn IS OLD FORM TYPE 14. REPLACE WITH NEW FORM.
An old form of type 14 table has been detected. This type was used only for a short time and is no longer valid. Recompute the table with version 3.5 or later of FEQUTL (Franz and Melching, in press), using the same input as for the original computation of the table. Only a single type 14 table can appear in a code 5, type 6 entry in the Network-Matrix Control Block (section 13.6).

ERR:66 TABLE TYPE MISMATCH. TABLE NUMBERS: nn AND nn MUST BE TYPE 14.
Input data requiring type 14 tables have been specified with table numbers *nn* and *nn*, but other table types have been found.

ERR:67 TABLE TYPE MISMATCH. TABLE NUMBERS: nn AND nn MUST BE TYPE 6 OR 13.
Input data requiring either type 6 or 13 tables have been specified with table numbers *nn* and *nn*, but other table types have been found.

ERR:68 Not in use.

ERR:69 Not in use.

ERR:70 ARGUMENT BELOW RANGE IN LKTAB
TABLE NUMBER = nn
TIME = ff
ARGUMENT = ff

Each table has a minimum value for arguments for which the function value is defined. In this error, an argument less than this minimum for the given table resulted at the given time. The user should check that the correct table has been specified and that it has an adequate range. This error results in some cases at the beginning of a simulation when roundoff error makes the starting time appear to be slightly earlier than the time given in one or more tables. The solution in this case is to either move the time earlier in the tables or move the start time slightly later. A change on the order of 1 second or less in the start time may eliminate the error.

ERR:71 ARGUMENT ABOVE RANGE IN LKTAB
TABLE NUMBER = nn
TIME = ff
ARGUMENT = ff
Each table has a maximum value of an argument for which the function value is defined. An argument greater than this maximum value has been detected. The input table should be checked for an adequate range.

ERR:72 EU < ED IN TWO-D TABLE# = nn EU = ff, ED = ff
In 2-D type 6 or 13 tables, the upstream water-surface elevation can never be less than the downstream water-surface elevation. A pair of water-surface elevations has been detected that does not satisfy this relation, and computations are terminated. This error indicates a problem with the table or a bug in the software.

ERR:73 UNEXPECTED TYPE IN TWO-D TABLE# = nn TYPE = nn EXPECTED TYPE = nn
An incorrect table type for a 2-D table has been detected. An incorrect table number has probably been given.

ERR:74 INVALID MONTH IN TABLE# = nn
An invalid month number has been detected during processing of the time argument for a table of type 7, 8, or 9. All months should be in the range of 1 to 12.

ERR:75 INVALID DAY IN TABLE# = nn
An invalid day of the month has been detected during processing of the time argument for a table of type 7, 8, or 9. All days should be in the proper range for the given month.

ERR:76 INVALID YEAR IN TABLE# = nn
The year for the time argument for a table of type 7, 8, or 9 must be greater than 1859 for the internal date computations to be valid.

ERR:77 FIRST VALUE OF P FOR CONSTANT HEAD LINE NOT ZERO. P = ff
In type 6 tables, the first value of P, the fraction of the drop to free flow, must always be zero. A table has been detected without this value.

ERR:78 P DECREASES. P = ff POLD = ff
In type 6 or 13 tables, the values of P, the fraction of the drop to free flow, must always be increasing. A table has been detected for which this is not true. POLD gives the previous value of P, and P gives the current value of P.

ERR:79 VALUES OF UPSTREAM HEAD ARE DECREASING IN 2-D TABLE.
In type 6 or 13 tables, the values of upstream head must always be increasing. A table has been detected for which this is not true.

ERR:80 INVALID ROADWAY FLOW. ZU = ff ZD = ff RDELEV = ff RS = ff
Flow over the roadway in the bridge-flow computations made with code 5, type 4 must always flow

from a higher elevation to a lower elevation. A case where the direction of flow and the relation of elevations for the computation of flow over the roadway are contradictory has been detected. If the problem cannot be resolved by modifying the input, then the error detection code will have to be modified to permit small discrepancies in the elevation-direction relation.

ERR:81 TABLE# = nn DOES NOT EXIST.
A table number not in the list of table numbers has been detected, an indication that the input to FEQ referenced a table number but the table for that number was not included in the input.

ERR:82 TABLE# = nn TOO LARGE. SET TO CURRENT MAXIMUM OF: nn
The current maximum table number has been exceeded by a table number given in the input to FEQ. The current maximum table number can be increased by changing MFTNUM (Maximum Function Table NUMber) in the INCLUDE file ARSIZE.PRM (appendix 3) to the maximum value desired. The new number does not take effect until FEQ is recompiled and linked. The only limit to the maximum is the available memory for the computer program and the space allowed for input of a table number. Table numbers appearing in the Network-Matrix Control Block (section 13.6) are currently limited to four digits and those for the cross-section table number in the Branch Description Tables are currently limited to five digits.

ERR:83 NODE FOR CHANNEL CONTROL DEPTH nn NOT ON A BRANCH
Channel control of the relation between flow and depth can be requested only for a node on a branch.

ERR:84 CHANNEL CONTROL NODE nn NOT AT DOWNSTREAM END OF BRANCH
Channel control of the relation between flow and depth can be requested only at the downstream node on a branch.

ERR:85 NUMERIC PROBLEM IN SUBROUTINE EXCON. 3nn,ff,6(ff)
This error message should not result because critical depth is currently computed in subroutine EXCON, ignoring the effect of the momentum-flux correction coefficient. If this restriction is removed, then it is possible that the critical flow cannot be computed because the variation of the momentum-flux coefficient with depth is such that the critical flow is mathematically undefined. This results because the currently available techniques for estimating the momentum-flux coefficient can be in error, especially for the computation of critical flow.

ERR:86 INVALID CROSS-SECTION INTERPOLATION REQUEST XL = ff XM = ff XR = ff UPS TABLE#=nn DNS TABLE#=nn
An interpolation request for an intermediate cross section has been detected with the station of the intermediate cross section falling outside of the station interval given by the next available upstream and downstream cross section. The table numbers for the cross-section tables involved are given to determine the station interval in a large stream network.

ERR:87 MORE THAN nn DEPTH VALUES BETWEEN TABLE#'S nn AND nn
The maximum number of depth values currently allowed for intermediate cross sections is nn. If this value too small, then the cross sections given by the table numbers in the message should be adjusted such that either fewer depth values are in each table or that a greater number of coincident depth values are in the tables. If this is not satisfactory, then the maximum must be increased by changing MNDEP in the INCLUDE file ARSIZE.PRM (appendix 3) to the desired value and recompiling and linking FEQ.

ERR:88 EXPECTED nn OPERATION BLOCK REFERENCES. ONLY nn FOUND.
The number of times that the Network-Matrix Control Block (section 13.6) refers to an operation block (section 13.12) is counted in FEQ. The number of operation block inputs also are counted, and

these two values disagree. Either a block is referenced in error in the Network-Matrix Control Block or a block is missing from the Operation of Control Structures Block.

ERR:89 EXTERIOR NODE = aa NOT FOUND IN EXTERIOR MATRIX CONTROL WITH CODE = 4
Computation of a one-node stage-discharge relation at the given flow-path end node was attempted, but no such relation appears in the Network-Control Matrix Block (section 13.6). The most likely source of the error is an incorrect specification of the options for the computation of the backwater profiles.

ERR:90 MIXED INPUT STYLE. NODE# = nn
A flow-path end node number without a prefix has been detected during processing of a new-style input requiring a U, D, or F prefix for each flow-path end node number.

ERR:91 Not in use.

ERR:92 Not in use.

ERR:93 Not in use.

ERR:94 DOWNSTREAM DEPTH MISSING FOR BRANCH nn
The downstream depth to start the computations of backwater appears to be missing. The depth may have been given or may have been obtained from a flow-path end node for which the depth has already been computed, but the bottom elevation of the downstream end of the current branch is greater than the water-surface elevation known at the source node.

ERR:95 MAXIMUM ITERATIONS EXCEEDED FOR BACKWATER ANALYSIS
The computations for the unknown depth in the backwater analysis have not converged within the allowed number of iterations. There are many potential explanations for the lack of convergence. The most common explanation is that the computational element is too long. Thus, most such convergence problems are corrected by interpolating one or more intermediate cross sections in the current computational element. The downstream node for the element in question is given by the last node printed for the branch in the Backwater Analysis Block (section 13.15). If the slope and roughness of the branch make supercritical flow possible, it may be necessary to request that the inertial terms be ignored in the branch by setting INERTIA to 0.0 in the branch heading information (section 13.2).

ERR:96 FATAL ERRORS ENCOUNTERED
This message is printed before the execution of the program is terminated because errors have been detected in the input, making continuation of the program computations impossible. One or more error messages will appear in the output describing the nature and location of the errors.

ERR:97 ELEVATION UNDEFINED AT EXN = aa
A flow-path end node has been referenced as a source of water-surface elevation in the Backwater Analysis Block (section 13.15) but the elevation at that node has not been defined. Some error has been made in either the node number or in the order of computation of the branches.

ERR:98 INVALID BRANCH NUMBER. BNUM = nn
A branch number detected in the Backwater Analysis Block (section 13.15) was not input in the Branch Description Block (section 13.2).

ERR:99 INVALID EXTERIOR NODE. EXN = aa
A flow-path end node number less than 1 or greater than the number of flow-path end nodes (section 13.1) has been detected in the Backwater Analysis Block (section 13.15).

ERR:100 EXN# WITH BRA = 0 AND ICODE = -3 IS NOT A FREE NODE.
EXN # = aa
A flow-path end node has been referenced as a free node in the Backwater Analysis Block (section 13.15), but the node is detected on a branch.

ERR:101 INVALID EXTERIOR NODE GIVEN IN C/ND WHEN BRA = 0.
C/ND = aa
A flow-path end node referenced in the Backwater Analysis Block (section 13.15) in the code/node column is either < 1 or greater than the number of flow-path end nodes (section 13.1).

ERR:102 INVALID EXTERIOR NODE GIVEN WHEN BRA = 0 EXN = aa
There is a flow-path end node referenced in the Backwater Analysis Block (section 13.15) where the branch column is zero and the flow-path end node is either < 1 or greater than the number of flow-path end nodes (section 13.1).

ERR:103 INCORRECT NO. OF EQUATIONS SPECIFIED FOR NETWORK MATRIX,
NO. OF EQUATIONS SPECIFIED = nn
NO. OF EQUATIONS REQUIRED = nn
Exactly twice as many equations as flow-path end nodes must be in the network matrix. Too few or too many equations may have been given. The schematic and the equations should be reviewed to find the error.

ERR:104 MIXED INPUT STYLE. NODE ID=aa
A flow-path end node has been detected with a prefix during processing of an input from a previous version of FEQ that does not allow a prefix on a flow-path end node number.

ERR:105 VALUE FOR BRANCH NODE ALREADY PRESENT. CHECK NODE NUMBER= aa IN CODE COLUMN.
A value for a node has been detected in the CODE column that is a likely error. Check to make sure that a value does not overlap two input fields in the Backwater Analysis Block (section 13.15).

ERR:106 CODE nn NOT IMPLEMENTED
A code found in the Network-Matrix Control Block (section 13.6) has not been implemented.

ERR:107 EXTERIOR NODES ARE THE SAME ON BRA = nn
The upstream and downstream nodes on the given branch have the same node number. These numbers must be distinct in FEQ.

ERR:108 CROSS SECTION TABLE NUMBER MUST BE > 0 AT FIRST NODE ON BRANCH
The first node on the branch must always have a cross-section table given that does not require interpolation.

ERR:109 INLET AREA GIVEN BUT INLET CUTOFF HEIGHT IS MISSING FOR NODE: nn
If the inlet area is given for a sewer, then the cutoff height also must be given.

ERR:110 Not in use.

ERR:111 TABLE# = nn DOES NOT EXIST.
The table number given in the Network-Matrix Control Block (section 13.6) is not detected.

ERR:112 CRITICAL DEPTH INVALID FOR THIS NODE: nn
Critical depth can be requested only at a node on a branch. The node given is not on a branch.

ERR:113 RESERVOIR TRIB AREA GIVEN FOR NODE aa BUT NODE IS NOT A RESERVOIR.
A flow-path end node has been referenced as a reservoir node in the Tributary Area Block (section 13.3). The node, however, is not a reservoir node.

ERR:114 CODE = 9 SELECTED WHEN NOT SUPPORTED
Code = 9 in the Network-Matrix Control Block (section 13.6) is not yet (1997) supported. It is under development in FEQ. The code number is reserved and will be made available in the future.

ERR:115 NODE = aa NOT ON A BRANCH FOR CODE = nn
The flow-path end nodes included with Code = *nn* in the Network-Matrix Control Block (section 13.6) must all be on a branch. The specified node is not on a branch and is, therefore, invalid.

ERR:116 CROSS SECTIONS NOT THE SAME FOR CODE = nn BETWEEN NODES aa AND aa
In Code = *nn* in the Network-Matrix Control Block (section 13.6), the cross sections at its upstream and downstream nodes must be identical and specified by the same table number. The table numbers are different, and the cross sections are assumed to also be different.

ERR:117 BOTTOM ELEVATIONS NOT THE SAME FOR CODE = nn BETWEEN NODES aa AND aa
In Code = *nn* in the Network-Matrix Control Block (section 13.6), the bottom elevations must be the same for the two flow-path end nodes included. The bottom elevations as specified by the user are not the same and, therefore, the input is invalid.

ERR:118 INVALID TYPE = nn IN POINT FLOW TABLE
Only point flow types 1 or 2 are valid. A type different than either of these valid numbers has been found during processing of a point-flow table.

ERR:119 TABLE NUMBER = nn OUT OF RANGE FOR POINT FLOW NO.: nn
The specified table number is out of the valid range for table numbers in the point-flow specification. The point-flow number is the current count of the point-flow specifications processed.

ERR:120 ONLY ONE FLOW PATH PERMITTED BETWEEN NODE aa AND NODE aa
Only one flow path is permitted for the table type applied in a two-node control structure. Tables from which flow may be estimated from the two heads can be summed in a code 5, type 6 entry in the Network-Matrix Control Block (section 13.6).

ERR:121 BOTTOM SLOPE FOR NORMAL DEPTH < 0. S = ff
The user-supplied bottom slope for the computation of normal depth is negative and, therefore, invalid.

ERR:122 UPSTREAM NODE: aa FOR CODE 5 TYPE 5 MUST BE ON DOWNSTREAM END OF A BRANCH.
In code 5, type 5 in the Network-Matrix Control Block (section 13.6), the upstream node must be on the downstream end of a branch. The node specified by the user is not on the downstream end of a branch and is invalid.

ERR:123 DOWNSTREAM NODE: aa FOR CODE 5 TYPE 5 MUST BE ON UPSTREAM END OF A BRANCH.
In code 5, type 5 in the Network-Matrix Control Block (section 13.6), the downstream node must be on the upstream end of a branch. The node specified by the user is not on the upstream end of a branch and is invalid.

ERR:124 FLOW NODE FOR CODE 5 TYPE 5 MUST BE THE UPSTREAM NODE
In code 5, type 5 in the Network-Matrix Control Block (section 13.6), the flow node must be the upstream node. The flow node given by the user is not the upstream node and is, therefore, invalid.

ERR:125 INVALID NODE NUMBER aa
A flow-path end node number < 0 or > NEX, the number of flow-path end nodes specified in the Run Control Block (section 13.1), has been detected in the Free-Node Initial Conditions Block (section 13.14).

ERR:126 INVALID INITIAL DEPTH ff DEPTH MUST BE POSITIVE NON-ZERO
A depth given in the Free-Node Initial Conditions Block (section 13.14) is ≤ 0 and is, therefore, invalid.

ERR:127 NUMBER OF EXTERIOR NODES GIVEN BY BRANCHES EXCEEDS EXPECTED NUMBER OF nn
The number of flow-path end nodes specified by twice the number of flow paths (branches, dummy branches, and level-pool reservoirs) is larger than the number of flow-path end nodes specified by the user in the Run Control Block (section 13.1). Either the number of flow paths given is too large or the specified number of flow-path end nodes is too small.

ERR:128 EXTERIOR NODE aa PREVIOUSLY INITIALIZED TO ff CHECK BRANCH-EXTERIOR NODE TABLE
A flow-path end node listed in the Free-Node Initial Conditions Block (section 13.14) has already been initialized. It is likely not a free node, or it has already appeared in the previous input for this block.

ERR:129 EXTERIOR NODE aa IS NOT FREE. IT IS ON BRANCH# nn
A flow-path end node listed in the Free-Node Initial Conditions Block (section 13.14) is not a free node and is, therefore, invalid.

ERR:130 MONTH= nn IN DATE < 1 or > 12.
A month value less than 1 or greater than 12 has been detected in a date.

ERR:131 INVALID USE OF AUXILIARY TABLE FILES.
FEQIN TABLE INPUT MUST BE COMPLETE BEFORE USING AUXILIARY FILES.
Auxiliary table files for the input of function tables can only be used after table input from the principal input (called FEQIN in the message) is complete. A function table in the principal input has been detected after an auxiliary table file has been processed. To correct the error, the function table or tables in the principal input should be moved above the first reference to an auxiliary table file.

ERR:132 ONLY nn EXTERIOR NODES GIVEN BUT AT LEAST 2*NBRA = nn REQUIRED.
The number of end nodes on branches exceeds the specified number of flow-path end nodes. Either the number of branches or the number of flow-path end nodes is in error.

ERR:133 NBRA = 0 INVALID. NBRA > 0 REQUIRED.
At least one branch must be present in FEQ.

ERR:134 BRANCH NUMBERS OUT OF RANGE OR SEQUENCE IN TRIB. AREA INPUT
The branch numbers in the Tributary Area Block (section 13.3) must be in ascending order with no omissions. A branch number in this input has been detected that is out of the valid range for branch numbers or is out of numerical sequence.

ERR:135 NODES ON A BRANCH OUT OF RANGE OR SEQUENCE IN TRIB. AREA INPUT
The nodes on a branch in the Tributary Area Block (section 13.3) must be in ascending order. A node has been detected that is either out of range or is not in ascending order.

ERR:136 TRIB AREA FOR RESERVOIRS REQUIRES nn
LOCATIONS IN QPVEC BUT ONLY nn ARE AVAILABLE.
Adequate space for storage of the tributary area for reservoirs is not available. The size of MEXTRA in the INCLUDE file ARSIZE.PRM (appendix 3) must be increased, and the program must be recompiled and linked.

ERR:137 YEAR= nn TOO EARLY FOR MODIFIED JULIAN DATE COMPUTATION. MUST BE 1859 OR LATER.
A year has been input preventing the proper computation of the modified Julian date. If such a date must be used, modification of FEQ is required to compute the modified Julian date over a wider range of years.

ERR:138 NUMBER OF GAGES < 1 OR > nn
The number of gages is out of range. If the number is correct, then MXGAGE must be changed in the INCLUDE file ARSIZE.PRM (appendix 3), and the program must be recompiled and linked.

ERR:139 GAGE NUMBER: nn OUT OF SEQUENCE
The gage numbers must be in ascending order (section 13.3), but a gage number has been detected that is out of order.

ERR:140 SUM OF GAGE LAND USES IN ERROR
The summation of land uses from the gages does not match the number of land uses specified in the input (section 13.3).

ERR:141 GAGE NUMBER: nn OUT OF RANGE
The gage number is outside the valid range. See ERR:138.

ERR:142 UPSTREAM STATION: ff NOT FOUND IN BRANCH: nn
The option to distribute tributary area by station interval has been invoked, but the upstream station of the interval cannot be detected in the list of stations specified in the Branch Description Block (section 13.2).

ERR:143 DOWNSTREAM STATION: ff NOT FOUND IN BRANCH: nn
The option to distribute tributary area by station interval has been invoked, but the downstream station of the interval cannot be detected in the list of stations specified in the Branch Description Block (section 13.2).

ERR:144 UPSTREAM NODE = nn >= DOWNSTREAM NODE = nn FOR TRIB. AREA.
The stations for the station-interval method (section 13.2) for distributing tributary area are specified in reverse order so that the node corresponding to the downstream station is upstream from the node corresponding to the upstream station. As a result, the tributary area cannot be distributed because the nodes are reversed.

ERR:145 NUMBER OF LAND USES < 1 OR > nn
The number of land uses must be at least 1 if the diffuse flows option is invoked in the Run Control Block (section 13.1). The number of land uses must not be larger than the maximum number of uses specified in the INCLUDE file ARSIZE.PRM (appendix 3). If the current maximum number is too small, MNDIFA must be changed in ARSIZE.PRM to the new maximum number, and FEQ must be recompiled and linked.

ERR:146 REQUEST FOR AUTOMATIC TRIB. AREA ALLOCATION INVALID. MUST BE FIRST AND ONLY ENTRY FOR THE BRANCH
If the automatic area allocation for the tributary area to a branch (Branch mode) is selected (section 13.2), then only one line of area information for each gage should appear for each branch. More than one line of information has been given for a branch, so allocation cannot be completed.

ERR:147 OUTPUT REQUESTED AT TOO MANY NODES. NOUT = nn
The limit for the number of output locations for the special-output option has been exceeded. If the number must be increased, the value of MNSOUT in the INCLUDE file ARSIZE.PRM (appendix 3) is changed to the new value and FEQ is recompiled and linked. Furthermore, the specification of the number of floating-point numbers to output in subroutine OUTSP in formats 10 and 11 must be changed to correspond to the new limit. The limit on the record size in the operating system used to control the computer system must be considered. Under MS-DOS the limit is 512 bytes. This constrains the number of special outputs to about 62 locations.

ERR:148 IN BRANCH nn THERE HAVE BEEN nn CONSECUTIVE INTERPOLATION REQUESTS. ONLY nn ARE ALLOWED.
More than the allowed number of consecutive interpolated sections have been requested in a branch. The allowed number is increased by changing the value of MNMID in the INCLUDE file ARSIZE.PRM (appendix 3) and recompiling and linking the program.

ERR:149 STAND FLD LVL AT OR BELOW MINIMUM POINT IN CHANNEL AT BRANCH nn NODE nn
The standard flood level given for the node has an elevation that is at or below the minimum point in the cross section and is invalid.

ERR:150 NUMBER OF INPUT FILES REQUESTED IS LARGER THAN THE CURRENT MAXIMUM OF: nn
Too many input files have been requested. The limit can be increased by changing MNFIN in the INCLUDE file ARSIZE.PRM (appendix 3) to the new limit and recompiling and linking FEQ. The parameter MRRBUF in the INCLUDE file ARSIZE.PRM also should be increased to allow more buffer space for the input files. The available buffer space is divided equally among the input files.

ERR:151 FILE NUMBER IN INPUT LIST NOT IN FORCED BOUNDARY LIST. UNIT = nn
An input file has been specified, but it has not been associated with any forced boundary condition (Code = 6 in the Network-Matrix Control Block (section 13.6)). All the files listed in the Input Files Block (section 13.10) must be used in the current run.

ERR:152 INPUT FILE nn HAS A START TIME = ff > RUN START TIME = ff
An input file has been found with a start time that is later than the start time of the run. Thus, the run cannot be made because the data in the file are incomplete.

ERR:153 FILE NUMBER IN FORCED BOUNDARY NOT IN INPUT LIST. UNIT = nn
A file number referenced in a forced boundary (Code = 6 in the Network-Matrix Control Block (section 13.6)) could not be detected in the list of input files. The missing file information must be supplied.

ERR:154 THERE ARE nn VALUES IN EACH RECORD OF THE DIFFUSE TSF BUT ONLY nn VALUES ALLOWED IN FEQ.
The record length of the diffuse time-series file being used is larger than the record length allowed in FEQ. The value of MNDIFA must be changed in the INCLUDE file ARSIZE.PRM (appendix 3) and FEQ must be recompiled and linked to change the number of values allowed.

ERR:155 THERE ARE nn DIFFUSE FLOWS IN THE DIFFUSE FLOW FILE BUT nn LAND USES IN FEQ.
The number of land uses specified in the Tributary Area Block (section 13.3) is not the same as the number of diffuse flows in the diffuse time-series file. These two numbers must be the same. If all the land uses in the diffuse time-series file are not of interest, then give a tributary area of zero for those land uses to be skipped. This will then result in the correct total for land uses and will yield the desired result.

ERR:156 ITEM FIELD MUST BE: “ELEV” OR “FLOW”
In the Output Files Block (section 13.11) the item to be output must be either water-surface elevation or flow. The spelling and spacing must be checked, and FEQ must be run again.

ERR:157 TYPE FIELD MUST BE: “PNT” OR “STAR”
In the Output Files Block (section 13.11) the recording type must be either a point or star value. The spelling and spacing must be checked, and FEQ must be run again.

ERR:158 NUMBER OF OUTPUT FILES REQUESTED LARGER THAN THE CURRENT MAXIMUM OF: nn
More than the current maximum number of output files has been requested. MNFOUT in the INCLUDE file ARSIZE.PRM (appendix 3) must be changed to the new maximum, and FEQ must be recompiled and linked.

ERR:159 MEAN VALUED ELEVATIONS CANNOT BE OUTPUT
It is meaningless to output mean values of water-surface elevations, and recording this information for water-surface elevations is not allowed.

ERR:160 ZERO DIVIDE. CHECK THAT ELEVATION > CREST ELEVATION AT EXTERIOR NODE: aa
The starting elevation for a stage-discharge rating in the Backwater Analysis Block (section 13.15) input has been detected at or below the crest elevation in the Network-Matrix Control Block (section 13.6). Increase the starting elevation so that a positive head results.

ERR:161 UNABLE TO SET UP VECTORS QPVEC, WSVEC IN SUBROUTINE LOAD
An error has been detected and reported in subroutine LOAD that makes it impossible to complete the subroutine properly. Therefore, the computations must stop. This error also occurs if the starting or ending item for the simulation is outside the time span defined for the table or file giving the conditions at a forced boundary.

ERR:162 EXECUTION TERMINATED BECAUSE OF PREVIOUS ERRORS
One or more errors requiring termination of the computations have been detected. The errors reported prior to this error message must be located and corrected.

ERR:163 CONVERGENCE CRITERION FOR SURCHARGE STORAGE IS TOO SMALL. MUST BE LARGER THAN 0.01
The value of EPSXS, the convergence criterion for cross-section tables, is not used in FEQ version 8.0 because the flow area is no longer used as the dependent variable. Thus, the string “EPSXS” should be replaced by the string “SSEPS” in the input for specifying the convergence criterion for surcharge storage. This convergence criterion is active whenever sewers are simulated and there is surcharge storage. The criterion should be in the range of 0.1 to 0.25. If the criterion is too large, then the water balance will have large errors; if it is too small, convergence problems will result.

ERR:164 TABLE NUMBERS EXHAUSTED IN NEXTN
All available table numbers have been used for the interpolation of cross-section tables. The maximum number of table numbers must be increased by increasing the parameter MFTNUM in the INCLUDE file ARSIZE.PRM (appendix 3) and recompiling and linking the program.

ERR:165 VALUE OF MORG WRONG IN ARSIZE.PRM. MUST BE 0 OR 1.
MORG is the origin for the retrieval of the command-line arguments when command-line arguments are retrieved by use of the standard function names from UNIX. Not all compilers support this option. Compilers that do support the option differ in how command-line arguments are counted. Some compilers label the command as argument 0 and the first argument labeled as 1. Other compilers will label the command as the first argument, so the first command-line argument is labeled as 2. In the first case, MORG = 0, and in the second case, MORG = 1. Any other value for MORG is invalid. MORG must be changed in the INCLUDE file ARCSIZE.PRM (appendix 3) to the correct value, and FEQ must be recompiled and linked.

ERR:166 WEIGHT COEFFICIENT FOR AVERAGING ELEVATION IS INVALID.
MUST BE >= 0 AND <= 1.0. WEIGHT= ff
The weight coefficient for averaging water-surface elevations in those Network-Matrix Control Block (section 13.6) options that use an average water-surface elevation is outside the valid range.

ERR:167 INCORRECT NUMBER OF BRANCHES GIVEN OR BNUM MISSPELLED.
The character string, “BNUM”, is read to determine that the proper point for input of branches has been reached in the processing of input. This string has not been found, perhaps because the string is misspelled or in lower case or because the incorrect number of branches is given in the Run Control Block (section 13.1).

ERR:168 SIDE NODE aa FOR SIDE WEIR MUST HAVE SIGN OF −1.
In the current side-weir option (section 13.6, code 14), a sign of −1 must be specified for the side-weir node in order for flows to be properly represented.

ERR:169 NUMBER OF GAGE LAND USES = nn EXCEEDS MAX OF nn AT GAGE nn.
Too many gage land uses have been assigned to a rain gage. MXGLU in the INCLUDE file ARSIZE.PRM (appendix 3) must be increased and the program must be recompiled and linked.

ERR:170 NUMBER OF GAGE LAND USES = nn IS INVALID.
The number of rain-gage land uses (section 13.3) is either negative or zero. Only positive numbers are meaningful.

ERR:171 FILE ACCESS NOT YET SUPPORTED FOR CONTROLLING LEVEL.
The controlling level for code 4, type 6 can come only from a time-series table at this point. Support for file access can be added if it is required.

ERR:172 EXTERIOR NODE aa DOES NOT HAVE TRIB. AREA OR IS NOT A RESERVOIR.
Output of the diffuse land-surface runoff value has been requested at a flow-path end node. The node specified must be a reservoir node with an assigned tributary area. The node given in the message does not meet these requirements.

ERR:173 TABLE#= nn AND TYPE=nn DOES NOT HAVE CRITICAL FLOW.
The cross-section table referenced for critical flow in a code 5, type 5 abrupt expansion does not include critical flow. The table must be recomputed to include critical flow (table types 22 and 25) or a table must be specified that does include critical flow.

ERR:174 TABLE#= nn OF TYPE= nn INVALID IN CURRENT CONTEXT. VALID TABLE TYPES ARE: nn
A cross-section table appears without the proper hydraulic characteristics. The table types that do include the proper hydraulic characteristics are listed.

ERR:175 TABLE TYPE= nn UNIMPLEMENTED IN FTABIN. VALID TYPES ARE: nn
A function-table type has been detected that is not currently supported by the Function Tables Block (13.13). Possible user error. Check the table type and supply a valid type.

ERR:176 INPUT STYLE OPTION: ISTYLE IS MISSING. ADD AFTER OPTION: CHKGEO
ISTYLE was added in FEQ version 7.0. The input does not have the option, so it must be added. The Run Control Block section (section 13.1) provides additional details.

ERR:177 Not in use.

ERR:178 BRANCH NUMBER nn REQUIRES NEAR-ZERO-DEPTH DATA BUT NONE FOUND.
The governing equation option STDW or STDCW has been invoked for a branch, but the branch number does not have the needed data in the Near-Zero-Depth Block (section 13.5). The missing input must be added.

ERR:179 BRANCH NUMBER = nn DOES NOT APPEAR IN NMC INPUT.
The given branch number has not been located in the Network-Matrix Control Block (section 13.6). All branches must be referenced exactly once in the input.

ERR:180 EXPECTED nn ENTRIES IN NMC INPUT BUT FOUND nn ENTRIES.
An incorrect number of entries are made in the Network-Matrix Control Block (section 13.6). The number of entries should be exactly twice the number of flow-path end nodes less the number of branches.

ERR:181 EXTERIOR NODE = aa APPEARS IN CODES 2 OR 6, MORE THAN ONCE.
An flow-path end node on a branch or a reservoir can appear only once in one of the specified codes.

ERR:182 NO SPACE TO ADD NODES. INCREASE MNBN IN ARSIZE.PRM AND RECOMPILE.
The ADDNOD option has increased the number of nodes to the extent that the current copy of the FEQ executable file does not have enough allocated space. The size of MNBN in the INCLUDE file ARSIZE.PRM (appendix 3) must be increased and the program recompiled and linked to create a new and larger capacity executable file.

ERR:183 EXTRAPOLATION TOLERANCE: EXTTOL IS MISSING.
ADD AFTER OPTION: MINBND.
The maximum depth in each cross section is recalled in FEQ before any extrapolation requested by the user is done. The resulting maximum elevations in each cross section are checked with the maximum depth in the cross sections, and a warning message is issued for each cross section that is overtopped by more than EXTTOL. EXTTOL should be picked so that minor extrapolations are not reported.

ERR:184 ELEVATION= ff ON BRANCH= nn AT NODE= nn HAS NO STATION
An elevation of the bottom profile has been given without a corresponding station. The station cannot be interpolated from the bottom elevation. A station must be supplied with the bottom elevation.

ERR:185 EXTERIOR NODE = aa NEVER APPEARS IN CODES 2, 4, 6, 7, OR 8.
A flow-path end node must appear at least once in one of the above codes to be properly connected.

ERR:186 TWO STATIONS MATCH IN BRANCH nn STATION = ff
Each station along a branch must be unique so that each element has a nonzero length.

ERR:187* KEY = aa INVALID OPTION. VALID OPTIONS ARE:
ELEV, QCON, or QVAR.

An invalid option for a monitoring point in an Operation of Control Structures Block (section 13.12) has been detected.

ERR:188 NUMBER OF NODES FOR KEY = QVAR <= 0.
A request for a variable null zone, QVAR, has an invalid number of nodes specified for the computation of the central value for the variable null zone (section 13.12). At least one flow-path end node must be given so that a flow rate about which to center the variable null zone can be computed.

ERR:189 NULL ZONE HALF WIDTH <= 0.
A null zone half width that is less than zero is requested. This is invalid because the null zone must always have a positive half width (section 13.12).

ERR:190 BRANCH NUMBER FOR KEY = QVAR > 0.
The node defining the control point for a variable null zone must be a flow-path end node defined by setting the branch number equal to zero (section 13.12). A positive branch number for the null zone has been detected.

ERR:191 SOURCE NODE = aa SAME AS SENSE NODE.
A source node for defining a variable null zone matches the node to be controlled. This is invalid because the monitored node must differ from all the source nodes used to define the variable null-zone location (section 13.12).

ERR:192 ONE OR MORE CROSS SECTION FUNCTION TABLES FOR BRANCH nn
DO NOT SUPPORT THE GEQ OPTION FOR THIS BRANCH.
A governing-equation (GEQ) option has been specified for a branch but at least one cross section in the branch does not have the needed cross-sectional hydraulic characteristics for the governing-equation option. The correct cross-section table type must be given for the GEQ option.

ERR:193 NUMBER OF INPUT FILES = nn NOT SAME AS NUMBER OF FILE REQUESTS = nn
The number of input files must be the same as the number of file requests. The same input file cannot be used at more than one flow-path end node. If the same flow sequence is to be applied at two nodes, then distinct file names containing the same flows must be given for proper simulation.

ERR:194 SQUARED RESIDUAL TOLERANCE: SQREPS IS MISSING.
ADD AFTER OPTION: EXTTOL.
The input is out of date relative to the version of FEQ. The new tolerance must be added as indicated. “Run Control Block” (section 13.1) contains complete details.

ERR:195 DATA DEFICIENCY IN TABLE# = nn VALID TYPES: nnnn
A table number has been given for a cross section; however, the required information is not available in that table type. The table for the cross section must be recomputed using the proper table type.

ERR:196 RESERVOIR AT NODE= nn HAS SURF. AREA = ff AT ELEV.= ff
UNABLE TO CONTINUE COMPUTATIONS WITH SURF. AREA <= 0.
A level-pool reservoir has been detected with a surface area of zero. Values for the reservoir cannot be computed in this case. If the reservoir is empty, dead storage below the minimum outlet point and a small surface area should be included in the input.

ERR:197 CROSS SEC. TAB. NUM.= nn NOT FOUND FOR STATION.
This error results in two cases. First, if table lookup is requested for either station or elevation in the branch input and the Function Tables Block (section 13.13) is placed before the Branch Description Block (section 13.2), this error indicates that the table given did not appear in the Function Tables Block. Second, if table lookup is requested for either station or elevation in the Branch Description

Block and the Function Tables Block is not placed before the Branch Description Block, this error indicates that the Function Tables Block must be moved immediately before the Branch Description Block. If this is not done, inadequate information on the function tables is available for FEQ processing of the branch tables.

ERR:198 CROSS SEC. TAB. NUM.= nn NOT FOUND FOR ELEVATION.
Same as *ERR:197 but for the bottom-profile elevation.

ERR:199 EXPECTED nn BRANCHES BUT FOUND nn IN GETIC FILE: aa
A Get-Initial-Conditions file has been specified (section 13.1), but the number of branches in the file does not match the number of branches in the model. Model simulation cannot continue because the initial condition is invalid. An incorrect file was probably specified.

ERR:200 EXPECTED nn EXTERIOR NODES BUT FOUND nn IN GETIC FILE: aa
A Get-Initial-Conditions file has been specified (section 13.1), but the number of flow-path end nodes in the file does not match the number of flow-path end nodes in the model. Model simulation cannot continue because the initial condition is invalid. An incorrect file was probably specified.

ERR:201 EXPECTED nn NODES ON BRANCHES BUT FOUND nn IN GETIC FILE: aa
A Get-Initial-Conditions file has been specified (section 13.1), but the number of nodes on branches in the file does not match the number of nodes on branches in the model. Model simulation cannot continue because the initial condition is invalid. An incorrect file was probably specified.

ERR:202 UNABLE TO CONTINUE BECAUSE MODEL IN GETIC FILE DIFFERS IN SIZE FROM CURRENT MODEL.
A difference in size between the schematization of the stream system used to define the initial conditions and the schematization of the stream system for simulation has been detected. Only three values are checked so that gross errors are detected. Only time-dependent data should vary in a Get-Initial-Conditions file (section 13.1) used for the starting values in the model simulation.

ERR:203 GET INITIAL CONDITIONS OPTION: GETIC IS MISSING. ADD AFTER OPTION: SQREPS.
In version 6.6 and later versions of FEQ, the GETIC option is required even if no file is specified. The string “GETIC=” must be added after SQREPS to obtain the default behavior. The PUTFC option must be added after GETIC (see error 204). “Run Control Block” (section 13.1) provides additional details.

ERR:204 PUT FINAL CONDITIONS OPTION: PUTFC IS MISSING. ADD AFTER OPTION: GETIC.
In version 6.6 and later versions of FEQ, the PUTFC option is required even if no file is specified. The string “PUTFC=” must be added after “GETIC=”. “Run Control Block” (section 13.1) provides additional details.

ERR:205 BRANCH= nn OUT OF RANGE.
A branch number greater than 999 has been detected. Branches are limited to numbers no greater than 999.

ERR:206 BRANCH= nn HAS UNDEFINED INTERNAL VALUE. BRANCH ID IS INVALID.
A branch number has been detected without a defined internal value. A branch number may have been referenced in the Network-Matrix Control Block (section 13.6) or in later input that no longer is available. A branch number must be in the Branch Description Block (section 13.2).

ERR:207 EXTERIOR NODE NUMBER= nn OUT OF RANGE.
A flow-path end node number under the previous input style (ISTYLE=OLD) is outside the interval 1 to 1998.

ERR:208 PREFIX= a IS INVALID.
A node identifier prefix other than U, D, or F has been detected.

ERR:209 NODE= aa HAS AN UNDEFINED INTERNAL VALUE. NODE ID IS INVALID.
A node identifier should have an internal value, but no value has been detected. If the prefix is U or D, the branch number in the identifier must be in the Branch Description Block (section 13.2). If the prefix is F, the identifier must appear in the Network-Matrix Control Block (section 13.6). If the identifier is in the previous input style (ISTYLE=OLD), the value must appear in the Branch-Exterior Node Block (section 13.4) or in the Network-Matrix Control Block. All nodes are defined at the completion of the Network-Matrix Control Block.

ERR:210 CONVERSION ERROR IN EXTERIOR NODE FIELD: aa
A flow-path end node field with a conversion error has been detected. Only the first character of the flow-path end node identifier in the new style may be non-numeric. In the previous style (ISTYLE=OLD), the entire field must be numeric.

ERR:211 NUMBER= nn OUT OF RANGE FOR EXTERIOR NODE. PREFIX= a
A current input style (ISTYLE=NEW) flow-path end node identifier with a numeric portion outside the range of 1 to 999 has been detected.

ERR:212 BRANCH NUMBER= nn DOES NOT EXIST. PREFIX= a FOR THE NODE ID.
A flow-path end node identifier has been detected with a prefix of U or D, but the identifier has no internal value. The branch number is missing from the Branch Description Block (section 13.2).

ERR:213 NODE= aa MAKES NODE COUNT > INPUT VALUE OF: nn
More flow-path end nodes than originally specified in the Run Control Block (section 13.1) are detected. Either there are more nodes than originally thought or there may be a reference to a node in the Network-Matrix Control Block (section 13.6) that is not in the model of the stream system. Any reference to a flow-path end node in the Network-Matrix Control Block that has not yet been read is considered a new node. Thus, a typographical input error could result in a new node not in the model of the stream system. If the node identifier starts with a U or a D, then an error message will be issued if the branch number for the node is not available. However, if the prefix for the node identifier is F or if there is no prefix, then the node will be considered new.

ERR:214 CONVERSION ERROR IN: aa
A field is detected in the Network-Matrix Control Block (section 13.6) that cannot be converted to a number. The field should not contain a flow-path end node identifier. Probable error is that a flow-path end node identifier has been placed in the wrong field of the input.

ERR:215 BRANCH NUMBER = nn <= 0 INVALID.
A negative branch number has been detected in the Network-Matrix Control Block (section 13.6). Negative numbers are invalid.

ERR:216 EXPECTED BNODE= aa BUT FOUND BNODE= aa IN GETIC FILE:aa
A beginning node has been detected in the initial-conditions file (section 13.1) that differs from the beginning node for the construction of the matrix in the input. The initial-conditions file is used only if a previous run of FEQ is applied as the initial conditions for a subsequent simulation. Disagreement in the beginning nodes indicates that an improper initial-conditions file has been specified.

ERR:217 DEPTH WHEN WX= 1 >= DEPTH WHEN WX=0.5.
In the Near-Zero-Depth Block (section 13.5), the depth input for application of a weight of 1 (0.98 in Version 7.0 and later) is larger than the depth input for application of a weight of 0.5. The depth for application of the centered approximation must always be greater than the depth for application of the off-center approximation.

ERR:218 INTERPOLATION REQUEST:',aa,' INVALID. MUST BE
linear, LINEAR, cubic, CUBIC, OR A BLANK.
An invalid interpolation request has been detected in the Near-Zero-Depth Block (section 13.5).

ERR:219 TOO MANY RELATIONS IN JUNCTION WITH NODES: aa
A junction connecting *m* flow-path end nodes can only have *m* specified relations. One of these relations is always a code 2, sum of flows is zero. Thus, $m-1$ other relations must be specified. More than this number has been detected in simulation.

ERR:220 NODE= aa NOT FOUND IN JUNCTION. NODE2=',aa
NODES AT JUNCTION ARE: nn nn . . .
A relation involving two nodes has been detected, but the two nodes are in different junctions. This is invalid; all relations must include nodes in the same junction.

ERR:221 NODE= aa NOT IN A JUNCTION OR BOUNDARY CONDITION.
Every flow-path end node simulated in the model must be part of a junction—that is, in a Code 2 relation—or it must be an external boundary condition. A flow-path end node in neither category has been detected in the Network-Matrix Control Block (section 13.6).

ERR:222 nn RELATIONS MISSING FOR JUNCTION WITH NODES: aa
For a junction connecting *m* flow-path end nodes, there must be $m-1$ relations in addition to the Code 2 condition defining the junction. A junction with one or more missing relations has been detected.

ERR:223 DUPLICATION OF RELATIONS INVOLVING NODES: aa AND aa
A duplicate relation has been detected including the two flow-path end nodes. Only one relation is possible between two flow-path end nodes.

ERR:224 RELATIONS WITH CODE= nn AND NODES aa AND aa NOT IN A JUNCTION.
A relation in the Network-Matrix Control Block (section 13.6) not belonging to a junction has been detected. Neither of the nodes in this relation are in a junction. A possible cause is failure to delete entries from the Network-Matrix Control Block when one or more branches have been deleted from the model.

ERR:225 NODE= aa, A FLOW NODE, IS USED IN A RELATION OTHER THAN THE ONE DEFINING IT AS A FLOW NODE.
A flow node cannot appear as a node in any relation other than the one defining it as a flow node. This error was not detected in previous versions of FEQ. The losses through a control structure are changed by this error because the flow through the structure is changed by the choice of the flow node.

ERR:226 MODEL REQUIRES nn OR MORE MATRIX BLOCKS BUT ONLY nn AVAILABLE.
Matrix blocks are used to divide the solution matrix into parts for efficient computation. More blocks are required for this stream system than are currently allocated in the compilation of FEQ. The value of MRMBLK in the INCLUDE file ARSIZE.PRM (appendix 3) must be increased, and FEQ must be recompiled and linked.

ERR:227 PENDING INSTRUCTION BUFFER OVERFLOW. NUMBER OF RECORDS= nn
Space is allocated in FEQ for the pending instructions when the pattern of equations in the solution matrix is developed. Space for this buffer is too small. This error usually results for very large models. The size of MLPDA must be increased in the INCLUDE file ARSIZE.PRM (appendix 3), and FEQ must be recompiled and linked.

ERR:228 BRANCH NUMBER= nn IS A DUPLICATE NUMBER.
A duplicate branch number has been detected in the Branch Description Block (section 13.2).

ERR:229 NUMBER OF MATRIX ELEMENTS REQUIRED= nn EXCEEDS NUMBER AVAILABLE= nn
The solution matrix for the current model is larger than the allocated space. The size of MLPDA in the INCLUDE file ARSIZE.PRM (appendix 3) must be increased, and FEQ must be recompiled and linked.

ERR:230 NO VALID BOUNDARY POINT EXISTS. MUST ADD DUMMY BOUNDARY TO MODEL.
Not all boundary conditions are valid for a beginning node. Boundaries at which elevation as a function of time is specified are invalid. If all boundary points are of this type, then a dummy boundary point of some other type must be added so that a valid starting point is available for simulation. Care must be taken because most models with all boundary points given as elevation as a function of time are inherently unstable.

ERR:231 NODE= aa NOT ON A BOUNDARY.
A beginning node has been detected that is not on a boundary. If the BNODE field (section 13.6) is blank and this message is issued, then a bug in the program is indicated. Otherwise, the beginning node given is not a boundary node. The beginning node must be a boundary node.

ERR:232 BEGINNING NODE INPUT MISSING. ADD BEFORE SPECIAL OUTPUT HEADING.
In FEQ Version 7.0, beginning node input must be given immediately after the completion of the Network-Matrix Control Block (section 13.6). The minimum input, starting in column 1, is BNODE=; that is, the beginning node field is left blank. “Network Matrix Control Block” (section 13.6) provides further details.

ERR:233 A CROSS SECTION FUNCTION TABLE AT ABRUPT EXPANSION BETWEEN NODES aa AND aa HAS WRONG TYPE.
For simulation of an abrupt expansion the first moment of area must be tabulated in the cross-section table at both the upstream and downstream nodes. The table given does not include this hydraulic characteristic. The table type must be 21, 22, 24, or 25.

ERR:234 A CROSS SECTION FUNCTION TABLE FOR CODE nn BETWEEN NODES nn AND nn HAS WRONG TYPE.
For application of the listed code, the first moment of area must be in the cross-section table. The type given does not include this hydraulic characteristic. Valid table types are 21, 22, 24, and 25.

ERR:235 OUTLET LOSS COEF. NOT 1.0 AT ZERO SUBMERGENCE IN TABLE# nn
The outlet-loss coefficient table for a variable-speed, variable-head pump must always be 1.0 at zero submergence of the outlet. This is required to maintain a continuous relation with head losses when the outlet conduit is not submerged.

ERR:236 OUTLET AREA FOR PUMP ff <= 0 INVALID.
User error in providing the outlet area. The area must always be positive.

ERR:237 BLOCK TYPE aa INVALID. MUST BE GATE OR PUMP.
The spelling for the Operation of Control Structures Block (section 13.12) must be checked. Only two types of Operation of Control Structures Blocks are permitted.

ERR:238 VARIABLE NULL ZONE FOR PUMP OPERATION INVALID.
A variable null zone cannot be used with pump control.

ERR:239 MAXIMUM GATE OPENING = ff <= 0 INVALID.
A maximum gate opening ≤ 0 has been detected. This is invalid. A correct value must always be positive.

ERR:240 IN TABLE#= nn OF TYPE 15 AT ARGUMENT= ff
2-D TABLE#= nn IS INVALID
A table number that is zero or negative has been detected in the body of a table of type 15. All table numbers must be greater than zero.

ERR:241 IN TABLE#= nn OF TYPE 15 AT ARGUMENT= ff
2-D TABLE#= nn IS MISSING
A table referenced in the body of a type 15 table cannot be found in the table system of FEQ. Either the table was not input or an error prevented completion of input.

ERR:242 IN TABLE#= nn OF TYPE 15 AT ARGUMENT= ff
2-D TABLE#= nn IS TYPE nn. TYPE MUST BE 6 OR 13.
All table numbers in type 15 used to represent a function with three arguments must be of type 6 or type 13. A table in the body of the type 15 table has been detected that is the wrong type.

ERR:243 MONTH=nn IS INVALID.
An invalid number for a month in a date has been detected.

ERR:244 DAY=nn IS INVALID FOR MONTH=nn
The given day does not occur in the month (and year if the month is February).

ERR:245 TYPE=nn AND NODES: aaaa NOT FOUND IN NETWORK-MATRIX CONTROL WITH CODE=5.
The entry for the code 5 control structure with the given TYPE and nodes cannot be found in the Network-Matrix Control Block (section 13.6). The request for computing initial conditions in the Backwater Analysis Block (section 13.15) should be checked.

ERR:246 FLOW AT FLOW NODE FOR CODE 5 TYPE 6 IS ZERO. UNABLE TO CONTINUE.
The flow at the flow node of the control structure must be nonzero to determine an initial condition for the control structure with the steady-flow initial-condition computations. If the flow at the flow node is zero, then the elevations are not uniquely defined in the flow tables.

ERR:247 AT DEPTH=ff AND FLOW=ff, RATE OF CHANGE OF FLOW WITH RESPECT TO UPSTREAM HEAD IS ZERO FOR CODE 5 TYPE 6.
CANNOT CONTINUE.
The initial elevations for the control structure currently being processed cannot be defined because no change in flow for a change in upstream head is computed for the control structure. There is no unique inversion of the flow rating for the structure in this case.

ERR:248 CODE FIELD MUST BE NON-ZERO IF BRA FIELD IS ZERO OR BLANK.
An invalid combination has been detected in the input for the Backwater Analysis Block (section 13.15). The error must be corrected and FEQ must be run again.

ERR:249 CODE=nn INVALID OPTION.
The given code is not supported in the Backwater Analysis Block (section 13.15).

ERR:250 BRANCH=nn PREVIOUSLY PROCESSED.
A branch-processing request has been encountered for the second time in the Backwater Analysis Block (section 13.15).

ERR:251 BOTTOM PROFILE ELEVATION MISSING AT FIRST NODE ON BRANCH:nn.
The first node on a branch must always be assigned an explicit elevation because interpolation is not possible at the first node on the branch.

ERR:252 BOTTOM PROFILE ELEVATION MISSING AT LAST NODE ON BRANCH:nn.
The last node on a branch must always be assigned an explicit elevation because interpolation is not possible at the last node on the branch.

ERR:253 YC <= 0 AT NODE: nn ON BRANCH: nn.
The distance from the invert of a storm sewer to the ground surface must always be positive. This error may result from the depth being placed in the improper position. If the first node on a branch has a nonzero value for YC, then the value is not YC but the elevation of the ground surface, ZC. The first node on a branch must be assigned a blank in the field for YC, and the first valid location for a YC value is the second node on the branch.

ERR:254 UPSTREAM NODE=aa FOR CODE nn IS NOT THE DOWNSTREAM NODE ON ITS BRANCH.
The upstream node must be the downstream node on the branch attached to the upstream end of the flow path described in the given code. The nodes must be reordered, and FEQ must be run again.

ERR:255 DOWNSTREAM NODE=aa FOR CODE nn IS NOT THE UPSTREAM NODE ON ITS BRANCH
The downstream node must be an upstream node on the branch attached to the downstream end of the flow path described in the given code. The nodes must be reordered, and FEQ must be run again.

ERR:256 LOSS COEFFICIENT FOR MECHANICAL ENERGY LOSS IS INVALID. MUST BE >=0 AND <=1.0.
The loss coefficient is a factor applied to the change in velocity head across the diversion of water described in code 13. The coefficient must be in the range shown and should be small enough so that problems with critical depth do not result.

ERR:257 GATE OR PUMP NAME=aa ALREADY IN USE.
Names given for a pump or gate must not be duplicated for any other pump or gate. A duplicate name has been detected. The duplication must be corrected, and FEQ must be run again.

ERR:258 GATE OPENING=ff > MAXIMUM=ff IN TABLE#=nn
A gate opening has been requested in a table of type 15 that exceeds the maximum gate opening in the table. Computations must stop. The largest gate opening in the table of type 15 must be at least as large as the maximum gate opening set for the underflow gate.

WRN:NN MESSAGES

Various conditions that may or may not cause problems in the computations are detected in FEQ. The user is made aware of these conditions so that corrective action, if required, can be taken.

WRN:01 EXTRAPOLATION NOT DONE. TOP WIDTH < 0. EXT = ff

An extrapolation request for a cross section results in a negative top width. The extrapolation is not done.

WRN:02 X-SECTION BELOW RANGE IN XLOOKY

TABLE NUMBER	= nn
STATION NUMBER	= ff
TIME	= ff
DEPTH	= ff

Depth input in subroutine XLOOKY is less than the minimum depth available in the cross section. This message is not issued if the CUTOFF option for cross sections has not been used. Use of the CUTOFF option defines an elevation in the cross-section table below which information in the cross-section table is discarded. If this warning is issued, the user should consider whether the CUTOFF option should be used.

WRN:03 X-SECTION ABOVE RANGE IN XLOOKY

TABLE NUMBER	= nn
STATION NUMBER	= ff
TIME	= ff
DEPTH	= ff

Depth input in subroutine XLOOKY is greater than the maximum depth in the cross section. If this warning is issued only a few times and if the resulting maximum depth at the conclusion of the run is less than the maximum, no action is required. However, if the warning is issued many times and the run terminates prematurely or the maximum depth at the conclusion of the run is the same as the maximum depth in the cross section, then either the cross section must be extended by use of the EXT option or additional data must be obtained to further define the cross section and recompute the table with FEQUTL (Franz and Melching, in press).

WRN:04 HU > HMAX IN TWO-D TABLE# = nn HU = ff HMAX = ff

The upstream head for a two-dimensional table of Type = 6 is greater than the maximum upstream head stored in the table. The maximum stored upstream head is used and computations continue in FEQ. If this message is issued many times and if the limitation on maximum upstream head appears to have affected the flow, then the table must be extended so that the message no longer is issued.

WRN:05 ABRUPT EXPANSION CANNOT ALLOW REVERSE FLOW AT TIME = ff WITH FLOW = ff UPS NODE nn, AND DNS NODE nn

The current implementation for an abrupt expansion cannot allow reverse flow, but such a flow has been simulated. The flow is forced to be zero and computations continue in FEQ.

WRN:06 INVALID CRITICAL DEPTH FACTOR AT UPS NODE = nn IN ABRUPT EXPANSION. DEPTH = ff

The computation of critical flow when the correction coefficient for momentum flux is included is not always possible. An invalid condition has been found at an upstream (UPS) node, but the computations will continue in FEQ if possible. This error will not result if the momentum-flux correction factor is a constant.

WRN:07 INVALID CRITICAL DEPTH FACTOR AT DNS NODE = nn IN ABRUPT EXPANSION. DEPTH = ff

The computation of critical flow when the correction coefficient for momentum flux is included is not always possible. An invalid condition has been found at a downstream (DNS) node, but the computations will continue in FEQ if possible. This error will not result if the momentum-flux correction factor is a constant.

WRN:08 EXP-CON WITH FLOW NODE = nn CANNOT ALLOW REVERSE FLOW AT TIME = ff WITH FLOW = ff AT UPS DEPTH = ff AND DNS DEPTH = ff

The computed flow in an expansion-contraction indicates reversal when the physical conditions are such that a reversal is impossible. The reverse flow is forced to zero, and computations continue in FEQ.

WRN:09 CONVEYANCE NONINCREASING AT DEPTH = ff

Conveyance should always be increasing with depth in an open channel with diverging walls. This message is issued if the drop in conveyance over the depth interval is less than 0.5 percent.

WRN:10 CRIT. FLOW NONINCREASING AT DEPTH = ff

Critical flow, computed without reference to the momentum-flux correction factor, is decreasing with an increase in depth. Computational problems may result in the region of decrease.

WRN:11 DROP OF ff PERCENT IN CONVEYANCE AT DEPTH = ff

Conveyance should always be increasing with depth in an open channel with diverging walls. This message is issued if the drop in conveyance is greater than 0.5 percent. Corrective action is to further subdivide the cross section so that the cross-section shape can be better represented by the hydraulic radius.

WRN:12 BOTTOM ELEVATION DIFFERENCE BETWEEN NODE nn, aa, AND NODE nn, aa, IS ff

The bottom-elevation difference may be correct but also may be the result of an error. If the difference is large enough relative to the flow, problems with critical or supercritical flow will result in simulation. This message also is issued in some cases when a branch discharges into a level-pool reservoir. In general, reservoirs are best represented such that the depth and the elevation of the water surface are the same. In simulation, the bottom of the reservoir is at the elevation datum, which is usually far below the bottom elevation of the branch. This warning is issued for each such case, but it can be ignored because the discontinuity in bottom elevation between a reservoir and a branch has a smaller effect than for a discontinuity in the bottom elevations at a junction among branches.

WRN:13 ELEVATION RELATION PATTERN MAY CAUSE PROBLEMS.
RECAST SO THAT CODE 3 RELATIONS ARE ALL TIED TO A SINGLE NODE. NODES AT JUNCTION ARE: nn

The pattern used for code 3 relations at a junction was previously given in round-robin style. For example, assume four nodes at a junction, nodes 3, 4, 6, and 9. Three elevation relations are needed to complete the junction specification. In FEQ versions prior to 7.0, the nodes would be paired as: 3 and 4, 4 and 6, and 6 and 9; that is, in a pattern that keeps the difference between node numbers as small as possible. This pattern is not efficient and should not be used because the matrix is not always successfully patterned in FEQ. The patterning of the matrix may fail, and a BUG message may be issued that no relation can be found to terminate a path at a node in a junction. The specified node number is in the junction in question. Limited experience has shown that the problem disappears if the relations including code 3 at a junction are rearranged so that all reference one node. In the current example, a valid pairing would be 3 and 4, 3 and 6, and 3 and 9. Any other node could be specified as the base node. This warning only arises if there are four or more nodes at a junction. If there are less than four nodes at a junction, the round-robin style and the one-node reference style are the same.

WRN:14 HEAD DATUM= ff DISAGREES WITH TABLE#= nn HAVING HEAD DATUM= ff

If a head datum is given in a table, agreement between the head datum used to compute the table in FEQUTL (Franz and Melching, in press) and the head datum given in FEQ when the table is applied will be checked. The head datum applied in the simulation is the head datum as given in FEQ and not in the table.

WRN:15 nn ITERATIONS REQUESTED FOR SYSTEM SOLUTION BUT ONLY nn ALLOWED.
More iterations have been requested for the maximum allowed for system solution than have been provided for in ARSIZE.PRM (appendix 3). The parameter MNITER in the INCLUDE file ARSIZE.PRM must be changed, and the program must be recompiled and linked.

WRN:16 OLD FORM OF TRIB AREA INPUT IS DISCONTINUED. PLEASE CONVERT OLD FILES TO NEW FORM TO ESTABLISH COMPATIBILITY WITH VERSION
The input uses the previous input style for Tributary Area Block (section 13.3). This form is no longer supported. Complete details on the current input format are given in section 13.3.

WRN:17 CONTINUITY ERROR OF ff IN INITIAL CONDITIONS AT NODES:
nn, nn
An error in continuity of flow in the initial conditions has been detected at a junction including the listed flow-path end nodes. If this error is large enough, computational problems may result. The error can be corrected by changing the flow assignments in the backwater computations (section 13.15).

WRN:18 WS ELEV DIF BTWN NODE nn, aa, & NODE nn, aa, : ff
A difference in water-surface elevation at the listed flow-path end nodes has been detected. A large difference relative to the depths of water could result in computational problems. The error can be corrected by making appropriate adjustments in the backwater computations.

WRN:19 Not in use.

WRN:20 Not in use.

WRN:21 Not in use.

WRN:22 FLOW DISCONTINUITY AT CONSTANT FLOW BOUNDARY AT NODE nn, aa
The flow assigned to a branch or a free node in the backwater computations does not match the flow forced at the node by Code = 6.

WRN:23 Not in use.

WRN:24 NON-CONVERGENCE. TIME STEP WILL BE REDUCED.
The stream-system solution has not converged in the number of iterations allowed. The time step will be reduced according to user instructions, and computations will continue in FEQ. Eventually this process will be terminated either by convergence or by reduction of the time step to a value less than a user-specified minimum value.

WRN:25 DF = 0.0 IN CHKGEO
The value of the derivative in the single-step, steady-flow computation in CHKGEO is zero. The value is reset to 1.0, and computations continue in FEQ.

WRN:26 COMPUTATIONAL PROBLEMS LIKELY: CUTOFF ELEVATION = ff SHOULD BE LESS THAN ELEVATION FOR HEAD = ff
For code 5, type 2 in the Network-Matrix Control Block (section 13.6), the elevation at which the pump is turned on considering conditions at the destination node should be less than the elevation used for computing the head for the diversion of water into the offline reservoir. If the cutoff elevation is higher than the elevation for head, then the pump is turned on while water is diverted into the offline-storage reservoir. If the two elevations are equal, the pumping of the water could cause the water-surface elevation at the destination node to exceed the cutoff elevation. This feedback can

result in convergence problems because the pump is turned on and off at successive iterations so that simulation convergence cannot be achieved.

WRN:27 XKNT= HAS BEEN REPLACED BY PAGE=. PLEASE CHECK RUN CONTROL BLOCK INPUT DESCRIPTION.
The XKNT option has not been available since version 4.41 of FEQ. The space used by XKNT in the input has now been replaced by a page size specification in the input for the special-output file so that headings can be placed in the file for easier use. Complete details are given in sections 13.1 (Run Control Block description) and 13.9 (Special-Output Locations Block).

WRN:28 FIRST DEPTH INCREMENT IN TABLE IS LARGE.
The first depth increment in a table to which a bottom slot is to be added is larger than 0.1. NRZERO in FEQUTL (Franz and Melching, in press) should be checked. Large NRZERO values increase the effect of adding the bottom slot to the cross section.

WRN:29 BACKW NO LONGER USED. REPLACED BY GEQOPT.
The backwater option has not been available since version 2.0 of FEQ. It has remained in the input for consistency. This option has been replaced by GEQOPT to provide the option of applying different sets of governing equations. Further details are given in section 13.1.

WRN:30 UNKNOWN OPTION = aa FOR GEQOPT. STDX OPTION ASSUMED.
An unknown value for the selection of governing equations has been detected for GEQOPT (section 13.1). The STDX option is selected, and computations continue in FEQ. A discussion of the governing equations options is given in section 6.2.

WRN:31 X-SECTION BELOW RANGE IN XLKTAL
TABLE NUMBER = nn
STATION NUMBER = ff
TIME = ff
DEPTH = ff
A depth less than the minimum depth available in the cross section has been detected in subroutine XLKTAL. This message is not issued if the CUTOFF option (section 13.13) for cross sections has not been applied.

WRN:32 X-SECTION ABOVE RANGE IN XLKTAL
TABLE NUMBER = nn
STATION NUMBER = ff
TIME = ff
DEPTH = ff
A depth greater than the maximum depth in the cross section has been detected in subroutine XLKTAL. If this warning is issued only a few times and if the resulting maximum depth at the conclusion of the run is less than the maximum, no action is required. However, if the warning is issued many times and the run terminates prematurely or the maximum depth at the conclusion of the run is the same as the maximum depth in the cross section, then the cross section should either be extended by use of the EXT option or additional data should be obtained to further define the cross section and recompute the table with FEQUTL (Franz and Melching, in press).

WRN:33 X-SECTION BELOW RANGE IN XLKTnn

TABLE NUMBER	= nn
STATION NUMBER	= ff
TIME	= ff
DEPTH	= ff

A depth less than the minimum depth available in the cross section has been detected in subroutine XLKT*nn*. This message is not issued if the CUTOFF option (section 13.13) for cross sections has not been applied.

WRN:34 X-SECTION ABOVE RANGE IN XLKTnn

TABLE NUMBER	= nn
STATION NUMBER	= ff
TIME	= ff
DEPTH	= ff

A depth greater than the maximum depth in the cross section has been detected in subroutine XLKT*nn*. If this warning is issued only a few times and if the resulting maximum depth at the conclusion of the run is less than the maximum, no action is required. However, if the warning is issued many times and the run terminates prematurely or the maximum depth at the conclusion of the run is the same as the maximum depth in the cross section, then the cross section should be extended by use of the EXT option or additional data should be obtained to further define the cross section and recompute the table with FEQUTL (Franz and Melching, in press).

WRN:35 X-SECTION BELOW RANGE IN LKTQC

TABLE NUMBER	= nn
STATION NUMBER	= ff
TIME	= ff
DEPTH	= ff

A depth less than the minimum depth available in the cross section has been detected in subroutine LKTQC. This message is not issued if the CUTOFF option for cross sections has not been applied. The critical flow in type 12 tables (section 11.1.5) is looked up in subroutine LKTQC.

WRN:36 X-SECTION ABOVE RANGE IN LKTQC

TABLE NUMBER	= nn
STATION NUMBER	= ff
TIME	= ff
DEPTH	= ff

A depth greater than the maximum depth in the cross section has been detected in subroutine LKTQC. If this warning is issued only a few times and if the resulting maximum depth at the conclusion of the run is less than the maximum, no action is required. However, if the warning is issued many times and the run terminates prematurely or the maximum depth at the conclusion of the run is the same as the maximum depth in the cross section, then the cross section should be extended by use of the EXT option or additional data should be obtained to further define the cross section and recompute the table with FEQUTL (Franz and Melching, in press). The critical flow in type 12 tables (section 11.1.5) is looked up in subroutine LKTQC.

WRN:37 TABLE OVERFLOW AT EXT. NODE = nn ARGUMENT = ff

An argument at a reservoir node exceeding the maximum value in the table has been computed. To prevent exceeding the maximum level in the table, the arithmetic average of the last valid argument and the maximum argument in the table is computed as the argument value to use. If the message persists and computations fail, the table may have to be extended to higher values. In some cases, these values will be false, but they must be supplied in any case. Occasionally, a very large value of storage in the table must be used for determining the solution. If the maximum value resulting from convergence of the iterations is below the maximum of the table, then the solution is valid. However, if the maximum value simulated is higher than the maximum in the table, then some changes must be made in the flows, cross-section hydraulic characteristics, and related input.

WRN:38 TABLE OVERFLOW AT: nn:nn ARGUMENT = ff
Same as *WRN:37 except that the problem is at a node on a branch.

WRN:39 DEPTH AT NODE: nn:nn OVERTOPS TABLE# = nn BY ff
The maximum depth in a cross-section table is recalled before any requested extrapolation in the EXT option is done. Cross sections are checked to determine overtopping by more than EXTTOL (section 13.1). EXTTOL should be set so that excessive use of extrapolated information is avoided.

WRN:40 BRANCH NUMBER = 0 HAS BEEN FOUND MORE THAN ONCE IN TRIB AREA INPUT.
A branch number of 0 is used in the Tributary Area Block (section 13.3) to reference input of the areas tributary to reservoirs. All reservoirs with tributary area must have the area specified when the branch number is 0. A second branch number of 0 has been detected. An error may result in the branch number sequence.

WRN:41 X-SECTION BELOW RANGE IN LKTK
TABLE NUMBER = nn
STATION NUMBER = ff
TIME = ff
DEPTH = ff
A depth less than the minimum depth available in the cross section has been detected in subroutine LKTK. This message is not issued if the CUTOFF option for cross sections has not been applied. The conveyance in type 12 tables (section 11.1.5) is looked up in subroutine LKTK.

WRN:42 X-SECTION ABOVE RANGE IN LKTK
TABLE NUMBER = nn
STATION NUMBER = ff
TIME = ff
DEPTH = ff
A depth greater than the maximum depth in the cross section has been detected in subroutine LKTK. If this warning is issued only a few times and if the resulting maximum depth at the conclusion of the run is less than the maximum, no action is required. However, if the warning is issued many times and the run terminates prematurely or the maximum depth at the conclusion of the run is the same as the maximum depth in the cross section, then the cross section should be extended by use of the EXT option or additional data should be obtained to further define the cross section and recompute the table with FEQUTL (Franz and Melching, in press). The conveyance in type 12 tables (section 11.1.5) is looked up in subroutine LKTK.

WRN:43 FLOW > MAXFLOW IN TWO-D TABLE# = nn FLOW = ff MAXFLOW = ff
A flow argument greater than the tabulated maximum has been detected in a table with an argument of flow. The maximum value from the table will be used, and computations will continue in FEQ, if possible. If this message is issued infrequently and if none of the output values appear to be limited by the maximum flow in the table, then this warning probably results because of an intermediate value of the solution. If, however, this message appears frequently, it is probable that the results have been affected by the flow limitation in the table. Therefore, the table must be recomputed to a higher value of flow.

WRN:44 HDN > MAXHDN IN TWO-D TABLE# = nn HDN = ff MAXHDN = ff

A table has been accessed with a value of downstream piezometric head that is greater than the maximum tabulated value. The maximum table value will be used, and computations will continue in FEQ if possible. If this message is issued frequently, then it is likely that the limitation on table extent affected the results. If it is issued infrequently, it is likely a transient problem detected in the search for the solution at some point. The output results should be checked to determine whether the downstream head for this table was limited to the tabulated maximum. If there is a limitation or the message is issued frequently, then the table must be recomputed to a higher value.

WRN:45 START TIME LEADS TIME IN GETIC FILE BY ff SECONDS.

The start time of the run differs from the time at which the information in the Get-Initial-Conditions file (section 13.1) was written. Computations will continue, but the times do not match. If the time given is positive, then there is a gap between the time at which the initial conditions were written and the time when the run starts. Otherwise, an overlap of the simulated periods results between the Get-Initial-Conditions file and the current run.

WRN:46 TYPE 3 TABLE INTEGRAL RELATIVE ERROR >.02 TABLE#= nn ARGUMENT= ff COMPUTED INTEGRAL USED

The integral for type 3 tables is computed in version 6.6 or later of FEQ, and the computed value is compared to the value specified in the table. If the difference is greater than 0.02, this warning is issued. In any case, the computed value of the integral and not the tabulated value of the integral is used in FEQ.

WRN:47 NODE= nn < 0 INVALID. FEQ TAKES ABSOLUTE VALUE.

Negative node numbers are not permitted in the Network-Matrix Control Block (section 13.6) in Version 7.0 and later versions of FEQ. The absolute value of the node is taken, and the processing of the model continues.

WRN:48 FOR NODE= aa INPUT SIGN DISAGREES WITH INTERNAL SIGN. INTERNAL SIGN USED.

The correct node sign is assigned to all free nodes in Version 7.0 and later versions of FEQ. The node sign input in the Free-Node Initial Conditions Block (section 13.14) is still retained and is checked. However, the internal sign based on the context of the free node in the Network-Matrix Control Block (section 13.6) is applied.

WRN:49 TWO-D TABLE OVERFLOWS HAVE OCCURRED

During the course of computations, a lookup in a 2-D table that goes above the maximum arguments will sometimes be attempted. This may just be an intermediate result in the iterative solution (the final solution is below the maximum argument values). However, in some cases, the tables are not large enough to contain the flows of interest. It is often difficult to distinguish the two cases from the warning messages issued as the computations progress. Therefore, at the conclusion of computations, the 2-D tables are checked against the extreme values computed. Two-dimensional tables with an inadequate argument range are listed in a table following this message. If the table is of type 6 or 13, the overflow is with respect to head; and if the table is of type 14, the overflow is with respect to flow and perhaps head. The entries in the warning table are filled as appropriate. When a 2-D table argument is exceeded, the maximum value in the table is used in the computations. No extrapolation of values is done. The table overflow can seriously distort computational results because table overflows yield results similar to obstructions to the flow. Thus, the water-surface elevations upstream from the location of a structure for which the table values are exceeded become too large. In general, the tabled values should be extended until this message is no longer issued.

WRN:50 FLOW DIRECTION AND HEAD DIFFERENCE INCONSISTENT. ADJUSTING UPSTREAM WATER LEVEL AND ATTEMPTING TO CONTINUE.

This message may be issued during computation of the initial conditions at a control structure represented with the code 5, type 6 option in the Network-Matrix Control Block (section 13.6). Nothing needs to be done if the computations converge. If the computations do not converge, then the reasons for the discrepancy between head difference and flow direction must be resolved. Perhaps the flow as assigned is in the wrong direction, or perhaps the control structure is reversed.

WRN:51 YE 2(I) <=0.0 IN CMPCOR AT EXNODE = aa

This message indicates that the standard method for correction of an estimated depth at the flow-path end node resulted in a nonpositive corrected depth. This is most likely an intermediate result in the solution process. However, if the solution does not converge, then the flow-path end node given and neighboring nodes in the model should be reviewed as potential sources for the cause of the nonconvergence.

BUG:nn MESSAGES

Code designed to detect mistakes in the program is contained in FEQ. Normally, these messages will not be issued, but there is always the chance that some programming change has not been properly made and the erroneous program code was not tested. A BUG message almost always requires a change in the program code. If a BUG message is encountered, please contact Linsley, Kraeger Associates, Ltd., Mountain View, California.