

- PN-AAZ-242

ISW
54508



**USERS MANUAL
FOR THE PASCAL VERSION OF THE
USU MAIN SYSTEM HYDRAULIC MODEL**



**WATER MANAGEMENT SYNTHESIS II PROJECT
WMS REPORT 75**

**USERS MANUAL FOR THE PASCAL VERSION OF THE
USU MAIN SYSTEM HYDRAULIC MODEL**

This study is an output of
Water Management Synthesis II Project
under support of
United States Agency for International Development
Contract AID/DAN-4127-C-00-2086-00

All reported opinions, conclusions or recommendations are the sole
responsibility of the author and do not represent the official or
unofficial positions of any agency of the United States
or Utah State University or the
Consortium for International Development

by

Gary P. Merkley - Irrigation Engineer

Utah State University
Agricultural and Irrigation Engineering Department
Logan, Utah 84322-4105

October 1987

WMS Report 75

PREFACE

This study was conducted as part of the Water Management Synthesis II Project, a program funded and assisted by the United States Agency for International Development through the Consortium for International Development. Utah State University, Colorado State University, and Cornell University serve as co-lead universities for the Project.

The key objective is to provide services in irrigated regions of the world for improving water management practices in the design and operation of existing and future irrigation projects and give guidance for USAID for selecting and implementing development options and investment strategies.

For more information about the Project and any of its services, contact the Water Management Synthesis II Project.

Jack Keller, Project Co-Director
Agricultural and Irrig. Engr.
Utah State University
Logan, Utah 84322-4105
(801) 750-2785

Wayne Clyma, Project Co-Director
University Services Center
Colorado State University
Fort Collins, Colorado 80523
(303) 491-6991

E. Walter Coward, Project Co-Director
Department of Rural Sociology
Warren Hall
Cornell University
Ithaca, New York 14853-7801
(607) 255-5495

USE OF THIS MANUAL

It is assumed that the reader is familiar with the basic principles and terminology of open channel hydraulics. Some knowledge of open channel flow is necessary for the correct entry of data and for the interpretation of the model results. Persons who lack this background will be able to run the model but will not be able to take full advantage of its capabilities and options.

No knowledge of computer programming languages or operating systems is required. However, as with the application of any software package it is useful to understand some of the basics about how files are managed on the discs by the operating system. This kind of information can be obtained by consulting the user's manuals for the computer itself. It is recommended that a potential user of the model read this manual and then try an example run with the program itself. It should be expected that some of the material presented in this manual will not be fully understood at first and that an attempt to actually use the program on the computer will both clarify some questions and create others. After some example runs are made the users manual can be referred to subsequently as needed.

ACKNOWLEDGEMENTS

Initial development of this model was based on modifications to a hydrodynamic model for surface irrigation by Dr. Wynn R. Walker of Utah State University. The approach to solving the theoretical equations has remained largely the same. Francis N. Gichuki and Gary P. Merkley did parallel development work on the USU Hydraulic Model in FORTRAN 77 and Pascal programming languages, respectively. Both the FORTRAN and the Pascal versions of the model are available, and they contain different features and user interfaces.

During installation and testing of this model in Northeast Thailand two Royal Irrigation Department engineers were actively involved and they provided many useful suggestions for improving the software. They are Kanching Kawsard and Charoon Pojsoontorn. Most of the figures contained in this manual were prepared by Titipan Chanvimol of the Northeast Small-Scale Irrigation Project in Khon Kaen, Thailand. Many thanks are extended to these people.

-- TABLE OF CONTENTS --

Section One: Hydraulic Model (CANAL)		Page
Preface.....		ii
Use of This Manual.....		iii
Introduction.....		2
Hydraulic Modeling.....		3
General Information.....		5
Screen Displays.....		7
Alpha Screen.....		7
Graphics Screen.....		13
Changing Screen Displays.....		18
Quick Reference Guide.....		19
Beginning a Hydraulic Simulation.....		20
Main Menu.....		20
Filling an Empty System.....		21
Continuing a Previous Simulation.....		22
Inflow Hydrograph.....		23
Operational Mode.....		24
Simulation Interrupts.....		27
Pausing the Simulation.....		27
Help File.....		27
Changing the Inflow Rate.....		27
Gate Scheduling.....		28
Change Turnout Demands.....		29
Change Control Structure Settings.....		30
Keep Current Profiles.....		31
View System Characteristics Data.....		31
Final Output of Simulation Results.....		33
Summary Menu.....		33
Tabular Summary.....		34
Graphical Summary.....		35
Hardcopy Printouts.....		39
Saving the Current Status.....		42

-- TABLE OF CONTENTS (cont) --

	Page
Definition of Terms.....	43
Control Structure.....	43
Turnout Structure.....	43
Seepage Rate.....	44
Data Files.....	44
Advance Phase.....	45
Grid Adjust.....	46
Gate Scheduling.....	46
Turnout Scheduling.....	51
Numerical Solution.....	53
 Section Two: File Editor (CDAT)	
Introduction.....	55
General Information.....	55
Changing the File Contents.....	57
Last Update.....	57
Reach Names.....	57
Turnout Names.....	58
Parameters.....	58
Warnings Flag.....	59
Definition of Terms.....	60
DOS Path.....	60
Branch Linkages.....	60
System Dimensions.....	63
Configuration Data.....	63
Control Structure Data.....	65
Turnout Structure Data.....	69
Allowable Input Ranges.....	72

-- TABLE OF CONTENTS (concl) --

Appendices

Appendix A:
Data Collection and Calibration..... 75

 Calibration of Controls and Turnouts..... 77
 Control Structure Equations..... 79
 Turnout Structure Equations..... 86
 Discharge Coefficients..... 90
 Seepage Loss Measurement..... 91
 Field Data Forms..... 91

Appendix B:
What Can Go Wrong ?..... 94

 Advance Too Fast..... 95
 Advance Just Beyond Reach End..... 96
 Inflow Rate Changes Too Fast..... 96
 Pre-Set Control Structure Adjustments..... 97
 Sluice Gate Submergence..... 97

Appendix C:
Error Messages..... 98

 File System Errors..... 99
 Data File Errors..... 99
 Data File Warnings..... 101
 Hydraulic Simulation Errors..... 103
 Messages On The Second Alpha Screen..... 104

-- LIST OF FIGURES --

Figure		Page
1.	First alpha screen display.....	8
2.	Second alpha screen display.....	9
3.	Third alpha screen display.....	10
4.	Fourth alpha screen display.....	11
5.	Fifth alpha screen display.....	12
6.	Sixth alpha screen display.....	13
7.	Graphics screen display with outflow bars.....	14
8.	Graphics screen display with outflow curves.....	17
9.	Main menu of the hydraulic model.....	20
10.	Screen format for entering the inflow hydrograph....	23
11.	Flow diagram indicating the possible paths to the five different gate scheduling status conditions for individual canal reaches.....	26
12.	Operating mode changes menu.....	28
13.	Screen display for changing turnout demands.....	30
14.	Summary menu.....	33
15.	Tabular summaries menu.....	34
16.	Tabular display for canal reach data.....	35
17.	Initial graphical display of simulation results.....	36
18.	Final graphical display of simulation results.....	38
19.	Illustrated example of steady-state reach discharge calculations during gate scheduling.....	48
20.	Illustrated example of how branches can be linked...	62
21.	Example of elevation changes across a control structure for both z_u and z_d positive.....	68
22.	Example of elevation changes across a control structure for z_u positive and z_d negative.....	69
23.	Example of elevation changes across a control structure for z_u zero and z_d positive.....	69
A-1.	Calibration data plots on logarithmic scale.....	78
A-2.	Circular culvert control structure.....	80
A-3.	Rectangular culvert control structure.....	81
A-4.	Rectangular weir control structure.....	82
A-5.	Rectangular sluice gate control structure.....	84
A-6.	Circular sluice gate control structure.....	85
A-7.	Circular and rectangular orifice turnouts.....	86
A-8.	Linear approximation of the stage-discharge relationship downstream of turnouts.....	88
A-9.	Adjustable Weir Turnout.....	89
A-10.	Wasteway Weir Turnout.....	90

-- LIST OF TABLES --

Table		Page
1.	Quick reference guide for keystrokes.....	19
2.	Sample printout for canal reach data.....	40
3.	Sample printout for canal turnout data.....	41
4.	Allowable input ranges.....	72
A-1.	Field data collection forms.....	92

SECTION ONE

HYDRAULIC MODEL (CANAL)

1

INTRODUCTION

Welcome to the USU Main System Hydraulic Model! This computer model performs hydrodynamic simulations of water flow in canals. Its applications are in the areas of canal operation, analysis, design, and operator training. As an operational tool the model can be used to help regulate a canal system through calculation of control structure settings which will help maintain constant water levels. By maintaining constant water levels the deliveries from turnouts are more stable, even when the flows through the system are changing. Applications of the model to canal operations can also significantly reduce delivery times and enhance the operational flexibility of the canal systems. This is because the hydraulic model simulates the transient flow conditions which exist in canals by treating the respective canal reaches as individual components in an integrated water distribution and delivery system.

The hydraulic model is very effective as an operator training tool since many simulations over a wide range of operating conditions can be performed in a short time. Manual operation of the control structures can be simulated and compared to settings which are calculated by the model, thus the canal system response to a variety of conditions can be evaluated without actually causing the conditions in the real canals. The operator becomes more familiar with the canal system and is better able to operate it effectively.

The software for this model has been designed with an interactive menu-driven user interface. Both graphical and tabular output are produced continuously by the model, and at the end of each simulation graphical and tabular summaries of the flow conditions and control structure settings can be displayed on the screen. Hardcopy printouts of the results of a hydraulic simulation can also be produced by the model if desired. In an operational application of the model the printouts can be distributed to the canal tenders in the field so that the control structures can be adjusted according to the optimal schedule.

HYDRAULIC MODELING

A hydraulic model can be physically or mathematically based. A small scale canal system with proportionally-sized control structures and turnouts would constitute a physical model and it might be located inside a hydraulic laboratory. Such a physical model could be used to test the operation of a real canal system over a wide range of operational scenarios. These tests could be performed in less time and with less difficulty than by using a real canal system in the field. Tests could be conducted with the physical model even before the real canal system is constructed. And, in using a physical model it would not be a catastrophic problem if the canals overflowed or if the turnout deliveries were fluctuating, since it is not a "real" canal system and there are no real water users downstream of the turnouts from the main canal. The training opportunities are obvious.

Of course the simulated flow in a physical model is not exactly the same as that which would occur in a real canal system. This is because the model is actually an idealization of a real canal system. However, if the resemblance in performance between the model and the real system is sufficiently close, then the results from the studies using the model can be usefully extrapolated to the real canal system. A mathematical model is similar to a physical model since it is also an idealization of a real system and it can be used to study the performance of a proposed or an existing canal system. The difference is that a mathematical model is based entirely on equations and numbers rather than on a physical replica of a canal system. However, a mathematical model can be used to do all of the things that a physical model can be used for in flow simulation and it can also be used to optimize the canal system operation. Furthermore, a mathematical model can be reconfigured to study a different canal system very easily and be used to examine a multitude of system configurations and operating conditions quickly and inexpensively.

The Utah State University (USU) Main System Hydraulic Model is a mathematical model capable of simulating actual hydraulic conditions in a canal system and of optimizing the operation of a system by calculating control structure settings which best maintain constant flow levels. This optimization feature is implemented during a hydraulic simulation by enabling the "Gate Scheduling" mode. Through gate scheduling, the model can be used to determine appropriate control structure settings which will optimize the daily operation of a canal system. When frequently changing demands are imposed on a canal system the actual supply can better meet these demands by implementing gate scheduling, and the water levels in the canals will remain as stable as possible.

The USU Hydraulic Model can also be very useful and effective as a training tool for canal operators. Many different operational schemes can be quickly and safely evaluated using the model and the canal operator can in this way become more familiar with the response of the real canal system to varying flow conditions and water distribution schedules. After spending some time with the model performing simulations, the canal operator should be better able to operate the real canal system effectively because he will be more familiar with the hydraulic behavior of the system. The canal operators' knowledge of the real canal system will extend beyond the normal operational modes of the canal system and thereby sensitize the operator to extra-ordinary flow conditions.

As a design tool the model can be used to evaluate proposed canal system designs under real-time operating conditions. The final design of a canal system can not only be based on static flow criteria, but also on the performance of the system in conveying and distributing water. Such a design is more likely to perform satisfactorily under actual operating conditions. Locations and types of control structures and turnouts can be evaluated, the need for canal lining can be assessed, and the ability of the system to deliver water according to proposed allocation schedules can be analyzed.

GENERAL INFORMATION

The hydraulic model CANAL is designed to simulate flow conditions which can exist in canal systems having cross-sections of trapezoidal shape and any layout of branches and reaches. The current version of the model limits the maximum number of branches to four, with each branch containing a maximum of nine reaches. This constraint is imposed by existing micro-computer memory capacities and can be relaxed as they become larger. Canal reaches are separated by control structures such as sluice gates and weirs. Each reach may contain up to nine turnouts which can be located anywhere along the length of the reach. The turnouts may be farm turnouts, lateral (branch) turnouts, or wasteway spills.

The data which define the system dimensions, branch linkages, physical configuration, control structures, and turnouts are stored in files on a disc drive (preferably a hard disc) which are read by the hydraulic model. These files can be created and modified only by using the program CDAT (see Section 2). The model CANAL only reads these data files, it does not change their contents. The inflow rate at the extreme upstream end of the canal system is always specified at the beginning of a simulation. Inflow rates are entered by the user in simulation time increments of five minutes with a maximum number of time steps of 150. This means that any one simulation can be of a maximum duration of twelve and one-half hours. However, simulations can be continued indefinitely by storing the results of one simulation and then beginning another from the same flow conditions as they existed at the time the previous simulation finished. This is explained in more detail in the section describing the inflow hydrograph.

The model has three modes of operation. The manual mode will simulate canal flow with user-specified operation of the control structures from the keyboard. The second mode involves pre-selected control structure settings for each five-minute simulation time interval. When the simulation begins the model will refer to the pre-defined control structure settings and operate them according to this schedule. Pre-setting the control structures is almost the same as manual operation of the control structures except that the movements are known before the simulation begins and it is not necessary to interrupt the simulation. The third operational mode is called "gate scheduling". With this mode the model operates the control structures automatically with the objective of maintaining actual flow levels at the target levels in each reach of the canal system. By default, target levels are equal to the operational supply levels, which can be specified for each individual canal reach.

The Pascal version of the model is written in MicroSoft Pascal 3.31 with assembly language routines to perform various screen display functions, including graphics primitives. The operating system can be either MicroSoft DOS 3.1 or 3.2. Hardware features include an IBM-AT compatible machine, an EGA (enhanced graphics adapter) card, a high-resolution color monitor, an 80287 coprocessor, at least 640 K RAM memory, and a fixed (hard) disc drive. The model will also function with a CGA or Hercules graphics card but the display will be monochrome. The model can be linked to non-80287 run-time libraries if the coprocessor is unavailable. However, the model will run much slower without the 80287 since the numerical calculations are intense. All data files are read and written to the disc drive and directory which is specified by the DOS (disc operating system) path name. This path name is "C:\CDATA\" by default, but the user can change this name to specify another disc drive or subdirectory (see Section 2). Also, it is more convenient to work from a hard disc drive since reading and writing operations will be performed much faster than with floppy disc drives.

SCREEN DISPLAYS

During simulations CANAL can output information about flow rates and flow levels in the canal system to either the alpha screen or the graphics screen. The alpha screen displays are useful for viewing the instantaneous flow levels, flow rates, control structure settings, and turnout data at any location in the canal system, and at any time during a simulation. The graphics screen displays are useful for viewing water surface profiles, overall system status, and continuous outflow hydrograph curves which show the changes in reach outflows since the beginning of a simulation. The user can toggle between the alpha and graphics screens during a simulation repeatedly as needed (see Changing Screen Displays).

Alpha Screen

When the alpha screen is being viewed during a simulation there will be one of six possible displays shown. The first display for the alpha screen (see Figure 1) shows the current flow conditions for each reach of the currently displayed branch. The data are displayed in ten rows with one column for each reach of the branch. The first three rows show flow depths in meters: Ldepth, Rdepth, and Tdepth. Ldepth is the flow depth at the upstream (left) end of a reach and Rdepth is the flow depth at the downstream (right) end of a reach. Tdepth is the gate scheduling (target) flow depth which is normally equal to the operational supply level. When gate scheduling is enabled CANAL attempts to maintain Rdepth equal to Tdepth by automatically adjusting the control structure settings at the downstream end of each reach.

The next five rows show flow rates in cubic meters per second: Lflow, Rflow, Wflow, Tflow, and Iflow. Lflow is the flow rate entering a reach and Rflow is the flow rate leaving a reach at the downstream end. Wflow is the flow rate passing over a weir control structure and is always equal to Rflow when the control consists of a weir only. When the control is a sluice gate with fixed side weirs Rflow will be equal to the combined discharge through the sluice gate and over the weir sills, and Wflow will be equal to the weir discharge alone. As with any weir structure, whether fixed or adjustable, the weir will only discharge water when the flow depth upstream of the structure is greater than the sill height. Tflow is the total of all turnout discharges in a reach and Iflow is the total seepage outflow from a reach. In the case of a net seepage inflow, the value of Iflow will be negative. Net seepage inflow will occur in the model when a negative seepage rate value is entered into a configuration data file by the user.

The ninth row shows the control structure setting in meters. For a sluice gate this is the gate opening which is the vertical distance from the lower gate frame to the bottom of the gate itself. The lower gate frame is taken to be at the same elevation as the bottom of the canal just upstream of the sluice gate. For a weir this is the vertical distance from the bottom of the canal to the top of the sill. Of course, this setting cannot change during a simulation if the control structure is non-adjustable. Non-adjustable control structures include fixed weirs, culverts, and inverted siphons.

(1) SYSTEM FLOW STATUS		Configuration Data File: TEST								
Branch 1 Step 123										
		Reach Number								
		1	2	3	4	5	6	7	8	9
Ldepth		1.978	1.960	1.922	1.745	1.200	0.134			
Rdepth		2.125	2.033	2.040	2.040	1.459	0.000			
Tdepth		2.125	2.083	2.040	2.040	1.900	1.750			
Lflow		1.000	0.984	0.966	0.962	0.302	0.095			
Rflow		0.984	0.966	0.962	1.032	0.095	0.000			
Wflow		0.000	0.035	0.011	0.005	0.000	0.000			
Tflow		0.000	0.000	0.000	0.000	0.000	0.000			
Iflow		0.016	0.018	0.004	0.017	0.005	0.000			
Setting		0.448	0.446	0.133	0.220	0.057	0.030			
Schedule		On	On	On	On	Wait	Fill			
All units are in meters and seconds.						System Inflow = 1.000				
						System Outflow = 0.229				

Figure 1. First alpha screen display.

The tenth and last row shows the gate scheduling status as one of the following five items: (1) On, (2) Off, (3) Fill, (4) Wait, and (5) Set. If the control structures are pre-set at the beginning of a simulation the gate scheduling status will be "Set". If gate scheduling is disabled or if the control structure is non-adjustable then the gate scheduling status will be "Off". If gate scheduling is enabled and water has not yet advanced to the end of the reach then the gate scheduling status will be "Fill". Or, if gate scheduling is enabled and some operational limitation has been encountered then the gate scheduling status will be "Wait". Finally, if gate scheduling is enabled and activated then the status will be "On". A detailed explanation of gate scheduling can be found at the end of Section 1 under Definition of Terms. Also see Figure 11.

The second alpha screen display is similar to the default

display except the some of the numerical solution details can be viewed in the lower part of the screen. These details include the flow regime and status, control structure type, computational node ranges, and various messages about the progress of the solution in each reach. All of the possible messages that could appear in this alpha screen display are described in Appendix C. In this second alpha screen display solution details are displayed for all reaches in all branches of the canal system which is being simulated, unlike the other alpha screen displays and the graphics display which only show the flow conditions for one branch at a time. An example of the second alpha screen display is shown in Figure 2.

(2) SYSTEM FLOW STATUS		Configuration Data File: TEST							
Branch 1 Step 123		Reach Number							
	1	2	3	4	5	6	7	8	9
Ldepth	1.978	1.960	1.922	1.745	1.200	0.134			
Rdepth	2.125	2.083	2.040	2.040	1.459	0.000			
Tdepth	2.125	2.083	2.040	2.040	1.900	1.750			
Lflow	1.000	0.984	0.966	0.962	0.302	0.095			
Rflow	0.984	0.966	0.962	1.032	0.095	0.000			
Wflow	0.000	0.035	0.011	0.005	0.000	0.000			
Tflow	0.000	0.000	0.000	0.000	0.000	0.000			
Iflow	0.016	0.018	0.004	0.017	0.005	0.000			
Setting	0.448	0.446	0.133	0.220	0.057	0.030			
Schedule	On	On	On	On	Wait	Fill			

NUMERICAL SOLUTION DETAILS	
Reach :	1
Branch:	1
Status:	Submerged Flow
Regime:	Post Advance
Contrl:	Rect Sluice
Nodes :	0 to 11

Figure 2. Second alpha screen display.

The third alpha screen display (see Figure 3) shows turnout flow demands for each reach of the currently displayed branch. These demand flow rates are specified by the user and not by the model itself. Each horizontal row from one to nine corresponds to that same turnout number in each reach. If a reach has less than the maximum number of nine turnouts then only the rows for the existing turnouts will show a discharge value, and the remaining rows will be blank for the column corresponding to the reach number. And, if a branch has less than the maximum number of nine reaches then only the

columns corresponding to existing reaches will show discharge values. Wasteway weirs are indicated by dashes instead of a number since they are not adjustable and do not have any demand discharge. Below each reach column the total turnout demand for the reach is displayed to the right of the word "SUM".

(3) TURNOUT FLOW DEMANDS		Configuration Data File: TEST									
Branch 1 Step 123		Reach Number									
	1	2	3	4	5	6	7	8	9		
1	-----	0.00%	0.000	0.100	0.100	-----					
2	0.000	0.000	0.000	0.100	0.100	0.000					
3	0.000	0.000	0.000	0.200	0.250	0.100					
4	1.200	0.000	0.000	-----	0.300	0.000					
5	0.000	0.100	0.000		0.000	0.125					
6	0.500	0.100	-----		0.000	0.000					
7	0.000	-----			0.150	0.125					
8	-----	0.100				-----					
9	0.000	0.000									
SUM	1.700	0.300	0.000	0.400	0.900	0.350					
Total Turnout Flow	3.649									System Inflow =	7.000
Total Seepage Flow	0.085									System Outflow =	5.082
Units are cubic meters per second.											

Figure 3. Third alpha screen display.

The lower part of the screen will show the branch total turnout flow and seepage flow, and the system inflow and total system outflow rates. The system inflow rate comes from the inflow hydrograph which the user enters at the beginning of a simulation. This inflow rate is always at the upstream end of the first reach of the first branch in the system. The total system outflow rate includes all seepage flows, turnout flows, and the discharge through control structures at terminal points in the canal system. If a turnout discharges into another branch then the flow rate through the turnout is not included in the total system outflow rate. This is because the flow through such a turnout is not leaving the canal system, but is still interior to the system. A terminal point in a system is defined to be the downstream end of the last reach in a canal branch, except when another branch is linked to this point (in which case it is an interior point in the system). Branch linkages are defined in Section 2.

When the total system outflow rate is equal to the system inflow rate for about three time steps or more the entire canal system will be at a steady-state flow condition. Conversely, the total system outflow rate will differ from the system inflow rate during filling and during transient flow conditions. All flow rates are shown in cubic meters per second.

The fourth alpha screen display (see Figure 4) is very similar to the third display. The difference is that the fourth display shows actual turnout flow rates. In this display the actual discharge from wasteway weirs is also shown, that is, wasteway weirs are not represented by dashes as in the other turnout displays. If the demand on a turnout does not exceed the turnout capacity then the actual discharge rate will usually be equal to the demand.

(4) ACTUAL TURNOUT FLOWS				Configuration Data File: TEST					
Branch 1 Step 123				Reach Number					
	1	2	3	4	5	6	7	8	9
1	0.000	0.000	0.000	0.100	0.100	0.000			
2	0.000	0.000	0.000	0.100	0.100	0.000			
3	0.000	0.000	0.000	0.200	0.250	0.100			
4	1.199	0.000	0.000	0.000	0.300	0.000			
5	0.000	0.100	0.000		0.000	0.125			
6	0.500	0.100	0.000		0.000	0.000			
7	0.000	0.000			0.150	0.125			
8	0.000	0.100				0.000			
9	0.000	0.000							
SUM	1.699	0.300	0.000	0.400	0.900	0.350			
Total Turnout Flow	3.649						System Inflow =	7.000	
Total Seepage Flow	0.085						System Outflow =	5.082	

Units are cubic meters per second.

Figure 4. Fourth alpha screen display.

The fifth and sixth alpha screen displays (see Figures 5 and 6) are also similar in format to the third display. The fifth display shows turnout settings in meters. Turnout settings will change automatically according to the demand and the flow depth in the canal at the turnout location. Wasteway weir turnouts are non-adjustable and they are shown as dashes. Adjustable weir turnout settings are shown as zero when the sill is fully raised. This is because the setting for adjustable weirs is measured from the top of the canal

bank down to the sill height. Therefore, for each of the three types of adjustable turnouts a setting of zero indicates that it is fully closed.

(5) TURNOUT SETTINGS				Configuration Data File: TEST					
Branch 1 Step 123									
	Reach Number								
	1	2	3	4	5	6	7	8	9
1	-----	0.000	0.000	0.045	0.046	-----			
2	0.000	0.000	0.000	0.038	0.041	0.000			
3	0.000	0.000	0.000	0.098	0.100	0.035			
4	0.167	0.000	0.000	-----	0.114	0.000			
5	0.000	0.034	0.000		0.000	0.056			
6	0.101	0.033	-----		0.000	0.000			
7	0.000	-----			0.057	0.058			
8	-----	0.030				-----			
9	0.000	0.000							
Total Turnout Flow			3.648	System Inflow =			7.000		
Total Seepage Flow			0.085	System Outflow =			5.082		
Units are in meters.									

Figure 5. Fifth alpha screen display.

The sixth display shows the status for individual turnouts. The turnout status is determined automatically by the model as it tries to match actual turnout flows to the demand flow rates. This status can be any one of the following eight types shown below:

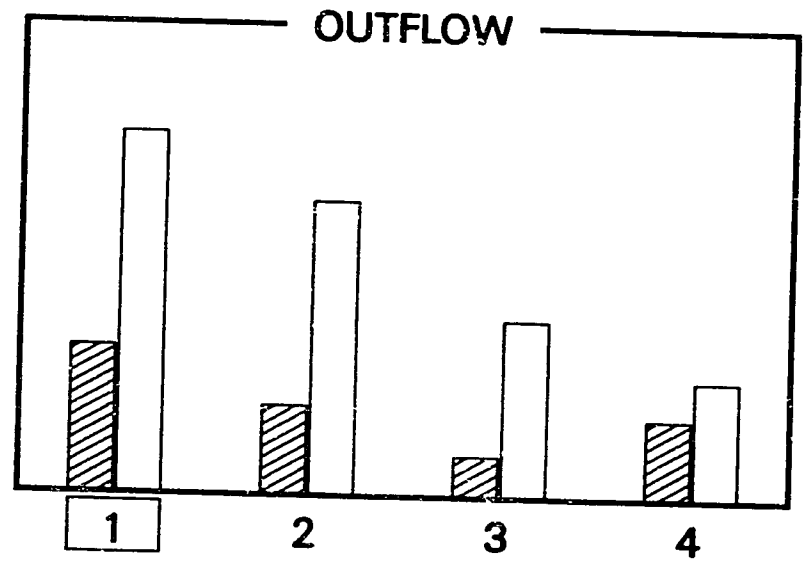
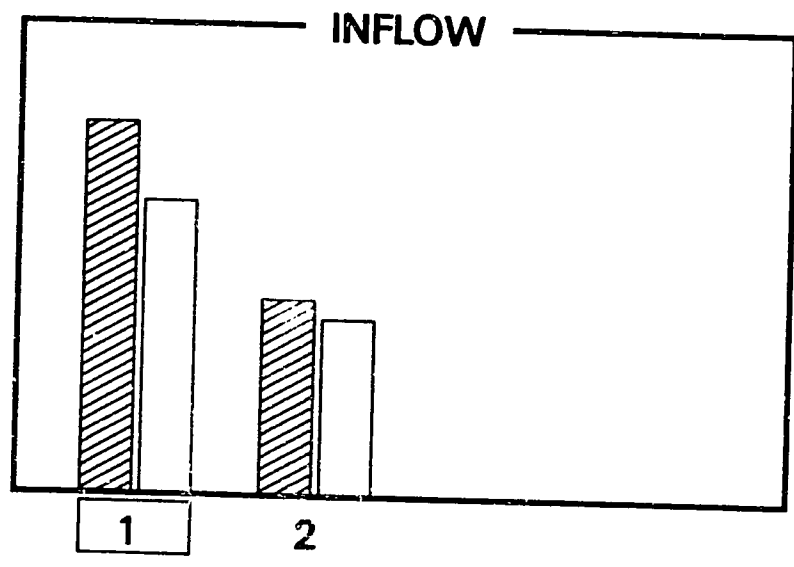
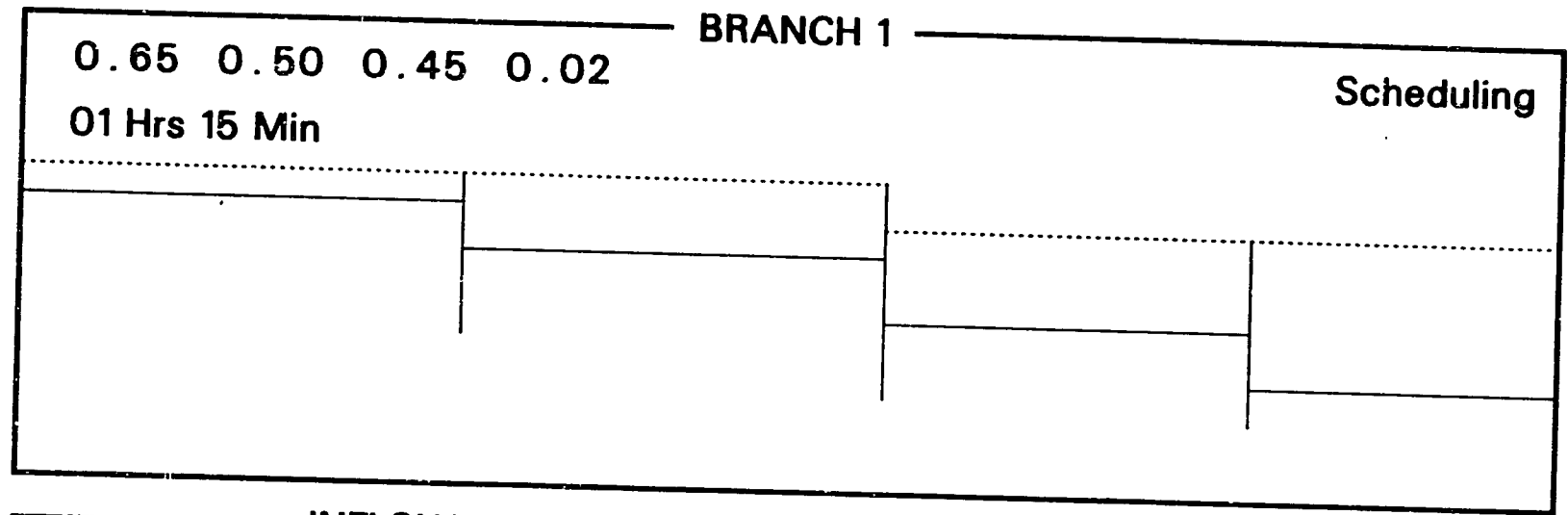
Status	Interpretation
Open	Turnout is completely open.
----	Turnout is a wasteway weir.
Wait	Turnout setting is changing.
Shut	Turnout is completely closed.
Back	Turnout backflow is impending.
Free	Turnout operates under free flow.
Subm	Turnout operates under submerged flow.
Surf	Turnout is opened to upstream flow level.

(6) TURNOUT STATUS		Configuration Data File: TEST							
Branch 1 Step 123		Reach Number							
	1	2	3	4	5	6	7	8	9
1	----	Shut	Shut	Free	Free	----			
2	Shut	Shut	Shut	Free	Subm	Shut			
3	Shut	Shut	Shut	Subm	Surf	Free			
4	Wait	Shut	Shut	----	Free	Shut			
5	Shut	Free	Shut		Shut	Free			
6	Subm	Free	----		Shut	Shut			
7	Shut	----			Free	Free			
8	----	Free				----			
9	Shut	Shut							
SUM	1.700	0.300	0.000	0.400	0.800	0.350			
Total Turnout Flow	3.648			System Inflow =			7.000		
Total Seepage Flow	0.085			System Outflow =			5.082		

Figure 6. Sixth alpha screen display.

Graphics Screen

During simulations CANAL can display flow profiles, outflow and inflow bargraphs, and outflow hydrographs on the graphics screen (see Figure 7). Only one branch can be displayed at a time but the branch display can be changed by typing an integer (1-4) which corresponds to a branch in the system. The graphics screen contains three subplots. The upper subplot shows schematic control structure positions and settings, and water depth profiles. There are two types of schematic representations of control structures in the upper subplot. For culverts and sluice gates the structure opening is at the bottom, and for weirs the opening is at the top. Therefore, the exact control structure type can never be distinguished by only looking at its schematic representation in the upper subplot of the graphics display. The longitudinal distance between control structures is proportional to the relative lengths of each reach in a branch. A dotted horizontal line above each reach shows the maximum canal depth for the reach. Above these dotted lines the control structure settings in meters are displayed for each reach in the currently displayed branch. At the upper left of the screen the elapsed simulation time is shown in hours and minutes, and at the upper right of the screen the current operational mode is displayed as being either Manual, Scheduling, or Pre-Set.



14

Figure 7. Graphics screen display with outflow bars.

The water depth profiles show the relative depths of water measured from the canal bottom but do not show the canal longitudinal slope. For this reason the flow profiles may seem to be incorrect since the water surface elevation can appear to increase in the downstream direction due to the influence of a control structure. In such a case the water surface elevation is actually decreasing in the downstream direction and it is only the flow depth relative to the canal bottom that increases. Also, the vertical and horizontal scales in the upper subplot are never equal since the horizontal scale is usually compressed on the graphics screen. Thus, relatively small water profile gradients usually appear to be much larger than they actually are. Elevation drops from the end of one reach to the beginning of another reach located downstream are taken into consideration in the flow calculations; however, the elevation drops are not displayed on the graphics screen. For this reason, downstream water levels may appear to be at higher elevations than upstream water levels across a control structure when the elevation drop from one reach to the next is large compared to the actual difference in water surface elevations. The reason that these elevation drops are not displayed is that for a general canal system configuration the vertical scaling must accommodate a large range of values and in some cases the displayed water surface profiles would be difficult to see. Therefore, the schematic flow profile display in the upper subplot of the graphics screen does not show the elevation drops from one reach to another.

The lower left subplot shows a flow rate bargraph. Two bars are shown for each branch in a canal system. The left bar shows the relative inflow rate to a branch. The inflow rate for branch number one always comes directly from the inflow hydrograph which is entered by the user at the beginning of the simulation. The right bar shows the relative average flow rate in a branch. During simulations the branch average flow rate will lag behind inflow rate fluctuations, but will approach the inflow rate as steady-state flow conditions are attained in the canal system. Of course, when flow is leaving the branch from turnouts or as seepage the average flow rate in the branch will be less than the branch inflow rate under steady-state conditions. This is because the branch average flow rate is the average of the flows that are internal to the canal system which is being simulated. For example, if a canal has no seepage losses and no turnouts then under steady-state conditions the flow rate is the same throughout the length of the canals and this is then equal to the average flow rate in the branch and the branch inflow rate. In such a simple case the canals are merely conveying water from a source to a single delivery point at the extreme downstream end of the system.

The default display in the lower right subplot shows two outflow bars for each reach of the currently displayed branch. The left bar shows the total turnout discharges for each reach, and the right bar

shows the outflow rate at the downstream end of each reach. The outflow rate from the end of a reach is the flow rate that passes through the control structure that demarcates the reach end. These bars can be replaced by outflow hydrograph curves for any or all of the reaches in the currently displayed branch (see Figure 8). Outflow hydrograph curves show the changes in reach outflows since the beginning of a simulation, but outflow bargraphs only show the instantaneous outflow rates. The vertical scale for bargraphs and outflow hydrograph curves in both of the lower subplots is the same. The horizontal axis in the lower right subplot is scaled to the duration of the simulation when outflow hydrograph curves are displayed. The duration of a simulation is always set by the number of values entered into the inflow hydrograph at the beginning of a simulation. This means that the outflow hydrograph curves will always reach the right edge of the lower right subplot just as the simulation is completed (unless the simulation is suspended prematurely by the user).

Below the lower left subplot the branch numbers are displayed and an indicator box shows the branch in which the model is currently performing flow calculations. When calculations are under way in a branch of the canal system that is not currently displayed, the number in the indicator box will change to correspond to the reach for which calculations are being performed. Below the lower right subplot the reach numbers for the currently displayed branch are shown and an indicator box shows the reach in which the model is currently performing calculations. When calculations are under way in a branch of the canal system which is not currently displayed the reach indicator box remains at reach number one. In this way the user can always see the branch and reach numbers for which flow computations are being performed during a hydraulic simulation.

When outflow hydrograph curves are displayed in the lower right subplot the curve colors are as follows:

Reach or Turnout Number	Curve Color
1	Blue
2	Green
3	Cyan
4	Red
5	Magenta
6	Yellow
7	White
8	Blue
9	Green

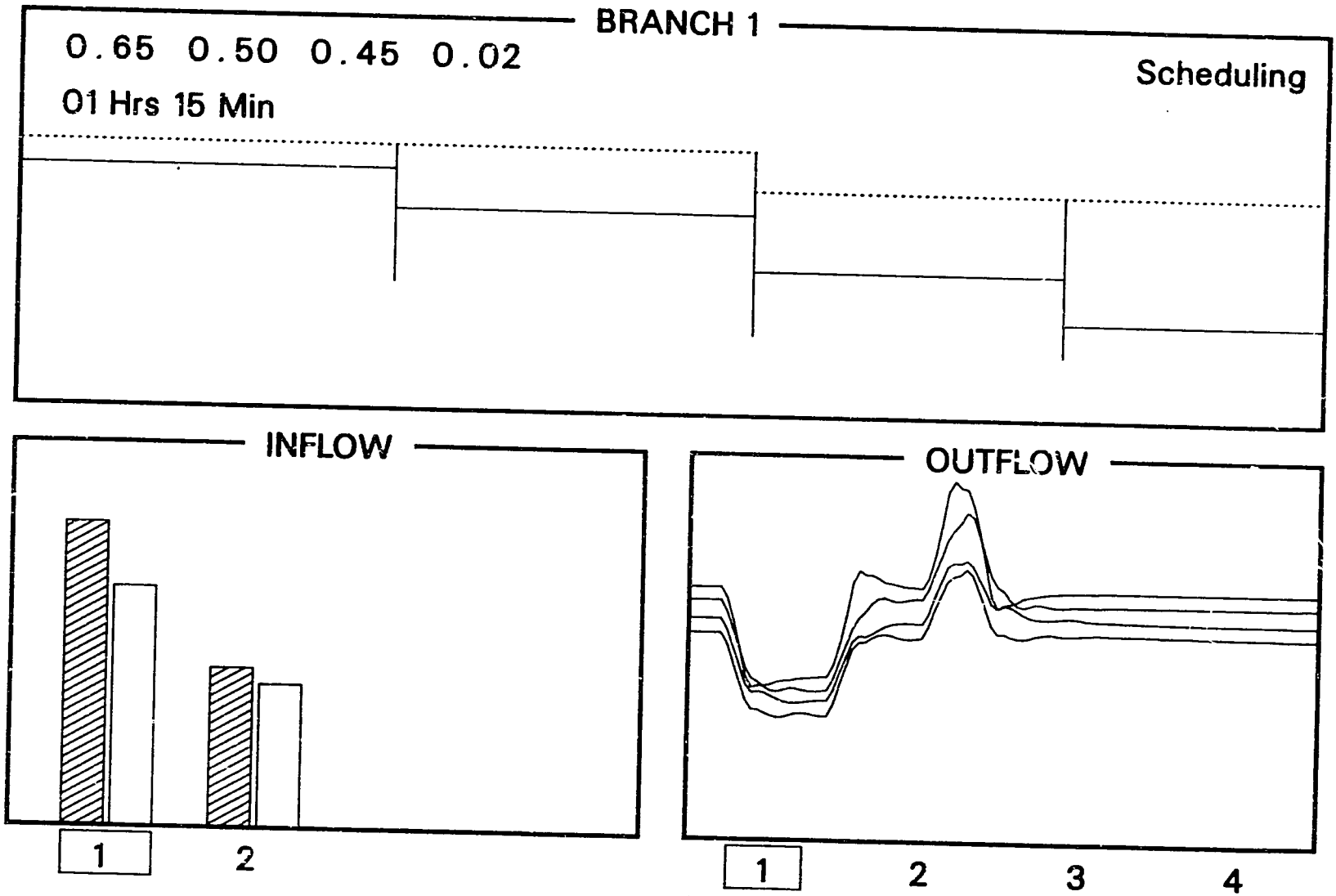


Figure 8. Graphics screen display with outflow curves.

The colors for the eighth and ninth curves are exactly the same as those of the first and second curves, respectively. If a canal branch has more than seven reaches and if there is any question about which curve corresponds to which reach then the user can choose to temporarily isolate individual curves by changing the display. In this way the curve for any reach can be positively identified in spite of the partial redundancy of curve colors.

Changing Screen Displays

The model continuously updates either the graphics or the alpha screen during a simulation. Either of these screen displays can be viewed one at a time by using the equal sign (=) to toggle between them. When a simulation begins the graphics screen will be displayed by default unless the user types the equal sign immediately after selecting the operational mode, just before the simulation begins. The equal sign can also be typed at any time during a simulation to immediately change from the graphics screen to the alpha screen, and vice versa.

Both the alpha and the graphics screens display information for only one branch of a canal system at a time. When a simulation begins the first branch is displayed but if a system has more than one branch then the branch display can be switched by typing the number of the branch. For example, if branch number one is currently displayed then by typing the number "2" the second branch will be displayed. This works regardless of whether the alpha or the graphics screen is currently displayed. If the graphics screen is on and the branch displayed is changed then outflow bargraphs will be displayed in the lower right subplot, even if outflow hydrograph curves were previously displayed. The lower left subplot remains the same for any branch display since it always displays information about all of the branches in a canal system.

The display in the lower right subplot of the graphics screen can be changed at any time by typing any combination of the ten function keys (F1-F10). The numerical values of the function keys will correspond to the reach numbers for the outflow hydrograph curves which are to be displayed in this subplot. For example, typing F2, F3, and F5 (in any order) will result in the display of outflow hydrograph curves for reaches two, three, and five of the currently displayed branch. Typing the F10 key will result in the display of outflow bargraphs in the lower right subplot of the graphics screen display.

If the alpha screen is currently displayed then typing the function keys F1 to F6 will cause the alpha screen display to change. The numerical value of the function key will correspond to the alpha

screen display to be shown on the screen. For example, typing the F4 key during a simulation will cause the fourth alpha screen display to be shown. If the fourth display is already being shown then nothing will happen. Changing any of the screen displays will never affect the hydraulic simulation itself, it only changes the information which is available to the user on the computer monitor.

Quick Reference Guide

The last page in the help file for the model is a quick reference guide containing abbreviated instructions on what keys to press to change the display or interrupt the simulation. These instructions are a summary of the descriptions contained in Changing Screen Displays and Simulation Interrupts. Table 1 shows the different keystrokes that are listed in the quick reference guide.

Table 1. Quick Reference Guide for Keystrokes.

Keystroke	Effect
-	Toggle between alpha and graphics screens
1	Display the first branch
2	Display the second branch
3	Display the third branch
4	Display the fourth branch
F1 to F6	Select from six alpha displays
F1 to F9	Display from 1 to 9 outflow hydrographs
F10	Display reach outflow bargraphs
Ctrl-End	Abort the simulation and go to DOS
I	Change the system inflow rate
T	Change the turnout demand discharges
C	Change the control structure settings
S	Change the operational mode
P	Pause the simulation
V	View the configuration data file
K	Terminate the simulation prematurely
H	Access the help file on the monitor
Q	Access the quick reference file

BEGINNING A HYDRAULIC SIMULATION

Main Menu

The main menu displays five items (see Figure 9). Menu items can be selected by typing the key of the first letter of the item description. Lower case letters are treated the same as upper case letters from the keyboard. From the main menu the user can type the letter "V" to view the contents of configuration data files which are created and modified through use of the program CDAT. These are the data files that are read by the hydraulic model CANAL prior to beginning a simulation for initial filling of a canal system. The program CANAL can also be exited from the main menu by typing an "E" and the help file can be accessed by typing an "I".

```
-----  
C A N A L  H Y D R A U L I C  M O D E L  
-----  
  
      Exit The Program  
  
      Fill An Empty Canal System  
  
      Information About The Program  
  
      Get A Suspended Simulation File  
  
      View The System Characteristics Data  
  
  
      To Select, Type the First Letter of the Item Description
```

Figure 9. Main menu of the hydraulic model.

There are two items in the main menu which can be selected to begin a hydraulic simulation. These two items correspond to (1) the filling of an empty canal system, and (2) the continuation of a previously suspended simulation. If the user chooses to fill an empty canal system by typing an "F" then a prompt will follow for a configuration data file name. This is the same type of file that is created and modified through use of the program CDAT.

Filling An Empty System

As the model reads a configuration data file it performs six checks on the turnout specifications. It first checks that the turnout locations for each reach are all within the reaches for which they are designated. This means that no turnout location can be greater than the length of the reach in which it resides. The second check is to confirm that the turnout locations are in the proper sequence. This means that the turnouts must be numbered beginning with the most upstream turnout and continuing consecutively in the downstream direction. For example, turnout number three in a canal reach cannot be located at 500 meters with turnout number four at 400 meters; the order must be reversed. The third check is for weir turnout structures only. The "Position" (sill height) of this type of turnout structure must be greater than one-half of the reach depth in which it resides. This restriction is meant to avoid sudden, large turnout flows just as the advance phase completes in a reach (only after completion of the advance phase are turnouts recognized by the model).

The fourth check is to ensure that turnouts are not located too close to each other in a reach. If a reach contains more than one turnout then all turnout locations for adjacent turnouts are checked. The difference in location for adjacent turnouts must be greater than twice the combined widths of the two turnout structures. For example, if two adjacent turnouts both have widths of one meter then the difference in their respective locations must be at least two meters. Turnout locations refer to the center of a turnout, therefore, the minimum distance between two adjacent turnouts is actually one gate width. The fifth check is to compare the "position" parameter with the depth of the canal reach in which the turnout resides. If the position is at or above the top of the canal bank the turnout cannot discharge any water unless the canal itself overflows. The sixth and final turnout check is for the downstream flow depth for orifice-type turnouts. The downstream depth at zero flow, which is referred to as "DS Depth" in the CDAT program, must be less than the depth of the canal on the upstream side of the turnout. Otherwise, backflow conditions will exist across the turnout and the model will set the discharge rate to zero.

At this time the model will also check the branch linkage data and compare it with the system dimensions data. For each branch other than branch one (if there is more than one branch in the canal system) the upstream linkage specification will be checked against the number of the upstream branch and its dimensions. Obviously, a branch cannot be linked to an upstream branch which is greater than the number of branches in the system. For example, branch two cannot be linked to branch three if there are only two branches in the system. Similarly, if a downstream branch is linked to a turnout in

an upstream branch, the turnout and reach must exist in the upstream branch. For example, branch two cannot be linked to the third turnout in a reach of an upstream branch if this particular reach has only two turnouts. Other linkage checks are made to verify that two different branches are not linked to the same upstream point, and that downstream branches are not linked to wasteway weir turnout structures.

If any of the above checks reveals an error the model will display a short message indicating what the error is and to which branch, reach, and turnout it belongs. If more than 18 separate errors are found then only the first 18 will be displayed. At this time the user can type any key to return to the main menu and the program CDAT (see Section Two) should then be used to correct the turnout or branch linkage data in the configuration data file. A complete list of all error messages from the CANAL model can be found in Appendix C.

Continuing A Previous Simulation

If the user chooses to continue a previous simulation the model also prompts for a data file name. However, this type of data file is created by the hydraulic model CANAL itself and not by the program CDAT. This type of data file is called a suspended simulation file and it contains the canal system configuration data which was originally read by CANAL from a configuration data file created by CDAT. The suspended simulation file will also contain all flow rates, flow levels, control structure settings, and turnout settings as they were at the time the previously suspended simulation terminated. Therefore, the suspended simulation file contains all of the necessary information to continue a previous simulation from exactly the same flow conditions as they existed before, without reading the configuration data from a separate file. In this way, simulations can in effect be continued indefinitely.

The model ensures continuity from the end of one simulation to its continuation in a later simulation by writing all configuration data to the suspended simulation file. This guarantees that none of the physical dimensions of the canal system will change when a simulation continues. If the configuration data are to be altered, the program CDAT must first be used to change the desired data file, and then the simulation must begin again by filling the empty canal system. For example, if the hydraulic roughness value for a reach in a canal system needs to be changed, the altered configuration data file must be read again by CANAL by filling the empty canal system. This is because the model only reads configuration data files when an empty canal system is going to be filled, not when a previous simulation is continued.

Inflow Hydrograph

After a main menu selection to either fill an empty canal system or continue a previous simulation the appropriate data file is read by the model and the program user is prompted to enter an inflow hydrograph (see Figure 10). The inflow hydrograph specifies the system inflow rate to the upstream end of the first reach of the first branch. The program user can enter up to 150 inflow rate values, with each individual value corresponding to one five-minute period of simulation time. This means that if the maximum of 150 values are entered, the simulation time will be twelve and one-half

Number of Inflow Values: 60		Initial Inflow Value: 0.22 m3/s									
(Five-Minute Time Steps)		Max Recommend Inflow: 21.61 m3/s									
1:	0.22	11:	3.00	21:	3.00	31:	3.00	41:	3.00	51:	3.00
2:	0.43	12:	3.00	22:	3.00	32:	3.00	42:	3.00	52:	3.00
3:	0.86	13:	3.00	23:	3.00	33:	3.00	43:	3.00	53:	3.00
4:	1.73	14:	3.00	24:	3.00	34:	3.00	44:	3.00	54:	3.00
5:	3.00	15:	3.00	25:	3.00	35:	3.00	45:	3.00	55:	3.00
6:	3.00	16:	3.00	26:	3.00	36:	3.00	46:	3.00	56:	3.00
7:	3.00	17:	3.00	27:	3.00	37:	3.00	47:	3.00	57:	3.00
8:	3.00	18:	3.00	28:	3.00	38:	3.00	48:	3.00	58:	3.00
9:	3.00	19:	3.00	29:	3.00	39:	3.00	49:	3.00	59:	3.00
10:	3.00	20:	3.00	30:	3.00	40:	3.00	50:	3.00	60:	3.00

Type space or Enter to repeat a value.
Type the End key to start all over again.
Type the Esc key to go back to the main menu.

Figure 10. Screen format for entering the inflow hydrograph.

hours. The duration of a hydraulic simulation is, therefore, implicitly defined by the number of values entered into the inflow hydrograph. The range of accepted inflow rate values is from a minimum of 0.01 up to a maximum of 200 cubic meters per second. This upper limit on the system inflow rate is based on the restrictions in the program CDAT on the maximum canal cross-sectional dimensions and other data which influence the flow capacity of an open channel. The first inflow rate value is always given by the model. If an empty canal system is being filled then the first inflow rate value will be set at two one-hundredths of the maximum recommended inflow rate. The maximum recommended inflow rate is calculated based upon the geometry of the first reach of the first branch. This calculation uses Manning's equation for normal flow depth at the operational

supply level. If a previously suspended simulation is being continued then the first inflow rate value will be equal to the last inflow rate value from the previous simulation. This is to ensure continuity from one simulation to the next.

Of course, the program user should be aware that if the system inflow rate exceeds the total outflows for a sufficient period of time after the system has filled, the canal can be expected to overflow during the simulation. Total outflows include turnout discharges, seepage losses, and reach outflows from terminal points in the canal system. Terminal points are located at the downstream end of the last reach in branches which do not have another branch linked to this point. Turnout discharges include flow over wasteway weirs. The system inflow and total outflow rates are shown at the bottom of all alpha screen displays (except for the second display).

As the user enters values into the inflow hydrograph the model checks the rate of change of inflow from one time step to the next. If the rate of change is too large a beep will be produced and the inflow rate will be set closer to the previous value. The limits for changes in the inflow rate from one time step to the next are based on the following criteria: (1) the inflow rate cannot be more than double or less than one-half of the previous inflow rate value, and (2) the inflow rate cannot either increase or decrease by more than one-sixth of the maximum recommended inflow rate for the first reach of the first branch in a single time step. It is possible that both of these two conditions could be simultaneously violated, in which case the most restrictive of the two conditions will be applied to reduce the rate of inflow change.

Operational Mode

After the inflow hydrograph is entered the user will select one of three operational modes. The first mode is to manually operate the control structures from the keyboard during the simulation. If this option is selected the control structure settings will not change unless the user interrupts the simulation to change them manually. Manual control structure adjustments can be made at any time during a simulation and they can be made as often as necessary.

The second operational mode is to pre-set the control structures before the simulation begins. If this mode is selected the user will be prompted to enter control structure settings for each reach for each five-minute time step. The number of five-minute time steps will be equal to the number of values already entered into the inflow hydrograph by the user. After the simulation begins CANAL will change the control structure settings exactly as entered by the user. The pre-set control structure movements can be manually over-ridden

at any time during the simulation if desired. Additionally, if the initial operational mode during a simulation is the pre-set mode then it can be changed to either the manual or the gate scheduling modes (described below) at any time during a simulation. However, the pre-set mode can only be selected at the beginning of a simulation and not after the simulation has already started.

This second operational mode is essentially the same as the first mode since both use manual control structure settings. The difference is that with the second operational mode the manual control structure movements are specified for the entire duration of the simulation before it begins. If the user already knows what the desired manual control structure adjustments are before a simulation begins then pre-setting the control structures is useful. This is because the simulation does not have to be interrupted each time that an adjustment needs to be made. A principle application of this pre-set mode is to test the difference between using averaged or "banded" control structure setting adjustments versus using the exact adjustments from a previous simulation for which the gate scheduling mode was in effect. Simulations with gate scheduling often produce slight fluctuations in control structure settings while the canal system changes from one steady-state condition and adjusts to another steady-state condition. These fluctuations can be eliminated or reduced by simplifying the exact gate scheduling settings according to the general trends in control structure responses to flow rate changes in the system. These trends can be identified most easily by viewing the summary plots at the end of a simulation (see Figure 8) and by obtaining printouts of the simulation results. Averaged or "banded" control structure setting adjustments are more practical for field operation and represent a compromise between exact model solutions and the feasible operation of a real canal system.

The third operational mode is to enable gate scheduling for target flow depths at the operational supply level in each reach of the canal system. If this mode is selected then the simulation will begin by computing appropriate control structure adjustments with the objective of establishing and maintaining downstream flow levels in each reach at the operational supply level. The operational supply level is specified for each reach independently with the configuration data file editing program CDAT. Gate scheduling can be disabled at any time during a simulation to cause the manual operational mode to take effect. The gate scheduling mode can also be re-enabled at any time during a simulation if desired by the user.

Figure 11 illustrates the possible paths to individual reach status conditions according to the current operational mode during a simulation. The operational mode is always selected by the user, and it can be changed during a simulation. However, the model itself determines the operational status for each canal reach. A canal reach can only be "activated" when the current mode is "scheduling"

and the reach's control structure is of an adjustable type. Further restrictions also apply as described under Gate Scheduling.

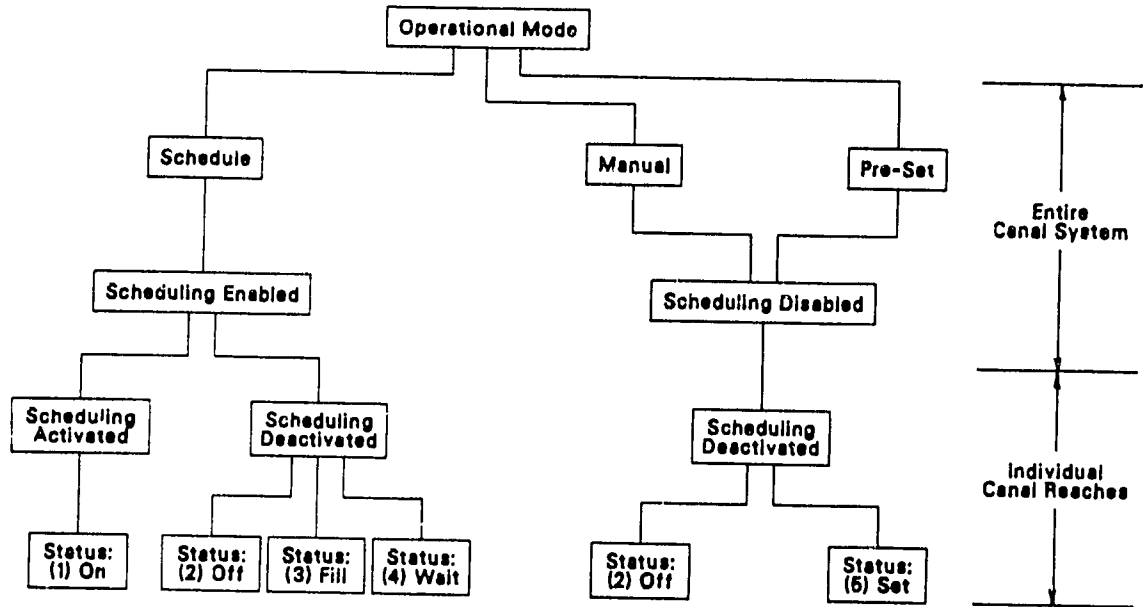


Figure 11. Flow diagram indicating the possible paths to the five different gate scheduling status conditions for individual canal reaches.

Before one of the three operational modes is selected the user can see the current gate scheduling status for the canal system on the screen. There will always be a message indicating whether gate scheduling is currently enabled or currently disabled. If an empty canal system is being filled the current gate scheduling status will be disabled. If a suspended simulation file is read to resume a previous simulation the gate scheduling status will be displayed as either enabled or disabled, depending on the gate scheduling status when the previous simulation terminated. Thus, it is not necessary that the user remember the gate scheduling status from a previous simulation.

SIMULATION INTERRUPTS

Pausing the Simulation

A simulation can be paused at any time by typing the letter "p". A period beeping can be heard, indicating that the simulation has been paused. Any key can be typed to continue the simulation again from exactly the same flow conditions as they existed at the time the simulation was paused.

Help File

The help file can be viewed at any time during a simulation by typing the letter "H". The instructions for viewing the help file are contained on the first page of the file which is displayed at the time the help file is first entered. When the user is finished viewing the help file, CANAL will continue the simulation as before. However, the alpha screen will be displayed automatically, even if the graphics screen was being displayed at the time that the help file was entered.

Changing the Inflow Rate

The system inflow rate can be changed at any time during a simulation by typing the letter "I". The model will respond by displaying the current system inflow rate at the upper left of the graphics screen. If the alpha screen is being displayed then it will be automatically replaced by the graphics screen at this time. The arrow keys can be used to change the current system inflow rate. The up/down arrow keys make changes in increments of 0.10 m³/s and the left/right arrow keys make changes in increments of 0.01 m³/s. As during entry of values into the inflow hydrograph at the beginning of a simulation the absolute magnitude of a change in inflow rate is limited. The new inflow rate cannot be more than double or less than one-half of the current value, and the inflow rate cannot change by more than one-sixth of the maximum recommended inflow rate for the first reach. As previously defined, the maximum recommended inflow rate is computed by the model using Manning's equation for uniform flow in the first reach of the first branch at the operational supply level. If the system inflow rate is changed in this way after the simulation has already begun, then from that point up until the end of the simulation the previously entered inflow hydrograph will be replaced by the new system inflow rate value. All of the remaining

inflow rate values will be set equal to the new inflow rate; however, if the current inflow rate is not changed then the existing inflow hydrograph will still be recognized as it was at the beginning of the simulation. Typing the End key causes the simulation to continue after the new inflow rate has been set using the arrow keys.

Gate Scheduling

The gate scheduling status can be changed at any time during a simulation. Typing the letter "S" causes the simulation to pause and a menu will appear on the screen as illustrated in Figure 12. Selecting the manual operating mode from this menu will cause the system scheduling status to be disabled and all reaches will be deactivated (i.e. status="Off"). If the operational mode is currently manual then nothing will happen and the simulation will continue as before. "Freezing" the current flow levels means that the target flow levels in each reach of the canal system will be set at the current downstream levels, which may or may not be at the operational supply levels as specified in the configuration data file. If the advance phase has not yet finished in all of the reaches in the system then freezing the current flow levels will not work - the command will be ignored by the model and the simulation will continue. Also, if the current mode is already scheduling and all of the reaches are at the operational supply level then freezing does nothing since the target levels remain at the operational supply levels.

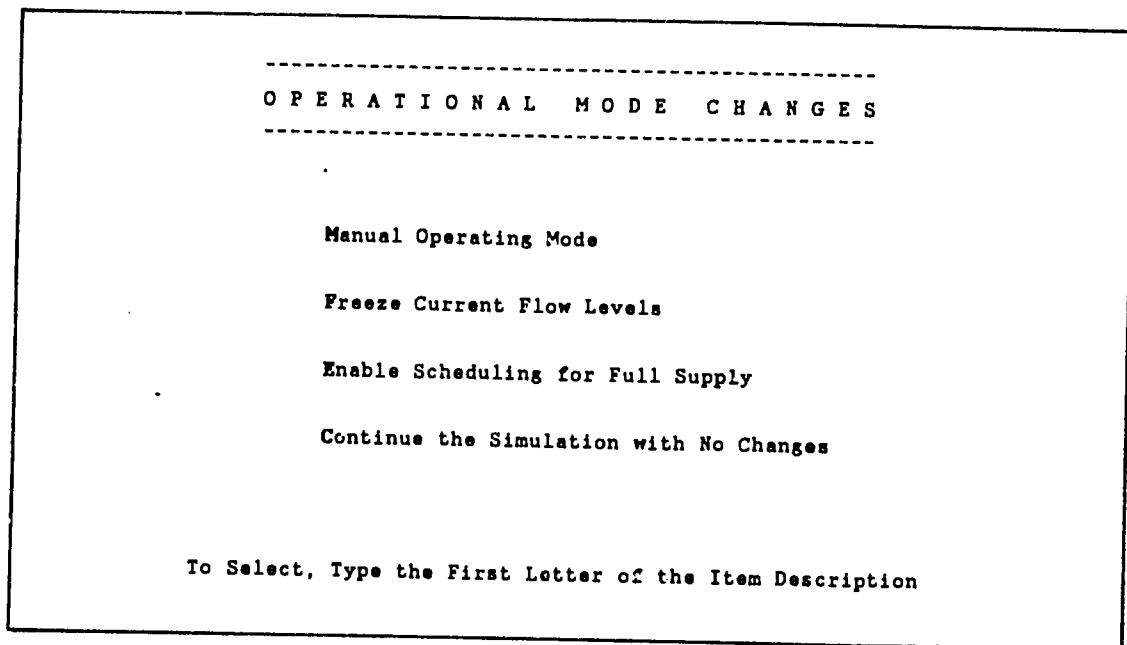


Figure 12. Operational mode changes menu.

The third option from this menu is to enable gate scheduling for the operational (full) supply level in each reach. In this case the model will set all target levels at the operational supply levels and will begin operating the adjustable control structures with the objective of matching actual flow levels in each reach to the operational supply levels. If the current mode is already scheduling and the target levels are at the operational supply levels then nothing will happen and the simulation will continue as before.

Change Turnout Demands

Typing the letter "T" during a simulation will cause the simulation to pause and the turnout demands for the currently displayed branch will appear on the screen. Turnout demands are shown in cubic meters per second and wasteway weirs are indicated by dashes instead of numbers. The format in this display is the same as in the fourth alpha screen display which shows the turnout demands during a simulation. Columns correspond to different reaches and rows correspond to different turnouts in a reach.

Turnout flow demands can be changed by typing in the flow rate value followed by Enter or the space bar. Typing either Enter or the space bar without first entering a number is interpreted by the model as a demand of zero. The maximum turnout demand accepted is 30 cubic meters per second and the minimum is zero. The arrow keys can be used to position the red highlight to other adjustable turnouts in the branch. The current position (branch, reach, and turnout number) is always shown at the lower right of the screen as illustrated in Figure 13. Turnout demands for other branches (if there is more than one branch in the system) can be changed by first typing the letter "B". This causes the display to toggle between each branch in the system, one branch at a time.

When a canal system is initially filled the model sets all turnout demands to zero. However, the user can change these demands at any time during a simulation, and as often as desired. If there are no adjustable turnouts in a canal system the model will ignore an attempt to change turnout demands (i.e. typing the letter "T" during a simulation). On the other hand, if a particular branch has no adjustable turnouts the message "No Adjustable Turnouts" will be shown on the screen instead of the turnout demands. Wasteway weirs are non-adjustable turnouts. Or, if a branch has no turnouts at all the message "No Turnouts in this Branch" will be displayed. If either of the above two messages appear then the user can only type a "B" to change to another branch, or type the End key to quit and continue the simulation. If the End key is typed at any time the simulation will immediately continue and any changes to the turnout demands will be in effect. See also Turnout Scheduling.

CHANGE TURNOUT DEMANDS									
	Reach Number								
	1	2	3	4	5	6	7	8	9
1	-----	0.000	0.000	0.100	0.100	-----			
2	0.000	0.000	0.000	0.100	0.100	0.000			
3	0.000	0.000	0.000	0.200	0.200	0.100			
4	1.200	0.000	0.000	-----	0.300	0.000			
5	0.000	0.100	0.000		0.000	0.125			
6	0.500	0.100	-----		0.000	0.000			
7	0.000	-----			0.150	0.125			
8	-----	0.100							
9	0.000	0.000							
SUM	1.700	0.300	0.000	0.400	0.900	0.350			

Branch 1 Reach 1 Turnout 2

Use arrow keys to move the cursor.
 Type "B" to change the branch number.
 Type the End key when finished with changes.

New Demand = ?
 Old Demand = 0.000

Figure 13. Screen display for changing turnout demands.

Change Control Structure Settings

Typing the letter "C" during a simulation will cause the control structure settings for the currently displayed branch to appear at the top of the screen. If the alpha screen is currently on then it will be automatically replaced by the graphics screen at this time. The control structure settings are displayed in meters and they can be changed by using the arrow keys. As a control structure setting is changed the exact opening in meters will change at the top of the graphics screen and the schematic representation of the structure will open or close in the upper subplot. If there is more than one reach in the currently displayed branch then the control structures corresponding to different reaches can be opened or closed at this time by typing the integer corresponding to the desired reach. When an integer is typed the indicator box will move to the control structure setting which corresponds to the integer value. Again, the arrow keys can be used to change the control structure setting.

If a branch contains non-adjustable control structures then these settings are constant and can be changed neither manually nor automatically through gate scheduling. For these control structures the setting will be shown in meters, the same as for adjustable

control structures, but any attempt to position the indicator box to the setting for these structures will instead produce a beep to remind the user that the setting is not adjustable.

Changing the control structure settings manually in this way can be done at any time during a simulation regardless of the existing operational mode. However, if gate scheduling is enabled then manual adjustment of the control structures will be of little use since the model will resume automatic operation of the structures in subsequent time steps in a continued effort to maintain downstream flow depths at the target levels. If the user wishes to change the operational mode to manual during a simulation then the F4 key should be typed.

The model will not allow sluice gate settings to be changed until the advance phase is finished in a reach. Nor will sluice gates be allowed to be raised above the water level just upstream of the structure (Rdepth). And if the gate is operating under submerged flow conditions then it cannot be raised above the water level on the downstream side of the gate either. Adjustable weirs cannot be raised above the upstream water level if water is already spilling into the next downstream reach. These restrictions of manual control structure adjustments are intended to prevent sudden computational instabilities which can cause the numerical solution to diverge and consequently fail. Typing the End key finishes the turnout setting changes and causes the simulation to continue with the graphics screen displayed.

Keep Current Profiles

Typing the letter "K" (for keep) during a simulation will cause the simulation to be suspended as soon as the current time step is updated for each reach of the canal system. At this time the simulation will stop and the summary menu (see Figure 14) will appear on the screen. From this menu the user can view summary data in both graphical and tabular form. Hardcopy printouts can be obtained and the current system flow status (e.g. flow rates, flow depths, gate settings, etc.) can be saved in a file.

View System Characteristics Data

Typing the letter "V" during a simulation will cause the contents of the configuration data file for the current simulation to be displayed on the screen. This data can only be viewed (not changed) from the CANAL program. The program CDAT must be used to create and modify configuration data files. If the current simulation is the continuation of a previous simulation then the name

of the configuration data file is saved in the suspended simulation data file and it is this file that is retrieved for viewing. Therefore, if the configuration data file has been modified since the canal system for the current simulation was first filled, the data viewed will not correspond exactly to the data which the hydraulic model CANAL is currently using. This is because suspended simulation files store all of the system characteristics data which was originally read from a configuration file, and resuming a suspended simulation does not cause the configuration data file to be read again.

The instructions for viewing the system characteristics data will be shown at the bottom of the screen. After viewing the data file contents the End key can be typed and the simulation will continue. The alpha screen display will be shown when the simulation continues, even if the graphics screen was displayed at the time that a "V" was typed.

FINAL OUTPUT OF SIMULATION RESULTS

Summary Menu

A hydraulic simulation can terminate normally in one of two ways. The first way is when the data from the inflow hydrograph are used up, and the second way is when the "Keep" option is selected by typing a "K" during a simulation (see Simulation Interrupts). Either way, termination is indicated by the display of the summary menu as illustrated in Figure 14. From this menu the user can view tabular and graphical displays of the simulation results, obtain hardcopy printouts, specify the date and starting time, and save the current simulation status in a disc file. Both the date and starting time are used on printouts, and the starting time is used on tabular displays. The starting time is only for the convenience of the program user and to facilitate field application of simulation results. By selecting a starting time the printouts and tabular displays can be made to coincide with the hours of the day so that the summary information is more relevant to the actual application of the simulation results. If a date and starting time are not specified the model will use January 1, 1987 as the date, and 00:00 (midnight) as the starting hour by default.

```
-----  
CURRENT SIMULATION FINISHED  
-----  
  
Return To The Main Menu  
  
Enter Date And Start Time  
  
Save The Current Flow Status  
  
Tabular Summary Of The Simulation  
  
Graphical Summary Of The Simulation  
  
Print Tabular Summary Of The Simulation  
  
To Select, Type the First Letter of the Item Description
```

Figure 14. Summary menu.

Tabular Summary

If more than five time steps have been completed in a simulation a tabular summary of the results can be obtained by selecting the corresponding item from the summary menu. A secondary menu will appear on the screen (see Figure 15) with five options. The user can return to the previous menu by typing a "G", or can enter the current date and starting time by typing an "E". Typing an "F" will cause the model to display reach data on the screen. Typing an "S" will display turnout settings, and typing a "D" will display turnout discharge rates. In either of the above two turnout displays the turnout names entered into the configuration data file from the CDAT program will be written directly below the turnout number. The screen format is always the same, so if a particular canal reach has less than the maximum of nine turnouts then only the columns with existing turnouts will show data values. For example, if a reach has eight turnouts the ninth column will be blank.

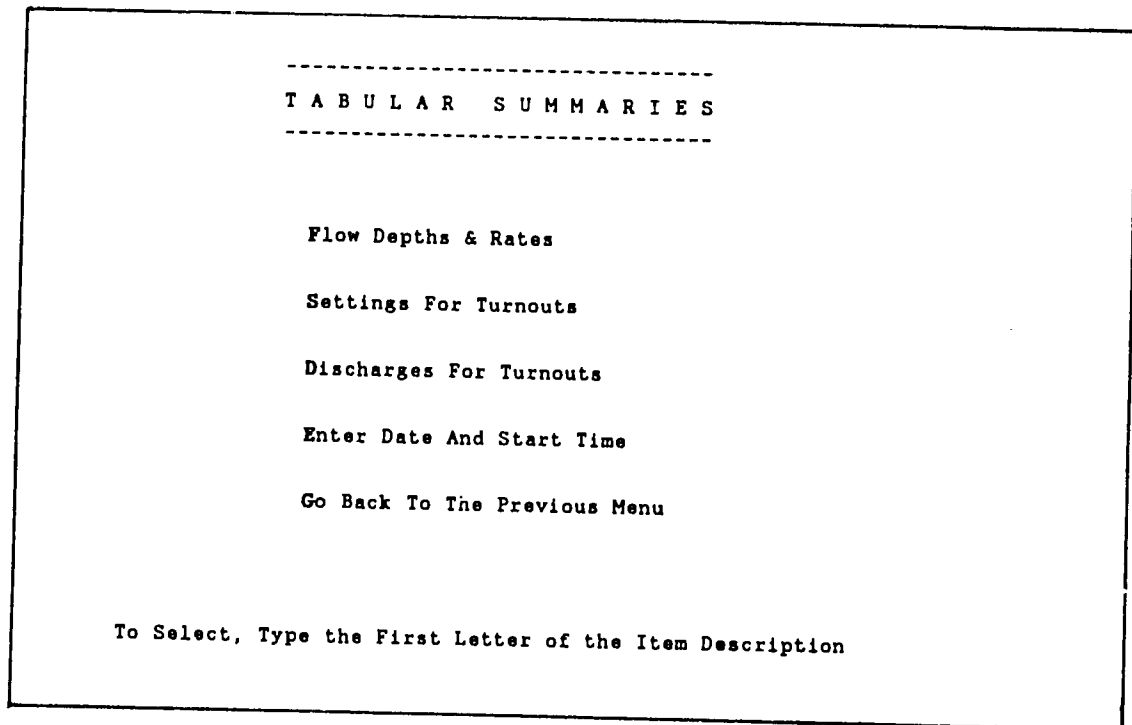


Figure 15. Tabular summaries menu.

The displays show fourteen lines of data for both reach and turnout data. The data can be scrolled up and down one line at a time by using the arrow keys. The PgUp and PgDn keys cause the display to jump to the top and bottom of the data lists, respectively. Initially, the first reach of the first branch will be displayed. The branch number can be changed by typing a "B" and the

reach number can be changed by typing an "R". If either of these two keys are pressed repeatedly the display will go through all branches in the system, or all reaches in a given branch, and then begin again with the first branch or reach. These instructions are always shown at the bottom of the screen while viewing tabular summary data, as illustrated in Figure 16. This data is the same that is displayed in graphical form for graphical summaries, and in hardcopy form for printouts. The end key is used to exit back to the tabular summaries menu.

CANAL HYDRAULIC MODEL							
Canal Reach Data for Branch 1 Reach 1							
Time (hrs)	Control Setting (m)	Upstream Flow Depth (m)	Downstream Flow Depth (m)	Reach Inflow (m ³ /s)	Reach Outflow (m ³ /s)	Branch Inflow (m ³ /s)	System Outflow (m ³ /s)
06:00	0.03	0.00	0.00	0.12	0.00	0.12	0.00
06:15	0.03	0.12	0.00	0.12	0.00	0.12	0.01
06:30	0.03	0.14	0.08	0.22	0.04	0.22	0.02
06:45	0.03	0.15	0.20	0.42	0.07	0.42	0.05
07:00	0.05	0.26	0.35	0.84	0.09	0.84	0.08
07:15	0.09	0.29	0.49	1.20	0.13	1.20	0.09
07:30	0.12	0.31	0.58	2.00	0.27	2.00	0.11
07:45	0.17	0.38	0.70	2.00	0.38	2.00	0.12
08:00	0.20	0.47	0.75	2.00	0.51	2.00	0.26
08:15	0.31	0.59	0.78	2.00	0.60	2.00	0.59
08:30	0.45	0.67	0.79	2.50	0.66	2.50	1.11
08:45	0.44	0.69	0.80	2.50	0.78	2.50	1.57
09:00	0.44	0.70	0.81	2.50	0.78	2.50	1.63
09:15	0.44	0.70	0.81	2.50	0.78	2.50	1.70

Use arrow keys, PgUp, PgDn, "B"branch, "R"reach, or type End to exit.

Figure 16. Tabular display for canal reach data.

Graphical Summary

If more than five time steps have been completed in a simulation a graphical summary of the results can be obtained by selecting the corresponding item from the summary menu. The summary plots are displayed one branch at a time for each branch of the canal system, beginning with the first branch (see Figure 17). The upper left subplot shows the inflow hydrograph to the first reach of a branch

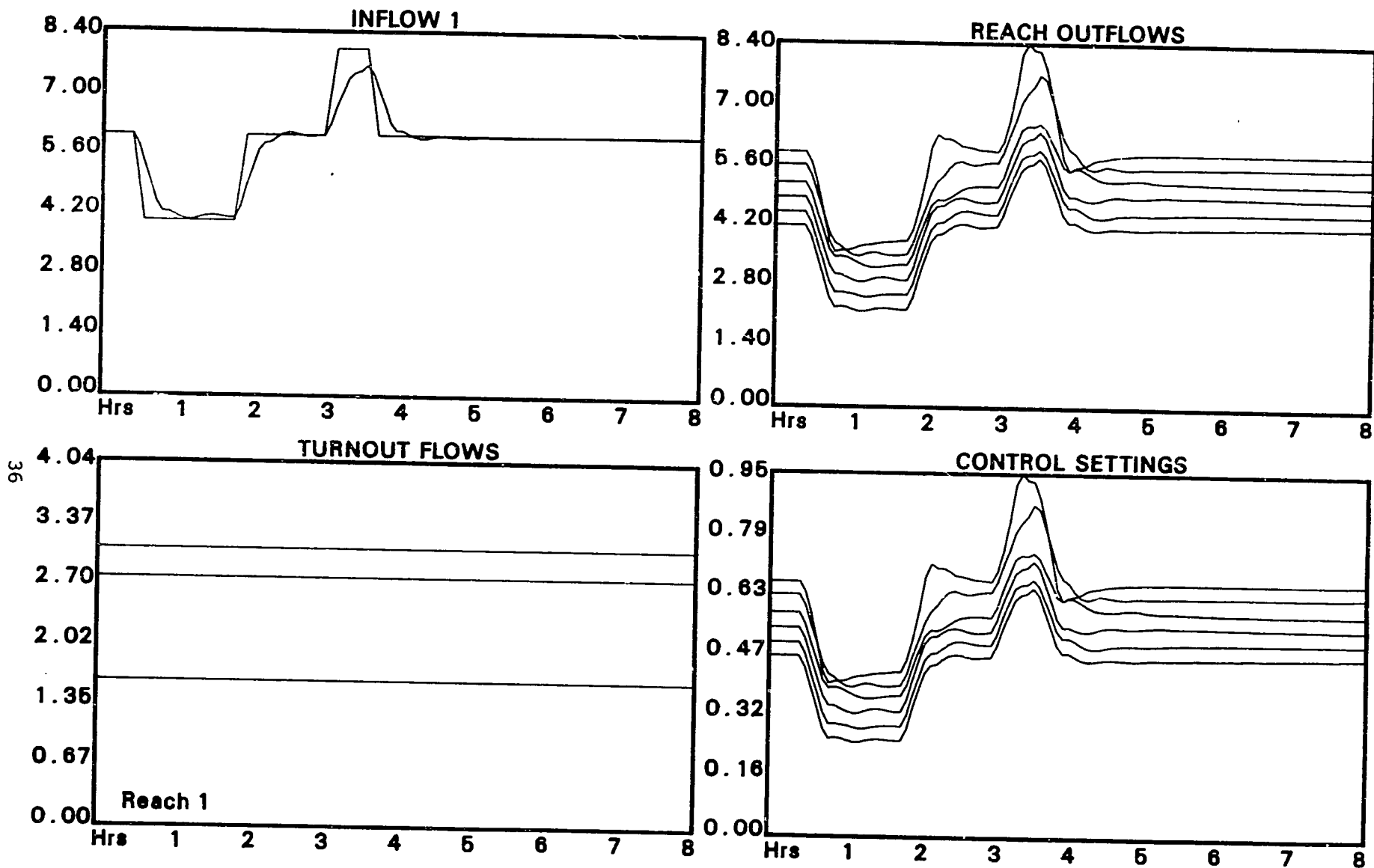


Figure 17. Initial graphical display of simulation results.

during the simulation. For the first branch this inflow hydrograph is constructed using the inflow hydrograph data that were entered by the user at the beginning of the simulation. If the system inflow rate was changed during the simulation by the user then the inflow hydrograph for the first branch will look different than it would if no changes had been made. For the first branch only, a curve showing the total system outflow rate will also be drawn in the upper left subplot in red. By comparing the system inflow and outflow curves the user can see when the system was at a steady-state condition (if at all during the simulation). Where the system inflow and total outflow curves coincide, the system was at a steady-state flow condition. And, where the vertical difference between the system inflow and total outflow curves is large, the system was in a highly transient flow condition. For branches two, three, and four (if they exist in a given canal system) the inflow hydrograph comes from either: (1) the outflow hydrograph from the last reach of the upstream branch, or (2) the discharge hydrograph through a turnout in the upstream branch to which this branch is linked. This depends on how the canal system is configured as specified by the branch linkage data. At the top of the upper left subplot the number of the branch to which the curves correspond will be displayed after the word "INFLOW".

The upper right subplot shows reach outflow hydrographs for each reach in a branch. These curves show the flow rates passing through control structures for the duration of the simulation. Thus, this subplot shows the flow rates passing from one reach to another, or leaving the system at a terminal point. The downstream end of the last reach in a branch to which no downstream branch is linked is a terminal point in the system. The lower left subplot shows turnout discharge hydrographs. The turnout discharge hydrographs will be displayed one reach at a time for each reach in a branch. After the turnout hydrographs for one reach have been displayed the output will stop and the user must type any key to erase the curves and draw the new ones for the next reach in the branch. The reach number for the turnout hydrograph curves displayed is shown in the lower left corner of the subplot. If all turnout discharges are zero for a given reach the model will skip past it and on to the next reach. When the turnout hydrograph curves for the last reach in a branch have been drawn, the control structure settings will be displayed for each reach of the branch in the lower right subplot.

The output will again stop at this point and the user must type any key to erase the lower two subplots and replace them with upstream and downstream flow depth curves for each reach of the branch (see Figure 18). The upstream flow depth curves correspond to the flow depths at the extreme upstream end of a canal reach and the downstream flow depth curves correspond to the flow depths at the extreme downstream end of each reach. As previously noted, the extreme downstream end of a reach is just upstream of the control

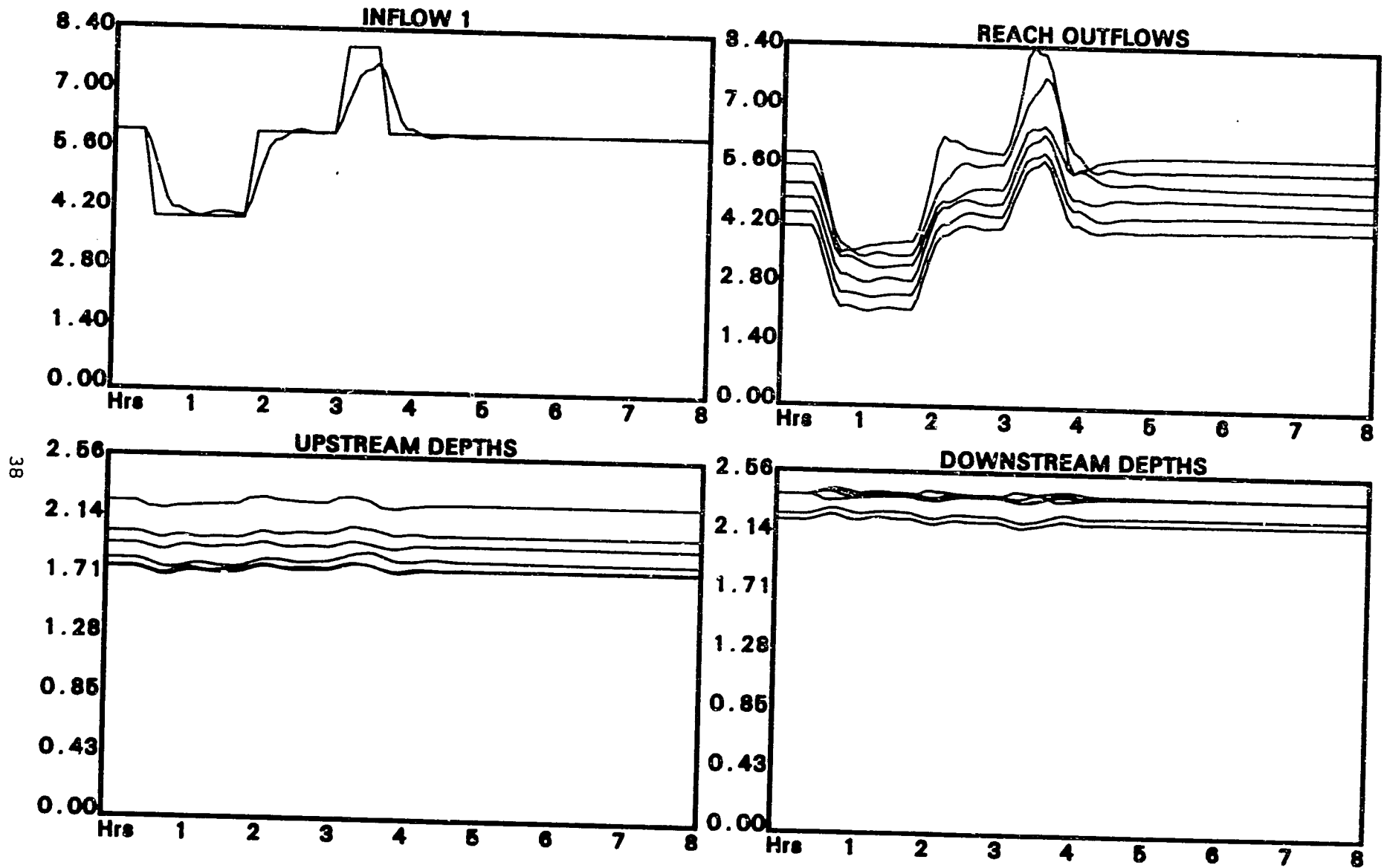


Figure 18. Final graphical display of simulation results.

structure for the reach. At this point the output will stop again and the user can type any key to erase the entire screen and begin again with the next branch in the system (if there is a next branch). All flow rates are in cubic meters per second and flow depths and control structure settings are in meters. The abscissa for all subplots is scaled to the duration of the simulation in hours.

For subplots two, three, and four, multiple curves can be displayed simultaneously. The order in which the curves are drawn is from reach (or turnout) number one on up to the highest numbered reach (or turnout) for the subplot. The curve colors are as follows:

Reach or Turnout Number	Curve Color
1	Blue
2	Green
3	Cyan
4	Red
5	Magenta
6	Yellow
7	White
8	Blue
9	Green

The colors for the eighth and ninth curves (if they are present) are exactly the same as those of the first two curves.

Hardcopy Printouts

The user may choose to have tables printed which show the simulation results. This is actually the same information displayed in the graphical and tabular displays, but as hardcopy printouts. These printouts can only be obtained if the simulation has completed at least five time steps. The tables will show canal system flow rates, flow depths, control structure settings, and turnout settings and discharges in fifteen-minute intervals for the duration of the simulation. Printouts can be obtained for all reaches in a canal system, or only for selected reaches. Printouts can be aborted at any time after they have begun by typing Ctrl-End. If the maximum simulation duration of 150 time steps is used then the printouts for each reach will fill one full page of paper. The correct date and starting time should be specified by the user before making hardcopy printouts. This can be done from the summary menu by typing an "E". An example of a printout from a hydraulic simulation is given in Tables 2 and 3.

Table 2. Sample printout for canal reach data.

CANAL HYDRAULIC MODEL
Canal Reach Data for Branch 1 Reach 1

Date: 7 Oct 1987
Reach: CHECK 4

Config File: LNOR
Suspnd File: LNOR

Time (hrs)	Control Setting (m)	Upstream Flow Depth (m)	Downstream Flow Depth (m)	Reach Inflow (m3/s)	Reach Outflow (m3/s)	Branch Inflow (m3/s)	System Outflow (m3/s)
08:00	0.65	2.15	2.46	6.00	10.44	6.00	2.47
08:15	0.31	2.06	2.52	6.00	5.42	6.00	2.77
08:30	0.31	2.06	2.48	7.00	5.43	7.00	2.77
08:45	0.35	2.06	2.43	7.00	5.75	7.00	2.77
09:00	0.35	2.04	2.42	7.00	5.59	7.00	2.77
09:15	0.34	2.02	2.40	7.00	5.35	7.00	2.77
09:30	0.43	2.11	2.36	10.00	6.43	10.00	2.79
09:45	0.51	2.14	2.33	10.00	7.28	10.00	2.81
10:00	0.52	2.14	2.33	10.00	7.38	10.00	2.82
10:15	0.53	2.14	2.34	10.00	7.40	10.00	2.84
10:30	0.53	2.14	2.34	10.00	7.41	10.00	2.85
10:45	0.53	2.14	2.34	10.00	7.41	10.00	2.87
11:00	0.53	2.14	2.34	10.00	7.41	10.00	2.88
11:15	0.54	2.14	2.34	10.00	7.42	10.00	2.90
11:30	0.54	2.14	2.34	10.00	7.42	10.00	2.92
11:45	0.54	2.14	2.34	10.00	7.42	10.00	2.93
12:00	0.54	2.14	2.34	10.00	7.42	10.00	2.95
12:15	0.54	2.14	2.34	10.00	7.42	10.00	2.96
12:30	0.54	2.14	2.34	10.00	7.42	10.00	2.98
12:45	0.54	2.14	2.34	10.00	7.42	10.00	2.99
13:00	0.54	2.14	2.34	10.00	7.43	10.00	3.01
13:15	0.54	2.14	2.34	10.00	7.43	10.00	3.02
13:30	0.52	2.14	2.34	10.00	7.15	10.00	3.40
13:45	0.51	2.13	2.34	10.00	7.12	10.00	3.43
14:00	0.51	2.13	2.34	10.00	7.10	10.00	3.44
14:15	0.51	2.13	2.34	10.00	7.09	10.00	3.46
14:30	0.55	2.16	2.33	11.00	7.59	11.00	3.48
14:45	0.57	2.18	2.32	11.00	7.73	11.00	3.49
15:00	0.58	2.18	2.32	11.00	7.85	11.00	3.51
15:15	0.58	2.18	2.33	11.00	7.89	11.00	3.53
15:30	0.59	2.19	2.33	11.00	7.92	11.00	3.55
15:45	0.59	2.19	2.33	11.00	7.95	11.00	3.56
16:00	0.59	2.19	2.33	11.00	7.97	11.00	3.58

Table 3. Sample printout for canal turnout data.

CANAL HYDRAULIC MODEL
Canal Turnout Data for Branch 1 Reach 1

Date: 7 Oct 1987
Reach: CHUCK 4

Config File: LNOR
Suspnd File: LNOR

Time (hrs)	Turnout Openings (m) and Discharges (m ³ /s)								
	1 MW 1	2 1-2	3 3-4	4 R-1L	5 5-6	6 R-2L	7 7	8 MW 2	9 8
08:00	--- / 0.00	0.04/ 0.10	0.02/ 0.05	0.10/ 0.90	0.03/ 0.08	0.04/ 0.10	0.07/ 0.09	--- / 0.00	0.07/ 0.10
08:15	--- / 0.00	0.04/ 0.10	0.02/ 0.05	0.14/ 1.19	0.03/ 0.08	0.04/ 0.10	0.07/ 0.10	--- / 0.00	0.07/ 0.10
08:30	--- / 0.00	0.05/ 0.10	0.02/ 0.05	0.14/ 1.20	0.03/ 0.08	0.04/ 0.10	0.08/ 0.10	--- / 0.00	0.07/ 0.10
08:45	--- / 0.00	0.05/ 0.10	0.02/ 0.05	0.14/ 1.20	0.03/ 0.08	0.04/ 0.10	0.08/ 0.10	--- / 0.00	0.07/ 0.10
09:00	--- / 0.00	0.05/ 0.10	0.02/ 0.05	0.14/ 1.19	0.03/ 0.08	0.04/ 0.10	0.08/ 0.10	--- / 0.00	0.07/ 0.10
09:15	--- / 0.00	0.05/ 0.10	0.03/ 0.05	0.14/ 1.20	0.03/ 0.08	0.04/ 0.10	0.08/ 0.10	--- / 0.00	0.07/ 0.10
09:30	--- / 0.00	0.05/ 0.10	0.03/ 0.05	0.14/ 1.20	0.03/ 0.08	0.04/ 0.10	0.08/ 0.10	--- / 0.00	0.07/ 0.10
09:45	--- / 0.00	0.05/ 0.10	0.02/ 0.05	0.14/ 1.20	0.03/ 0.08	0.04/ 0.10	0.09/ 0.10	--- / 0.00	0.07/ 0.10
10:00	--- / 0.00	0.05/ 0.10	0.02/ 0.05	0.14/ 1.20	0.03/ 0.08	0.04/ 0.10	0.09/ 0.10	--- / 0.00	0.07/ 0.10
10:15	--- / 0.00	0.05/ 0.10	0.02/ 0.05	0.14/ 1.20	0.03/ 0.08	0.04/ 0.10	0.09/ 0.10	--- / 0.00	0.07/ 0.10
10:30	--- / 0.00	0.05/ 0.10	0.02/ 0.05	0.14/ 1.20	0.03/ 0.08	0.04/ 0.10	0.09/ 0.10	--- / 0.00	0.07/ 0.10
10:45	--- / 0.00	0.05/ 0.10	0.02/ 0.05	0.14/ 1.20	0.03/ 0.08	0.04/ 0.10	0.09/ 0.10	--- / 0.00	0.07/ 0.10
11:00	--- / 0.00	0.05/ 0.10	0.02/ 0.05	0.14/ 1.20	0.03/ 0.08	0.04/ 0.10	0.09/ 0.10	--- / 0.00	0.07/ 0.10
11:15	--- / 0.00	0.05/ 0.10	0.02/ 0.05	0.14/ 1.20	0.03/ 0.08	0.04/ 0.10	0.09/ 0.10	--- / 0.00	0.07/ 0.10
11:30	--- / 0.00	0.05/ 0.10	0.02/ 0.05	0.14/ 1.20	0.03/ 0.08	0.04/ 0.10	0.09/ 0.10	--- / 0.00	0.07/ 0.10
11:45	--- / 0.00	0.05/ 0.10	0.02/ 0.05	0.14/ 1.20	0.03/ 0.08	0.04/ 0.10	0.09/ 0.10	--- / 0.00	0.07/ 0.10
12:00	--- / 0.00	0.05/ 0.10	0.02/ 0.05	0.14/ 1.20	0.03/ 0.08	0.04/ 0.10	0.09/ 0.10	--- / 0.00	0.07/ 0.10
12:15	--- / 0.00	0.05/ 0.10	0.02/ 0.05	0.14/ 1.20	0.03/ 0.08	0.04/ 0.10	0.09/ 0.10	--- / 0.00	0.07/ 0.10
12:30	--- / 0.00	0.05/ 0.10	0.02/ 0.05	0.14/ 1.20	0.03/ 0.08	0.04/ 0.10	0.09/ 0.10	--- / 0.00	0.07/ 0.10
12:45	--- / 0.00	0.05/ 0.10	0.02/ 0.05	0.14/ 1.20	0.03/ 0.08	0.04/ 0.10	0.09/ 0.10	--- / 0.00	0.07/ 0.10
13:00	--- / 0.00	0.05/ 0.10	0.02/ 0.05	0.14/ 1.20	0.03/ 0.08	0.04/ 0.10	0.09/ 0.10	--- / 0.00	0.07/ 0.10
13:15	--- / 0.00	0.05/ 0.10	0.02/ 0.05	0.14/ 1.20	0.03/ 0.08	0.04/ 0.10	0.09/ 0.10	--- / 0.00	0.07/ 0.10
13:30	--- / 0.00	0.05/ 0.10	0.05/ 0.10	0.18/ 1.50	0.04/ 0.10	0.04/ 0.10	0.09/ 0.10	--- / 0.00	0.07/ 0.10
13:45	--- / 0.00	0.05/ 0.10	0.05/ 0.10	0.18/ 1.50	0.04/ 0.10	0.04/ 0.10	0.09/ 0.10	--- / 0.00	0.07/ 0.10
14:00	--- / 0.00	0.05/ 0.10	0.05/ 0.10	0.18/ 1.50	0.04/ 0.10	0.04/ 0.10	0.09/ 0.10	--- / 0.00	0.07/ 0.10
14:15	--- / 0.00	0.05/ 0.10	0.05/ 0.10	0.18/ 1.50	0.04/ 0.10	0.04/ 0.10	0.09/ 0.10	--- / 0.00	0.07/ 0.10
14:30	--- / 0.00	0.05/ 0.10	0.05/ 0.10	0.18/ 1.50	0.04/ 0.10	0.04/ 0.10	0.09/ 0.10	--- / 0.00	0.07/ 0.10
14:45	--- / 0.00	0.04/ 0.10	0.05/ 0.10	0.18/ 1.50	0.04/ 0.10	0.04/ 0.10	0.09/ 0.10	--- / 0.00	0.07/ 0.10
15:00	--- / 0.00	0.04/ 0.10	0.05/ 0.10	0.18/ 1.50	0.04/ 0.10	0.04/ 0.10	0.09/ 0.10	--- / 0.00	0.07/ 0.10
15:15	--- / 0.00	0.04/ 0.10	0.05/ 0.10	0.18/ 1.50	0.04/ 0.10	0.04/ 0.10	0.09/ 0.10	--- / 0.00	0.07/ 0.10
15:30	--- / 0.00	0.04/ 0.10	0.05/ 0.10	0.18/ 1.50	0.04/ 0.10	0.04/ 0.10	0.09/ 0.10	--- / 0.00	0.07/ 0.10
15:45	--- / 0.00	0.04/ 0.10	0.05/ 0.10	0.18/ 1.50	0.04/ 0.10	0.04/ 0.10	0.09/ 0.10	--- / 0.00	0.07/ 0.10
16:00	--- / 0.00	0.04/ 0.10	0.05/ 0.10	0.18/ 1.50	0.04/ 0.10	0.04/ 0.10	0.09/ 0.10	--- / 0.00	0.07/ 0.10

Saving the Current Status

When the user types an "S" from the summary menu the model will prompt for a file name under which the current flow rates, flow depths, control structure and turnout settings, etc., will be saved in a disc file. This file name can be from one to five characters long and it will be automatically appended with a & symbol before being opened in the data directory. The model always appends suspended simulation data files with the & symbol. As previously stated the data directory is specified by the DOS path from the CDAT program. If the model finds a file with the same name in the data directory a warning message will be displayed on the screen giving the user the option of overwriting the existing contents or selecting a new name. This save operation must be performed if the user wishes to continue the simulation at a later time, otherwise the current status will be lost. If the user attempts to save the data after having saved it already for a given simulation the model will display a message indicating that it has already been saved. Therefore, the status for a particular simulation can only be saved once.

DEFINITION OF TERMS

Control Structure

A control structure is defined as any physical structure in the canal system that significantly affects the flow regime upstream and or downstream of its location. Control structures are in-line structures only and are different than turnout structures. Control structures may have adjustable settings or they may be nonadjustable. Examples of adjustable control structures are sluice gates and weirs. Examples of non-adjustable control structures are fixed-sill weirs, culverts, and inverted siphons. In the model canal reaches within a branch are always separated by control structures.

The model treats sluice gates as either rectangular or circular orifices which may be either operating in the free-flow or submerged flow regimes, depending on the gate opening, the upstream and downstream water levels, and the elevation changes across the structure. However, the model does not do a good job of simulating flow through a sluice gate when the gate opening is above the upstream water level, that is, when the bottom of the gate does not touch the water. Sluice gates include overflow side weirs with fixed sill height by default in the hydraulic model. If a real control structure is a sluice gate with no overflow side weirs, the sill height of the weir can be set to the maximum canal depth or higher. If the weir sill height is at or above the top of the canal then there will never be any weir flow unless the canal itself is overflowing. Thus, the weir is effectively non-existent. Sluice gates are not allowed to be completely closed since this would cut off the flow to downstream reaches.

Weirs are assumed to operate under free-flow conditions at all times. The weir equations in the model do not take into account flow contractions. Furthermore, the weir equations are for sharp-crested weirs. Weirs can be either adjustable or non-adjustable, the difference being that non-adjustable (fixed) weirs are not eligible for gate scheduling or manual operation. Other types of control structures can be added to the model if necessary since the computer program is structured in modular form and can be modified as needed by the program developers.

Turnout Structure

A turnout structure is similar to a control structure, except instead of being an "in-line" structure it is an "off-line"

structure. Turnouts can be circular or rectangular orifices, adjustable weirs, and non-adjustable wasteway overflow weirs. Turnouts may be small individual farm turnouts or they may be large lateral turnouts. Any place that water can potentially be removed from the main channel is defined to be a turnout. However, seepage outflow and evaporation losses are not considered to be turnout flow since they are usually not point discharges and cannot be managed in normal operation.

Turnouts can be located at any position along a canal reach, and on either side of the canal. As noted previously, two turnouts may not exist at the same location even if one is on the left canal bank and one on the right. They must be entered into the model at different locations. In this case, one of the turnouts should be located slightly upstream of the true location in the real canal system, and the other slightly downstream.

Seepage Rate

The model calculates the seepage rate as a flow rate in cubic meters per second. The seepage rate may also reflect evaporation losses and small leakages from closed turnouts. If a canal reach is in a cut section below the adjacent ground surface it is possible to have negative seepage due to seepage inflow to the canal reach. The seepage rate entered by the program user is in millimeters per day. The model converts this rate into meters per second and multiplies it by the canal reach length and the average wetted perimeter to yield a flow rate in cubic meters per second. Point calculations of the wetted perimeter along a canal reach are made at computational nodes. The seepage outflow (or inflow) is re-calculated during each time step, and during a simulation this seepage rate for a canal reach will change only with changes in flow depth. As the flow depth approaches zero the seepage rate also approaches zero; that is, there is no base flow associated with seepage for very small flow depths (even though the wetted perimeter approaches the canal base width and not zero). This is to prevent seepage losses from overwhelming the advance of water in an empty canal reach which can result in a convergence failure of the numerical solution of the hydraulic equations.

Data Files

There are two kinds of external data files that are used by the hydraulic model. The first kind is called a "configuration data file" and is created and modified with the program CDAT. Configuration data files are formatted as random access files and

contain canal system dimensions, channel geometry, control structure and turnout types and locations, and seepage rates. The model reads this type of data file before simulating the filling of an empty canal system, but never writes to it or modifies it in any way. Configuration data files are stored on a disc in a directory that is specified by the user-inputted DOS path. These file names are appended automatically with a @ symbol to reduce the possibility of confusion with other data files in the same disc directory.

The second kind of external data file is called a "suspended simulation data file" and it is both created and read by the hydraulic model CANAL. The program CDAT does not access this second type of data file. Suspended simulation files are formatted as sequential access files and they contain all of the information from a configuration data file that was read by the model at a previous time when a canal system was initially filled. They also contain data describing the flow profiles and status of a previously terminated simulation. At the completion of every simulation the program user can choose to save the current flow profiles and system status in an external data file. This external file is a suspended simulation file and can be read at the beginning of a new simulation to continue a previous simulation. Suspended simulation data files are also stored on a disc in a directory that is specified by the DOS path, and their names are automatically appended with a & symbol to reduce the possibility of confusion with other data files. The same name can be used for a configuration data file and for a suspended simulation file because the model can distinguish between the two by checking whether a @ or a & symbol was appended to the name.

Advance Phase

The advance phase is the period of simulation time during which a canal reach is being filled with water. As soon as water gets to the end of a canal reach the advance phase terminates for the reach. If the control structure at the end of the reach is a sluice gate then the phase becomes equivalent to a blocked end condition. The blocked end condition will change to "grid adjust" (see Grid Adjust) for one time step as soon as the downstream water level is high enough to ensure that upstream submergence will be maintained. After this the phase will become post-advance and the model will allow water to pass through the sluice gate into the next downstream reach. If the control structure at the end of the reach is a weir then the "grid adjust" phase will immediately follow the advance phase. After this the weir can discharge water into the next downstream reach as soon as the water level upstream of the weir is high enough to spill over the sill.

Grid Adjust

The grid adjust phase involves modification of the internal node arrangement for the hydraulic calculations in a canal reach. Internal nodes are imaginary points which the model places along the length of each canal reach during the advance phase. Calculations for flow rate and flow depth are made only at these nodes, which should not be spaced too close together, nor too far apart. During grid adjust nodes may be added or deleted automatically by the hydraulic model to provide a computationally favorable node arrangement. This is because the original placement of nodes depends on the duration of the advance phase, which in turn depends on a number of system configuration and operational parameters during a hydraulic simulation. At this time turnout nodes are also inserted into the grid if there are any turnouts in the reach. For this reason there are no turnout flows until the grid adjust phase is completed (since the model does not recognize the presence of turnouts until this time).

Gate Scheduling

Gate scheduling is one of the three operational modes of the hydraulic model CANAL. When gate scheduling is enabled the model operates the adjustable control structures automatically in an effort to maintain constant water levels on the upstream side of each structure (the upstream side of each structure is the downstream end of the reach in which the structure is located). The target water level during gate scheduling may be either the operational supply level or the current water levels at the time that gate scheduling is enabled (see Simulation Interrupts). The operational supply level is specified for each reach individually in the configuration data file for a canal system. This level is defined as a percentage of the canal depth for each reach.

When gate scheduling is enabled the entire canal system will be in the gate scheduling mode; however, individual reaches may be either activated or deactivated. The default alpha screen display will show a scheduling status of "On" for reaches which are activated during a simulation. Non-adjustable control structures (e.g. culverts and fixed weirs) can never be activated since the model "knows" that their setting cannot be changed. Reaches with adjustable control structures may also be deactivated when gate scheduling is enabled if: (1) the advance phase is not yet completed for the reach, or (2) some operational limitations have been encountered. The operational limitations which will cause a reach to be deactivated are described below.

When gate scheduling is enabled during the filling of a canal system the adjustable control structure openings will remain at their initial settings at least until the next downstream reach has completed the advance phase. For example, while the water in reach four is still advancing to the end of the reach, the control structure setting in reach three will not change from its initial value. After reach four has completed the advance phase the model will begin operating the control structure in reach three with the objective of changing the actual water level upstream of the structure so that it approaches the target level. During filling this usually means that the actual water level must be raised up to the target level, and that the flow through the structure and into reach four must be reduced (using the above example). If the system inflow rate is sufficiently high, the actual water level will eventually come close to the target level. When this happens, the reach will be activated and in subsequent time steps the model will try to maintain the actual water level at the target level.

The model tries to maintain the actual water level on the upstream side of each adjustable control structure at the target level for all activated reaches. This is performed in an indirect way. The model will calculate a steady-state discharge rate for each reach based upon the system inflow rate and all system outflow rates from each structure in the upstream direction. For example, if the model is currently updating the flow conditions in reach two of branch one, and if this reach is currently activated, the steady-state discharge rate from reach two into reach three will be equal to the system inflow rate minus all turnout and seepage flows in reaches one and two. Figure 19 illustrates this example with some flow rate values. For the purpose of calculating these steady-state discharge rates the system inflow rate is taken to be the average of the system inflow rates from the past three time steps. This averaging slightly delays the reaction of the model to inflow changes to allow the reaches to adjust to the new inflow rate more smoothly.

After calculating the steady-state discharge rate for a reach the model will check the difference between the current actual water level and the target level on the upstream side of the reach's control structure. The model will then calculate a new discharge rate for the reach by either increasing or decreasing the steady-state value according to the absolute magnitude of the difference in actual and target water levels. If this difference is large compared to the canal depth then the adjustment on the steady-state discharge value will be relatively large. And of course, if the actual water level is above the target level, the new discharge rate must be higher than the steady-state rate since the actual water level needs to be lowered. The opposite is true when the actual water level is below the target level. When the difference between actual and target water levels becomes small, the new discharge rate will be

close to the steady-state discharge rate, and the reach will be near a steady-state flow condition. During gate scheduling the model is always "aiming" at a system-wide steady-state flow condition even though the system flow rate and distribution of flows changes.

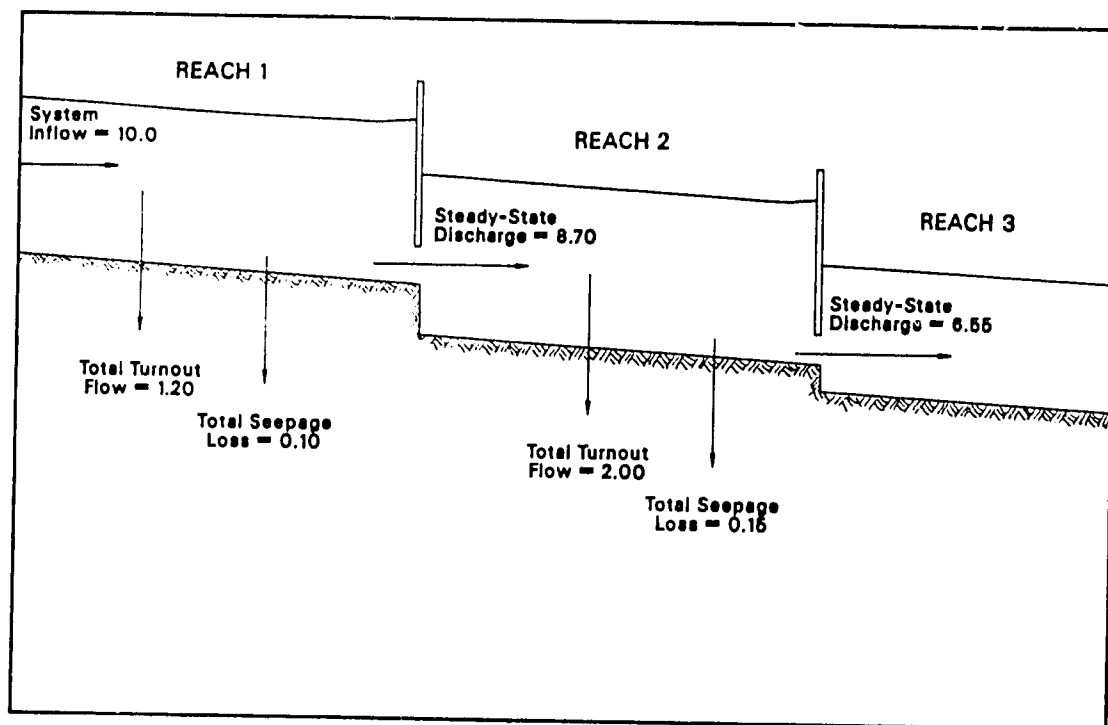


Figure 19. Illustrated example of steady-state reach discharge calculations during gate scheduling.

Once the model calculates this new discharge rate based on the steady-state discharge rate and the deviation between actual and target water levels, a series of checks will be made. These checks are to compare the current reach discharge rates with the discharge rates from previous time steps so that potentially problematic flow conditions can be avoided if possible. The first check is to determine whether the reach outflow is fluctuating. This check is performed by comparing the reach discharge rates from the past two time steps to the new discharge rate. If the reach discharge rate from the previous time step is greater than both the new rate and the rate from two time steps before, the model will attempt to stabilize the discharge rate by averaging these three rates and setting the new discharge rate equal to this average. Similarly, if the reach discharge rate from the previous time step is less than both the new rate and the rate from two time steps before the model will average these three discharge rates and set the new reach discharge rate

equal to this average value. This averaging will always be performed for the above two conditions in which a positive or negative "spike" exists in the reach discharge hydrograph, except when the differences in flow rates between consecutive time steps is very small.

The second check is to prevent the reach discharge rate from changing too fast in a single time step. If the new discharge rate is less than one-half of the discharge rate from the previous time step, the model will set the new rate equal to one-half of the rate from the previous time step. And, if the new discharge rate is more than nine-fifths of the rate from the previous time step the model will set the new rate equal to nine-fifths of the rate from the previous time step. This second check tends to prevent the model from over-reacting to a sudden large change in the flow conditions. The third check is to compare the new discharge rate to the extreme upper and lower flow rate limits for the reach as defined in the model. The extreme lower limit is 0.01 m³/sec. Therefore, if possible, the model will not allow a reach discharge rate to be lower than this minimum value during gate scheduling. Nor will the model allow a reach discharge rate to exceed the design flow capacity of a reach while that reach is activated during gate scheduling. The design flow capacity is defined using Manning's equation under uniform flow conditions at the operational supply level for a canal reach.

After the model has calculated the new reach discharge rate and performed the above three checks this new rate will be fixed in the hydraulic equations for the current canal reach and time step. The solution to the hydraulic equations will then yield a flow depth and a control structure setting that will accommodate the fixed discharge rate. If the flow depth on the upstream side of the structure is not at the target level then it should at least be closer to the target level than it was at the previous time step since the fixed discharge rate through the structure was calculated in such a way as to reduce the difference between actual and target water levels. However, this may not always be true because during transient conditions reaches may in effect be competing with one another for the available water in the canal system. If the system inflow rate is enough to satisfy total system outflows the canal system will eventually reach a steady-state condition, even though during one or two time steps the actual water level may not be approaching the target level for some of the reaches.

It was mentioned previously that the model tries to maintain the actual water level at the downstream end of each reach at the target level in an indirect way. The method is indirect because instead of fixing the flow depth at the downstream end of a reach to be equal to the target flow level, the reach discharge through the control structure at the downstream end of the reach is fixed. Scheduling the structure movements indirectly as such tends to significantly

reduce the sensitivity of the models' reaction to sudden changes in upstream or downstream flow conditions. Scheduling in this way also tends to dampen oscillations of structure settings much quicker than if the flow depth is fixed at the downstream end of a reach. Furthermore, since the reach discharge rate is being fixed during gate scheduling it is possible to reference the entire system back to the system inflow rate at the extreme upstream end of the canal system. Doing this integrates the operation of the entire system and eliminates the amplification of local disturbances in the downstream direction which can result from successive over-reactions to system flow changes with fixed downstream flow depths.

Before a new control structure setting is accepted by the model during gate scheduling some final conditions must be met. First of all, if the structure is a weir the model will not allow the sill height to raise above the upstream flow depth if the weir has already begun to discharge water into the next downstream reach. This is to prevent the flow from being cut off in the downstream direction. If the structure is a sluice gate the model will check whether the current flow regime is free flow or submerged flow. If the structure is operating under free flow conditions the model will not allow the bottom of the gate to raise above the upstream water level. And, if the structure is operating under submerged flow conditions the model will not allow the bottom of the gate to raise above the downstream water level. These conditions are intended to preserve the current flow regime if possible since changes from one regime to another can cause abrupt changes in the computed discharge rate, which can then cause the numerical solution of the hydraulic equations to fail. Next, the model checks the hydraulic head differential for sluice gate structures. If the downstream water level is at or above the upstream water level across a sluice gate operating under submerged flow conditions, the model will set the flow rate through the structure equal to zero. This is to prevent the occurrence of "back flow" in which water may temporarily go upstream through the sluice gate. Finally, the new setting is compared to the extreme maximum and minimum settings which define the operational range of the structure. If the new setting is lower than the extreme minimum setting the new setting will be set equal to this minimum value. And, if the new setting is higher than the extreme maximum setting the new setting will be set equal to this maximum value. If any of the above conditions are not met the scheduling status for the current canal reach will be deactivated since the model is unable to accommodate the fixed discharge rate for which the new control structure setting was calculated. The model will then re-compute the current flow conditions for the current reach before moving on to the next reach in the system. When gate scheduling is enabled and a reach is deactivated the scheduling status for this reach on the default alpha screen display will be shown as "Wait". This means that the gate scheduling mode has been temporarily deactivated for the current reach and that the model will try to reactivate

scheduling for the reach in subsequent time steps by opening or closing the structure incrementally. When the actual water level upstream of the reach's control structure is near the target level the scheduling status will be automatically reactivated and the scheduling status will be shown as "On".

Turnout Scheduling

The model calculates turnout settings continuously based on turnout demand discharges and canal flow levels. During each time step the model uses the previous time step's flow levels to estimate the required turnout setting. After solving all of the hydraulic equations the model will then calculate the actual discharge through the turnouts in a reach. The actual discharge may not be the same as the demand discharge since the flow levels may have changed in the canal reach. Nevertheless, the actual discharge rate will usually be very close to the demand after the flow levels have stabilized. During a simulation turnout demands, actual discharges, settings, and status indicators can be viewed on the third, fourth, fifth, and sixth alpha screens, respectively.

The sixth alpha screen display shows individual turnout status indicators which can be any of the following eight values:

1. "-----"

This status indicator means that the turnout is a wasteway weir which is non-adjustable and, therefore, has a constant status. Wasteway weirs cannot have demand discharges since their function is only as a safety device.

2. "Shut"

This means that the turnout is completely closed. For an adjustable weir turnout it means that the weir sill is fully raised to the top of the canal bank (see Appendix A, Turnout Structure Equations). When the demand is zero the turnout will always be closed, except possibly for a few time steps during which the model incrementally closes the turnout (see "Wait") after a non-zero demand existed. Orifice turnouts will be closed when the current water level in the canal is below the bottom of the opening, even if there is a demand on the turnout. That is, if the current flow depth at the turnout location is less than the turnout "position" then no discharge can occur and the turnout will not open. When this happens it is usually a temporary condition because the water level in the canal is increasing.

3. "Open"

This means that the turnout is fully open. Usually when this happens the turnout demand is too high and the actual discharge is less than the demand. It could also mean that the flow level in the canal is too low, or that the downstream flow level is too high.

4. "Wait"

The model will not change a turnout setting by more than 0.10 meters in a single time step. Thus, if a large demand is placed on a turnout the model will open it incrementally in 0.10-meter steps until the appropriate setting is attained, during which the status will be "Wait". This can also occur when a turnout demand is reduced by the user and the turnout closes incrementally in 0.10-meter steps.

5. "Free"

This means that the turnout is currently operating under free-flow conditions. The actual discharge is probably at or very close to the demand discharge.

6. "Subm"

This means that the turnout is currently operating under submerged flow conditions. The actual discharge is probably at or very close to the demand discharge.

7. "Surf"

This means that the water level in the canal is below the top of an orifice-type turnout's opening. The actual discharge through the turnout is less than the demand, but opening the turnout above the current water surface will not increase the actual discharge. Therefore, the model waits until the water level increases and leaves the bottom of the turnout gate at the current water level. As the water level in the canal increases the turnout will open until the demand is matched or until the turnout is fully opened.

8. "Back"

This means that the downstream water level is at a higher elevation than the upstream water level, causing backflow conditions. When this happens the model will set the turnout discharge to zero rather than allow flow to enter the canal through the turnout. The upstream water level at a turnout is in the canal and it is calculated by the model based on the hydraulic equations. The downstream water level at a turnout is outside of the canal system and is approximated by a linear stage-discharge relationship (see Figure A-8).

Numerical Solution

The model solves the equations of momentum and continuity simultaneously for each computational node in a canal reach. These hydrodynamic equations are sometimes referred to as the St. Venant Equations in the open channel hydraulics literature. A deformable control volume approach is used to mathematically reduce the equations from first-order, non-linear hyperbolic differential equations to a system of non-linear algebraic equations. The algebraic equations are solved by applying the Newton-Raphson technique along with a form of Gaussian elimination for a sparse Jacobian matrix. The solution to these equations can diverge when relatively large changes in the flow conditions occur during a hydraulic simulation. However, the model makes numerous checks and cross-checks during the course of a simulation which are intended to prevent the solution from failing. In spite of these checks it is still possible that the solution will fail at some point during a simulation. If the solution does fail during a simulation the message "Solution Diverged" will be displayed on the screen and the simulation will be aborted.

SECTION TWO

FILE EDITOR (CDAT)

INTRODUCTION

This program is designed to be used with the model CANAL which performs hydraulic simulations of water flow in canal systems. CDAT is a file editing program which is designed specifically for use with configuration data files that are read by the program CANAL. This editing program is used to create and initialize new configuration data files. Like the program CANAL, CDAT is interactive and menu-driven. The program CDAT allows flexibility in the entry of data to configuration data files because values can be entered in any order and in different sessions. CDAT also includes an on-line help file which can be accessed to answer questions about the different features available in the program.

GENERAL INFORMATION

Each configuration data file contains the hydraulic characteristics of a particular canal system. These data include turnout and control structure definitions, seepage rates, canal cross-sectional geometry, and other hydraulic parameters. Once a hydraulic simulation begins the data read from this type of file by the program CANAL become constant; these values never change during a simulation. If a configuration data file is modified the changes will be in effect during a hydraulic simulation only by re-filling the canal system again.

Configuration data files are formatted as random access files. File names may be up to five characters in length, and all file names are automatically appended with the symbol @ before they are opened on a disc. This is intended to minimize the risk of confusion with other files which may reside in the same directory. Valid data file names may use any combination of letters from A to Z and integers 0 to 9. Lower-case letters are automatically converted to upper-case and entered file names longer than five characters are truncated to only five characters. Configuration data files are always stored in a directory that is specified by the DOS path (see DOS Path).

New configuration data files can be created by entering a file name which does not already exist on the hard disc in the data directory. The program CDAT will try to find a data file with this name in the specified directory, and if it cannot be found then CDAT will ask the user if a new file should be created with the entered file name. When new files are created CDAT will fill the data file with default values which can subsequently be changed if desired.

The default values with which a configuration data file is initialized are always the same.

Configuration data files always contain a full compliment of data values as if every canal system were of the maximum allowable dimensions. That is, every configuration data file contains data for four branches, with nine reaches in each branch and nine turnouts in each reach. The actual canal system dimensions may be less than this, and in this case CANAL will only read the data values which pertain to the actual dimensions. This means that CANAL first looks at the system dimensions from a configuration data file and then extracts only the pertinent information from the file, ignoring the rest of the data file contents. When CDAT is used to change the contents of a configuration data file the data is automatically updated on the disc on a continuous basis. For this reason it is not necessary to explicitly save a modified data file - it is done automatically by CDAT. All modifications to a configuration data file will remain in effect until they are changed again. Configuration data files may be copied to a backup disc to protect the contents in case the working file is accidentally altered or erased. This can be accomplished by using the COPY command provided by the DOS operating system (see the DOS manual).

Each individual parameter in a configuration data file has high and low limits. Whenever CDAT is used to change data values in a file the newly entered values are compared with the high and low limits. If a new value is above the high limit or below the low limit then a beep will be heard and the value will be set at either the high or the low limit, depending on which was violated. However, CDAT only checks the high and low limits for each individual data parameter. No cross-checking is done to verify whether the data is consistent, as it must be in a realistic canal system. For example, CDAT will not check the turnout locations and compare them with reach lengths to make sure that the turnout locations are not erroneously placed beyond the downstream end of a reach. The CANAL model does perform such cross-checks when reading a configuration data file in preparation of filling an empty canal system.

The program user should check the data carefully in a configuration data file before running the model since a simulation may proceed normally even though the data are not for the canal system that the user anticipated. If an incorrect hydraulic parameter is entered into a configuration data file the hydraulic simulation will not reflect the true flow conditions as they would be in a real canal system. Such an error may be difficult to identify during a simulation.

CHANGING THE FILE CONTENTS

Last Update

A date can be entered for a configuration data file to record the last time that the file contents were modified. This date's associated with the entire contents of a configuration data file and it must be set manually. When new configuration data files are created the date is initially set at 01 Jan 1987. This last update can be changed from either the configuration, control structure, or turnout structure menus by first positioning the indicator arrow at the words "Last Update". When the indicator is positioned as such the program user can type Enter to position the cursor at the day. Then the arrow keys can be used to change the day, and when the day is correct Enter can be typed again to move the cursor to the month. Then the arrow keys can be used to change the month. After the month is correct the Enter key can be typed to move the cursor to the year. Then the arrow keys can be used again to change the year. When the year is correct the Enter key can be typed once again and the indicator arrow can then be moved to another item on the menu.

Reach Names

Names can be specified for individual reaches to assist the program user in identifying the canal reaches. When new configuration data files are created there are default names assigned to each reach. These names can be changed from either the configuration, control structure, or turnout structure menus. First, the branch and reach should be set to the desired values and then the indicator arrow can be positioned at the words "Reach Name". Then any key can be typed to erase the current name and subsequent keystrokes will be echoed on the screen as characters in the new reach name. After entering the new reach name the End key is typed by the program user to accept the name, and the indicator arrow can be positioned at another menu item.

Reach names cannot be more than twenty characters in length; however, any character can be used in defining a name (including blank spaces). If a mistake is made in entering a reach name then entry must begin again by typing End and then typing another key to erase the incorrect name. The backspace key does not erase typed characters when reach names are being entered.

Turnout Names

Names can be specified for individual turnouts to assist the program user in identifying the canal turnouts. When new configuration data files are created there are default names assigned to each turnout. These names can be changed from the turnout structure menu. First, the branch, reach, and turnout should be set to the desired values and then the indicator arrow can be positioned at the words "Trnout Name". Then any key can be typed to erase the current name and subsequent keystrokes will be echoed on the screen as characters in the new turnout name. After entering the new turnout name the End key is typed by the program user to accept the name, and the indicator arrow can be positioned at another menu item.

Turnout names cannot be more than five characters in length; however, any character can be used in defining a name (including blank spaces). If a mistake is made in entering a turnout name then entry must begin again by typing End and then typing another key to erase the incorrect name. The backspace key does not erase typed characters when turnout names are being entered.

Parameters

Individual data parameters can be changed by positioning the indicator arrow at the desired menu item and typing the new value. If the parameter is a single-digit integer (such as the branch or reach number) then the new value will be immediately accepted, otherwise the Enter key must be typed to accept a new value. Data parameter values for different reaches and branches can be viewed and changed by changing the branch and reach numbers as necessary. The same is true for turnout data by changing the turnout number. From the menus for configuration, control structure, and turnout structure data, the branch, reach, and turnout numbers can be changed without affecting the data file contents. Changing branch, reach, and turnout numbers only changes the screen display to show data for different branches, reaches, and turnouts.

The active number of branches, reaches, and turnouts in a canal system are changed from the system dimensions menu. Changing the branch number under "Active Reaches" only changes the display on the next line which shows the active number of reaches in a branch; it does not change the data file contents. Similarly, changing the branch or reach number under "Active Turnouts" only changes the display on the last line which shows the active number of turnouts in a reach; it does not change the data file contents. The data file contents will be changed if a new value is entered for the number of branches, reaches, or turnouts.

Typing Enter without first typing a number will result in a default entry of zero. For some data parameters a value of zero is not allowed by CDAT (e.g. the canal depth cannot be zero); therefore, if Enter is typed without first typing a number then a beep may be heard, indicating that the value of zero is not accepted. If this happens then the minimum allowable value for the parameter will be displayed and updated on the data file.

Warnings Flag

The warnings flag can be turned on or off by the user from either the configuration, control structure, or turnout menus. The F1 key toggles between on and off, which is displayed in the lower right corner of the screen. When new configuration data files are created the flag is initially turned on.

When the CANAL model reads configuration data files before filling a canal system it performs various cross-checks on the data (see Section One, Filling an Empty System). If the warnings flag for the data file is turned on the model will perform some additional checks for potential problems with the data before beginning the simulation. Unlike error messages, warnings will not prevent a simulation from beginning. Warning messages only indicate an unusual canal system configuration which may be incorrect, but it is up to the user to decide whether or not to proceed with the simulation. Initially, it is a good idea to turn the warnings flag on from CDAT and then run the CANAL model to see if any messages are displayed. If there are warnings and the user wishes to accept the data as it is then CDAT can again be used to turn the flag off so that the messages are not displayed every time the canal is filled. All possible warning messages and their meanings are listed in Appendix C.

DEFINITION OF TERMS

DOS Path

The DOS (Disc Operating System) path determines where the program CDAT and the hydraulic model CANAL will attempt to access configuration and suspended simulation data files. The DOS path also tells these two programs where to look for help files. The default path is "C:\CDATA\" and this path name is stored in a file called "DPATH.DAT" in the directory from which either CDAT or CANAL are being run. If CDAT or CANAL cannot find this file with the DOS path the file will be created automatically and the default path will be used. If either of these programs is running and they try to access a file which does not exist in the current DOS path, or if the current DOS path is invalid (e.g. the path is to a non-existent directory or sub-directory), there will be an error message displayed on the screen indicating that the file was not found. Such error messages are always accompanied by the current DOS path which the program is using.

The DOS path can be changed only from the CDAT program by selecting the "DOS Path" item from the menu. The program will display the current path and the user can then type in a new path definition, or type Enter to keep the current path. CDAT will look at paths typed in by the user and will try to identify obvious errors if they exist. For example, the first letter of the path definition must be the drive letter which can only be A, B, C, D, E, or F. If the user omits a colon (:) after the drive letter the program will insert a colon in accordance with DOS protocol. CDAT will also insert backslashes (\) if they are missing. The path definition can include any level of sub-directories, separated by backslashes, provided that the entire definition is not more than twenty characters in length.

Branch Linkage

The model simulates canal flow for branches containing a maximum of nine reaches. Control structures such as sluice gates always define the location of the end of a canal reach. If what might be considered a single branch in a real canal system has more than nine reaches the model will necessarily treat this branch as being two or more branches. This is because of the restriction on the number of reaches in a single branch for the purpose of simulation. In such a case a downstream branch would be connected to the end of the last reach in an upstream branch, creating a number of reaches in series

which behave as a single branch. The only difference is that the model requires that they be labeled as two separate branches. System bifurcations can also be simulated by linking a branch to a turnout in a reach of an upstream branch. The only restriction imposed by the model in linking branches is that the first branch (i.e. branch number one) must be the most upstream branch. This means that the upstream end of the first reach in the first branch is connected to the water source. Therefore, the inflow hydrograph to the canal system always determines the flow rate into the first reach of the first branch. Branches other than branch number one can be linked together in any order.

Branch linkage information for a canal system is entered into a configuration data file by selecting the "Branch Linkage" item from the CDAT menu. When this item is selected the user will be able to view and modify the branch linkages by specifying the upstream branch, reach, and turnout. Branch linkage information does not need to be specified for non-existent branches. For example, if a canal system has only two branches the linkage information for branches three and four is irrelevant and will be ignored by the model. The default branch linkage is for a canal system with no bifurcations--each branch is linked to the last reach in the upstream branch. This means that in the default configuration branch two is connected to the last reach in branch one, branch three is connected to the last reach in branch two, and so on. The upstream branch, reach, and turnout for branch number one is always set at 0, 0, 0, meaning that it is linked to the system source. When other branches are linked to the last reach in an upstream branch only the upstream branch number is specified, the upstream reach and turnout numbers are both zero. However, if a branch is linked to a turnout in an upstream branch the upstream branch number, reach number in that branch, and turnout number in that reach must be specified to uniquely identify the linkage point.

The following example is for a canal system with four branches. Branch one has nine reaches, branch two has two reaches, branch three has four reaches, and branch four has three reaches (see Figure 20). In the real canal system branches one and two appear as one branch; however, in the model they constitute two branches in series since the maximum number of reaches per branch is nine. Branch three is linked to the second turnout in reach five of branch one. Branch four is linked to the first turnout in reach nine of branch one. For branch two both the upstream reach number and the upstream turnout number are entered in the program CDAT as zeros since branch two does not connect to a turnout. CANAL interprets these two zeros to mean that branch two connects to the end of the last reach in an upstream branch. Hence, only the upstream branch number is needed since the number of reaches per branch is specified in the system dimensions data. The number of the upstream reach and of the upstream turnout

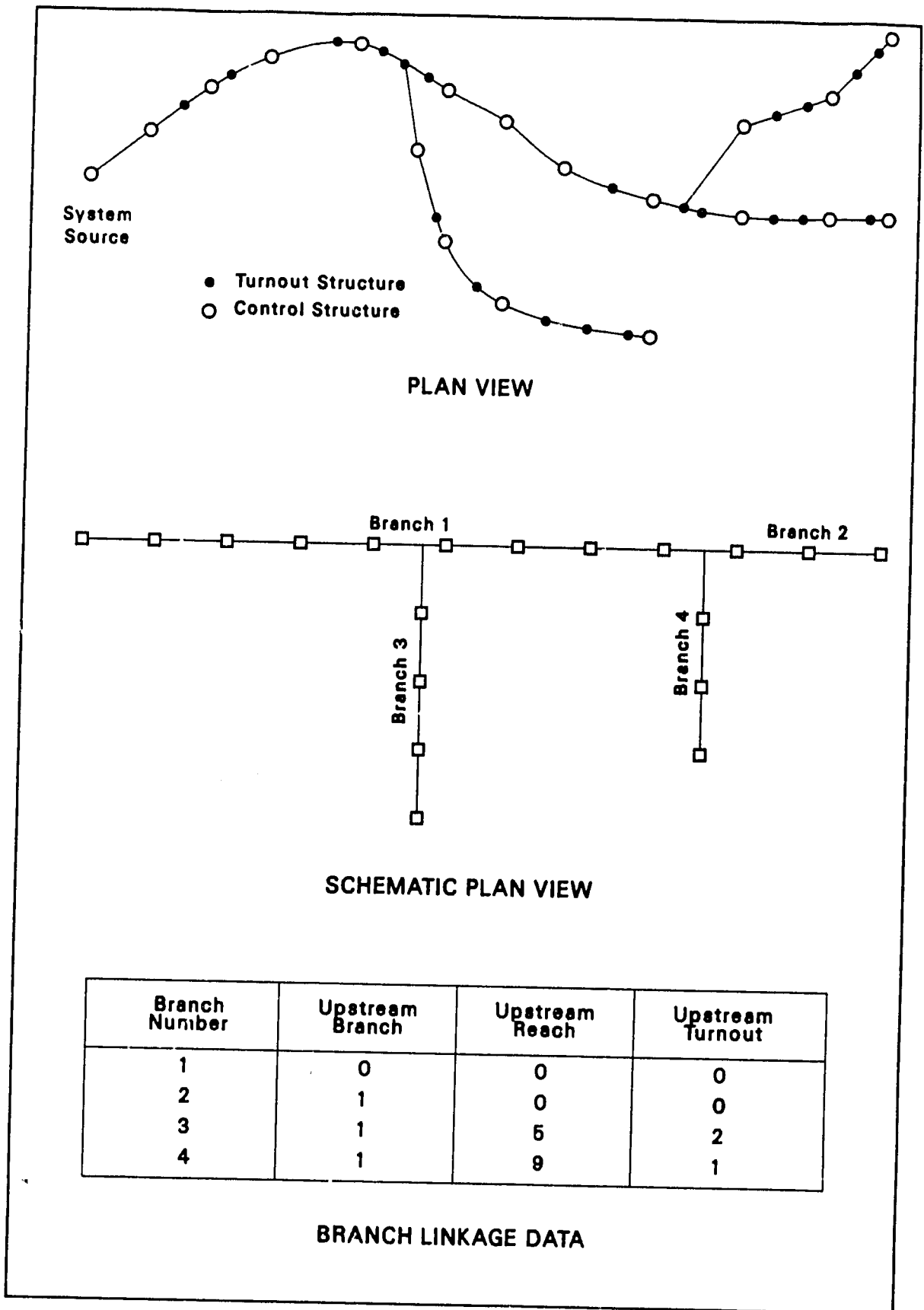


Figure 20. Illustrated example of how branches can be linked.

should always be either both zero or both non-zero (e.g. if the upstream reach number is non-zero this implies that the linkage is to a turnout, in which case the upstream turnout number must also be non-zero).

The model will cross-check branch linkage and system dimensions data when reading a configuration data file before filling an empty system. These checks are to ensure the validity of the linkage information for a given canal system. The program CDAT will allow any branch linkage data to be entered into a configuration data file without performing any cross-checks since the user may enter linkage data before entering system dimensions data. However, CDAT will not allow the user to modify the default linkage for branch number one since this branch is always connected to the system source. And, CDAT will not allow a branch to be linked to itself.

System Dimensions

The system dimensions data include the number of branches, the number of reaches in each branch, and the number of turnouts in each reach. The maximum number of branches is four and the maximum number of reaches per branch is nine. Therefore, the largest canal system that can be simulated is one with thirty six reaches. Canal systems are divided into branches with a maximum of nine reaches each in order to reduce the systems to a tractable level during hydraulic simulations. Screen displays in the program CANAL are only shown for one branch at a time because too much information outputted simultaneously is difficult to interpret by the program user.

The maximum number of turnouts per reach is nine. This restriction limits the total possible number of turnouts in a canal system to 324. Large numbers of turnouts in a canal system significantly increase the number of computational cells required for hydraulic simulation, and this tends to reduce the simulation speed.

Configuration Data

BASE WIDTH:

This is the width of the bottom of a trapezoidal canal cross-section in meters. The base width will increase if sediment is deposited in the bottom of a canal reach (unless the sides of the canal are vertical). Thus, if there is sediment in the bottom of a canal reach then the base width in the configuration data file should be entered as the width at the top of the sediment layer.

SIDE SLOPE:

This is the inverse side slope of a trapezoidal canal cross-section in meters per meter. This means that a rectangular cross-section has a side slope of zero. If a canal section is not trapezoidal then it must be approximated as such by choosing an appropriate value for the side slope.

REACH LENGTH:

This is the length of a canal reach in meters. The end of a canal reach is always demarcated by the presence of a control structure. Thus, each reach length corresponds to the distance between successively located control structures in a canal branch.

MAXIMUM DEPTH:

This is the vertical distance from the bottom of a canal cross-section to the top of the canal lining in meters. If sediment is uniformly deposited in the bottom of a canal reach the maximum depth will decrease according to the depth of deposited sediment. And it is this reduced depth that should be entered into the configuration data file.

HYDRAULIC ROUGHNESS:

Manning's equation is used in the model to compute friction gradients for the momentum equation during a hydraulic simulation. The hydraulic roughness will tend to increase as canals age and the lining deteriorates. Deterioration of the canal lining or growth of weeds in the canal can be simulated by increasing both the hydraulic roughness and the seepage outflow rate. Listed below are some examples of the hydraulic roughness for use in the Manning equation:

Condition	Roughness
Smooth Concrete.....	0.013
Bricks or Masonry.....	0.015
Rough Stone Set in Concrete.....	0.019
Smooth Earth-Lined with no Weeds.....	0.020
Rough Earth-Lined with Many Weeds.....	0.040

LONGITUDINAL SLOPE:

This is the slope of the bottom of a canal reach in the direction of flow. If a reach has a compound slope then the weighted average slope should be entered. The longitudinal slope is entered in meters per 100 meters.

OPERATIONAL SUPPLY LEVEL:

The operational supply level (OSL) is the percent of the maximum reach depth at which the downstream flow depth is at the desired operational level. This may also be referred to as the "full supply level". When gate scheduling for OSL is enabled the model attempts to maintain reach downstream flow depths (i.e. just upstream of each adjustable control structure) at OSL. However, gate scheduling can also be enabled for flow levels other than OSL by selecting the "freeze" option from the operational mode changes menu during a hydraulic simulation (see Section One, Simulation Interrupts).

SEEPAGE RATE:

This is the rate in millimeters per day of seepage outflow from a reach. The program user may wish to include evaporation losses in the value entered for the seepage rate if these losses are considered significant. Seepage rates may be negative if a canal reach is below the adjacent ground surface and water seeps into the canal. During hydraulic simulations the seepage rate is converted to meters per second and multiplied by the wetted perimeter and length of each computational cell to yield a seepage outflow in units of cubic meters per second for the entire reach.

Control Structure Data

CONTROL TYPE:

The program CDAT displays six types of control structures. The first two types are circular and rectangular culverts, respectively. The third type is a sharp-crested rectangular weir with a fixed sill height. This type of structure is assumed to always operate under free-flow conditions and contraction effects (if any) are assumed to be taken into account by the discharge coefficient. Control structure type 4 is exactly the same as type number 3 except that the sill height is adjustable by both manual control during a simulation and by automatic control through gate scheduling. Control structure type 5 is a rectangular sluice gate which operates under free-flow or submerged flow conditions, depending on the gate opening and upstream and downstream flow depths. Control structure type 6 is the same as type 5 except that the gate opening is circular.

Sluice gates are assumed to have overflow side weirs with fixed sill heights. If a sluice gate in a canal reach does not have overflow side weirs then the weir sill height should be set to the maximum canal depth or higher so that the model will not simulate any weir flow (unless the canal is overflowing). Control structure

types 1,2, and 3 are non-adjustable and types 4,5, and 6 are adjustable.

NO. OF CONTROLS:

The number of controls at the downstream end of a reach can be from a minimum of one to a maximum of nine. Multiple control structures at a single location and in parallel are assumed to be of identical type and dimensions. Also, multiple control structures are assumed to operate identically, that is, the setting for one is always the same as the setting for the other adjustable control structures at the same location.

CD1 - FREE FLOW:

This is the sluice gate or culvert discharge coefficient for free-flow conditions. Free-flow conditions exist when the upstream side of a sluice gate or culvert is submerged and the downstream water level is below the bottom of the gate or culvert. Free-flow conditions normally exist only when there is a significant drop in elevation from one reach to the next. When free-flow conditions exist the downstream water level does not affect the discharge through the control structure. This coefficient does not need to be entered if the control structure is type 3 or 4 since weirs are assumed to operate under free-flow conditions at all times and the only discharge coefficient used for these is $Cd3$.

CD2 - SUBMERGED FLOW:

This is the sluice gate or culvert discharge coefficient when both the upstream and the downstream water levels are above the bottom of the sluice gate, or above the top of the culvert opening on both the upstream and downstream sides of the structure. When submerged flow conditions exist at a control structure any changes in either the upstream or the downstream water levels will affect the discharge through the structure. This coefficient does not need to be entered if the control structure is type 3 or 4 since weirs are assumed to operate under free-flow conditions at all times and the only discharge coefficient used for these is $Cd3$.

CD3 - WEIR OVERFLOW:

This is the weir discharge coefficient for free-flow conditions over a sharp-crested rectangular weir. This coefficient applies to all control structure types; however, if the control structure is type 5 or 6 (sluice gates) and the weir sill height is set at the maximum canal depth, or higher, then the value of $Cd3$ is irrelevant to the hydraulic simulation and it does not need to be set. This is because there will be no weir flow if the sill height is above the top of the canal (unless the canal overflows).

For culverts this discharge coefficient is used in a discharge equation for a weir operating under free flow conditions. When the upstream flow depth at a culvert is below the top of the culvert opening a weir equation is used to approximate the discharge. This approximation is made under the assumption that culverts normally operate under submerged flow conditions.

WIDTH OR DIAMETER:

This is the width of the opening of a rectangular sluice gate or a rectangular culvert in meters. It can also be the diameter of a circular sluice gate or the diameter of a circular culvert in meters. If a control structure is type 3 or 4 (weirs) then this value does not need to be entered since it will only be ignored by the model.

WEIR SILL WIDTH:

This is the width of the sill of a sharp-crested rectangular weir in meters. If there is more than one weir at the control structure then this value should be the total sill width. If the control structure is of type 5 or 6 (sluice gates) and the sill height is set at the maximum canal depth or higher then this value does not need to be entered since it will not be used by the model. This value does not need to be entered for control types 1 and 2 because unlike sluice gates, culverts do not include side weirs.

SILL OR CULVERT HEIGHT:

This is the height of the sill of a sharp-crested rectangular weir measured from the bottom of the canal in meters. This value does not need to be entered if the control structure is type 4 because the hydraulic model CANAL will automatically determine the initial sill height before filling an empty canal. If the control structure is type 5 or 6 (sluice gates) then the weir sill height should be set at the value which corresponds to the fixed sill height of overflow side weirs. Also, if the control structure is a sluice gate and no weirs exist then this value should be set at the maximum canal depth or higher so that no weir flow will be computed (unless the canal is overflowing).

For rectangular culverts this parameter represents the height of the culvert opening in meters. Culverts are treated as non-adjustable control structures and the opening on the upstream side is assumed to be of the same dimensions as the opening on the downstream side of the structure. For circular culverts this value does not need to be entered since the maximum height and width are both equal to the structure diameter.

UPSTREAM DELTA Z:

This is the elevation drop (z_u) in meters going into a control structure. It may also be part of a channel transition in which the lowest point of the control structure frame departs from the upstream canal bed elevation. This parameter is used for calculating upstream flow levels relative to control structures for free flow conditions. If the flow regime across the structure is submerged then it is added to the downstream elevation drop (z_d) to calculate the head differential across the control structure. For adjustable weir control structures (which are assumed to always operate under free-flow conditions) this parameter can be used to specify the minimum sill height. See Figures 21, 22, and 23 for examples of z_u .

DOWNSTREAM DELTA Z:

This is the elevation drop (z_d) in meters on the downstream side of a control structure. It may be part of an exit transition in which the lowest point of the control structure frame is not at the same elevation as the beginning of the canal bed on the downstream side of the structure. The model ignores this value for weir control structures since they are always assumed to operate under free-flow conditions. The model uses this parameter to determine whether free or submerged flow conditions exist at a sluice gate, and if the regime is submerged flow it will be combined with z_u to compute the head differential across the structure. See Figures 21, 22, and 23 for examples of z_d .

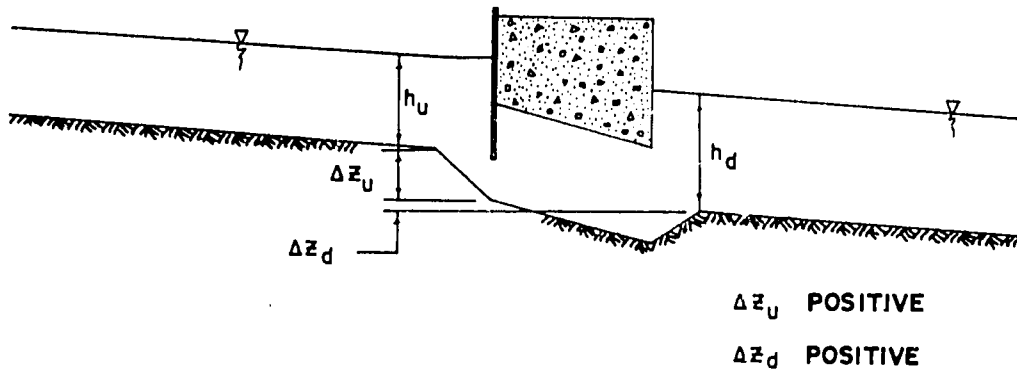


Figure 21. Example of elevation changes across a control structure for both z_u and z_d positive.

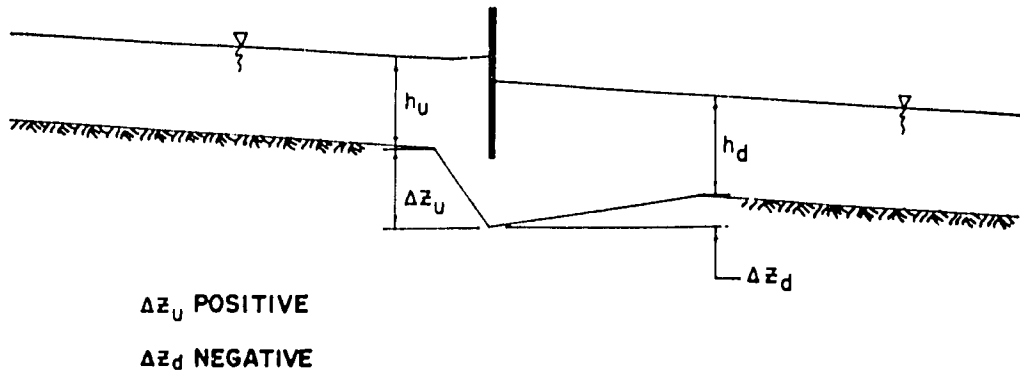


Figure 22. Example of elevation changes across a control structure for z_u positive and z_d negative.

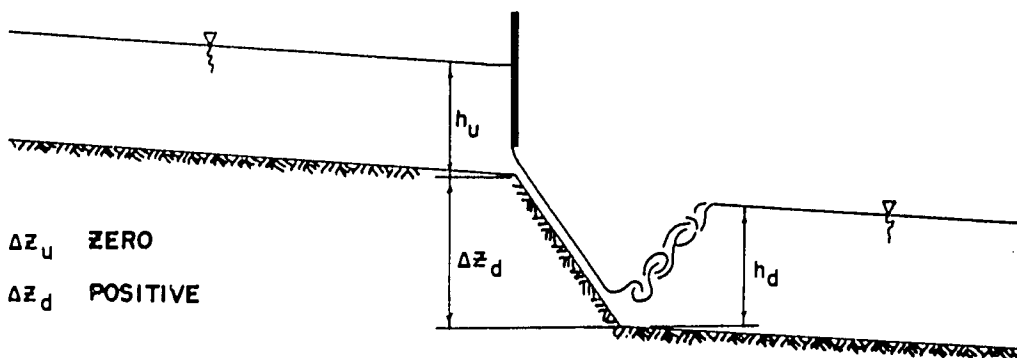


Figure 23. Example of elevation changes across a control structure for z_u zero and z_d positive.

Turnout Structure Data

TURNOUT TYPE:

The program CDAT displays four types of turnout structures. The first type is a circular orifice. The depth downstream of orifice-type turnouts is approximated by a linear stage-discharge relationship as described in Appendix A under Turnout Structure Equations. This relationship must be approximated since it is external to the canal system which is being simulated. The second type of turnout structure is the same as the first type except that the turnout

opening is rectangular. The third type is a sharp-crested rectangular weir with an adjustable sill height. This type of structure is assumed to operate under free-flow conditions whenever the canal water level is high enough to spill over the sill. The fourth type of turnout structure is a sharp-crested rectangular weir with a fixed sill height. This type of structure is intended to simulate wasteway overflow weirs and if water spills over this type of structure during a hydraulic simulation the flow regime is assumed to be free-flow. If more than one structure exists at a single turnout location then the combined widths of the multiple structures should be used. Compound turnout structures such as sluice gates with overflow weirs are not allowed. However, such structures could be simulated by placing two structures near each other in a canal reach as if they were independent.

DISCHARGE COEFFICIENT:

This is the discharge coefficient which is specified for any of the four types of turnout structures. The exact discharge equations are given in Appendix A.

WIDTH:

This is the width of the opening of a turnout structure which may be of more than a single piece. For example, a wasteway overflow weir may have two distinct weirs in parallel. If this is the case then the value entered for the width should be the total of all the separate pieces. This value is entered in meters.

HEIGHT:

This is the height of the control structure in meters. This value only needs to be specified for turnout type number 2 since type number 4 has a fixed sill height and type number 3 is assumed to operate up to the top of the canal lining. Turnout structure type 1 has the height fixed at the diameter of the opening, therefore, the height does not need to be specified for this type of structure. This value is equivalent to the maximum opening of a rectangular orifice turnout structure.

DISTANCE:

This is the distance measured from the beginning of a canal reach to the location of a turnout structure in meters. This location is defined to be the center of the turnout so that half of its width is upstream of this point and half is downstream. Turnouts must be located in ascending order. This means that the first turnout in a reach (if there are any turnouts in the reach) must be the farthest upstream turnout in the reach. The last turnout in a reach must be the farthest downstream turnout in the reach and all other turnouts

must be at intermediate locations and in order from upstream to downstream. For example, it would not be acceptable to the model to have turnout number one in a canal reach located at 300 meters and turnout number two located at 200 meters; the order must be reversed before a simulation can begin.

Turnout locations must be unique so that no two adjacent turnouts overlap. This means that right and left sides of a canal are not recognized (because the hydraulic equations are one-dimensional) and two turnouts at one location must be entered as if they were really at different locations. This can be accomplished by placing one turnout slightly upstream of the actual location and one turnout slightly downstream. It is possible that two turnouts could physically be at or nearly at the same location if they are on opposite sides of the canal bank. All turnouts in a given reach must be located within the reach, that is, they must have locations less than or equal to the length of the canal reach.

POSITION:

This is the height of the lowest turnout structure setting measured from the bottom of the canal at the turnout location in meters. The position value for turnout type 1 is equal to the difference in elevation between the bottom of the canal at the turnout location and the bottom of the circular opening. The position value for turnout structure type 2 is equal to the difference in elevation between the bottom of the canal at the turnout location and the bottom of the rectangular opening.

The position value is equal to the lowest sill height for turnout structure type 3 and is equal to the fixed sill height for turnout structure type 4. The position value for either turnout type 3 or type 4 must be greater than or equal to one-half of the maximum canal depth. The fully closed setting for turnout type 3 (adjustable weir) corresponds to the sill being flush with the top of the canal lining.

DS DEPTH:

This is the depth of flow on the downstream side of an orifice-type turnout when the discharge through the turnout is zero. It is measured from the bottom of the canal on the upstream side of the turnout (see Appendix A, Turnout Structure Equations).

DS SLOPE:

This is the slope of the linear stage-discharge relationship which is used to approximate flow depths on the downstream side of orifice-type turnouts (see Appendix A, Turnout Structure Equations).

ALLOWABLE INPUT RANGES

The upper and lower limits for configuration data file values are defined in the program CDAT. The upper or lower limit for a data value will be used instead of an entered value from the keyboard if an entered value is outside of the allowable input range. Also, when the allowable input range is violated during data entry a beep will be produced by the model to alert the user. The model always interprets the typing of Enter or the space bar as a zero entry, unless a number precedes Enter or the space bar. These limits are listed below in Table 4.

Table 4. Allowable Input Ranges.

System Dimensions	Minimum	Maximum	Units
Number of Branches	1	4	-
Number of Reaches per Branch	1	9	-
Number of Turnouts per Reach	0	9	-
Configuration Data			
Base Width	0.10	10.00	m
Side Slope	0.00	10.00	m/m
Reach Length	50.00	10000.00	m
Maximum Depth	0.10	10.00	m
Hydraulic Roughness	0.005	0.10	-
Longitudinal Slope	0.005	1.00	m/100m
Operational Supply Level	50.00	98.00	%
Seepage Rate	-1000.00	5000.00	mm/day
Control Structure Data			
Control Type	1	6	-
No. of Controls	1	9	-
Cd1 - free flow	0.20	2.00	-
Cd2 - submerged flow	0.20	2.00	-
Cd3 - weir overflow	0.20	8.00	-
Width or Diameter	0.10	10.00	m
Weir Sill Width	0.00	10.00	m
Sill or Culvert Height	0.10	10.00	m
Upstream Delta Z	-5.00	15.00	m
Downstream Delta Z	-5.00	15.00	m

Table 4 (concl). Allowable Input Ranges.

Turnout Structure Data

Turnout Type	1	4	-
Disch Coeff	0.20	2.00	-
Width	0.10	25.00	m
Height	0.10	10.00	m
Distance	0.00	10000.00	m
Position	-2.00	10.00	m
DS Depth	-10.00	10.00	m
DS Slope	0.00	100.00	-

APPENDICES

APPENDIX A

DATA COLLECTION AND CALIBRATION

The collection of field data and calibration of flow control structures are important activities associated with the application of hydraulic modeling. The accuracy of a hydraulic simulation by computer modeling is significantly affected by the quality of the field data which are used by the model. The field data and calibration requirements of the model are not such that data must be collected on a frequent basis. However, as the physical infrastructure of a real canal system changes with time the pertinent data parameters for use in the model must be updated. This is necessary to insure that the hydraulic simulations reflect the true field conditions with a high level of accuracy. Changes in the infrastructure that affect the accuracy of computer modeling include lining deterioration, deposition of sediment in the canals, and the proliferation of aquatic growth in the canals and around the flow control and turnout structures. Similarly, rehabilitation and maintenance work on the canals and flow control structures will change the hydraulic characteristics of the canals. Other things can also significantly affect the hydraulic characteristics of the canals. Therefore, it is important to update the configuration data file values for a canal system periodically as needed.

The files which contain field data for the model are referred to here as "configuration data files" (the other type of data file used by the hydraulic model is called a "suspended simulation file"). Configuration data files are organized into the following five separate categories listed below:

- (1) Branch Linkages
- (2) System Dimensions
- (3) Configuration Data
- (4) Control Structure Data
- (5) Turnout Structure Data

The first category contains the definitions for the physical layout of a canal system in terms of branch connections. Branch connections are specified as upstream locations at which downstream branches begin. Downstream branches may begin at (1) the end of the last reach in an upstream branch, or (2) a turnout in a reach of an upstream branch. The data needed for the second category consist of the number of branches, reaches, and turnouts which are present in a canal system. The maximum system dimensions for this version of the model are four branches with nine reaches in each branch and nine turnouts in each reach. Canal reaches must be separated by flow control structures. A change in the canal cross-section alone does

not constitute the beginning of a new reach.

The data needed for the third, fourth, and fifth categories include physical system characteristics, of which some can be measured directly and others require calibration activities. Much of this data can be obtained directly from the canal design specifications. However, if the design specifications are not available or if the actual construction did not conform exactly to the original design then it is necessary to obtain the pertinent data from field measurements. Any doubt about whether the finished canal system conforms to the design specifications should be resolved with field measurements to verify the values which may be in question. It is also recommended that field measurements be compared to design specifications whenever possible. This is because obvious field measurement errors can sometimes be identified by checking whether there are any significant deviations between the design specifications and the field measurements.

Calibration of Controls and Turnouts

Control structures and turnouts need to be calibrated so that the value for the coefficient of discharge (Cd) can be determined. Calibration activities for control structures and turnouts should take place under steady flow conditions. In other words, the flow levels and the flow rate should not be changing in the vicinity of the structure which is being calibrated. Each calibration measurement should include a sketch of the structure, the flow rate through the structure, the flow levels upstream and downstream of the structure, and the structure setting (if it is an adjustable structure).

With each set of calibration data for a control structure or turnout a single point can be generated on a calibration curve. Usually the relationship between discharge and flow level at a control structure or turnout is linear throughout the normal operational range when plotted on logarithmically-scaled paper. This means that it is possible to calculate a discharge coefficient for a structure from a single set of calibration data since the slope of the calibration curve can usually be assumed by noting the form of the stage-discharge relationship (flow equation) for the type of structure which is being calibrated. For example, the discharge through a submerged sluice gate is proportional to the square root of the water elevation differential across the gate, and the discharge over a sharp-crested rectangular weir is proportional to the flow depth over the crest raised to the power of 1.5.

Nevertheless, it is preferable to make at least three separate calibration measurements which will generate three different points

on a calibration curve. Separate sets of field data for a single structure provide a greater degree of confidence in the accuracy of the field work when the results are consistent, and help to identify invalid data due to field measurement or calculation errors. Furthermore, a calibration curve with field data points at either end of the flow range for which the structure can operate will give a more representative discharge coefficient. This means that it is better to have field data points to define a calibration curve which are spread out than to have points which are clustered together near a small segment of the curve. Examples of clustered and non-clustered data points are shown in Figure A-1.

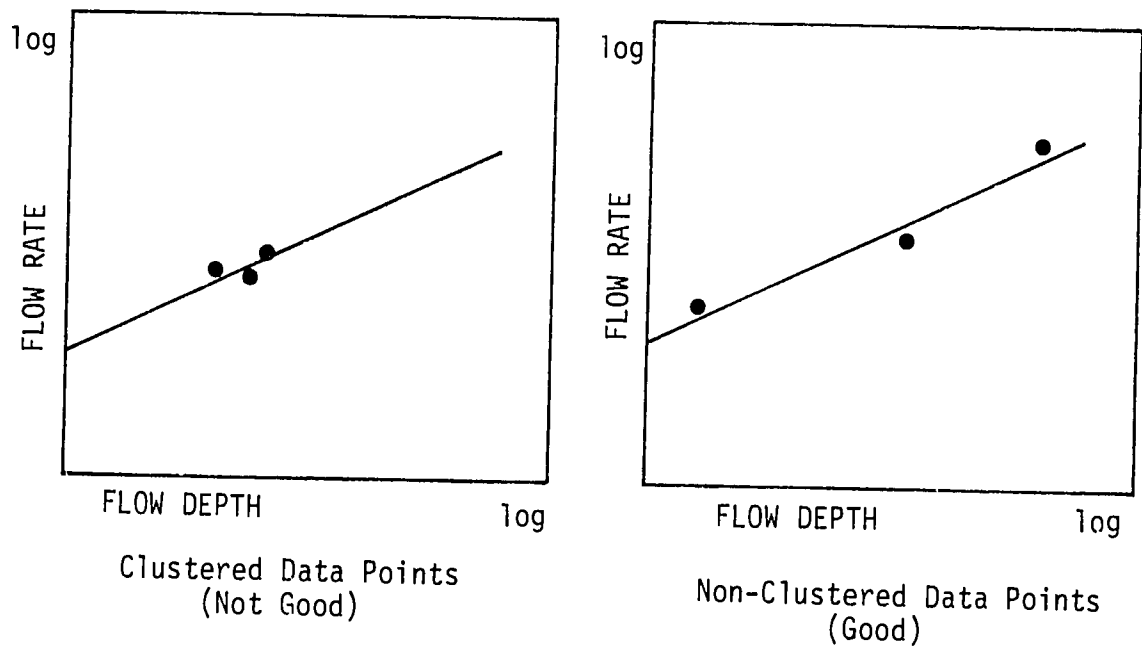


Figure A-1. Calibration data plots on logarithmic scale.

Because of the field measurement inaccuracies and other errors the field data points for a calibration curve will not usually fall exactly on a straight line when plotted on logarithmically-scaled paper. Therefore, an average of the field data points can be used to determine the value of the discharge coefficient. Graphically, the discharge coefficient is directly proportional to the value of the flow rate on the ordinate at which the calibration curve passes through the value of unity on the abscissa.

It is useful to plot the field calibration points on logarithmically-scaled paper because it can help to identify large

errors in the field data. However, it is not essential to plot the calibration data. The equations and figures on the following pages can be used to calculate values for the coefficient of discharge directly from each set of field calibration data. The form of these equations is exactly the same as in the model for the different types of flow regimes possible (i.e. free flow, submerged flow, and open flow) with each structure. "Open flow" refers to the flow regime when an orifice structure operates such that the upstream water level is below the top of the opening, in which case the structure behaves more like a channel constriction than an orifice.

Control Structure Equations

1- Circular Culverts

Free Flow:	$Q = Cd_1 * N * A * \sqrt{2g(h_u + z_u - W/2)}$	for, $(h_u > W)$, $(h_d < W + z_u + z_d)$
Submerged Flow:	$Q = Cd_2 * N * A * \sqrt{2g(h_u - h_d + z_u + z_d)}$	for, $(h_u > W)$, $(h_d > W + z_u + z_d)$, $(h_u > h_d - z_u - z_d)$
Open Flow:	$Q = Cd_3 * N * (h_u + z_u)^{2.5}$	for, $(h_u < W)$, $(h_u > h_d - z_u - z_d)$
Zero Flow:	$Q = 0$	for, $(h_u \leq h_d - z_u - z_d)$

where,

- g = 9.81 m/s²
- W = culvert diameter (m)
- N = number of culverts in parallel
- Cd_1 = free flow discharge coefficient
- Cd_2 = submerged flow discharge coefficient
- Cd_3 = weir flow discharge coefficient
- h_u = upstream flow depth (m)
- h_d = downstream flow depth (m)
- z_u = upstream elevation drop (m)
- z_d = downstream elevation drop (m)
- Q = discharge rate (m³/s)

and, $A = \pi W^2 / 4$

Notes: The structure setting is non-adjustable and it equals the diameter ($b = W$). And, "Open Flow" is approximated with a V-notch weir equation.

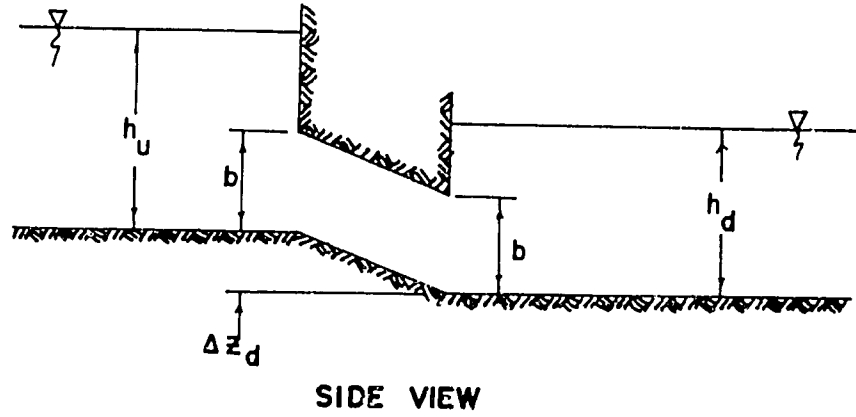
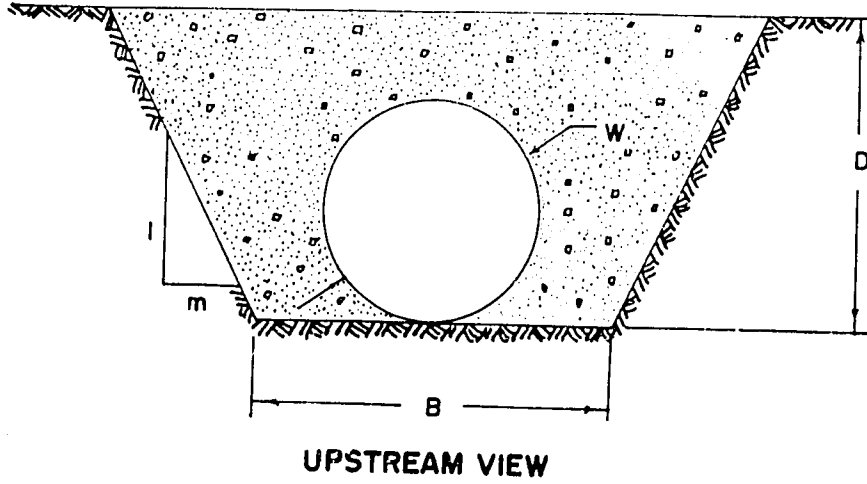


Figure A-2. Circular culvert control structure.

2- Rectangular Culverts

- Free Flow: $Q = Cd_1 * N * A * \sqrt{2g(h_u + z_u - b/2)}$ for, $(h_u > b)$,
 $(h_d < b + z_u + z_d)$
- Submerged Flow: $Q = Cd_2 * N * A * \sqrt{2g(h_u - h_d + z_u + z_d)}$ for, $(h_u > b)$,
 $(h_d > b + z_u + z_d)$,
 $(h_u > h_d - z_u - z_d)$
- Open Flow: $Q = Cd_3 * N * W * (h_u + z_u)^{1.5}$ for, $(h_u < b)$,
 $(h_u > h_d - z_u - z_d)$
- Zero Flow: $Q = 0$ for, $(h_u \leq h_d - z_u - z_d)$

2- Rectangular Culverts (cont)

- where,
- g = 9.81 m/s²
 - W = culvert width (m)
 - b = culvert height (m)
 - N = number of culverts in parallel
 - Cd_1 = free flow discharge coefficient
 - Cd_2 = submerged flow discharge coefficient
 - Cd_3 = weir flow discharge coefficient
 - h_u = upstream flow depth (m)
 - h_d = downstream flow depth (m)
 - z_u = upstream elevation drop (m)
 - z_d = downstream elevation drop (m)
 - Q = discharge rate (m³/s)

and, $A = b \cdot W$

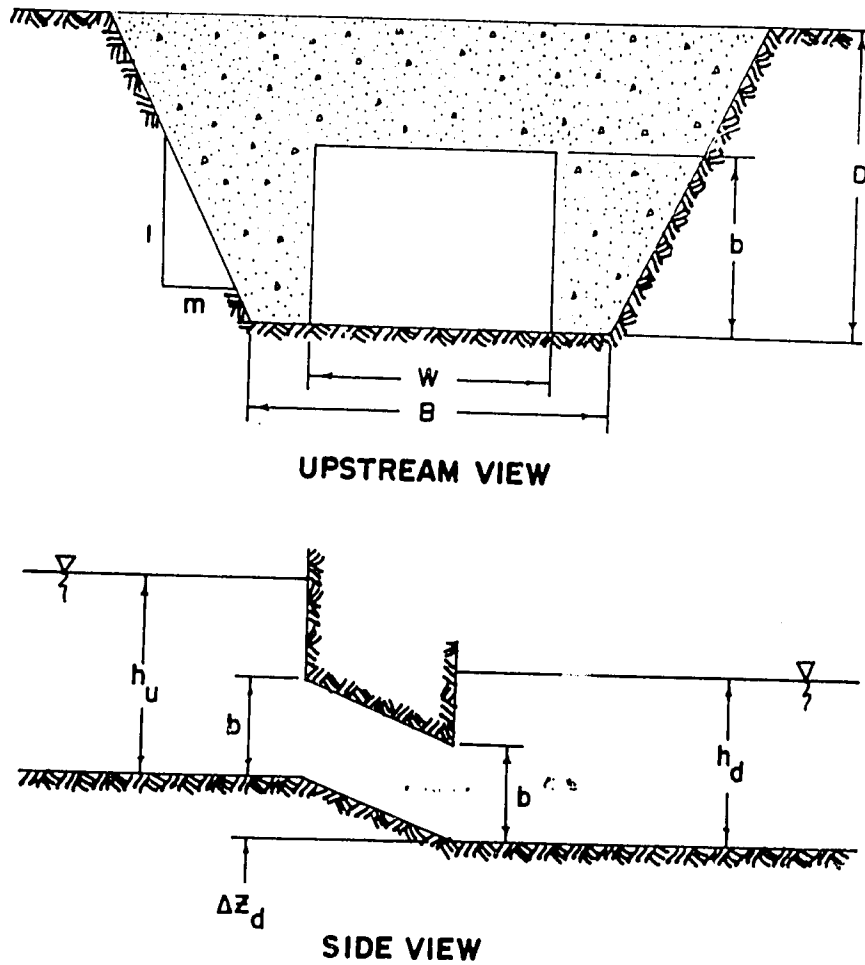


Figure A-3. Rectangular culvert control structure.

3,4- Rectangular Weirs (fixed and adjustable)

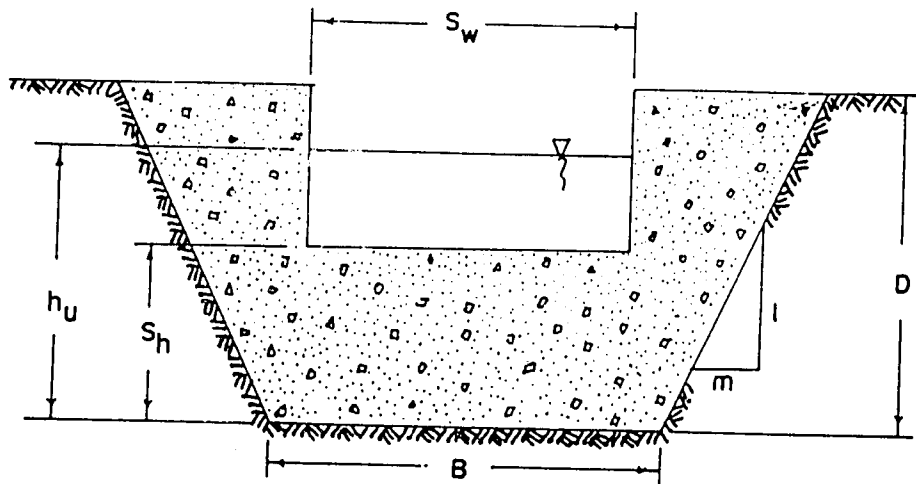
Free Flow: $Q = C_{d3} * N * S_w * (h_u + z_u - S_h)^{1.5}$

for, $(h_u > S_h + z_u)$

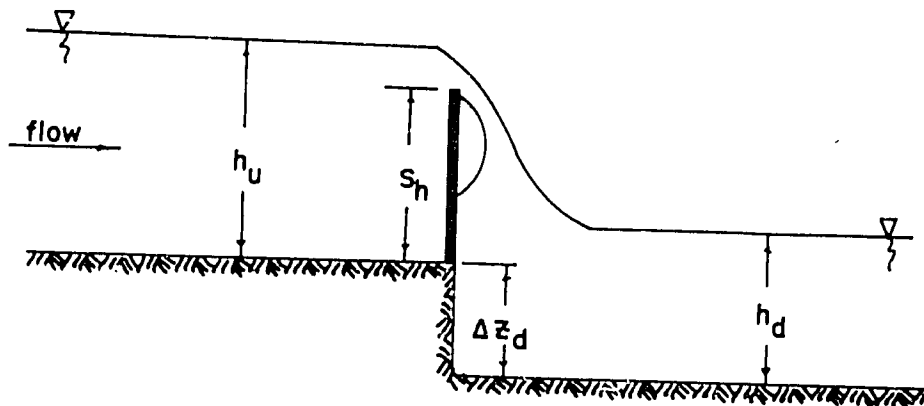
Zero Flow: $Q = 0$

for, $(h_u \leq S_h + z_u)$

- where,
- S_w - weir sill width (m)
 - S_h - weir sill height (m)
 - N - number of weirs in parallel
 - C_{d3} - weir flow discharge coefficient
 - h_u - upstream flow depth (m)
 - z_u - upstream elevation drop (m)
 - Q - discharge rate (m³/s)



DOWNSTREAM VIEW



SIDE VIEW

Figure A-4. Rectangular weir control structure.

5,6- Sluice Gates with Side Weirs

Sluice Gate Discharge, Q_1

Free Flow: $Q_1 = Cd_1 * N * A * \sqrt{2g(h_u + z_u - b/2)}$ for, $(h_u > b)$,
 $(h_d < b + z_u + z_d)$

Submerged Flow: $Q_1 = Cd_2 * N * A * \sqrt{2g(h_u - h_d + z_u + z_d)}$ for, $(h_u > b)$,
 $(h_d > b + z_u + z_d)$,
 $(h_u > h_d - z_u - z_d)$

Zero Flow: $Q_1 = 0$ for, $(h_u \leq h_d - z_u - z_d)$

Side Weir Discharge, Q_2

Free Flow: $Q_2 = Cd_3 * N * S_w * (h_u + z_u - S_h)^{1.5}$ for, $(h_u > S_h + z_u)$

Zero Flow: $Q_2 = 0$ for, $(h_u \leq S_h + z_u)$

Total Discharge, $Q = Q_1 + Q_2$

where,

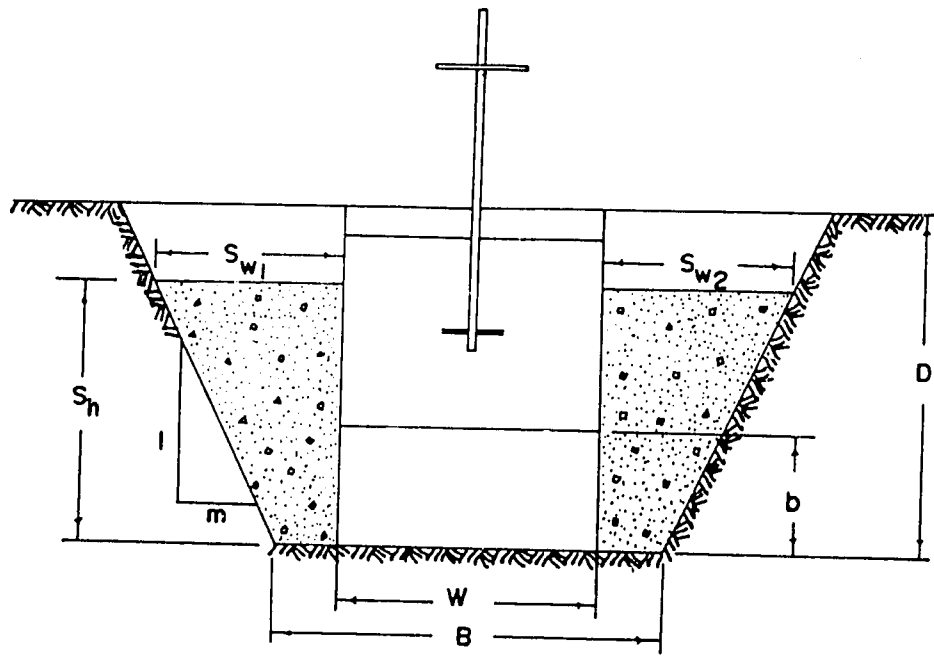
- g = 9.81 m/s²
- b = sluice gate opening (m)
- W = sluice gate diameter (m)
- N = number of gates in parallel
- Cd_1 = free flow discharge coefficient
- Cd_2 = submerged flow discharge coefficient
- Cd_3 = weir flow discharge coefficient
- S_w = weir sill width (m)
- S_h = weir sill height (m)
- h_u = upstream flow depth (m)
- h_d = downstream flow depth (m)
- z_u = upstream elevation drop (m)
- z_d = downstream elevation drop (m)
- Q_1 = sluice gate discharge rate (m³/s)
- Q_2 = side weir discharge rate (m³/s)
- Q = total discharge rate (m³/s)

and,

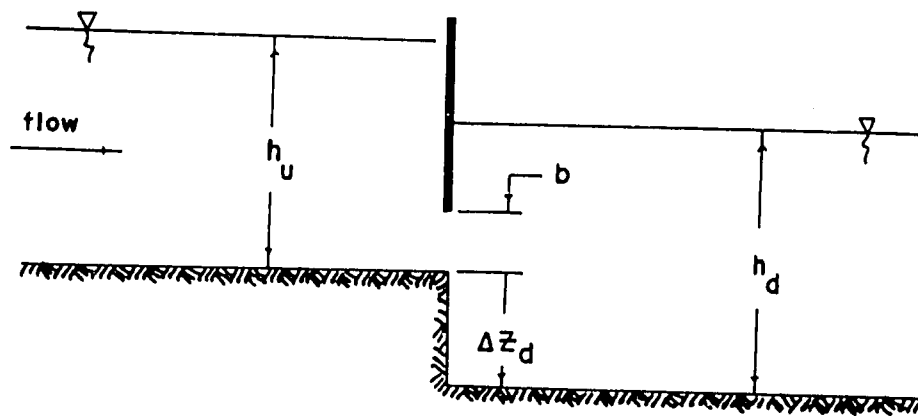
- $A = b * W$ (Rectangular Gate)
- $A = W^2(\beta - \sin\beta)/8$ (Circular Gate)

where,

$$\beta = 2 * \arcsin((2 * b - W) / W) + \pi$$

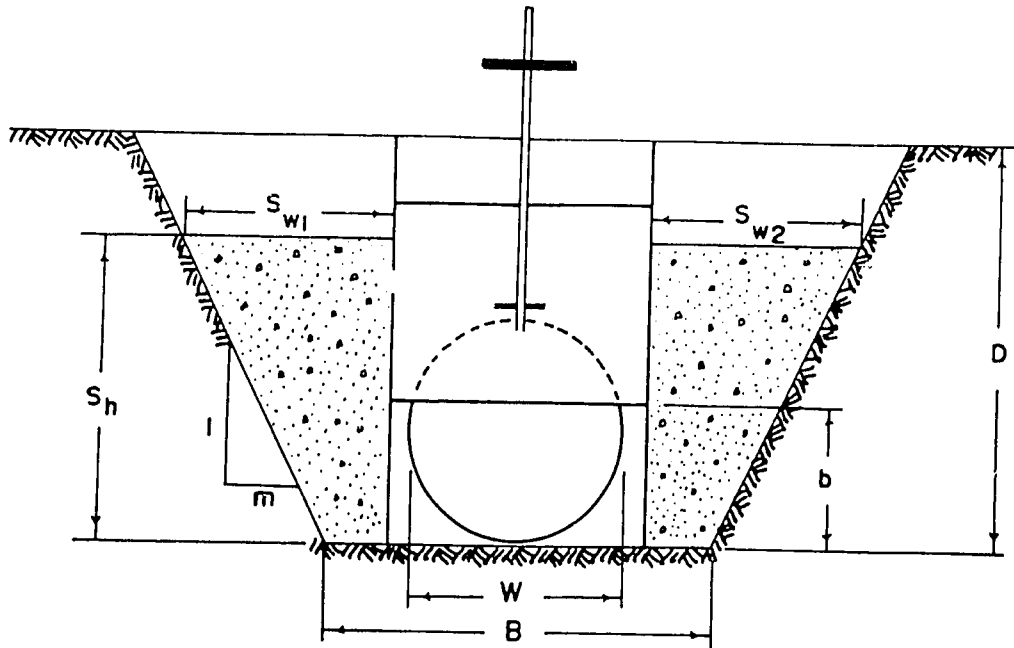


UPSTREAM VIEW

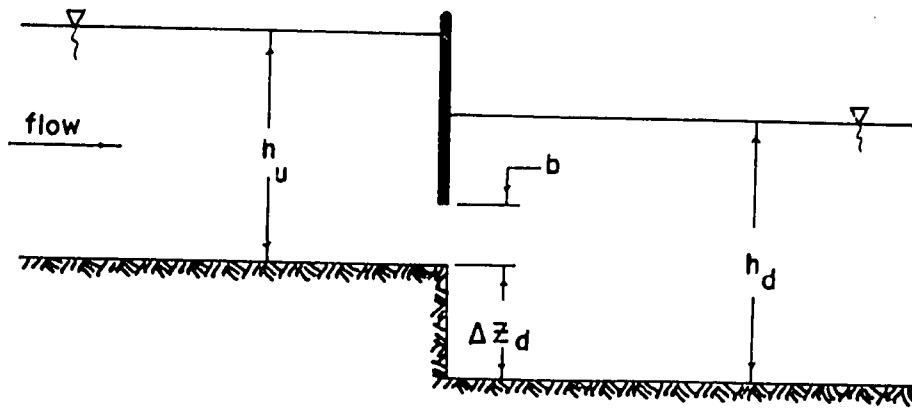


SIDE VIEW

Figure A-5. Rectangular sluice gate control structure.



UPSTREAM VIEW



SIDE VIEW

Figure A-6. Circular sluice gate control structure.

Turnout Structure Equations

1,2- Orifice Turnout Structures

Free Flow:	$Q = C_d * A * \sqrt{2g(h_u - P - b/2)}$	for, $(h_u > b + P),$ $(h_d < b + P)$
Submerged Flow:	$Q = C_d * A * \sqrt{2g(h_u - h_d)}$	for, $(h_u > b + P),$ $(h_d > b + P)$
Open Flow:	$Q = C_d * A * \sqrt{2g(h_u - P)/2}$	for, $(h_u > P),$ $(h_u \leq b + P)$
Zero Flow:	$Q = 0$	for, $(h_u \leq P)$

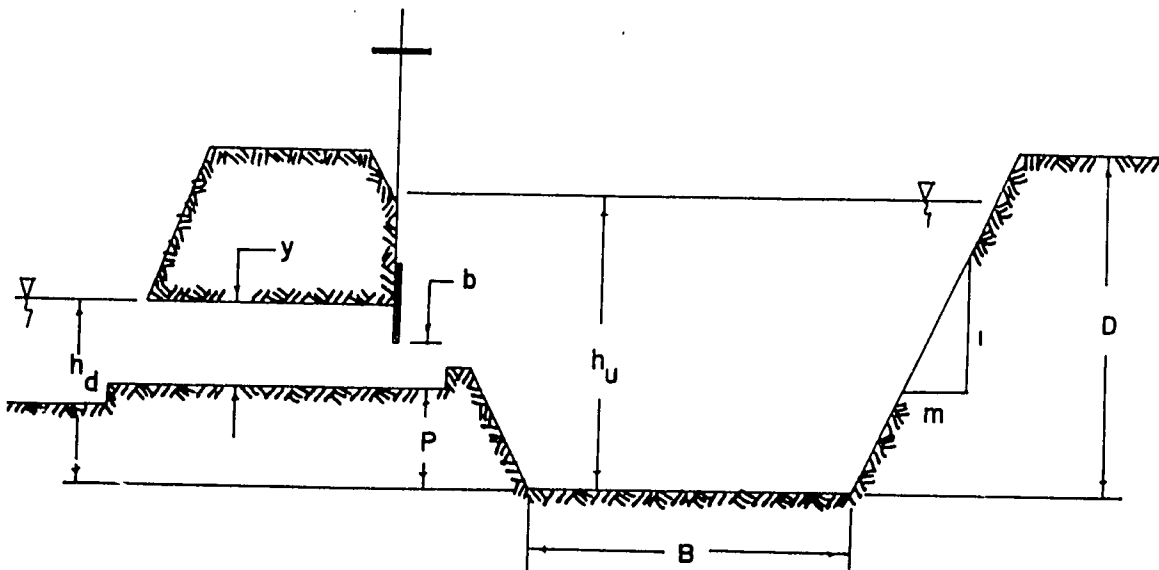


Figure A-7. Circular and rectangular orifice turnouts.

1,2- Orifice Turnout Structures (concl)

where, $g = 9.81 \text{ m/s}^2$
 b = turnout opening (m)
 y = turnout height or diameter (m)
 W = turnout width (rectangular) (m)
 P = turnout position (m)
 C_d = discharge coefficient
 h_u = upstream flow depth (m)
 h_d = downstream flow depth (m)
 Q = turnout discharge rate (m^3/s)

and, $A = b \cdot W$ (Rectangular Turnout)

$A = y^2(\beta - \sin\beta)/8$ (Circular Turnout)

where,

$$\beta = 2 \cdot \arcsin((2 \cdot b - y)/y) + \pi$$

The downstream flow depth (h_d) for orifice-type turnouts must be inputted to the model since it is external to the canal system being simulated. It is estimated by a linear stage-discharge relationship as illustrated in Figure A-8. The parameter P_1 is referred to as "DS Depth" in the turnout menu of the CDAT program. This is the difference in elevation between the bottom of the canal upstream of the turnout and the water surface downstream of the turnout when the discharge through the turnout is zero. If this water surface is lower than the bottom of the canal then P_1 will be a negative value, otherwise it is positive. The parameter P_2 is referred to as "DS Slope" in the turnout menu of the CDAT program. This is the slope of the approximating line as shown in Figure A-8. Physically, it represents the rate of water level increase on the downstream side of the turnout as the turnout discharge increases.

The value of P_1 can be measured in the field. Then the value of P_2 can be calculated by measuring the water surface elevation downstream of the turnout for a typical discharge rate. This will give two points from which the approximating stage-discharge relationship can be defined. Additional points could be obtained by measuring water level elevations at other discharge rates and using linear regression to compute P_1 and P_2 . The hydraulic model CANAL uses this stage-discharge relationship when calculating the flow rate from orifice-type turnouts during a simulation. The downstream flow depth, h_d , is first computed using the turnout flow rate from the previous time step. If this depth is enough to cause submerged flow conditions through the turnout then the new discharge rate is calculated based on the difference between h_u and h_d , and the current turnout setting (see Figure A-7). Otherwise, the discharge rate is

calculated based on free-flow conditions. The flow regime through an orifice-type turnout can be seen on the sixth alpha screen display (see Figure 6) during a simulation.

APPROXIMATION OF STAGE — DISCHARGE
RELATIONSHIP DOWNSTREAM OF TURNOUTS

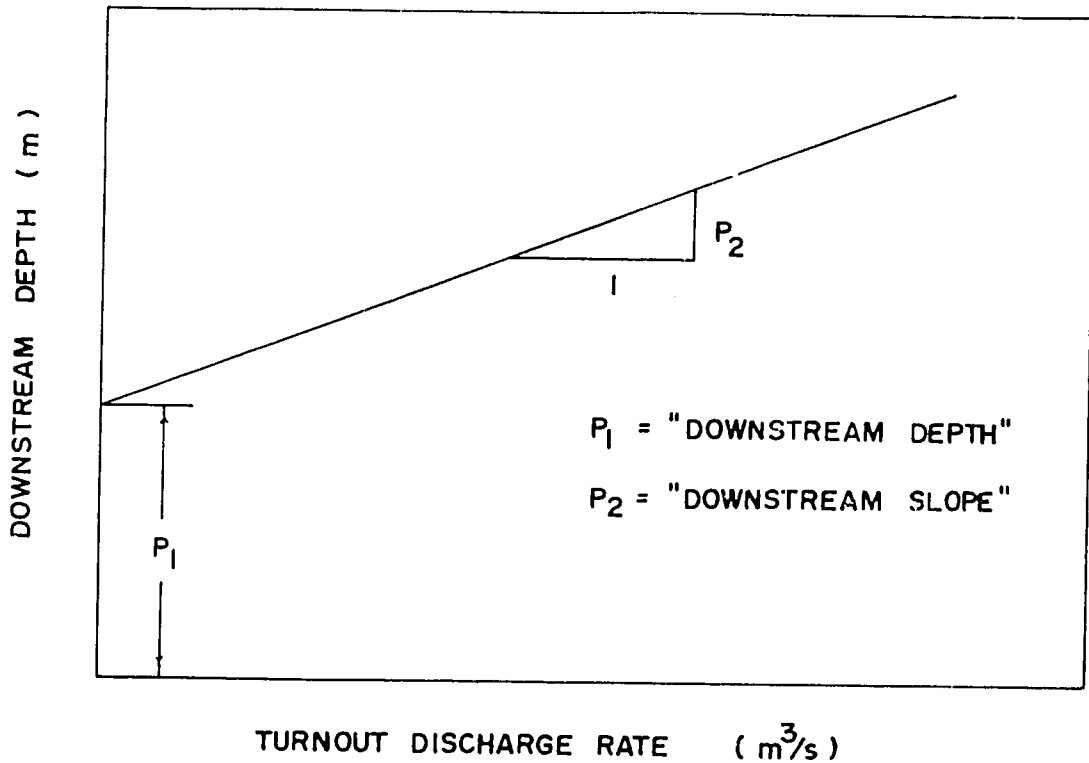


Figure A-8. Linear approximation of the stage-discharge relationship downstream of turnouts.

3 - Adjustable Weir Turnout Structures

Free Flow: $Q = C_d * W * (h_u - D + b)^{1.5}$ for, $(h_u > b + P)$
 Zero Flow: $Q = 0$ for, $(h_u \leq b + P)$

where,
 b - weir sill height (m)
 W - weir sill width (m)
 P - turnout position (m)
 C_d - discharge coefficient
 h_u - upstream flow depth (m)
 D - upstream canal depth (m)
 Q - turnout discharge rate (m³/s)

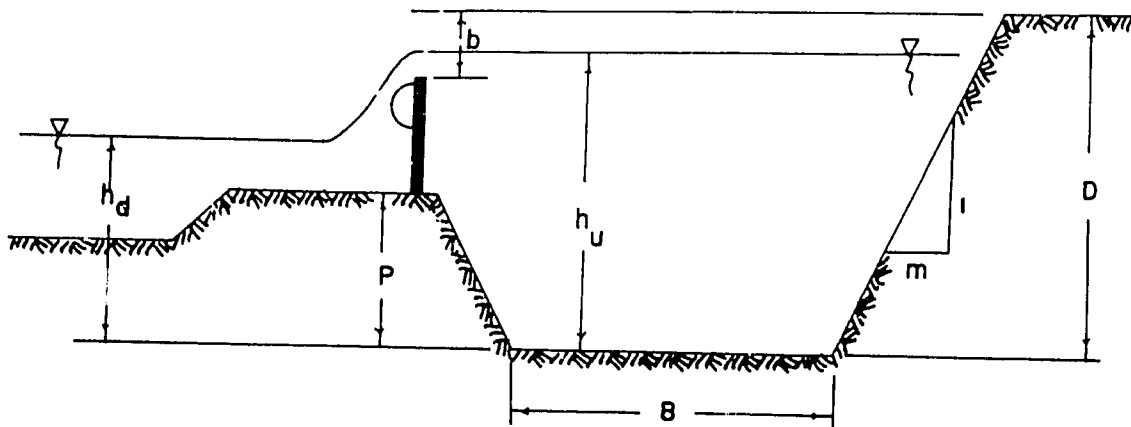


Figure A-9. Adjustable weir turnout.

4 - Wasteway Weir Turnout Structures

Free Flow: $Q = C_d * W * (h_u - P)^{1.5}$

Zero Flow: $Q = 0$

for, $(h_u > P)$

for, $(h_u \leq P)$

where,

- W = weir sill width (m)
- P = turnout position (m)
- C_d = discharge coefficient
- h_u = upstream flow depth (m)
- Q = turnout discharge rate (m^3/s)

Note: All weir turnout and control structures are assumed to operate under free flow ("modular") conditions at all times.

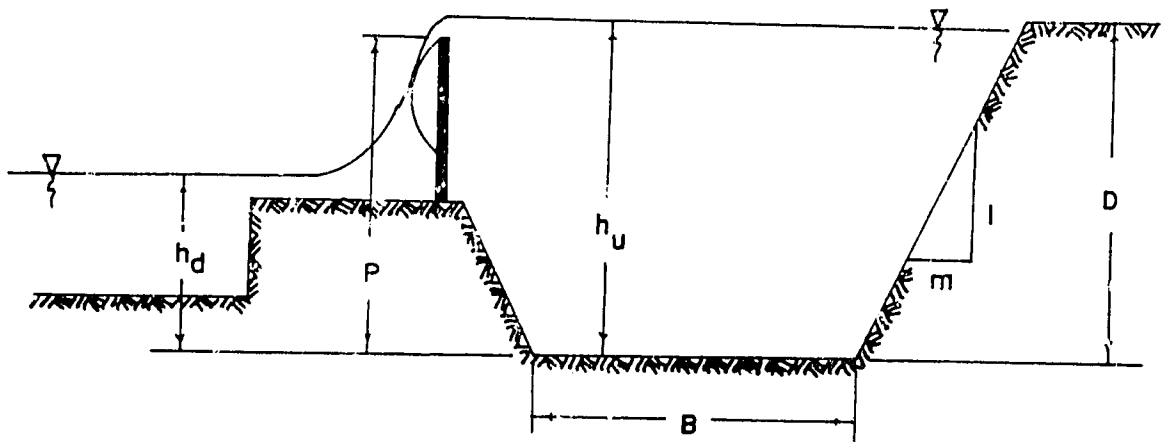


Figure A-10. Wasteway weir turnout.

Discharge Coefficients

The approximate expected range of coefficient of discharge (C_d) values is as follows:

Sluice Gate - free flow	$0.50 < C_{d1} < 0.80$
Sluice Gate - submerged flow	$0.50 < C_{d2} < 0.80$
Rectangular Weir - free flow	$1.00 < C_{d3} < 2.50$

The expected range of C_d values for orifice turnout structures is the same as the expected range of C_{d2} values above for sluice gates under submerged flow conditions. Also, the expected range of C_d values for both weir turnout structures and wasteway weirs is the same as the expected range of C_{d3} values above for rectangular weir control structures under free flow conditions. The model always assumes that weir control and turnout structures operate under free flow ("modular") conditions.

If field calibration measurements yield coefficient of discharge values which are outside of the above expected ranges then the data should be checked carefully for any errors which may have been present. Theoretically, the coefficient of discharge values for Cd1 and Cd2 should be equal to 0.611; however, field conditions can cause the actual values to deviate from this theoretical value slightly. The value for Cd3 is not as well defined since the equation assumes that the weir sill is sharp-crested and no terms are included to take into account contraction effects. Many weir structures in actual canal systems are not strictly sharp-crested and contracted flow is common. Therefore, the value of Cd3 can be expected to vary more than the values for Cd1 and Cd2. Nevertheless, for rectangular sharp-crested weirs operating under suppressed (i.e. no end contractions) flow conditions the discharge coefficient is expected to be about 1.83 (given the above equation forms and metric units).

Seepage Loss: Measurement

For large canal sections the seepage loss rate is best determined using the inflow-outflow method. For small canal sections it is more accurate to use the ponding method since the small flow rates are difficult to measure precisely. When the inflow-outflow method is used the flow must be steady in the canal section for which the measurement is being made and any turnout flow from the section must be taken into account as non-seepage outflow. The measured seepage outflow rate should be divided by the average wetted perimeter of the canal section and by the length of the section to give a value in units of length per unit time. This value should then be converted to units of millimeters per day so that it can be entered directly into a configuration data file for the canal system. Descriptions of the inflow-outflow and ponding methods for measuring seepage rates can be found in United States Bureau of Reclamation (USBR) publications (e.g. Lining for Irrigation Canals, USBR, 1963).

Field Data Forms

The following two pages (Table A-1) are forms which may be used to record field data for each reach of a canal system. It is not necessary to use the forms exactly as they appear here; however, it is recommendable to adopt some standard form for recording the field data and calibration results so that all of the data is organized and easily referenced when needed. Such data forms should be filled out and they should be saved, even after the data is entered into a configuration data file. It is also useful to make a sketch of the control and turnout structures to accompany the data forms.

TABLE A-1. FIELD DATA COLLECTION FORMS

**MAIN SYSTEM
CALIBRATION AND HYDRAULIC DATA**

Date _____ Time _____ Location _____

Branch _____ Reach _____ Reach Name _____

Names of persons making measurements:

CONFIGURATION:

Base Width (m) _____ Roughness (Manning) _____

Side Slope (m/m) _____ Canal Slope (m/100m) _____

Reach Length (m) _____ Canal Depth (m) _____

Seepage Rate (mm/day) _____

CONTROL STRUCTURE:

Type of Structure _____

Number of Structures _____ Width or Diameter (m) _____

Weir Sill Width (m) _____ US Elevation Drop (m) _____

Weir Sill Height (m) _____ DS Elevation Drop (m) _____

Flow Regime	b (m)	h_u (m)	h_d (m)	Q (m^3/s)	C_d
Free Flow					
Submerged					
Weir Flow					

TABLE A-1. FIELD DATA COLLECTION FORMS (concl)

MAIN SYSTEM
CALIBRATION AND HYDRAULIC DATA

No.	Name	Type	Width (m)	Height (m)	Position (m)	Distance (m)	C _d	P ₁	P ₂
1									
2									
3									
4									
5									
6									
7									
8									
9									

APPENDIX B

WHAT CAN GO WRONG ?

During a simulation the solution to the hydrodynamic equations may fail and the simulation stopped. A message will appear on the screen to indicate to the program user that this has occurred. No recovery can be made from this kind of error when the message is displayed - the model has already detected the problem with the numerical solution and has taken a series of steps to attempt to avoid a solution failure. When these steps are unsuccessful the solution will fail and the simulation will stop.

Failure of the numerical solution can occur as a result of relatively abrupt changes in the flow conditions or due to unusual flow conditions for which the model is incapable of performing satisfactory simulations. However, numerical problems are almost always related to what are called "boundary conditions". The most critical boundary conditions are those which exist at control structures and turnouts. Sudden changes in the flow status at a boundary condition can cause the hydraulic model to lose track of the flow levels and flow rates from one time step to the next. Local hydraulic transients near the boundary condition may be severe enough for the model to generate impossible flow conditions in the vicinity of the boundary (e.g. negative flow depths).

Listed below are some of the possible causes of a failure of the numerical solution during a simulation. Observance of these potentially troublesome conditions can help to avoid solution failures.

Advance Too Fast

When a real canal is filled with a high inflow rate it is difficult to control the water levels before a steady-state flow condition is achieved. High filling rates during a hydraulic simulation can also cause problems with the numerical solution. The canals may overflow before a control structure can open sufficiently, and if the control structure does open quickly, the next downstream reach will be subjected to large hydraulic transients. Similarly, the numerical solution may become unstable if the rate of advance during canal filling is very high. Thus, as with real canal systems it is advisable to hydraulically simulate the filling of canals with a flow rate that is significantly lower than the maximum flow capacity of the channels.

Advance Just Beyond Reach End

When the water arrives at the end of a reach at the completion of the advance phase a significant change occurs in the mathematical representation of the downstream boundary condition. Usually this transition is made smoothly by the model. However, if the advancing water just barely arrives at the downstream end of the canal reach, this transition may result in negative flow rates and flow depths at the boundary. This can happen because of the internal characteristics of the mathematical representation of the flow conditions (obviously there can never be negative flow depths in a real canal). If this situation is encountered during a simulation the model will attempt to correct the numerical solution artificially. If this correction fails then the numerical solution itself will probably fail. The only way for the program user to correct such an error is to begin filling of the canal system again with a slightly different inflow hydrograph. This kind of error is not very common.

Inflow Rate Changes Too Fast

Abrupt changes in the system inflow rate can cause large hydraulic transients in the canals and result in failure of the numerical solution. The model attempts to avoid abrupt system inflow changes by restricting the rate of inflow change during entry of values into the inflow hydrograph at the beginning of a simulation. These restrictions also apply to the change of system inflow rate from the keyboard during a simulation. However, since the new inflow rate will not be in effect until all of the canal reaches are updated for the current time step, it is possible to make multiple inflow rate changes before the new inflow rate is actually used in the simulation. In this way the user could cause a change in the system inflow rate which is larger than that which would be allowed during entry of values into the inflow hydrograph.

The inflow rate to all reaches other than the most upstream reach in a canal system (reach one of branch one) is always equal to the outflow through the control structure at the downstream end of the next upstream reach. In other words, the outflow from one reach is equal to the inflow to the next downstream reach since the reaches are separated only by a control structure. If the change in inflow rate through a control structure is abrupt from one time step to the next the numerical solution may fail. During gate scheduling the hydraulic model will attempt to avoid this problem by restricting the absolute magnitude of a control structure adjustment from one time step to the next. However, if either the manual or the pre-set operational modes are enabled such restrictions will not be applied. In this case the only restrictions are the extreme upper and lower

adjustment limits. Therefore, it is advisable that large rates of change for control structure settings be avoided by the program user during a hydraulic simulation.

Pre-Set Control Structure Adjustments

When the program user selects the pre-set operational mode at the beginning of a simulation the control structure movements will be defined for the duration of the simulation. In this case the model will only check the extreme upper and lower adjustment limits. As the simulation begins the model will perform the control structure adjustments exactly as specified. No checks are made to ensure that sluice gate submergence is maintained or that flow regimes do not change from submerged flow to free flow. Adjustable weir structures are not checked to make sure that they do not raise above the upstream water level and cut-off the flow to the downstream reaches. In effect, the control structure adjustments are made "blindly" by the model without checking the upstream or downstream flow levels and comparing them to the control structure settings. For this reason it is possible that unfavorable boundary conditions may be encountered during a simulation with pre-set control structures which could cause the numerical solution to diverge and fail.

Sluice Gate Submergence

During gate scheduling, the model will attempt to maintain the water level on the upstream side of a sluice gate above the bottom of the gate opening. This is to maintain upstream submergence. When the control structures are pre-set or when the manual operational mode is in effect it is possible for the water level upstream of a sluice gate to drop below the bottom of the gate opening. This can occur as a result of a sluice gate adjustment, a dropping water level upstream of a sluice gate, or both. If during a simulation the water level upstream of a sluice gate does drop below the bottom of the gate the structure no longer behaves like an orifice but rather as a channel constriction. If this happens then the flow equation must be changed to a different form to accurately model the hydraulic characteristics of the structure under such conditions. This change in equations can often cause abrupt changes in the computed discharge through the structure, which in turn may cause the numerical solution to fail.

APPENDIX C

98'

ERROR MESSAGES

File System Errors

Help File Not Found.

Indicates that the model CANAL or the editor CDAT unsuccessfully tried to access the help file in the disc directory specified by the current DOS path. There is a separate help file for each of the two programs: HELP1R.DAT is used by the CDAT program, and HELP2R.DAT is used by the CANAL model. Either (1) the DOS path is wrong, or (2) the help file(s) are missing or defective.

Data File Errors

Turnouts 1 and 2 in Reach 3 of Branch 1 Are Not in Order.

Indicates that the model has located an error in the turnout data after comparing turnout numbers and turnout locations in a canal reach. Turnouts are "not in order" if a turnout with a higher number is located upstream of a turnout of a lower number. For example, turnout 1 cannot be located downstream of turnout 2 in any given canal reach.

Turnout 2 in Reach 1 of Branch 1 is Located Beyond Reach End.

Indicates that the model has located an error in the turnout data after comparing turnout locations with a canal reach length. No turnout can be located beyond the end of the canal reach in which the turnout resides. For example, a turnout cannot be located at 2000 meters from the beginning of a reach if the reach length is only 1500 meters (this would imply that the turnout actually resides in the next downstream reach - this is unacceptable to the model).

Turnout 4 in Reach 4 of Branch 2 Has Sill Height Too Low.

Indicates that the model has located an error in the turnout data after having identified a turnout as a weir structure and comparing the sill height to the canal depth. The fixed sill height for a wasteway weir, and the minimum sill height for an adjustable weir turnout structure, must be at least one-half of the canal depth for the current reach. This restriction prevents sudden large turnout discharges during filling of a canal system.

Turnouts 2 and 3 in Reach 3 of Branch 3 Located Too Close.

Indicates that the model has found the locations of two adjacent turnout structures to be too close together. The difference between the respective locations must be at least equal to twice the sum of the turnout structure widths. For example, if turnout 1 is one meter wide and turnout 2 is two meters wide, the minimum allowable difference in their locations is $2*(1+1) = 6$ meters. This restriction prevents computational nodes from being placed too close to each other. This check is only made for adjacent turnouts in the same canal reach (i.e. it is not made for the last turnout in one reach and the first turnout in the next downstream reach).

Branch Number 2 is Linked to a Non-Existent Upstream Branch.

Indicates that the model has identified a branch linkage error after checking the upstream connection for the branch number shown. In this example, it means that the specified upstream branch from which branch two is to be linked to is higher than the number of branches in the canal system. The number of branches in the system is specified in the system dimensions data. For example, branch two cannot be linked to branch four if there are only three branches in the system.

Branch Number 2 is Linked to a Non-Existent Upstream Reach.

Indicates that the model has identified a branch linkage error after checking the upstream connection for the branch number shown. In this case the upstream linkage for the branch number shown in the error message is presumably to a turnout in an upstream branch (otherwise the reach number would be zero and this error would not occur). However, in checking the system dimensions the model found that the specified reach in the upstream branch is of a higher number than the number of reaches in the branch. For example, branch number two cannot be linked to reach four of branch one if branch one has only three reaches. Branches can be linked to (1) a turnout in an upstream branch, or (2) the end of the last reach in an upstream branch.

Branch Number 2 is Linked to a Non-Existent Upstream Turnout.

Indicates that the model has identified a branch linkage error after checking the upstream connection for the branch number shown. In this case the upstream linkage for the branch number shown in the error message is presumably to a turnout in an upstream branch (otherwise the turnout number would be zero and this error would not occur). However, in checking the system dimensions the model found that the specified turnout in the upstream reach of the upstream branch is of a higher number than the number of turnouts in the

reach. For example, branch number three cannot be linked to turnout five of reach one of branch one if this reach has only four turnouts. Branches can be linked to (1) a turnout in an upstream branch, or (2) the end of the last reach in an upstream branch.

Branch Number 2 Linked to a Wasteway Weir Turnout Structure.

Indicates that the model has identified a branch linkage error after checking the turnout type of the specified upstream linkage point for the branch number shown in the error message. Downstream branches cannot be linked to wasteway weir turnout structures.

Branch Numbers 2 and 3 Have the Same Upstream Linkage.

Indicates that the model has identified a branch linkage error after checking the linkage specifications for the different branches in the canal system. Branch linkages must be unique, that is, no two branches can share the same upstream linkage point.

Turnout 3 in Reach 2 of Branch 1 has "Position" Too High.

This means that the position parameter for a turnout is at or above the top of the canal bank. This creates a condition in which the turnout can only discharge water if the canal is overflowing at the turnout location. Therefore, the position parameter must be less than the canal depth for the reach in which the turnout resides.

Turnout 3 in Reach 2 of Branch 1 DS Flow Depth is Too High.

This means that the "DS Depth" parameter which defines the flow depth downstream of an orifice-type turnout for zero discharge is at or above the top of the canal. Therefore, there will always be a negative head differential across the turnout unless the canal is overflowing. Turnout downstream flow depth is measured from the bottom of the canal upstream of the turnout (i.e. the canal reach from which the turnout takes water). This downstream depth must not exceed the top of the canal bank (see Appendix A).

Data File Warnings

Structure is Wider Than Canal Base at Branch 1 Reach 2.

The total control structure width is the width multiplied by the number of structures as specified in the configuration data file. This warning message is the result of a comparison between the canal base width and the total control structure width.

Structure Diameter More Than Canal Depth at Branch 1 Reach 2.

The diameter of a circular control structure exceeds the depth of the canal reach. This involves a situation in which the control structure is taller than the top of the canal, unless the upstream elevation drop is sufficient at the control.

Weir Sill More Than 90% of Canal Depth at Branch 1 Reach 2.

The sill height for a fixed weir is above 90% of the canal depth. This could be a non-adjustable weir structure or fixed side weirs on a sluice gate structure. In this situation the weir may be too restrictive to allow flow to leave the reach without causing the banks to overtop.

Weir Sill Less Than 30% of Canal Depth at Branch 1 Reach 2.

The sill height for a fixed weir is below 30% of the canal depth. This could be a non-adjustable weir structure or fixed side weirs on a sluice gate structure. In this situation the weir may allow too much flow to pass out of the reach, reducing the utility of having the structure there at all.

Net Drop in Elevation is Negative at Branch 1 Reach 2.

The sum of the upstream and downstream elevation changes at a control structure is a negative value. This means that the elevation of the bottom of the downstream reach is higher than that of the upstream reach at the control structure. These two parameters are referred to as "US Delta Z" and "DS Delta Z" in the control structure menu of the CDAT program.

The Downstream Canal Reach is Wider at Branch 1 Reach 2.

This means that the canal base width in a downstream reach is greater than the base width in an upstream reach. This warning can occur for successive reaches in a canal branch, and it can occur for the last reach in a branch where another branch is linked to this reach.

The Downstream Canal Reach is Deeper at Branch 1 Reach 2.

This means that the canal depth in a downstream reach is greater than the depth in an upstream reach. This warning can occur for successive reaches in a canal branch, and it can occur for the last reach in a branch where another branch is linked to this reach.

Upstream Target Lower Than Downstream at Branch 1 Reach 2.

The product of reach depth and operational supply level (OSL) for an upstream reach is less than the same product for a downstream reach (this also takes into account elevation changes in the canal bottom across the control structure). This creates a situation in which the canal system may be incapable of meeting target levels during a simulation in which gate scheduling is enabled. When the downstream reach attains OSL the upstream reach will have to go above its OSL in order to prevent backflow through the control. That is, the head differential across the upstream control structure will be negative. The result is that the operational requirements of the two reaches are such that they fight each other in an attempt to match actual downstream water levels to the target levels.

Hydraulic Simulation Errors

First Advance Failed

Indicates that the solution failed during the initial time step for which water entered a canal reach at the beginning of the advance phase. The probable cause is that the initial discharge rate into the reach was either very small or very large. When this error is shown the simulation has been aborted and the user can type any key to go back to the main menu.

Advance Stopped

Indicates that the solution failed during the advance phase in a canal reach. This error occurs when the inflow rate to a reach is small compared to the seepage loss in the same reach. When this is the case the advance of water in the reach will stop if the inflow rate to the reach is matched or exceeded by the seepage loss rate (because there is no "extra" water to push the advancing front forward toward the end of the canal reach). When this error is shown the simulation has been aborted and the user can type any key to go back to the main menu.

Too Many Cells

Indicates that the number of computational cells in a branch has exceeded the maximum of 275. This upper limit on the number of cells in a single branch can be reached when the advance phase in the last reaches of the branch are taking a long time to complete. Computational node numbering always begins with zero in each branch and continues through to the end of the last reach in the branch. Therefore, the limit of 275 nodes is for each individual branch only, not for the entire canal system. The user can always see the

beginning and ending computational node numbers for each reach at the bottom of the second alpha screen display. When this error is shown the simulation has been aborted and the user can type any key to go back to the main menu.

Solution Diverged

Indicates that the solution to the hydraulic equations has failed after twenty iterations of the Newton-Raphson method for the current reach and time step. The model may have taken some corrective steps to avoid a solution failure before this message is displayed on the screen (see Appendix B). When this error is shown the simulation has been aborted and the user can type any key to go back to the main menu.

Messages On The Second Alpha Screen

Above Target Level

Indicates that during gate scheduling the actual downstream water level in a canal reach is more than one centimeter above the target water level. In this case the model is trying to lower the water level to the target level.

Activate Scheduling

Indicates that the scheduling status for the current reach has just been activated (or re-activated) and that the actual downstream water level in the reach is near the target level. When this happens the scheduling status for the reach will be shown as "On" in the default alpha screen display after having been "Wait" previously (see Gate Scheduling in Section 1). The scheduling status for a canal reach may activate and deactivate any number of times during a simulation, especially if the flow is highly transient.

Adjusting The Grid

Indicates that the advance phase has just completed in the current reach and that the model is in the process of inserting computational nodes at turnout locations (if there are any turnouts in the reach). Some existing nodes may be deleted by the model at this time if they are close to the locations of the turnout nodes. This grid adjustment is automatic and the results of it are essentially transparent to the user.

Advanced Beyond DS Boundary

Indicates that the advance phase has just been completed for the current reach. The model knows that the advance phase is finished because it has compared the new advance distance with the reach length and has found that the water would have advanced beyond the downstream boundary had there not been a control structure (there is always a control structure at the downstream reach boundary). In the following time step the computational grid for this reach will be adjusted and possibly thinned or expanded.

Below Target Level

Indicates that during gate scheduling the actual downstream water level in a canal reach is more than one centimeter below the target water level. In this case the model is trying to raise the water level up to the target level.

Canal Overflowing

Indicates that the flow depth at some point in the current reach (probably the downstream end of the reach) exceeds the canal depth. If the real canal is lined and there is an earthen berm above the top of the lining on the canal banks, then the real canal would not necessarily be overflowing. The model does not calculate spillage of this form, therefore, when this message is shown the hydraulic simulation is probably losing accuracy since in a real canal system water may be spilling over the canal banks. However, if there is a wasteway weir in the reach, the spillage over this structure will be calculated by the model. And, if the control structure is a sluice gate, the spillage over the side weirs will also be calculated (if the sill is not placed too high (see Section 2, Control Structure Data). Bulk flow over the canal banks when the canal is overflowing is not calculated.

Circular Failed To Converge

Indicates that the current reach has a circular sluice gate control structure at the downstream end, and that during gate scheduling the model was unable to calculate the new structure setting for the current time step. This may happen if the setting is very small or is much different than the previous setting. Due to the nature of the circular sluice gate's flow area vs. gate setting equation the new settings must be computed iteratively, unlike rectangular sluice gate settings which are computed directly. This is why the convergence may fail. When this happens the setting will remain where it is for the current time step and the scheduling status for this reach will deactivate until the actual water level is near the target level again. This can also happen for circular orifice turnouts.

C_Culv Neg Head

Indicates that the model has detected a negative head differential across a circular culvert control structure. This means that the water elevation on the downstream side of the structure is higher than on the upstream side, and this implies that the water should be flowing upstream through the structure. The model does not allow "backflow" through a control structure, so when this happens the model will set the discharge through the structure to be equal to zero. This may cause related problems which could eventually cause the simulation to be aborted by the model.

Expanding The Grid

Indicates that the advance phase has just been completed for the current reach and that the model is inserting additional computational nodes into the internal grid. Such grid expansion typically occurs for relatively short canal reaches and fast advance times (the faster the advance of water, the lower the number of computational nodes). Strategic insertion of additional nodes helps avoid solution failures of the hydraulic equations by providing a numerically stable grid arrangement.

Flow Changing Too Fast

Indicates that during gate scheduling the model has detected a potential "over-reaction" by a control structure to a sudden change in the flow rates nearby. When this happens the model will restrict the actual rate of change and the scheduling status for the reach will be temporarily deactivated. The model will usually be able to reactivate the reach in one or two time steps (see Section 1, Gate Scheduling).

High Control Limit

Indicates that during gate scheduling the model has discovered that the new control structure setting is above the extreme upper limit on the operational range of the structure. When this happens the scheduling status will be deactivated until the setting can be reduced.

Lost DS Submergence

Indicates that during gate scheduling a comparison of the downstream flow level and the gate setting has shown that a change in sluice gate flow regimes from submerged to free flow is impending. The model will try to avoid this regime change from taking place by lowering the gate a small amount and temporarily deactivating the scheduling status.

Lost US Submergence

Indicates that during gate scheduling a comparison of the upstream flow level and the gate setting has shown that the upstream water level is about to drop below the bottom of the sluice gate. This would probably cause a large change in the computed discharge of the structure and is, therefore, desirable to avoid if possible. When this happens the model will lower the gate a small amount and the scheduling status will temporarily deactivate.

Low Control Limit

Indicates that during gate scheduling the model has discovered that the new control structure setting is below the extreme lower limit on the operational range of the structure. When this happens the scheduling status will be deactivated until the setting can be increased.

Matching Cd1 to Cd2

Indicates that the model has been unable to avoid a change in flow regime across a sluice gate from submerged to free flow. In such a case the model will try to ensure a successful continuation of the simulation by recomputing the free flow discharge coefficient (Cd1) so that the current discharge is matched to the new discharge when a change in flow equations occurs (see Appendix A, Control Structure Equations). Usually this action will cause a smooth transition from submerged flow to free flow.

Neg Head Ctrl Setter

Indicates that during gate scheduling the model detected a negative head differential across a sluice gate or culvert. This means that the water elevation downstream of the structure is higher than the water elevation upstream of the structure. When this happens the flow should reverse at the structure and go upstream (at least temporarily). However, this flow reversal is not allowed by the model and this situation will instead cause the flow to be set at zero across the structure and the scheduling status will temporarily deactivate. This may cause related problems which could eventually cause the simulation to be aborted by the model.

Qfixed Exceeds DFC

Indicates that during gate scheduling the model has discovered that the new reach discharge rate (see Section 1, Gate Scheduling) exceeds the design flow capacity (DFC) of the reach. The design flow capacity is computed at the beginning of a simulation using Manning's equation for uniform flow at the operational supply level. When this happens the model will set the new discharge rate equal to the design

flow capacity; however, the scheduling status will remain activated.

Qfixed Is Too Low

Indicates that during gate scheduling the model has discovered that the new reach discharge rate (see Section 1, Gate Scheduling) is less than the minimum allowable rate of 0.01 cubic meters per second. In this case the model will set the new discharge rate equal to 0.01 cubic meters per second and the scheduling status will remain activated.

R_Culv Neg Head

Indicates that the model has detected a negative head differential across a rectangular culvert control structure. This means that the water elevation on the downstream side of the structure is higher than on the upstream side, and this implies that the water should be flowing upstream through the structure. The model does not allow "backflow" through a control structure, so when this happens the model will set the discharge through the structure to be equal to zero. This may cause related problems which could eventually cause the simulation to be aborted by the model.

Sluice Gate Dry

Indicates that the bottom of a sluice gate control structure has raised above the upstream water level, or that the upstream water level has dropped below the bottom of the gate, or a combination of the two. This situation is desired to be avoided since it tends to cause abrupt changes in the flow rate through the structure. Furthermore, the model does not do a good job of simulating the flow through a sluice gate under these conditions. Therefore, this message may indicate impending failure of the solution to the hydraulic equations.

Sluice Neg Head

Indicates that the model has detected a negative head differential across a sluice gate control structure. This means that the water elevation on the downstream side of the structure is higher than on the upstream side, and this implies that the water should be flowing upstream through the structure. The model does not allow "backflow" through a control structure, so when this happens the model will set the discharge through the structure to be equal to zero. This may cause related problems which could eventually cause the simulation to be aborted by the model.

Stabilizing The Discharge

Indicates that during gate scheduling the model has detected a fluctuating discharge rate through a control structure. In this case the model will set the new discharge rate equal to the average of the discharge rates from the previous three time steps in an attempt to stabilize the discharge (see Section 1, Gate Scheduling). This message does not necessarily mean that anything is wrong, only that the model is taking some corrective action to stabilize the canal system. This situation can occur when the model successively "overshoots" the target level, causing the actual water to oscillate above and below the target level.

Thinning The Grid

Indicates that the advance phase has just been completed for the current reach and that the model is erasing some computational nodes from the internal grid. Such grid thinning typically occurs for relatively long canal reaches and slow advance times (the slower the advance of water, the larger the number of computational nodes). Strategic erasure of selected nodes helps avoid solution failures of the hydraulic equations by providing a numerically stable grid arrangement.

Weir Sill Too High

Indicates that during gate scheduling the model has discovered that the new setting for an adjustable weir control structure is above the upstream water level. This situation is automatically corrected by lowering the sill a small amount so that some water still passes downstream. This message can only occur after the reach containing the weir has first begun to discharge water into the next downstream reach.

WATER MANAGEMENT SYNTHESIS PROJECT REPORTS

WATER MANAGEMENT SYNTHESIS PROJECT REPORTS

- WMS 1 Irrigation Projects Document Review
- Executive Summary
 Appendix A: The Indian Subcontinent
 Appendix B: East Asia
 Appendix C: Near East and Africa
 Appendix D: Central and South America
- WMS 2 Nepal/USAID: Irrigation Development Options and Investment
 Strategies for the 1980's
- WMS 3 Bangladesh/USAID: Irrigation Development Options and Investment
 Strategies for the 1980's
- WMS 4 Pakistan/USAID: Irrigation Development Options and Investment
 Strategies for the 1980's
- WMS 5 Thailand/USAID: Irrigation Development Options and Investment
 Strategies for the 1980's
- WMS 6 India/USAID: Irrigation Development Options and Investment
 Strategies for the 1980's
- WMS 7 General Asian Overview
- WMS 8 Command Area Development Authorities for Improved Water Management
- WMS 9 Senegal/USAID: Project Review for Bakel Small Irrigated
 Perimeters Project No. 685-0208
- WMS 10 Sri Lanka/USAID: Evaluation Review of the Water Management
 Project No. 383-0057
- WMS 11 Sri Lanka/USAID: Irrigation Development Options and Investment
 Strategies for the 1980's
- WMS 12 Ecuador/USAID: Irrigation Sector Review
- WMS 13 Maintenance Plan for the Lam Nam Oon Irrigation System in
 Northeast Thailand
- WMS 14 Peru/USAID: Irrigation Development Options and Investment
 Strategies for the 1980's
- WMS 15 Diagnostic Analysis of Five Deep Tubewell Irrigation Systems in
 Joydebpur, Bangladesh
- WMS 16 System H of the Mahaweli Development Project, Sri Lanka: 1980
 Diagnostic Analysis

- WMS 17 Diagnostic Analysis of Farm Irrigation Systems on the Gambhiri Irrigation Project, Rajasthan, India: Volumes I-V
- WMS 18 Diagnostic Analysis of Farm Irrigation in the Mahi-Kadana Irrigation Project, Gujarat, India
- WMS 19 The Rajanqana Irrigation Scheme, Sri Lanka: 1982 Diagnostic Analysis
- WMS 20 System H of the Mahaweli Development Project, Sri Lanka: 1983 Diagnostic Analysis
- WMS 21 Haiti/USAID: Evaluation of the Irrigation Component of the Integrated Agricultural Development Project No. 521-0078
- WMS 22 Synthesis of Lessons Learned for Rapid Appraisal of Irrigation Strategies
- WMS 23 Tanzania/USAID: Rapid Mini Appraisal of Irrigation Development Options and Investment Strategies
- WMS 24 Tanzania/USAID: Assessment of Rift Valley Pilot Rice Project and Recommendations for Follow-On Activities
- WMS 25 Interdisciplinary Diagnostic Analysis of a Work Plan for the Dahod Tank Irrigation Project, Madhya Pradesh, India
- WMS 26 Prospects for Small-Scale Irrigation Development in the Sahel
- WMS 27 Improving Policies and Programs for the Development of Small-Scale Irrigation Systems
- WMS 28 Selected Alternatives for Irrigated Agricultural Development in Azua Valley, Dominican Republic
- WMS 29 Evaluation of Project No. 519-0184, USAID/El Salvador, Office of Small-Scale Irrigation - Small Farm Irrigation Systems Project
- WMS 30 Review of Irrigation Facilities, Operation and Maintenance for Jordan Valley Authority
- WMS 31 Training Consultancy Report: Irrigation Management and Training Program
- WMS 32 Small-Scale Development: Indonesia/USAID
- WMS 33 Irrigation Systems Management Project Design Report: Sri Lanka
- WMS 34 Community Participation and Local Organization for Small-Scale Irrigation
- WMS 35 Irrigation Sector Strategy Review: USAID/India; with Appendices, Volumes I and II (3 volumes)

- WMS 36 Irrigation Sector Assessment: USAID/Haiti
- WMS 37 African Irrigation Overview: Summary; Main Report; An Annotated Bibliography (3 volumes)
- WMS 38 Diagnostic Analysis of Sirsia Irrigation System, Nepal
- WMS 39 Small-Scale Irrigation: Design Issues and Government Assisted Systems
- WMS 40 Watering the Shamba: Current Public and Private Sector Activities for Small-Scale Irrigation Development
- WMS 41 Strategies for Irrigation Development: Chad/USAID
- WMS 42 Strategies for Irrigation Development: Egypt/USAID
- WMS 43 Rapid Appraisal of Nepal Irrigation Systems
- WMS 44 Direction, Inducement, and Schemes: Investment Strategies for Small-Scale Irrigation Systems
- WMS 45 Post 1987 Strategy for Irrigation: Pakistan/USAID
- WMS 46 Irrigation Rehab: User's Manual
- WMS 47 Relay Adapter Card: User's Manual
- WMS 48 Small-Scale and Smallholder Irrigation in Zimbabwe: Analysis of Opportunities for Improvement
- WMS 49 Design Guidance for Shebelli Water Management Project (USAID Project No. 649-0129) Somalia/USAID
- WMS 50 Farmer Irrigation Participation Project in Lam Chamuak, Thailand: Initiation Report
- WMS 51 Pre-Feasibility Study of Irrigation Development in Mauritania: Mauritania/USAID
- WMS 52 Command Water Management - Punjab Pre-Rehabilitation Diagnostic Analysis of the Niazbeg Subproject
- WMS 53 Pre-Rehabilitation Diagnostic Study of Sehra Irrigation System, Sind, Pakistan
- WMS 54 Framework for the Management Plan: Niazbeg Subproject Area
- WMS 55 Framework for the Management Plan: Sehra Subproject Area
- WMS 56 Review of Jordan Valley Authority Irrigation Facilities

- WMS 57 Diagnostic Analysis of Parakrama Samudra Scheme, Sri Lanka: 1985 Yala Discipline Report
- WMS 58 Diagnostic Analysis of Giritale Scheme, Sri Lanka: 1985 Yala Discipline Report
- WMS 59 Diagnostic Analysis of Minneriya Scheme, Sri Lanka: 1986 Yala Discipline Report
- WMS 60 Diagnostic Analysis of Kaudulla Scheme, Sri Lanka: 1985 Yala Discipline Report
- WMS 61 Diagnostic Analysis of Four Irrigation Schemes in Polonnaruwa District, Sri Lanka: Interdisciplinary Analysis
- WMS 62 Workshops for Developing Policy and Strategy for Nationwide Irrigation and Management Training. USAID/India
- WMS 63 Research on Irrigation in Africa
- WMS 64 Irrigation Rehab: Africa Version
- WMS 65 Revised Management Plan for the Warsak Lift Canal, Command Water Management Project, Northwest Frontier Province, Pakistan
- WMS 66 Zimbabwe Joint Field Workshop
- WMS 67 Variations in Irrigation Management Intensity: Farmer-Managed Hill Irrigation Systems in Nepal
- WMS 68 Experience with Small-Scale Sprinkler System Development in Guatemala: An Evaluation of Program Benefits