COMPUTER PROGRAM

HYDRAUX

A MODEL FOR SIMULATING ONE-DIMENSIONAL, UNSTEADY, OPEN-CHANNEL FLOW

By Lewis L. DeLong and David H. Schoellhamer

U.S. GEOLOGICAL SURVEY

WATER-RESOURCES INVESTIGATIONS REPORT 88-4226

Reston, Virginia
1989

DEPARTMENT OF THE INTERIOR

DONALD PAUL HODEL, Secretary

U.S. GEOLOGICAL SURVEY

Dallas L. Peck, Director

For additional information
write to:

Chief, Office of Surface Water
U.S. Geological Survey
415 National Center
12201 Sunrise Valley Drive
Reston, Virginia 22092

Copies of this report can be
purchased from:

U.S. Geological Survey
Books and Open-File Reports
Federal Center, Building 810
P.O. Box 25425
Denver, Colorado 80225

CONTENTS

Abstract ---------- 1
Introduction ---------- 1
Governing equations ---------- 2
Description of the HYDRAUX flow model ---------- 3
  Numerical solution technique ---------- 3
    Finite-element formulation in space ---------- 3
    Finite-difference formulation in time ---------- 7
    Iterative solution ---------- 7
  Boundary conditions ---------- 9
Application of HYDRAUX ---------- 10
  Spacial and temporal discretization ---------- 12
  Simulating abrupt floods ---------- 13
  Simulating floods in meandering rivers ---------- 14
  Example 1--Tidal network ---------- 15
  Example 2--Upland flood ---------- 22
Summary ---------- 26
References ---------- 27
Appendices ---------- 29
  A. Preprocessors for HYDRAUX ---------- 29
      XHYDRP ---------- 30
      INTRPH ---------- 31
      EXPAND ---------- 32
      BNDRAW ---------- 33
  B. The CNTRL and CNTRLE files ---------- 35
  C. The BOUND file ---------- 41
  D. The BNDCON file ---------- 43
  E. Input and output for Example 1 ---------- 45
      CNTRL input file ---------- 45
      XHY.DAT input file ---------- 46
      BOUND input file ---------- 47
      PRT output file ---------- 48
  F. Input and output for Example 2 ---------- 53
      CNTRL input file ---------- 53
      XHD.DAT input file ---------- 54
      PRT output file ---------- 55

TABLES

Table 1. Files used with HYDRAUX and related programs- 10
2. HYDRAUX and related programs ---------- 11
3. Hydraulic variables for example 1 ---------- 17

FIGURES

Figure 1. Graph showing Hermitian shape functions----- 5

2. Schematic representation of a branched network, example 1-------------------------- 16

3. Graph showing simulated discharge at the downstream end of branches 3 and 4, example 1---------------------------------- 21

4. Graph showing model results at 50,000 ft using 10,000-ft cross-sectional spacing, example 2---------------------------------- 23

5. Graph showing model results at 50,000 ft using 5,000-ft and 10,000-ft cross-sectional spacing, example 2-------- 25

CONVERSION FACTORS

For use of readers who prefer to use International System (SI) units, conversion factors for terms used in this report are listed below:

Multiply	By	To obtain
foot (ft)	0.3048	meter (m)
square foot (ft^2)	0.0929	square meter (m^2)
cubic foot per second (ft^3/s)	0.02832	cubic meter per second (m^3/s)

COMPUTER PROGRAM HYDRAUX, A MODEL FOR SIMULATING ONE-DIMENSIONAL, UNSTEADY, OPEN-CHANNEL FLOW

By Lewis L. DeLong and David H. Schoellhamer

ABSTRACT

HYDRAUX is a FORTRAN program for numerically solving a form of the unsteady, one-dimensional, open-channel flow equations extended for flow simulation in meandering channels with flood plains. An orthogonal-collocation, finite-element method is used to solve the governing equations. HYDRAUX is particularly adept at simulating abrupt floods, such as those commonly associated with dam breaks and debris flows and is capable of simulating flows in complex networks of channels.

INTRODUCTION

HYDRAUX is a FORTRAN program for numerically solving the unsteady, one-dimensional, open-channel flow equations. It solves a complete form of the one-dimensional shallow-water flow equations, extended to allow for complex cross sections and changes in effective channel lengths that occur when meandering rivers inundate adjacent floodplains (DeLong, 1985). The model has been used successfully to simulate the extremely abrupt floods and debris flows associated with volcanic activity (Laenen and Hansen, 1988), potential moraine-dam failures (Laenen and others, 1987; 1988), and is capable of simulating flow in branched networks of interconnected channels. The flow equations are solved by a method combining finite elements in space and finite differences in time. The finite elements use Hermitian interpolation, and the finite differences are weighted in time. The equations are solved iteratively, in terms of incremental changes in the dependent variables.

The purpose of this report is to document governing equations, numerical methods, and data required for the HYDRAUX flow model, and to demonstrate, through two simple examples, the general use and characteristics of the model.

Manuscript approved for publication December 9, 1988

GOVERNING EQUATIONS

Equations describing one-dimensional, unsteady flow in rigid open channels may be written (DeLong, 1985)

$$\frac{\partial}{\partial t}(AM_a) + \frac{\partial Q}{\partial x} - q = 0, \tag{1}$$

and

$$\frac{\partial}{\partial t}(QM_q) + \frac{\partial}{\partial x}\left(\frac{\beta Q^2}{A}\right) + gA\left(\frac{\partial Y}{\partial x}\Big|_A + \frac{\partial Y}{\partial A}\Big|_x \frac{\partial A}{\partial x} + \frac{Q|Q|}{K^2}\right) = 0, \tag{2}$$

in which t = time,
A = cross-sectional area,
M_a = area-weighted sinuosity coefficient,
Q = volumetric discharge,
x = distance in a reference coordinate unchanging in time,
q = lateral inflow per unit of x coordinate length,
M_q = discharge-weighted sinuosity coefficient,
β = momentum coefficient,
g = acceleration due to gravity,
Y = distance of water surface above a given datum, and
K = total conveyance.

The area-weighted sinuosity coefficient (M_a) may vary both with depth of flow and distance and is defined by

$$M_a = \frac{1}{A}\int_A m\, dA\ , \tag{3}$$

in which, for the increment of cross-sectional area (dA), m is the ratio of channel length (s) to reference length (x), expressed as

$$m = \frac{ds}{dx}\ . \tag{4}$$

When the reference coordinate is aligned with the main-valley length, M_a may be thought of as sinuosity. If M_a is exactly 1, equations 1 and 2 reduce to the more commonly observed form of the one-dimensional flow equations (Strelkoff, 1969; Cunge and others, 1980).

The discharge-weighted sinuosity coefficient (M_q) also may vary with depth and distance and is defined by

$$M_q = \frac{1}{Q}\int_Q m\, dQ\ , \tag{5}$$

in which dQ is an increment of discharge corresponding to the area, dA.

The momentum coefficient (β) is defined as

$$\beta = \frac{\int_A v^2 dA}{V^2 A} , \tag{6}$$

in which v = velocity and V = mean velocity in the cross section.

In the use of equations 1 and 2 it is assumed that: (1) flow is one dimensional to the extent that the momentum coefficient can sufficiently account for non-uniform velocity distribution, (2) streamline curvature and vertical accelerations are negligible, (3) effects of turbulence and friction are adequately described by the resistance laws used for steady flow, and (4) the channel slope is sufficiently mild so that the cosine of its angle with the horizontal is close to unity. To the extent that steady flow resistance laws and the momentum coefficient can approximate hydraulic properties during unsteady flow conditions, hydraulic properties of complex cross sections can be represented by allowing momentum coefficients, conveyance, and sinuosity (M_a and M_q) to vary appropriately with depth of flow.

DESCRIPTION OF THE HYDRAUX FLOW MODEL

HYDRAUX is written in standard FORTRAN (American National Standards Institute, 1978) in compliance with the coding convention established by the Office of Surface Water, Water Resources Division, U.S. Geological Survey (J. L. Kittle, AquaTerra Consultants, written commun., 1988). Other than in the addition of gradients as unknowns, the solution technique is similar to the four-point-implicit, finite-difference scheme used in other models such as BRANCH (Schaffranek and others, 1981) and DAMBRK (Fread, 1984). In fact, if linear interpolation had been used rather than Hermitian (cubic) interpolation, this approach would result in working equations and numerical characteristics similar to the four-point-implicit finite-difference scheme.

Numerical Solution Technique

Finite-Element Formulation in Space

Equations 1 and 2, in general, are not amenable to analytical solution. The orthogonal-collocation, finite-element method used herein to solve the governing equations is described in this section. A general description of the numerical technique is in texts such as Lapidus and Pinder (1982) and an application to one-dimensional transport (similar in some aspects to the method presented here) is made by Pinder and Shapiro (1979).

The stream is divided longitudinally into discrete sections or reaches referred to as elements. Dependent variables are approximated within each element by Hermite polynomials in terms of values of the dependent variables and spacial gradients of the dependent variables at the nodes located at the the ends of each element. To accommodate gradients in addition to nongradient values as unknowns, nongradient values typically occupy odd-numbered nodes and odd-numbered positions of program arrays, and gradient values typically occupy even-numbered nodes and even-numbered positions of program arrays. For example, a single channel with 20 cross sections will require 40 nodes. Discharge at cross sections 1 and 2 would correspond to nodes 1 and 3 or positions 1 and 3 of the discharge array. Gradients of discharge with respect to channel reference length would correspond to nodes 2 and 4 or positions 2 and 4 of the discharge array.

A local coordinate ξ may be defined such that over an element ξ varies exactly from -1 to 1. In local coordinate ξ, the approximation for a typical variable, such as cross-sectional area (A), may be written

$$A(\xi,t) \cong \hat{A}(\xi,t) = \sum_{j=-1,1} \left\{ N_{0j}(\xi)\; A_j(t) + N_{1j}(\xi)\; \frac{\partial A_j(t)}{\partial \xi} \right\}, \qquad (7)$$

where the subscript j refers to location in local coordinates. The Hermite polynomials (N) shown in figure 1 are

$$N_{0,-1} = \frac{1}{4}\,(\xi - 1)^2\,(\xi + 2), \qquad (8a)$$

$$N_{0,1} = -\frac{1}{4}\,(\xi + 1)^2\,(\xi - 2), \qquad (8b)$$

$$N_{1,-1} = \frac{1}{4}\,(\xi + 1)\,(\xi - 1)^2, \qquad (8c)$$

and $$N_{1,1} = \frac{1}{4}\,(\xi - 1)\,(\xi + 1)^2. \qquad (8d)$$

In global coordinate x, this approximation becomes

$$\hat{A}(x,t) = \sum_{j=-1,1} \left\{ N_{0j}(\xi)\; A_j(t) + N_{1j}(\xi)\; \frac{dx_j}{d\xi}\,\frac{\partial A_j(t)}{\partial x} \right\}. \qquad (9)$$

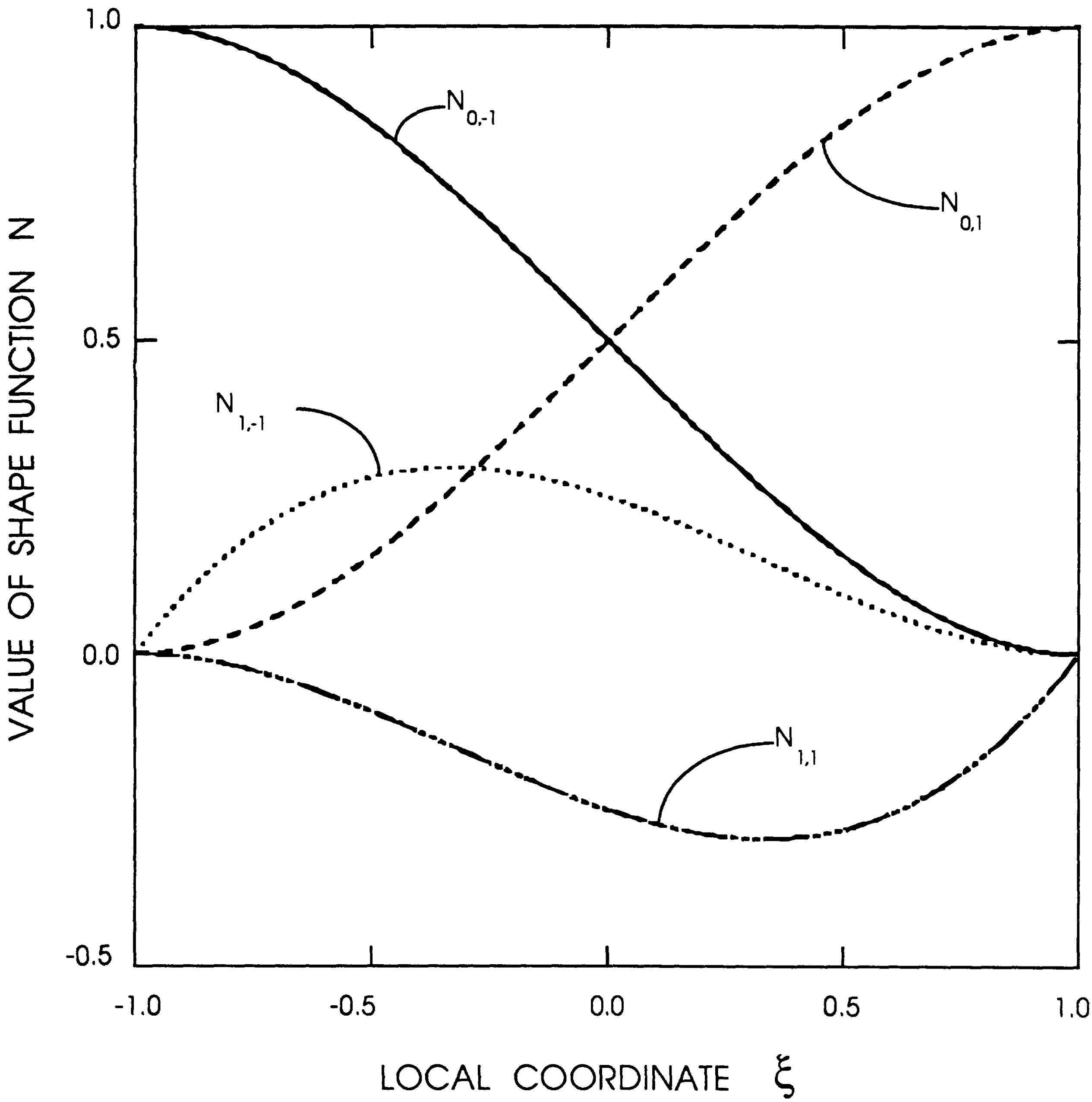

Figure 1. Hermitian shape functions.

The spacial derivative of a typical dependent variable is approximated as

$$\frac{\partial \hat{A}}{\partial x}(x,t) = \frac{d\xi(\xi)}{dx} \sum_{j=-1,1} \left\{ \frac{\partial N}{\partial \xi}_{0j}(\xi)\; A_j(t) + \frac{\partial N}{\partial \xi}_{1j}(\xi)\; \frac{dx_j}{d\xi}\, \frac{\partial A_j}{\partial x}(t) \right\}. \quad (10)$$

The collocation method is a special case of the more general method of weighted residuals which requires that over the domain of the solution

$$\int R_1\; w(x)\; dx = 0, \quad (11)$$

$$\int R_2\; w(x)\; dx = 0, \quad (12)$$

where the residuals (R_1 and R_2) are

$$R_1 = \frac{\partial}{\partial t}\,(M_a\; \hat{A}) + \frac{\partial}{\partial x}\,\hat{Q} - q \quad , \quad (13)$$

and

$$R_2 = \frac{\partial}{\partial t}\,(M_q\; \hat{Q}) + \frac{\partial}{\partial x}\left(\frac{\beta \hat{Q}^2}{\hat{A}}\right) + g\,\hat{A}\left(\left(\frac{\partial Y}{\partial x}\right)_A + \left(\frac{\partial Y}{\partial A}\right)_x \frac{\partial \hat{A}}{\partial x} + \frac{\hat{Q}|\hat{Q}|}{K^2}\right) . \quad (14)$$

In the collocation method, the arbitrary weighting function (w) is a unit-impulse or Dirac-delta function. Properties of this function (Lapidus and Pinder, 1982) reduce the above integrals to the integrands

$$R_1 = 0 \ , \quad (15)$$

and $R_2 = 0$, (16)

evaluated at the collocation points (k). In the orthogonal-collocation method, the collocation points are located at quadrature points normally associated with numerical integration. Use of Hermite polynomials results in two unknowns at each node for each dependent variable and, thus, requires two collocation points within each element. In the method presented, collocation points are optionally located at Gaussian quadrature points (±0.577350269) or Lobatto quadrature points (± 1.0 and 0.0). Use of Lobatto quadrature points is often more efficient; however, as later shown in example 2, use of Gaussian quadrature points may result in superior numerical properties.

Finite-Difference Formulation in Time

Time discretization of equations 15 and 16 is accomplished through a time-weighted finite-difference scheme resulting in

$$\left(\frac{M_a\ \hat{A}}{\Delta t}\right)^+ - \left(\frac{M_a\ \hat{A}}{\Delta t}\right)^- + \theta\ \frac{\partial \hat{Q}^+}{\partial x} + (1 - \theta)\ \frac{\partial \hat{Q}^-}{\partial x} - q = 0 \quad , \tag{17}$$

and

$$\left(\frac{M_q\ \hat{Q}}{\Delta t}\right)^+ - \left(\frac{M_q\ \hat{Q}}{\Delta t}\right)^- + \theta\left(\frac{\partial}{\partial x}\left(\frac{\beta \hat{Q}^2}{\hat{A}}\right) + g\hat{A}\ \left(\left(\frac{\partial Y}{\partial x}\right)_A + \left(\frac{\partial Y}{\partial A}\right)_x \frac{\partial \hat{A}}{\partial x} + \frac{\hat{Q}|\hat{Q}|}{K^2}\right)\right)^+$$

$$+ (1 - \theta)\left(\frac{\partial}{\partial x}\left(\frac{\beta \hat{Q}^2}{\hat{A}}\right) + g\hat{A}\ \left(\left(\frac{\partial Y}{\partial x}\right)_A + \left(\frac{\partial Y}{\partial A}\right)_x \frac{\partial \hat{A}}{\partial x} + \frac{\hat{Q}|\hat{Q}|}{K^2}\right)\right)^- = 0 \ , \tag{18}$$

where θ is a time weighting factor, the superscript "+" indicates evaluation at the new or unknown time, and "-" indicates evaluation at the old or known time.

Iterative Solution

Equations 17 and 18 are solved iteratively in terms of the incremental change in the dependent variables. Equation 17 is written as

$$\left(\frac{M_a}{\Delta t}\right)^* \Delta A + \theta\,\Delta\left(\frac{\partial Q}{\partial x}\right)^* = q + \left(\frac{M_a\ A}{\Delta t}\right)^- - (1-\theta)\frac{\partial Q^-}{\partial x} - \left(\frac{M_a\ A}{\Delta t} + \theta\frac{\partial Q}{\partial x}\right)^* \ , \tag{19}$$

where the superscript "*" indicates evaluation at the new or unknown time level approximated from current values obtained from the preceding iteration. The superscript "^" has been dropped in this and all following equations to simplify notation.

Two of the nonlinear terms in equation 18 are approximated by truncated Taylor series, expanded on the dependent variables A, $\frac{\partial A}{\partial x}$, Q, and $\frac{\partial Q}{\partial x}$, resulting in

$$\left(A\frac{Q|Q|}{K^2}\right)^+ = \left(A\frac{Q|Q|}{K^2}\right)^* + \left(\frac{Q|Q|}{K^2}\left(1-2\frac{A}{K}\frac{\partial K}{\partial A}\right)\right)^*\Delta A + 2\left(\frac{A|Q|}{K^2}\right)^*\Delta Q, \qquad (20)$$

and

$$\frac{\partial}{\partial x}\left(\frac{\beta Q^2}{A}\right)^+ = \frac{\partial}{\partial x}\left(\frac{\beta Q^2}{A}\right)^*$$
$$+\left(\frac{Q}{A}\right)^*\left(\frac{Q}{A}\left(\frac{\partial A}{\partial x}\left(\frac{2\beta}{A}-\frac{\partial\beta}{\partial A}\right)-\frac{\partial\beta}{\partial x}\right)+\left(2\frac{\partial Q}{\partial x}\left(\frac{\partial\beta}{\partial A}-\frac{\beta}{A}\right)+Q\frac{\partial}{\partial x}\left(\frac{\partial\beta}{\partial A}\right)\right)\right)^*\Delta A$$
$$-\left(\frac{\beta Q^2}{A^2}\right)^*\Delta\left(\frac{\partial A}{\partial x}\right)+\frac{2}{A^*}\left(Q\frac{\partial\beta}{\partial x}+\beta\frac{\partial Q}{\partial x}-\frac{\beta Q}{A}\frac{\partial A}{\partial x}\right)^*\Delta Q$$
$$+\left(\frac{2\beta Q}{A}\right)^*\Delta\left(\frac{\partial Q}{\partial x}\right). \qquad (21)$$

Substituting equations 20 and 21 into equation 18 and rearranging terms results in

$$\theta C_1^*\,\Delta A + \theta C_2^*\,\Delta\left(\frac{\partial A}{\partial x}\right) + \left(\frac{M_q}{\Delta t}+\theta\,C_3\right)^*\Delta Q + \theta\,C_4^*\,\Delta\left(\frac{\partial Q}{\partial x}\right)$$
$$= -\frac{M_q^*Q^*}{\Delta t} - \theta\,F^* + \frac{M_q^-Q^-}{\Delta t} - (1-\theta)\,F^-, \qquad (22)$$

$$F = \frac{Q}{A}\left(Q\frac{\partial\beta}{\partial x}+2\beta\frac{\partial Q}{\partial x}-\frac{\beta Q}{A}\frac{\partial A}{\partial x}\right)+gA\left(\frac{Q|Q|}{K_2}+\left(\frac{\partial Y}{\partial x}\right)_A+\left(\frac{\partial Y}{\partial A}\right)_x\frac{\partial A}{\partial x}\right), \qquad (23)$$

and the coefficients

$$C_1 = \frac{Q}{A}\left(2 \frac{\partial Q}{\partial x}\left(\frac{\partial \beta}{\partial A} - \frac{\beta}{A} \right) + Q \frac{\partial}{\partial A}\frac{\partial \beta}{\partial x}\right) + \frac{Q^2}{A^2}\left(\frac{\partial A}{\partial x}\left(\frac{2\beta}{A} - \frac{\partial \beta}{\partial A}\right) - \frac{\partial \beta}{\partial x}\right)$$
$$+ g \left(\left(\frac{\partial Y}{\partial x}\right)_A + \frac{Q|Q|}{K_2}\left(1 - 2 \frac{A}{K} \frac{\partial K}{\partial A}\right) \right), \tag{24a}$$

$$C_2 = - \beta \frac{Q^2}{A^2} + g A \left(\frac{\partial Y}{\partial A}\right)_x , \tag{24b}$$

$$C_3 = 2 \left(\frac{1}{A}\left(Q \frac{\partial \beta}{\partial x} + \beta \frac{\partial Q}{\partial x} - \beta \frac{Q}{A}\frac{\partial A}{\partial x} \right) + g A \frac{|Q|}{K^2} \right), \tag{24c}$$

and

$$C_4 = \frac{2\beta Q}{A} . \tag{24d}$$

Boundary Conditions

Equations 19 and 22 are evaluated at each collocation point. For a single channel of N elements, there are (4)(N + 1) degrees of freedom. The use of collocation points located at Lobatto quadrature points results in (2)(N + 1) + (2)(N) equations, requiring two external boundary conditions--one at each end of the channel for subcritical flow, or two at the inflow end of the channel for supercritical flow. The boundary conditions may be known values of the dependent variables. The use of collocation points located at Gaussian quadrature points results in (2)(2)(N) equations requiring two conditions in addition to the known boundary values. These conditions are supplied through additional equations (eqs. 19 and 22, or an algebraic combination) evaluated at the channel end nodes. In branched or looped networks, interior boundary conditions are obtained from appropriate constraint equations enforcing continuity of mass and equal water-surface elevations. Other relations such as those describing flow through breaches and gates and over dams and spillways have been adapted from Fread (1984) and may be used in place of equations constraining water-surface elevations (see appendices B and D).

APPLICATION OF HYDRAUX

For ease of processing, input to HYDRAUX is separated into individual files according to the type of input. Most of the files may be created by other programs used as pre-processing programs to HYDRAUX. Coding instructions for input are contained in appendices to this report. The sequential creation and use of these files is summarized in table 1 and a brief description of the programs used is contained in table 2. Only three of the files in table 1 (XHD.DAT, CNTRL, and BOUND) are created directly by the user.

Table 1.--Files used by HYDRAUX and related programs.

Input Files	Program	Output Files
XHD.DAT*	XHYDRP	PRT,CXH,DRMX
CXH	INTRPH**	DRM,CXA,ASEG,ERROR
CNTRL*,DRM,CXA, DRME,CXE,ASEGE,CNTRLE, ASEG,	EXPAND	PRT
ASEGE,CNTRLE	BNDRAW**	BNDCON,BND.SCH,ERROR, BND.DAT
DRME,CXE,ASEGE, CNTRLE, BNDCON, BOUND*	HYDRAUX	PRT,PUN,PLT

* created directly by user.

** require graphics routines.

Table 2.--HYDRAUX and related programs.

Program	Description
XHYDRP	Computes geometry and hydraulic properties from station-elevation input file, XHD.DAT.
INTRPH	Interpolates XHYDRP output and tabulates properties in a format suitable for EXPAND. (Presently requires DISSPLA[1] graphics library.)
EXPAND	Computes and tabulates gradients of cross-sectional properties, and inserts additional computational cross sections as desired.
BNDRAW	Creates and/or modifies BNDCON file that describes connections among channel branches and types of boundary conditions applied. (Presently requires DISSPLA graphics library.)
HYDRAUX	Solves the one-dimensional open-channel flow equations.

[1]Trade names are for identification purposes only and do not constitute endorsement by the U.S. Geological Survey.

Spacial and Temporal Discretization

As with most flow models, choice of distance between cross sections, length of time steps, and their ratio will affect results. Primarily, the user should select spacial increments that adequately define channel characteristics such as volume and resistance to flow. Time increments should be selected to adequately represent time-series data in both input (boundary conditions) and output.

Secondly, spacial and temporal increments should be selected that will cause the approximate numerical solution to converge to the true solution of the flow equations for a given application. This can be ascertained by reducing time and distance increments in successive executions of the model and comparing results. The approximate numerical solution (model result) will have converged to the true solution to the extent of the difference between successive results. Obvious indications of the lack of convergence might include oscillations in dependent variables with respect to space and/or time. A test to ascertain convergence should be an integral part of an attempt to simulate flow and should be planned for accordingly. Finer spacial discretization can easily be obtained through the use of EXPAND, and the time step may be reduced in input to HYDRAUX.

Simulating Abrupt Floods

HYDRAUX is particularly adept (in comparison to the four-point-implicit scheme) at simulating abrupt hydrographs such as those that result from dam breaks or mudflows often associated with volcanic activity. Often large floods (large in comparison to floods normally occurring in a particular channel) will reach supercritical velocities along parts of the channel simulated. It is suggested that when supercritical flows are expected that Gaussian quadrature points be selected on input (see appendix A). While the governing flow equations do not apply under conditions where vertical acceleration and water-surface curvature are significant, these conditions generally do not persist over a significant portion of the channel length. For example, a hydraulic jump may occur over tens of feet while the distance between model cross sections is on the order of hundreds of feet. The model would conserve mass and momentum over the distance between cross sections, but would not be expected to correctly depict conditions at the jump. Similarly, the model may be driven by known discharge at the upstream boundary when flow at that boundary changes from subcritical to supercritical. Hydraulic theory requires the specification of two upstream boundary conditions when the flow is supercritical at that boundary. In practice, other information such as stage or cross-sectional area may not be known, and the model will generally continue to function without it. In most cases, results at a few cross sections downstream from the boundary are not affected because the volume of flow introduced at the boundary is correctly determined by the known discharge, and errors in momentum introduced at the boundary are often insignificant to the effects of channel slope and flow resistance within the first few cross sections. However, results (Froude number, stage, and cross-sectional area) within the first few cross sections may not be meaningful.

Simulating Floods in Meandering Rivers

Simulation of unsteady flows in natural rivers is often complicated by significant variation in hydraulic properties and effective lengths of wetted channels as rivers inundate flood plains. The use of constant channel lengths in models to represent base and flood flows may introduce errors in travel times and timing of discharge peaks. For example, McDonald and Sanders (1984) found it necessary to reduce invariant channel lengths in a model by as much as 41 percent during flood flows to match timing of observed discharge peaks. Reducing errors in simulations of flood flows by this method, however, increases errors in base-flow simulations. Fread (1976, 1984) modified the governing flow equations to account for effective shortening of channels at flood stages but timing errors persisted (DeLong, 1987), caused by a failure of Fread's method to conserve mass. The governing equations solved in HYDRAUX have been shown to conserve mass (DeLong, 1988). HYDRAUX allows the user to vary effective channel lengths automatically during simulations by specifying sub-section sinuosities as a function of stage.

Example 1--Tidal Network

The application of HYDRAUX to the tidal network introduced by Jobson and Schoellhamer (1987) will be presented in this section. The upstream end of branch 1 of the network, shown in figure 2, has a constant riverine inflow of 1,059 ft^3/s (cubic feet per second), branch 2 is a dead-end canal, branches 3 and 4 split around an island, and the downstream ends of branches 5 and 6 are tidal boundaries with varying stage. The distance downstream from the upstream end of the branch, width, Manning's friction coefficient, initial discharge, initial water-surface elevation, and initial cross-sectional area at each of the 18 cross sections in the simulation are listed in table 3. All the channels are rectangular with a bed elevation of 31.53 ft. The water-surface elevations at the tidal boundaries of branches 5 and 6 are given by

$$Y = 38.642 + 0.984 \sin\left(\frac{2\pi(t - t_1)}{24}\right) , \qquad (25)$$

in which Y is the water-surface elevation in feet, t is the time in hours, and t_1 is the lag time in hours. The value of t_1 was assumed to be 1.0 and 0.0 for branches 5 and 6, respectively (Jobson and Schoellhamer, 1987).

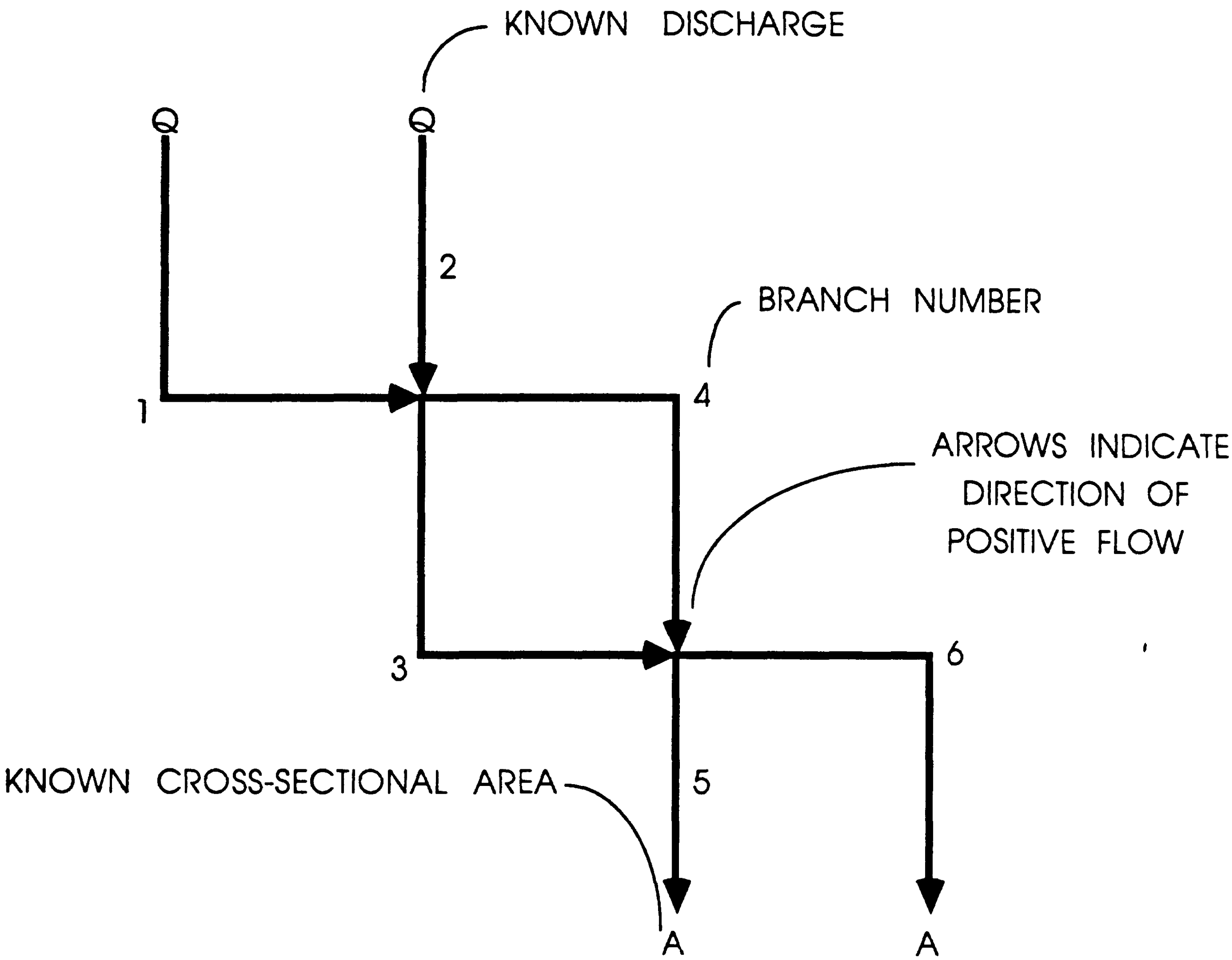

Figure 2. Branched network, example 1.

Table 3.--Hydraulic variables for example 1.

Branch	Cross section	Downstream distance (miles)	Width (feet)	Manning's n	Initial discharge (cubic feet per second)	Initial surface elevation (feet)	Initial cross-sectional area (square feet)
1	1	0.000	528	0.025	1059	38.62	3816
	2	0.994	538	0.025	1038	38.62	3816
	3	1.988	538	0.025	1017	38.62	3816
2	1	0.000	197	0.025	0.00	38.62	1396
	2	4.660	197	0.025	-35.31	38.62	1396
3	1	0.000	312	0.0374	720.4	38.62	2216
	2	1.553	344	0.0374	692.2	38.58	2423
	3	3.107	375	0.0374	642.7	38.55	2633
	4	4.660	406	0.0374	568.6	38.55	2852
	5	6.214	437	0.0374	459.1	38.55	3071
	6	7.767	469	0.0374	314.3	38.55	3291
4	1	0.000	118	0.035	264.9	38.62	838
	2	2.796	118	0.035	236.6	38.55	829
	3	5.592	118	0.035	180.1	38.55	829
5	1	0.000	164	0.045	512.1	38.55	1152
	2	3.418	164	0.045	346.1	38.39	1126
6	1	0.000	191	0.045	-17.66	38.55	1344
	2	3.418	191	0.045	-215.4	38.65	1363

The first step for applying the HYDRAUX model is to generate geometric and hydraulic properties with the program XHYDRP (W.H. Kirby, U.S. Geological Survey, written commun., 1983). Sample input and output files for example 1 are shown in Appendix E, and detailed instructions for data entry are available in a computer file accompanying the computer code and are not repeated here. Data input to the program for this example is only briefly described herein. The first line ($OPTS 1) of XHD.DAT, the input file to XHYDRP, requests that the input data be printed as echo output for the first cross section. The second line contains the branch number (optional, not used by XHYDRP), the number of miles the section is downstream from the upstream boundary of the branch in columns 4 through 8, and label information used in the output for branch 1, section 1. The succeeding lines are pairs of ground-point locations (station, elevation) in the cross section which are read in free-field format terminated with a slash (/). The next free field is the subarea-division stations of which there are none in this simulation. The next two free fields indicate Manning's friction coefficient and sinuosity (0.025 and 1.0 for branch 1, section 1). The final free field for branch 1, section 1 indicates that properties are to be calculated from elevation 35.0 to 45.0 at increments of 1.0 feet. The file XHD.DAT contains similar information for every other cross section in the example network, proceeding in the assumed downstream direction within each branch and sequentially by branch number within the file. By convention, positive discharge flows in the downstream direction.

The second step in this HYDRAUX application is to run the INTRPH preprocessor in order to interpolate and tabulate XHYDRP output and prepare the initial conditions. INTRPH fits a series of piece-wise Hermitian-cubic equations to the curves of properties versus cross-sectional area and tabulates corresponding equation coefficients.

The third step in this HYDRAUX application is to run the EXPAND preprocessor in order to compute and tabulate longitudinal gradients of cross-sectional properties and to insert additional cross sections as desired. EXPAND requires the user-written input file CNTRL described in Appendix B. The first two lines in the CNTRL file contain text information describing the simulation. The following lines specify 36 node points (twice the number of cross sections), 6 branches, output for transport modeling will not be performed, Lobatto collocation points will be used because they are more efficient for network applications, normal areas and channel fill will not be used (NORM = 0), and gradient values are not present in file CXA which is the case when the preprocessors are used. The next two free-format input fields give the minimum and maximum distances between interpolated cross sections in each branch. The selected numbers were set large enough to prevent the addition of interpolated cross sections in this simulation. The following parameters specify that printed output will be minimized (NSKPRT = 0), results will be printed every twelfth time step

(PRT = 0, PRTINC = 12), spacial-series results will not be written to the PUN file (PUN = 0, PUNINC = 9999), time-series results for every time step will be written to the PLT file (PLT = 0, PLTINC = 1), the time-weighting factor equals 0.6, 96 time steps will be run, simulation time will start at zero, the output units for time are hours (3600 seconds), the maximum number of iterations per time step is five, the allowable closure error for iterative flow calculations is 0.01 (1 percent), the three equation coefficients are zero (not used in this simulation), there are no dams in the system, and four boundary conditions will be read from the BOUND file during the first time step of 900 seconds followed by the reading of two boundary conditions at each of the next 95 time steps of 900 seconds.

After the EXPAND preprocessor has been run, the BNDRAW program is used to create the coded file (BNDCON). This file, required input to the flow model, describes external boundary conditions and connections among branches of the network. The BNDCON file is automatically created during execution of BNDRAW by graphically drawing the network and locating and specifying external boundary conditions. A BNDRAW schematic of this system, including branch numbers, assumed positive flow directions, and boundary-condition codes, is shown in figure 2.

The third and final input file prepared directly by the user is the BOUND file which contains time series of boundary values. Reading of this file during model execution is controlled by the variables NBC, BNDINC, and NEQ in the last lines of the CNTRL file as explained in appendix B. Appendix C describes the format of the BOUND file, and appendix D describes how boundary-condition codes contained in the BNDCON file control boundary conditions in the model using values read from the BOUND file.

NBC is initially 4. As a result, 4 lines are read from the BOUND file just prior to executing the first time step. Values read correspond to the starting time plus one time increment which has been set equal to 900.0 seconds. Boundary values remain constant until replaced by new values. In this example, new values will be read after one time step has been completed (BNDINC = 1, and NEQ = 0). The first 4 lines in conjunction with boundary codes contained in the BNDCON file will, at the end of time step 1, force discharge at node 1 to be equal to 1,059.44 ft^3/s, discharge at node 7 to be equal to 0.0 ft^3/s, area at node 31 to be equal to 1,125.019 ft^2, and area at node 35 to be equal to 1,361.198 ft^2. Note that as explained in appendix D, discharge values are entered in the BOUND file at the even-numbered node following the node to which the value is to be applied. (This allows discharge and either stage or area to be specified at the same node when a supercritical boundary is required.) Location of nodes can be seen in the schematic produced by BNDRAW and corresponding coordinate distances read from the DRME or PRT files.

Immediately following completion of the first time step, another control line is read from CNTRLE. On the second and subsequent readings of the BOUND file, NBC is equal to 2 causing only 2 values to be read each cycle. Because NEQ is equal to 2 and NBCINC is equal to 9,999, the BOUND file will be read each time step without reading any more control cards from CNTRLE until 9,999 time steps have expired or program execution ends.

Successful execution of the XHYDRP, INTRPH, EXPAND, and BNDRAW preprocessors is followed by execution of the HYDRAUX flow model to determine the time-varying hydraulic properties of the system. The printed output file PRT is shown in appendix E. The first section of PRT contains the input parameters in file CNTRL. This section is followed by tables of area, discharge, stage, velocity, and Froude number at every cross section. A table is printed when indicated by the print control variables PRT and PRTINC. For this example, a table was printed every twelfth time step (3 hours). The number of iterations needed to solve the flow equations, the maximum error in closure, and the node with the maximum error are also printed.

Reversing flows caused by the tidal boundaries extended into branches 3 and 4 some of the time. Stages are nearly level and Froude numbers are low. Figure 3 is a plot of discharge at the downstream boundaries of branches 3 and 4 that was generated with data stored in PLT. Most of the discharge around the island is seen to occur in branch 3. The initial ebb tide quickly reverses to a flood tide which reverses back to an ebb tide after six hours. The ebb tide remains throughout the rest of the simulation.

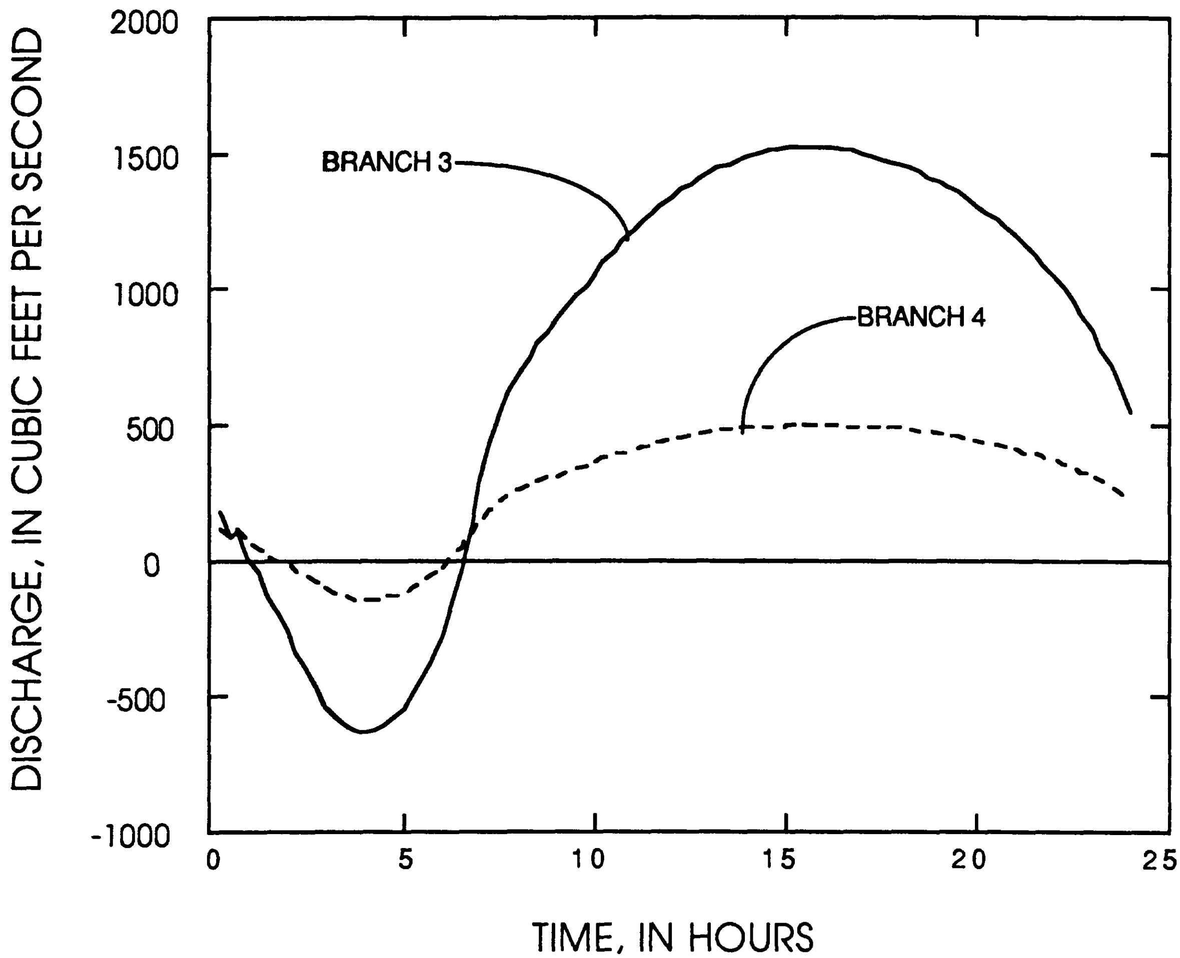

Figure 3. Simulated discharge at the downstream end of branches 3 and 4, example 1.

Example 2--Upland Flood

This hypothetical, single-channel example was previously used to test numerical algorithms (P.E. Smith, U.S. Geological Survey, written commun., 1981, and R.W. Schaffranek, U.S. Geological Survey, written commun., 1982) and is included here to provide a very simple example and to demonstrate a numerical problem common to numerical-solution schemes used in many models.

A flood hydrograph is introduced at the upstream end of a rectangular channel 70,000 ft long, 100 ft wide, with a slope of 0.001, and Manning's n of 0.045. Initially, flow is 250 ft^3/s and the depth of flow is 1.69 ft (cross-sectional area of 169 ft^2). The flood hydrograph assumed at the upstream end of the channel is described by

$$Q = 250 + \frac{477.465}{2}\left(1 - \cos\left(\pi \frac{t}{75}\right)\right) , \qquad (26)$$

where Q is discharge in cubic feet per second and t is time in seconds.

This upstream boundary condition is simulated using an equation-type boundary option by selecting NEQ equal to 1 (the node at which it is to be applied) for 30 time steps (300 seconds each) and setting the equation parameters

EQ1 = 250.0,

EQ2 = 477,465,

and EQ3 = 75.0,

in the CNTRL or CNTRLE files. These parameters control the base, amplitude, and period of a sine or cosine curve as explained in appendix B. The model uses the value computed from the equation as a discharge value because the boundary-condition code in the BNDCON file for the first cross section is set equal to 2 specifying a known discharge boundary. A self-setting downstream boundary is selected by setting the boundary-condition code for the node corresponding to the last cross section equal to 4.

Because the channel is prismatic, only the upstream and downstream cross sections need be coded for input to XHYDRP. Intervening cross sections may be interpolated and inserted by EXPAND. Initially, cross sections are placed at 10,000-ft intervals. This is accomplished by setting DXMIN and DXMAX equal to 10,000 ft in the CNTRL file prior to executing EXPAND.

Hydrographs at the upstream boundary and 50,000 ft downstream are shown in figure 4. Differences in the hydrographs at 50,000 ft are a result of location of collocation points. An oscillation occurs when Lobatto collocation points are used in combination

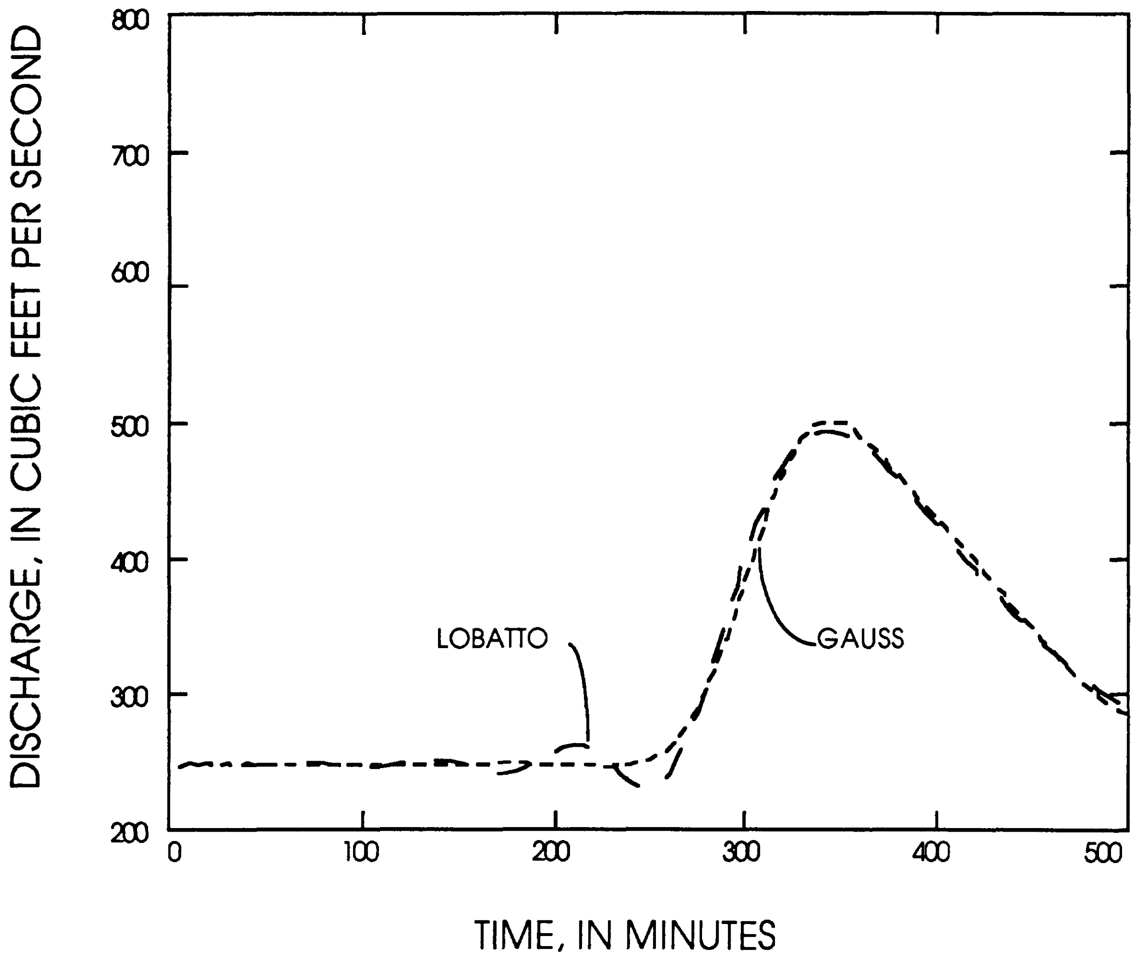

Figure 4. Model results at 50,000 ft using 10,000-ft cross-sectional spacing, example 2.

with a 10,000-ft cross-sectional spacing. This numerical artifact leads or travels faster than the flood peak and is sometimes referred to as leading-phase error. Leading-phase error present in other numerical algorithms such as the four-point-implicit scheme used in the BRANCH and DAMBRK one-dimensional unsteady flow models has been demonstrated with this example hydrograph by Smith and Schaffranek (P.E. Smith, U.S. Geological Survey, written commun., 1981; R.W. Schaffranek, U.S. Geological Survey, written commun., 1982). Use of Lobatto collocation points (by setting NFLOW equal to 1) results in more efficient solutions, especially when simulating large or complex networks of channels. However, use of Gauss collocation points (by setting NFLOW equal to 2) eliminates the leading-phase-error problem in the numerical scheme used in HYDRAUX. Absence of leading-phase error is especially advantageous when simulating abrupt hydrographs preceded by low flows such as often occurs with dam breaks, debris flows, and large floods. Oscillations leading the flood peak typically cause the computed water surface to dip below the channel bottom causing solutions to fail.

In this simple example it may be obvious that the oscillation leading the flood peak is a numerical artifact. In many cases however, numerical problems may not be as apparent. To assure that model results have converged to an acceptable solution of the underlying flow equations, cross-sectional spacing was reduced to 5,000 ft. Example model input and output with this spacing is contained in appendix F. As can be seen in figure 5, there is little difference between results obtained with 5,000-ft or 10,000-ft spacing in conjunction with Gauss collocation points. This demonstrates convergence with respect to the space. In practice, it would also be necessary to vary the time increment to assure convergence with respect to time. Notice also that at a cross-sectional spacing of 5,000 ft, the use of Lobatto and Gauss collocation points give virtually identical results .

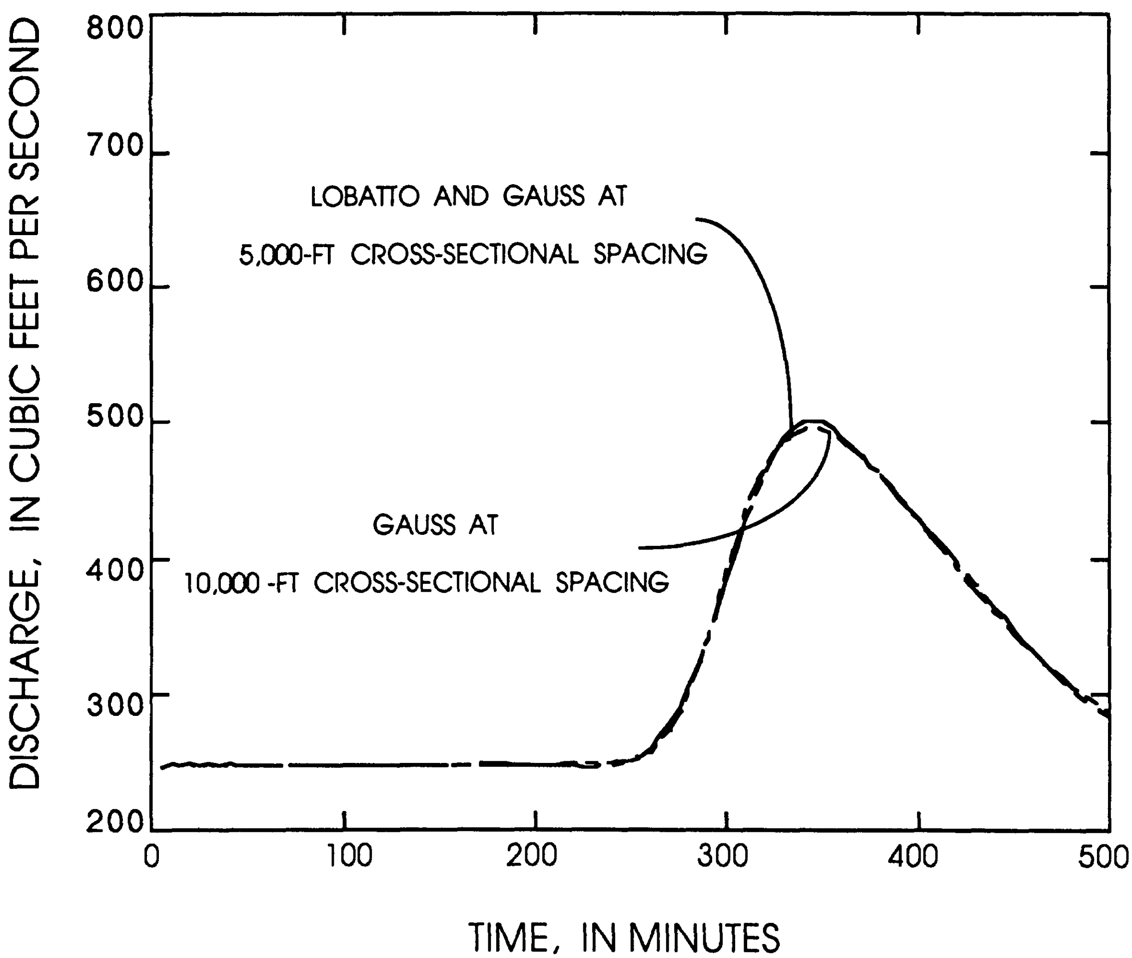

Figure 5. Model results at 50,000 ft using 5,000-ft and 10,000-ft cross-sectional spacing, example 2.

SUMMARY

The HYDRAUX flow model is governed by a form of the differential equations describing one-dimensional, unsteady, open-channel flow extended to account for overbank flows and channel lengths that vary with stage. The governing equations are solved numerically by an iterative method combining a finite-element-collocation scheme in space, using Hermitian interpolation, with a finite-difference scheme in time. Preparation of data input is simplified through the use of separate preprocessing programs. The model may be applied to complex networks of interconnected channels as is particularly useful in simulating abrupt floods, such as those resulting from dam breaks or debris flows.

REFERENCES

American National Standards Institute, 1978, American National Standard Programming Language FORTRAN: New York, 350 p.

Cunge, J.A., Holly, F.M., and Verwey, A., 1980, Practical aspects of computational river hydraulics: London, Pitman Publishing Ltd., 420 p.

DeLong, L.L., 1985, Extension of the unsteady one-dimensional open-channel flow equations for flow simulation in meandering channels with flood plains, in Selected papers in the hydrologic sciences, December 1985: U.S. Geological Survey Water-Supply Paper 2220, 101-105.

_____ 1989, Mass conservation: 1-D open-channel flow equations: Journal of Hydraulic Engineering, v. 115, no. 2, p. 263-269.

Fread, D. L.,1976, Flood routing in meandering rivers with flood plains, in Rivers 76, v. 1: 3d Annual Symposium on Inland Waterways for Navigation, Flood Control, and Water Diversions, Colorado State University, August 1976, 16-35 p.

_____ 1984, DAMBRK: The NWS Dam Break Flood Forecasting Model: Office of Hydrology, National Weather Service, Silver Spring, Md.

Jobson, H.E., and Schoellhamer, D.H., 1987, Users manual for a branched Lagrangian transport model: U.S. Geological Survey Water-Resources Investigations Report 87-4163, 73 p.

Laenen, Antonius, and Hansen, R.P, 1988, Simulation of three lahars in the Mount St. Helens Area, Washington, using a one-dimensional, unsteady state streamflow model: U.S. Geological Survey Water-Resources Investigations Report 88-4004, 20 p.

Laenen, Antonius, Scott, K.M., Costa J.E., and Orzol, L.L., 1987, Hydrologic hazards along Squaw Creek from a hypothetical failure of the glacial moraine impounding Carver Lake near Sisters, Oregon: U.S. Geological Survey Water-Resources Investigations Report 87-41, 45 p.

Laenen, Antonius, Scott, K.M., Costa J.E., and Orzol, L.L., [in press], Modeling flood flows from a hypothetical failure of the glacial moraine impounding Carver Lake near Sisters, Oregon, in Selected papers in the hydrologic sciences: U.S. Geological Survey Water-Supply Paper 2340.

Lapidus, Leon, and Pinder, G.F., 1982, Numerical solution of partial differential equations in science and engineering: New York, John Wiley, 677 p.

McDonald, B.B., and Sanders, C.L., 1984, Simulated flood discharges and elevations for the Savannah River, South Carolina and Georgia, using an unsteady streamflow model: U.S. Geological Survey Water-Resources Investigations Report 84-4158, 34 p.

Pinder, G. F., and Shapiro, Allen, 1979, A new collocation method for the solution of the convection-dominated transport equation: Water Resources Research, v. 15, no. 2, p. 1177-1182.

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, bk. 7, ch. C3, 110 p.

APPENDIX A

PREPROCESSORS FOR HYDRAUX

Preprocessor programs are available to help prepare the necessary input data for the HYDRAUX flow model. The preprocessor programs supplied with the HYDRAUX flow model are written in ANSI77 FORTRAN. Two of the programs, INTRP and BNDRAW, use graphics routines that presently require DISSPLA software. Table 1 lists the preprocessor programs and the HYDRAUX flow model in order of execution and the input and output files for the programs. XHD.DAT, CNTRL, and BOUND are the only three input files that must be prepared directly by the user. Three of the programs have an output file called PRT that contains information for the user. The remainder of this section will describe the preprocessor programs.

XHYDRP

XHYDRP computes geometric and hydraulic properties using station ground points in the file XHD.DAT (W.H. Kirby, U.S. Geological Survey, written commun., 1983). The user may specify up to 100 pairs of ground points, up to 19 ground points to delineate subareas in the cross section within which hydraulic properties are assumed to be laterally homogeneous, Manning's friction coefficient and sinuosity for each subarea, and the elevation increments at which the geometric and hydraulic properties are to be tabulated. The section option record and section identification record are fixed-format records, all other records are free format. The cross sections must be given, beginning with the assumed upstream end of each branch, and the branches must be ordered sequentially. For example, the first cross section listed will be the assumed upstream section in branch 1 and the last cross section will be the assumed downstream cross section in the last branch. This input file must be prepared carefully, because any geometric errors will be carried through the rest of the preprocessor programs and the HYDRAUX flow model.

INTRPH

INTRPH interpolates XHYDRP output, tabulates properties in a format suitable for the EXPAND preprocessor and HYDRAUX flow model, and prepares the initial condition file DRM. INTRPH is an interactive program that prompts the user for the desired task and, if necessary, the number of branches and the number of cross sections in each branch. The routine uses piece-wise cubic polynomials to interpolate and extrapolate, when needed, stage, conveyance, the momentum coefficient, and sinuosity coefficients. These properties are tabulated versus cross-sectional area. The range of cross-sectional area is common to all cross sections within a branch and is chosen by the user. To assist the user in selecting the range of area to be considered, the common range of area for all cross sections in the branch and the minimum and maximum areas in the branch are given to the user based on the output from XHYDRP. The user also selects the number (3 to 10) of knots (i.e., computational points) to be used for interpolation and extrapolation within the selected range of area. Each pair of knots bounds a separate cubic equation maintaining continuity of the first derivative between adjacent equations. More knots increase the accuracy of the interpolation and, subsequently, the size of cross-sectional properties files. The ratio of knot spacing, a multiplicative factor for the distance between knots, is also specified by the user. A ratio of one gives even spacing within the selected range of area, a ratio smaller than one causes knot spacing to decrease as area increases, and a ratio larger than one causes knot spacing to increase as area increases. In order to check whether the number of knots and the ratio of knot spacing is sufficiently accurate, the user may select an option to plot the interpolated and extrapolated curves over the data provided by XHYDRP. INTRPH also prepares the initial conditions file DRM. The initial conditions required are discharge and area for each cross section. The initial condition specification is also an interactive procedure.

EXPAND

The preprocessor program EXPAND computes and tabulates longitudinal gradients of cross-sectional properties and inserts additional cross sections as desired. EXPAND reads the user-supplied input file CNTRL which is described in table 3. This file contains the control information for the HYDRAUX simulation that is passed to HYDRAUX by the EXPAND output file CNTRLE. The number of nodal points in the simulation (variable NUMNP) is twice the number of user-supplied cross sections because HYDRAUX places two nodes at each cross section, one for the geometric and hydraulic properties and one for the longitudinal gradients of those properties. The minimum and maximum distances allowed between interpolated cross sections inserted by EXPAND are specified by the variable arrays DXMIN and DXMAX in CNTRL which are read by a free format. Insertion of interpolated cross sections can be prevented by specifying a sufficiently large DXMIN. User-supplied cross sections are never removed by EXPAND. Other information in CNTRL controls HYDRAUX computations and output.

 FOLLOWED BY P. 35.

APPENDIX B

The CNTRL and CNTRLE files

The CNTRL file is input to the preprocessing program EXPAND. The CNTRLE file is created by EXPAND and is very similar to CNTRL. The CNTRLE file is input to the 1-D flow model HYDRAUX. Consequently, control information contained in CNTRL and passed to CNTRLE may ultimately control the execution of HYDRAUX. Unless otherwise stated, all input described below is required, one parameter per line.

Parameter	Explanation	Format
MESAGE	Text -- user information, analysis title, etc.	80A1/80A1
NUMNP	Number of nodal points, 2x(total number of cross sections)	10X,I10
NBCH	Number of branches.	10X,I10
NTRANS	Control for transport computations. 0) No output for transport modeling. 1) Output files for Lagrangian transport. (not presently implemented, but input required)	10X,I10
NFLOW	Control for flow computations 1) Use Lobatto collocation points. 2) Use Gauss collocation points. NOTE: Supercritical flow generally requires Gauss points, Lobatto points are more efficient than Gauss points for network applications.	10X,I10
NORM	Control for start-up approximation When input to EXPAND (program for interpolating and inserting additional cross sections), non-zero integer causes approximate "normal areas" to be substituted for initial cross-sectional areas. When input to HYDRAUX (flow model) : Positive integer causes channel to be sufficiently filled to eliminate adverse surface slopes. Negative integer causes channel to be filled to maximum stage determined from initial conditions.	10X,I10

Parameter	Explanation	Format
INIT	Control for reading cross-sectional geometry and hydraulic properties file (CXA) : 1) Gradient of properties (with respect to channel distance) are contained in CXA. 2) Gradient values are not present in CXA.	10X,I10
DXM1N	Minimum distance, in feet, at which interpolated cross sections are inserted between user supplied cross sections.	*
DXMAX	Maximum distance, in feet, allowed between user supplied cross sections with out insertion of interpolated cross sections.	*
	* free field entry, spaces and commas "," are delimiters. NBCH values each for DXM1N and DXMAX are required. Terminate entry with a "/" .	
NSKPRT	Control for printed output: 0-9 (normally less than 2) Larger numbers create increasingly voluminous output useful mainly in debugging.	10X,I10
PRT	initial value of printed output counter in time steps.	10X,I10
PRTINC	Interval between printed output, in time steps	10X,I10
PUN	Initial value of counter controlling output to PUN file, in time steps.	10X,I10
PUNINC	Interval between output to PUN file: >0) Tabulate area and discharge and corresponding gradients in format identical to the initial conditions file DRME. <0) Tabulates area and discharge at cross sections and intervening collocation points.	10X,I10
PLT	Initial value of counter controlling output to PLT file, in time steps.	10X,I10

Parameter	Explanation	Format
PLTINC	Interval between output to PLT file, in time steps. >0) Tabulate stage, area, and discharge, versus time at stations listed in NSTA array. <0) Tabulate discharge, area, Courant number, hydraulic depth, and top width versus time at each cross section.	10X,I10
NUMSTA	Number of nodes at which values vs time output is desired, 40 maximum	10X,I10
NSTA	Array of node numbers for which results will be tabulated in PLT file. * free field entry, NUMSTA values required. Numbers are odd integers corresponding to nodes.	*
THETA	Time weighting factor: Value may range from >0.5 to 1.0 . Larger values increase numerical dampening. A value of 0.60 is often used.	10X,E10.2
MNI	Number of time steps to be executed.	10X,I10
TIME	Starting time. in TUNITs	10X,E10.2
TUNITS	Time units for output, in seconds	10X,E10.2
MNIU	Maximum number of iterations allowed per time step for execution of flow computations Value may range from 1 to 9; 5 is often used.	10X,I10
UERROR	Closure criteria for iterative flow computations. At the conclusion of each iteration a ratio for each variable is computed by dividing the increment of change in the variable by its current value. Iteration stops when each ratio is less than UERROR or when MNIU iterations have occurred.	10X,E10.2

NOTE: The next three parameters are used in conjunction with "equation" type boundary condition. (See "NEQ" index below.)

Parameter	Explanation	Format
EQ1	Base value.	10X,E10.2
EQ2	Magnitude from base to maximum value	10X,E10.2
EQ3	Time from base to maximum value, in TUNITs.	10X,E10.2
NDAMS	Number of dams to be simulated	10X,I10
	As many as 5 dams may be simulated. The following 15 coefficients should be supplied only if NDAMS > 0. (The basic equations used in this implementation are adapted from the National Weather Service dambreak forecasting model [FREAD, 1984].)	
TTB	Time of breach formation	10X,5E10.3
YD	Elevation of top of dam	10X,5E10.3
YBM	Final elevation of bottom of breach	10X,5E10.3
BT	Final breach width	10X,5E10.3
Z	Side slope of breach (horizontal/vert)	10X,5E10.3
YBRCH	Water-surface elevation initiating breach	10X,5E10.3
SC	Uncontrolled spillway discharge coefficient	10X,5E10.3
SL	Spillway length	10X,5E10.3
SH	Spillway crest elevation	10X,5E10.3
GC	Gate discharge coefficient	10X,5E10.3
GA	Gate area	10X,5E10.3
GH	Center-line elevation of gated spillway	10X,5E10.3
DC	Discharge coefficient for flow over dam	10X,5E10.3
DL	Length of dam less uncontrolled spillway	10X,5E10.3
DH	Dam crest elevation	10X,5E10.3

(this completes dam-related coefficients)

NOTE: The following "control" lines direct the reading of the BOUND file during flow model execution and, consequently, control boundary information used for flow computations. One "control" line is read initially when execution of the flow model begins, and one additional "control" line is read each time BNDINC time steps have been executed.

NBC BNDINC DT NEQ (2I10,F10.3,I10)

NBC Number of lines to be read from BOUND file each time information is read from BOUND file.

Parameter	Explanation	Format
BNDINC	Number of time steps for which parameters in current "control" line will apply.	
DT	Time step, in seconds.	
NEQ	Control index.	

0) Read information from BOUND file only once until next "control" line is read.

2) Read information from BOUND file each time step.

Odd integer) Apply "equation" type boundary condition at node specified by NEQ.

If NEQ > 0,

$$Q(NEQ) = EQ1 + \left(\frac{EQ2}{2}\right) \quad (1.0 - \cos\left(\pi \frac{TIME}{EQ3}\right)$$

If NEQ < 0,

$$Q(-NEQ) = EQ1 + \left(\frac{EQ2}{2}\right) \sin\left(\pi \frac{TIME}{EQ3}\right)$$

NOTE: When "equation" type boundary is selected, Q(NEQ) computed above will become either the DISCHARGE or AREA at node NEQ depending upon NCODE(NEQ) contained in BNDCON file.

page 41 follows

APPENDIX C

The BOUND file

The BOUND file contains boundary information used in the execution of HYDRAUX (flow model). Control instructions for reading this file to update boundary conditions during execution of HYDRAUX are supplied by the user at the end of the CNTRL file and are passed on to the CNTRLE file by EXPAND to be used during execution of HYDRAUX.

	Parameters	Format
	Q(I) QV(I) YW(I)	(I10,4E10.3)
I	Node number	
Q	Boundary value. The type of boundary represented depends on the value of NCODE in the BNDCON file (see appendix D).	
QV	Lateral inflow, in cfs per foot of channel, if I is positive or change in lateral inflow, in cfs per foot, per foot, if I is even.	
YW	Elevation at which overbank flow commences. This flow is simulated by simple weir flow and is lost from the channel. Total volume of water lost in this way is summarized in printed output file PRT. If YW is blank or "0", no weir flow will occur.	

page 43 follows

APPENDIX D

The BNDCON file

This file is created automatically by execution of BNDRAW. However a brief description is listed here to allow manual coding. The purpose of this file is to describe external boundary condition codes and connections among channel branches for a particular model application.

Parameters	Format
I NODE1(I) NODE2(I) NODE3(I) NCODE(I)	(I10,15X,3I5,I10)

I — Node to which NCODE(I) applies. Nodes referred to in this file will be located at the ends of channel branches.

NODE1(I) — First node to which node I is connected.

NODE2(I) — Second node to which node I is connected.

NODE3(I) — Third node to which node I is connected.

NCODE(I) — Boundary condition code. (Q(I) and Q(I+1) listed below are input from BOUND file). In the present implementation NCODE = 4 should be applied only at downstream boundaries sufficiently removed from any reach of interest (infinite boundary assumption). Upstream end of a channel branch is, by convention, the end from which positive discharge will flow in to the channel.

NCODE(I)	CONDITION
1)	Area at node I = Q(I).
2)	Discharge at node I = Q(I+1).
3)	Combination of 1) and 2).
4)	Ax(I) = (A(I)-A(I-1))/DX (if NFLOW = 1) Ax(I) = Qx(I) = 0.0 (if NFLOW = 2)

Parameters	Format
5)	Sum of discharges at nodes I, NODE1, NODE2, and NODE3 = Q(I+1). By convention, positive signs on NODE1, NODE2, and NODE3 indicate flow out of connection. Positive Q(I+1) is a source to the connection.
6)	Stage-discharge relation: AREA = Q(I)*DISCHARGE**Q(I+1)
7)	DA/DQ = dambreak function (Trapezoidal breach, enlarging with time. Basic equations are adapted from the National Weather Service dambreak forecasting model [Fread,1984])
8)	STAGE(I) = STAGE(NODE1)
9)	NULL (no condition)
10)	STAGE at node I = Q(I)
11)	AREA(I) = AREA(NODE1)

APPENDIX E

Input and output for example 1

[This is a sample of the input and output files. Complete machine-readable files are available on request]

CNTRL input file

```
HYDRAUX EXAMPLE NETWORK APPLICATION
FROM JOBSON AND SCHOELLHAMER, BLTM USERS MANUAL, WRI 87-4163
NUMNP     36
NBCH      6
NTRANS    0
NFLOW     1
NORM      0
INIT      2
88888.8 88888.8 88888.8 88888.8 88888.8 88888.8 /
99999.9 99999.9 99999.9 99999.9 99999.9 99999.9 /
NSKPRT    0
PRT       0
PRTINC    12
PUN       0
PUNINC    9999
PLT       0
PLTINC    1
NUMSTA    2
21 27 /
THETA     0.6
MNI       96
TIME      0.0
TUNITS    3600.0
MNIU      5
UERROR    0.01
EQ1       0.0
EQ2       0.0
EQ3       0.0
NDAMS     0
4         1         900.0     0
2         95        900.0     2
```

XHD.DAT input file

```
$OPTS 1
1  0.000 BRANCH 1, SECTION 1, RIVER DISCHARGE BOUNDARY
0.0 50.0
0.0 31.529
538.189 31.529
538.189 50.0 /
/
0.025 /
1.00 /
35.0 1.0 45.0 /
$OPTS 1
1  0.994 BRANCH 1, SECTION 2
0.0 50.0
0.0 31.529
538.189 31.529
538.189 50.0 /
/
0.025 /
1.00 /
35.0 1.0 45.0 /
$OPTS 1
        ........
       ( complete machine-readable files available on request )
       .........
$OPTS 1
6  0.000 BRANCH 6, SECTION 1
0.0 50.0
0.0 31.529
191.371 31.529
191.371 50.0 /
/
0.045 /
1.00 /
35.0 1.0 45.0 /
$OPTS 1
6  3.418 BRANCH 6, SECTION 2, TIDAL STAGE BOUNDARY
0.0 50.0
0.0 31.529
191.371 31.529
191.371 50.0 /
/
0.045 /
1.00 /
35.0 1.0 45.0 /
```

BOUND input file

```
 2  1059.44
 8  0.0
31  1125.019
35  1361.198
31  1135.309
35  1373.517
31  1145.733
35  1385.784
31  1156.248
35  1397.945
31  1166.808
35  1409.948
31  1177.368
35  1421.743
31  1187.882
35  1433.279
31  1198.307
35  1444.506
31  1208.596
35  1455.377
31  1218.707
35  1465.844
31  1228.595
35  1475.863
31  1238.219
35  1485.391
31  1247.537
35  1494.387
31  1256.509
35  1502.813
31  1265.098
35  1510.632
31  1273.265
35  1517.812
31  1280.976
35  1524.321
31  1288.199
35  1530.131
.............
.............
( complete machine-readable files available on request )
  ............
  ............
```

PRT file

```
HYDRAUX EXAMPLE NETWORK APPLICATION
FROM JOBSON AND SCHOELLHAMER, BLTM USERS MANUAL, WRI 87-4163
    NUMNP=        36
     NBCH=         6
   NTRANS=         0
    NFLOW=         1
     NORM=         0
     INIT=         1

      DXMIN, in feet---
     88888.8        88888.8        88888.8        88888.8
     88888.8        88888.8

       DXMAX, in feet---
     99999.9        99999.9        99999.9        99999.9
     99999.9        99999.9

   NSKPRT=         0
      PRT=         0
   PRTINC=        12
      PUN=         0
   PUNINC=      9999
      PLT=         0
   PLTINC=         1
   NUMSTA=         2
 NODES AT WHICH OUTPUT OF TIME SERIES DATA IS REQUESTED
      21      27

    THETA=      0.60
      MNI=        96
     TIME= 0.000E+00
    TUNIT= 0.360E+04
     MNIU=         5
   UERROR= 0.100E-01
      EQ1= 0.000E+00
      EQ2= 0.000E+00
      EQ3= 0.000E+00
```

INCREMENT= 12 TIME= 3.00
ITERATIONS FOR SOLUTION OF FLOW EQUATIONS = 3(last time step)
Largest ratio of change over the last iteration = 0.722E-03,
occurred in discharge at node 25

______Branch 1______

NODE	AREA	DISCHARGE	STAGE	VELOCITY	FROUDE NUMBER
1	0.402612E+04	0.105944E+04	39.01	0.26	0.02
3	0.402322E+04	0.916213E+03	39.00	0.23	0.01
5	0.402167E+04	0.768302E+03	39.00	0.19	0.01

______Branch 2______

NODE	AREA	DISCHARGE	STAGE	VELOCITY	FROUDE NUMBER
7	0.146913E+04	0.000000E+00	38.99	0.00	0.00
9	0.147096E+04	-0.276195E+03	39.00	-0.19	-0.01

______Branch 3______

NODE	AREA	DISCHARGE	STAGE	VELOCITY	FROUDE NUMBER
11	0.233492E+04	0.389834E+03	39.00	0.17	0.01
13	0.256712E+04	0.244408E+03	39.00	0.10	0.01
15	0.280222E+04	0.803207E+02	39.00	0.03	0.00
17	0.303760E+04	-0.106103E+03	39.01	-0.03	0.00
19	0.327393E+04	-0.311413E+03	39.01	-0.10	-0.01
21	0.351298E+04	-0.536589E+03	39.02	-0.15	-0.01

______Branch 4______

NODE	AREA	DISCHARGE	STAGE	VELOCITY	FROUDE NUMBER
23	0.882575E+03	0.102272E+03	39.00	0.12	0.01
25	0.883705E+03	0.636945E+01	39.01	0.01	0.00
27	0.885281E+03	-0.951314E+02	39.02	-0.11	-0.01

______Branch 5______

NODE	AREA	DISCHARGE	STAGE	VELOCITY	FROUDE NUMBER
29	0.122950E+04	-0.498984E+02	39.02	-0.04	0.00
31	0.123822E+04	-0.228743E+03	39.08	-0.18	-0.01

______Branch 6______

NODE	AREA	DISCHARGE	STAGE	VELOCITY	FROUDE NUMBER
33	0.143436E+04	-0.581822E+03	39.02	-0.41	-0.03
35	0.148539E+04	-0.783586E+03	39.29	-0.53	-0.03

INCREMENT= 24 TIME= 6.00
ITERATIONS FOR SOLUTION OF FLOW EQUATIONS = 3(last time step)
Largest ratio of change over the last iteration = 0.519E-03,
occurred in discharge at node 29

______Branch 1______

NODE	AREA	DISCHARGE	STAGE	VELOCITY	FROUDE NUMBER
1	0.436564E+04	0.105944E+04	39.64	0.24	0.02
3	0.436315E+04	0.925191E+03	39.64	0.21	0.01
5	0.436106E+04	0.791068E+03	39.63	0.18	0.01

______Branch 2______

NODE	AREA	DISCHARGE	STAGE	VELOCITY	FROUDE NUMBER
7	0.159520E+04	0.000000E+00	39.63	0.00	0.00
9	0.159510E+04	-0.242481E+03	39.63	-0.15	-0.01

______Branch 3______

NODE	AREA	DISCHARGE	STAGE	VELOCITY	FROUDE NUMBER
11	0.253197E+04	0.422427E+03	39.63	0.17	0.01
13	0.278161E+04	0.296993E+03	39.62	0.11	0.01
15	0.303320E+04	0.163812E+03	39.62	0.05	0.00
17	0.328393E+04	0.232964E+02	39.61	0.01	0.00
19	0.353401E+04	-0.124408E+03	39.61	-0.04	0.00
21	0.378404E+04	-0.275810E+03	39.60	-0.07	0.00

______Branch 4______

NODE	AREA	DISCHARGE	STAGE	VELOCITY	FROUDE NUMBER
23	0.957058E+03	0.126160E+03	39.63	0.13	0.01
25	0.955330E+03	0.476096E+02	39.62	0.05	0.00
27	0.953580E+03	-0.242246E+02	39.60	-0.03	0.00

______Branch 5______

NODE	AREA	DISCHARGE	STAGE	VELOCITY	FROUDE NUMBER
29	0.132436E+04	-0.143005E+02	39.60	-0.01	0.00
31	0.131970E+04	-0.110464E+03	39.57	-0.08	-0.01

______Branch 6______

NODE	AREA	DISCHARGE	STAGE	VELOCITY	FROUDE NUMBER
33	0.154502E+04	-0.285734E+03	39.60	-0.18	-0.01
35	0.154915E+04	-0.366606E+03	39.62	-0.24	-0.01

....
....

(complete machine-readable files available on request)
....

page 53 follows

APPENDIX F

Input and output for example 2

CNTRL input file

```
  Hypothetical example, Swamprat Creek, Near Pete's Byou
             n = 0.045, slope = 0.001
    NUMNP=          4
     NBCH=          1
   NTRANS=          0
    NFLOW=          1
     NORM=          0
     INIT=          2
5000.0 /
10000.0 /
   NSKPRT=          1
      PRT=          0
   PRTINC=         12
      PUN=          0
   PUNINC=         24
      PLT=          0
   PLTINC=          1
   NUMSTA=          2
     1   21 /
    THETA=       0.55
      MNI=        100
     TIME= 0.000E+00
    TUNIT= 0.600E+02
     MNIU=          5
   UERROR= 0.500E-02
      EQ1= 250.0
      EQ2= 477.465
      EQ3= 75.0
    NDAMS=          0
        00         30    300.00          1
        00       9999    300.00          0
```

XHD.DAT input file

```
$OPTS 1
 1 0.000 UPSTREAM EXTERNAL BOUNDARY
0.0 90.0
0.0 70.0
100.0 70.0
100.0 90.0 /
/
0.045 /
1.00 /
70.0 0.5 90.0 /
$OPTS 1
02 13.26 DOWNSTREAM EXTERNAL BOUNDARY
0.0 20.0
0.0 0.0
100.0 0.0
100.0 20.0 /
/
0.045 /
1.00 /
0.0 0.5 20.0 /
```

PRT output file

```
 Hypothetical example, Swamprat Creek, Near Pete's Byou
            n = 0.045, slope = 0.001
   NUMNP=        30
    NBCH=         1
  NTRANS=         0
   NFLOW=         2
    NORM=         0
    INIT=         1

     DXMIN, in feet---
    5000.00

      DXMAX, in feet---
    10000.0

  NSKPRT=         1
     PRT=         0
  PRTINC=        12
     PUN=         0
  PUNINC=        24
     PLT=         0
  PLTINC=         1
  NUMSTA=         2
NODES AT WHICH OUTPUT OF TIME SERIES DATA IS REQUESTED
       1      21

   THETA=      0.55
     MNI=       100
    TIME= 0.000E+00
   TUNIT= 0.600E+02
    MNIU=         5
  UERROR= 0.500E-02
     EQ1= 0.250E+03
     EQ2= 0.477E+03
     EQ3= 0.750E+02
```

INCREMENT= 12 TIME= 60.00
ITERATIONS FOR SOLUTION OF FLOW EQUATIONS = 2(last time step)
Largest ratio of change over the last iteration = 0.214E-02,
occurred in discharge at node 3

______Branch 1______

NODE	AREA	DISCHARGE	STAGE	VELOCITY	FROUDE NUMBER
1	0.305904E+03	0.681450E+03	73.06	2.23	0.22
3	0.233712E+03	0.450566E+03	67.34	1.93	0.22
5	0.177322E+03	0.271113E+03	61.77	1.53	0.20
7	0.170092E+03	0.247865E+03	56.70	1.46	0.20
9	0.169996E+03	0.247482E+03	51.70	1.46	0.20
11	0.169999E+03	0.247487E+03	46.70	1.46	0.20
13	0.169999E+03	0.247487E+03	41.70	1.46	0.20
15	0.169999E+03	0.247487E+03	36.70	1.46	0.20
17	0.169999E+03	0.247487E+03	31.70	1.46	0.20
19	0.169999E+03	0.247487E+03	26.70	1.46	0.20
21	0.169999E+03	0.247487E+03	21.70	1.46	0.20
23	0.169999E+03	0.247487E+03	16.70	1.46	0.20
25	0.169999E+03	0.247487E+03	11.70	1.46	0.20
27	0.169999E+03	0.247487E+03	6.70	1.46	0.20
29	0.169999E+03	0.247487E+03	1.70	1.46	0.20

INCREMENT= 24 TIME= 120.00
ITERATIONS FOR SOLUTION OF FLOW EQUATIONS = 2(last time step)
Largest ratio of change over the last iteration = 0.151E-02,
occurred in discharge at node 9

______Branch 1______

NODE	AREA	DISCHARGE	STAGE	VELOCITY	FROUDE NUMBER
1	0.245923E+03	0.414800E+03	72.46	1.69	0.19
3	0.300067E+03	0.611059E+03	68.00	2.04	0.21
5	0.303286E+03	0.652957E+03	63.03	2.15	0.22
7	0.255032E+03	0.515762E+03	57.55	2.02	0.22
9	0.188074E+03	0.303878E+03	51.88	1.62	0.21
11	0.170916E+03	0.250488E+03	46.71	1.47	0.20
13	0.170008E+03	0.247523E+03	41.70	1.46	0.20
15	0.169998E+03	0.247479E+03	36.70	1.46	0.20
17	0.169998E+03	0.247479E+03	31.70	1.46	0.20
19	0.169998E+03	0.247479E+03	26.70	1.46	0.20
21	0.169998E+03	0.247479E+03	21.70	1.46	0.20
23	0.169998E+03	0.247479E+03	16.70	1.46	0.20
25	0.169998E+03	0.247480E+03	11.70	1.46	0.20
27	0.169998E+03	0.247479E+03	6.70	1.46	0.20
29	0.169998E+03	0.247479E+03	1.70	1.46	0.20

....
....
(complete machine-readable files available on request)
....