

 $\underbrace{Users'\ Guide}_{\mathrm{for\ the\ macOS^{TM}\ Version}}$



Copyright and Disclaimer

This users' guide and the *mCanal* application may contain inaccuracies and typographical errors. Updates will be periodically made to both the users' guide and software, and incorporated in future editions. These changes may not be announced by Natural Resources Consulting Engineers, Inc. (NRCE).

NRCE does not warrant mCanal for any specific purpose and does not assume any liability resulting from use of the software. The hydraulic model in this app includes variations of commonly reported procedures and algorithms for simulating the hydraulics of open-channel and closed-conduit water conveyance and distribution systems. However, application of the model to specific situations may involve parameter combinations that occasionally violate one or more assumptions or may require interpretations beyond the limits of the various algorithms, resulting in incorrect or inaccurate simulations that cannot be anticipated by NRCE.

Copyright © 2015-2019 Natural Resources Consulting Engineers, Inc. All rights reserved.

 $Contact \ us \ at \ gmerkley @nrce.com.$

https://www.nrce.com

mCanal version 2.13 for macOSTM. Requires macOS 10.13 or later.

December 2019

This document was created using IAT_EX .

Content

I.	Introduction	1
II.	System Layout	13
III	Reach Configuration	27
IV.	Structure Configuration	49
V.	Structure Calibration	63
VI.	Pump and Valve Calibration	95
VII.	Scheduling Operations	101
VIII.	Numerical Parameters	111
IX.	Performing Hydraulic Simulations	117
Х.	Simulation Results	131

Annexes

Α.	Weir Submerged Flow	141
В.	Momentum Function	145
С.	USBR Radial-Gate Calibration	147
D.	Horseshoe Cross-Section Geometry	151
E.	Transitional Flow at Underflow Gates	157
F.	References and Bibliography	163

Nomenclature

- $A \quad {\rm cross-sectional \ area}$
- A_o gate opening area
- B pump curve coefficient
- C pump curve coefficient
- C_c coefficient of flow contraction
- C_d value or gate discharge coefficient
- C_{df} free-flow coefficient of discharge
- C_{ds} submerged-flow coefficient of discharge
- C_e rectangular-weir discharge coefficient
- C_f free-flow coefficient
- C_{me} units conversion coefficient (Manning equation)
- C_o hydraulic-seal coefficient
- C_v hydraulic head loss coefficient (pumps)
- d inside diameter
- D hydraulic drag term, equal to $S_f A$
- DS downstream
- E modulus of elasticity
- f pipe hydraulic head-loss factor (Darcy-Weisbach)
- g ratio of weight-to-mass $(9.81 \ m/s^2)$
- G_o gate opening
- G_r gate leaf radius
- G_w gate width
 - h water depth (h_u = upstream; h_d = downstream)
- \bar{h} vertical distance from channel bed to area centroid
- $h_c \quad {\rm vertical \ distance \ from \ area \ centroid \ to \ water \ surface}$
- h_e effective head (weirs)
- h_f hydraulic head loss
- h_v valve hydraulic head loss
- h_z static lift (pumps)
- H reservoir geometry variable
- K bulk modulus; minor loss coefficient
- K_h V-notch-weir head adjustment factor
- K_L rectangular-weir length adjustment
- K_s submerged-flow coefficient (weirs)
- L reach length; weir crest length; pipe length
- L_e effective weir-crest length
- m inverse side slope
- m_c inverse side slope in the sill section of a BCW
- M momentum function
- n_f free-flow exponent
- \dot{P} pressure; upstream weir crest height
- Q flow rate, or discharge
- Q_p pump flow rate
- r_g gate diameter (round gate)
- r_o gate opening diameter (round gate)
- R hydraulic radius
- R_e Reynolds number
- S submergence (h_d/h_u)

- S_f hydraulic loss gradient
- S_o longitudinal bed slope
- t elapsed time
- T water surface width; water temperature
- TDH total dynamic head
 - US upstream
 - V average cross-sectional velocity

 - $w \quad \text{side-slope length} \quad$
 - W throat width
 - W_p wetted perimeter
 - x distance
 - $y \quad \text{weir crest-referenced depth} \\$
 - $z \quad {\rm crest \ or \ sill \ height}$
- α, β, ξ various calibration parameters
 - χ $\,$ transitional-flow-range parameter $\,$
 - ΔE_d reach drop
 - ΔE_s upstream invert drop

 - Ω_o free-submerged orifice threshold
 - ρ water density
 - θ V-notch-weir angle
 - ν kinematic viscosity

Note: Each chapter begins on an odd-numbered page, so the last page of some chapters is blank.

I. Introduction

I.1. Overview of *mCanal*

mCanal is a software application (app) based on a mathematical model for simulating one-dimensional, incompressible, unsteady-flow hydraulics of water in open channels. It is designed for analyzing the hydraulic performance of system designs and redesigns, changes in system infrastructure, operational variations in existing and planned systems, and in assessing operating decisions in training and analysis situations.

The app is especially suited to performing simulations of both simple open channels and branching open-channel networks with regular channel cross sections and flow-control structures, such as gates, weirs, flumes, and pumps. Through the app, a simulated canal system can be easily configured to model a variety of different cross-sectional channel shapes, including linearly-varying channel cross sections and compound longitudinal bed slopes.

Governing Equations The St-Venant equations of motion (energy or momentum) and mass balance are combined with control-structure equations at boundaries to perform hydraulic simulations. At some locations, flow rate or water depth may be specified, and everywhere else they are calculated. The St-Venant equations are derived from the Navier-Stokes equations, and with turbulent-flow capability.

The app is interactive and has comprehensive internal data checking and cross-checking features, but it is, nevertheless, intended for use by specialists in hydraulics who are knowledgeable about the operation of open-channel conveyance and delivery systems. The most useful and applicable results from the app can best be obtained from such specialists who understand the governing hydraulic equations and their limitations. Thus, this Users' Guide includes a wealth of technical information to define the details of hydraulic model implementation. Moreover, the more complete the field calibration data for a given canal system and flow-control structures, the more representative the modeling results – it is important to base simulations on extensive calibration data from real systems whenever possible. This is still true, to a certain extent, for the design of new systems, or modifications and extensions to existing systems. To this end, the app contains a wide variety of calibration features, including several pre-calibrated structures based on laboratory and or field data.

When using a mathematical model like mCanal it is often helpful to think creatively and flexibly to successfully simulate the hydraulic conditions of interest. For example, even though there are many different structure (gates, weirs, and others) types included in the app, some adaptation may be required to simulate variations not specifically covered herein, but that are found in practice.

In many instances, this adaptation can be achieved by modifying calibration parameters, but in any case, an engineering knowledge of hydraulics and the calibration of flow control structures is highly advantageous. The app should be thought of as a computational tool which can be set up to accomplish your hydraulic simulation objectives, and the most successful users will realize that there is often more than one way to approach a simulation scenario, and to produce practically-applicable results.

Herein, "the app" and *mCanal* are used synonymously.

I.2. Perform a Simulation!



After you download and install the *mCanal* app from the App StoreTM you can execute it immediately and perform a hydraulic simulation. The app has a default data set with one canal reach, so you can press \Re -1 (hold down the command key and press the 1 key) or click on the green-dot icon in the toolbar, to run a simulation. Results are updated on the three views of the main window during a simulation. The results can also be written to text and CSV (comma-separated values) files, which you can open with a text editor,

spreadsheet application, or word processor. A CSV file is actually a type of text file, and easy to import to a spreadsheet app.

I.3. Getting Started

If you are familiar with the Mac OS^{TM} , you will intuitively understand most of the interface features of mCanal because they mostly follow standard conventions. But if you need help getting started, you can browse through the following general interface descriptions.

The main window has a toolbar at the top, and three views: (1) the System Layout View at the left; (2) the Branch View at the upper right; and, (3) the Graphs View at the lower right. The figure below is an example of the main window. You can dynamically resize the three views within the main window, and even hide one or two of the views, by dragging the vertical or horizontal dividers as far as they will go. The Branch View shows a schematic of a selected branch, with water levels as blue curves, in which water flows (generally) right to left. That is, the abscissa is distance along the branch, and the ordinate is relative elevation. The Graphs View shows one or more curves of simulation results, in which the abscissa is elapsed time, and a legend can be optionally displayed. The system layout is a plan view, as seen from above, and a schematic representation of the layout configuration – you can interact with, edit, and specify display options for the system layout using the mouse and several editing modes (Chapter II).



The main menu is always shown at the top of the screen, duplicating many of the commands that are available through the toolbar, popup menus, and keyboard shortcuts.

System layouts consist of two or more node types (there are five different layout node types that can be used), and one or more channel reaches, separated by control structures called *regulators*. Reaches can be organized in series and in branches. *Turnouts* are another type of control structure, through which lateral inflows and or outflows can be simulated along the length of a reach. Most of the structure types that can be used as regulators can also be used as turnouts.

Regulators and Turnouts Control structures are boundary conditions that include both regulators and turnouts. Each type (e.g. gates, weirs, flumes, pumps, and so on) has one or more equations that define the relationship between water depths (upstream and downstream), volumetric flow rate, and structure setting. Some structures are non-adjustable, and so they have a fixed setting, but most are adjustable, and can be opened and closed. These boundary conditions are combined with the governing St-Venant equations in hydraulic simulations.

Each reach has at least one regulator at the upstream end, and another at the downstream end. There can be multiple regulators in parallel at either reach end. Turnouts allow the simulation of lateral flows along the length of a reach, and there can be multiple turnouts in each reach.

All available structure types can be used as regulators, in general, and most can also be used as turnouts.

I.3.1. Clicking, Menus, and Toolbars

When you read "click on..." in this Users' Guide, it means moving the cursor (using the mouse, trackpad, or other pointing device) over some area of a window or data entry sheet and pressing the left mouse button. When you see "right click..." it means that you move the cursor over a region of a window or sheet and press the right mouse button. Alternatively, you can control-click (hold down the **control** key and click the left mouse button) instead of the right mouse button. An Apple mouse might not have visible left and right buttons, but you can usually click on the left or right sides of the mouse to accomplish the respective click operations.

Some of the sheets, and views within the sheets, have popup menus that can be displayed by rightclicking over an area of a view, or over a specific feature on a view. These are described for each case in this Users' Guide for the app, but in general you should experiment by right-clicking over windows to discover otherwise hidden commands and options that will help you set up system layouts, specify modeling options, perform hydraulic simulations, and view simulation results.

Besides the popup menus, there is a main menu that appears at the top of the screen. Click on an item in the main menu to display a drop-down menu with one or more commands. Note that many of the main menu commands have keyboard shortcuts (e.g. \Re -O to open a file) and many have corresponding toolbar buttons that do exactly the same thing as clicking on a menu command.

I.3.2. Data Entry Sheets

The app includes several sheets (popup windows over the main window) for entering and editing data, and for choosing among options. These sheets are not resizable, meaning that you cannot change their width or height, as you can with the main window. Many of the sheets have tabs near the top, each with a short text description of what is contained in the corresponding tab. You can click on a tab to display the contents of a tab. Each of the data entry sheets has two buttons: **OK** and **Cancel**. You can click over either of the buttons to invoke the corresponding action, or you can simply press the **Enter** key for the default button, which is **OK**. The buttons are used as follows:

OK Accepts all changes that you have made to data values and or options in the sheet, including those values on any and all of the tabs that might be found on the sheet. If you have only looked at the values on the window without making any changes, clicking on **OK** is the same as clicking on **Cancel**. The **OK** button is usually the default button, the one that would be invoked if you press the **Enter** key. However, when a simulation is running or paused, most data entry sheets will have the **OK** button disabled, because you can only view, but cannot change, the physical configuration of a system during a simulation.

Cancel If you have made some changes to the data values and or options in the sheet and do not want to keep any of those changes, click on the **Cancel** button. When you do this, all data values and options will revert to those that existed before you opened the sheet. As mentioned above, if you have not made any changes, clicking on either **OK** or **Cancel** will have the same effect.

I.3.3. Return and Tab Keys

In this Users' Guide, if you are instructed to type \mathbb{H} -**R** (for example), it means to depress and hold down the command key while you depress the **R** key, and so on for any other key in combination with the **command** key. The same procedure applies to the **option**, **control**, and **shift** keys, where applicable in the app. In some cases, you may need to depress combinations of **option**, **shift**, and **control** along with another key.

When typing numerical or text values in a data-entry sheet, do not press the **return** key. This will usually be interpreted as a click on the **OK** button and will cause an exit from the sheet. Instead, you can click over another part of the sheet, or press the **tab** key to scroll to the next item on the sheet. You can also press **shift-tab** to scroll backwards among the items.

I.4. Preferences

Open the **Preferences** sheet by selecting **mCanal**, **Preferences** (\Re -,) from the main menu. The sheet has six tabs, and each is described in the following subsections.

I.4.1. Options

These include both general and layout-specific preferences. The general preferences will remain in effect for all layout configuration files that you work with in the app, not just for the current system layout, and they will be the defaults for any new layouts. Once you set them, they will remain the same on a given computer until you change them again.

Options	Nodes	Frame	Curves	Arrows	Notes]	
General preference	es		Preferences for the current layout				
Units Metric English Font size Small			Units N E Sort re	Aetric Inglish eaches JS to DS			
 Medium Large Show warn 	ings		Show	ulphabetica w warnings ultaneous s re supercri	illy solution	me	
 Auto-open Simultaneo Allow track 	last file ous solutior pad gestu	res	Slow	Speed	Fa	ast	
General preferences	are default	values for a	all new layout	t files.			
				ОК	Canc	el	

Units

You can set the default system of units to be metric or English. In the app, metric units use meters, cubic meters per second, hectares, Celsius degrees, and so on. English units use feet, cubic feet per second (cfs), acres, and Fahrenheit degrees. However, some parameters are shown only as millimeters and they do not change with the selected system of units. In this Users' Manual, all of the sample figures and simulation examples are presented in metric units.

Parameter	Metric System	English System
Angle	degrees	degrees
Length	meters (m)	feet (ft)
Depth, head	meters (m)	feet (ft)
Elevation	meters (m)	feet (ft)
Evaporation	millimeters per day (mm/day)	millimeters per day (mm/day)
Seepage	millimeters per day (mm/day)	millimeters per day (mm/day)
Flow rate	cubic meters per second (m^3/s)	cubic feet per second (cfs)
Slope	meters per meter (m/m)	feet per foot (ft/ft)
Temperature	$^{\circ}C$	$^{\circ}F$
Viscosity	square meters per second (m^2/s)	square feet per second (ft^2/s)

In the app, flow rate, volumetric flow rate, and discharge are used synonymously.

Font size

Choose the font size that is used for layout node labels (*System Layout View*), and for labels on the graphical results in the *Branch* and *Graphs* views of the main window. This also affects the font size for the legend in the *Graphs View*.

Show warnings

When you check this box, warning messages may be shown when checking the data and when beginning a simulation. If the box is unchecked, it means that warnings will not be displayed and will not prevent a simulation from running. However, it is recommended that the box be checked because addressing (and eliminating) warnings can help prevent issues during a simulation.

Auto-open last file

Layout configuration files contain layout details, calibration parameters and all other values that are needed to uniquely define a canal system for simulation purposes. When this box is checked, the app will attempt to open the last configuration data file each time that mCanal is opened. The last configuration data file is that which was most recently opened or saved before closing the application. This is an especially convenient option when you work on a single data configuration data file over an extended period of time.

Simultaneous solution

This is an option to solve the hydraulic equations simultaneously each time step for each branch in the layout (all layouts have at least one branch with one or more reaches). This means that the calculations are potentially performed across some reach boundaries. In some situations, and with certain structure types, the simultaneous solution will perform better (numerically) than the reach-by-reach solution. This is because there is no time lag in the solution to upstream and downstream conditions at a reach end, so it is more realistic when submerged-flow occurs at a reach end, especially for a relatively large simulation time-step size.

With the simultaneous-solution option, live simulation updates on the main window may not be as smooth as they would otherwise be, especially in branches with many reaches.

Allow Trackpad Gestures

For computers with a trackpad or analogous input device, checking this box will allow scrolling (two fingers) vertically and horizontally, rotating, and zooming based on the standard "gestures" for these actions. Scrolling can also be done with an $Apple^{TM}$ mouse. Gestures apply to the system layout view.

Sort reaches

You can choose to order canal reach names in lists, which appear in many data entry and options sheets throughout the app, either alphabetically (by name) or in an approximate upstream-to-downstream order. This also applies to the text and CSV file outputs of simulation results. The upstream-to-downstream ordering is exactly correct if you have only one branch (one or more reaches in series) in the system layout, otherwise it is just one ordering among two or more possibilities. If you choose the upstream-to-downstream option, the app will sort the reaches according to their position in the system layout. Each reach must have a unique name.

Ignore supercritical regime

The app does not currently handle supercritical flow, so this option (to ignore the possibility of supercritical flow) is checked and cannot be changed.

Speed

In some cases, to more carefully observe a running simulation in the main window, you may wish to artificially slow down the calculations. This is accomplished by moving the *Speed* slider setting to the left. The furthest left position will cause a 2-s delay between time steps, while a simulation will run as fast as possible in the right-most position. The right-most position is the default for new system layouts.

I.4.2. Nodes

These are general preferences for the five system layout node types, comprising attribute features of the nodes in the system layout view. Here, you can specify the shape, color, and size of each type of layout node, as seen below. These attributes will apply to all configuration data files in the app on a given computer.

There are five different shapes, almost any number of colors, and 16 sizes to choose from for each node type. You can see previews of the symbols at the right side of the sheet. To select the node color, click on the corresponding **Color** bar to open a standard color-selection sheet.

Reach nodes Shape Color Square Image: Size in the state of the shown at the right side. Primary source node Shape Color Size in the state of the shown at the right side.		Options	Nodes	Frame	Curves	Arrows	Notes	
Shape Color Size 10 Square Image: Size Image: Size Image: Size Image: Size Image: Size Primary source nodes Shape Color Size Image: Size Image: Size Shape Color Size Image: Size Image: Size Image: Size Bifurcation nodes Shape Color Size Image: Size Image: Size Image: Size Image: Size Up triangle Image: Size Image: Size<	Read	ch nodes						
Square Primary source node Shape Circle Secondary source nodes Shape Color Size Diamond Or Size Bifurcation nodes Shape Color Size Output riangle Shape Color Size	Sha	pe		Color	Siz	e	10	
Primary source node Shape Color Size 6 Circle Circle Secondary source nodes Shape Color Size 6 Diamond Color Size 6 Diamond Color Size 6 Up triangle Color Size 6 Up triangle Color Size 6 Layout node attributes (general), with a sample of each shown at the right side.	Sc	quare	\diamond				-	
Shape Color Size 6 Circle Image: Color Size 6 Secondary source nodes Shape Color Size 6 Diamond Image: Color Size 6 Bifurcation nodes Shape Color Size 6 Up triangle Image: Color Size 6 Confluence nodes Shape Color Size 6 Down triangle Image: Color Size 6 Down triangle Image: Color Size 6 Color Size 6 Image: Color Confluence nodes Shape Color Size 6 Layout node attributes (general), with a sample of each shown at the right side.	Daire		de					
Shape Color Size 6 Circle Image: Color Size 6 Secondary source nodes Shape Color Size 6 Diamond Image: Color Size 6 Bifurcation nodes Shape Color Size 6 Up triangle Image: Color Size 6 Confluence nodes Shape Color Size 6 Down triangle Image: Color Size 6 Layout node attributes (general), with a sample of each shown at the right side.	Prim	ary source no	ae	0.1	e.			
Circle Secondary source nodes Shape Diamond Diamond Shape Color Shape Color Size 6 Up triangle Confluence nodes Shape Color Shape Color Size Own triangle Color Size Color Size Confluence nodes Shape Color Size Size Color C	Sha	pe		Color	SIZ	e	6	
Secondary source nodes Shape Color Size 6 Diamond Color Size 6 Bifurcation nodes Shape Color Size 6 Up triangle Color Size 6 Confluence nodes Shape Color Size 6 Down triangle Color Size 6 Down triangle Color Size 6 Down triangle Color Size 6 Color Color Size 6 Color Color Color Size 6 Color Color C	Ci	rcle						
Shape Color Size 6 Diamond Image Image Image Bifurcation nodes Shape Color Size 6 Up triangle Image Image Image Confluence nodes Shape Color Size 6 Down triangle Image Image Image Layout node attributes (general), with a sample of each shown at the right side.	Seco	ondarv source	nodes					
Diamond Bifurcation nodes Shape Color Size 6 Up triangle Color Size 6 Confluence nodes Shape Color Size 6 Down triangle Color Size 6 Down triangle Color C	Sha	ne		Color	Siz	20	6	
Bifurcation nodes Shape Color Size 6 Up triangle Color Size 6 Confluence nodes Shape Color Size 6 Down triangle Color Size 6 Down triangle Color		amond						•
Bifurcation nodes Shape Color Size 6 Up triangle Confluence nodes Shape Color Size 6 Up triangle Color Size 6 V Cancel CK Cancel		amonu	<u> </u>					•
Shape Color Size 6 Up triangle Image Image Image Image Confluence nodes Shape Color Size 6 Down triangle Image Image Image Image Image Layout node attributes (general), with a sample of each shown at the right side. Image I	Bifu	rcation nodes						
Up triangle Confluence nodes Shape Color Down triangle Color Layout node attributes (general), with a sample of each shown at the right side. OK Cancel	Sha	ре		Color	Siz	e	6	
Confluence nodes Shape Color Size 6 Down triangle Color Layout node attributes (general), with a sample of each shown at the right side. OK Cancel	U	o triangle	\$		-	-0		
Confluence nodes Shape Color Size 6 Down triangle Color Layout node attributes (general), with a sample of each shown at the right side. OK Cancel								
Shape Color Size 6 Down triangle Image: Color Size 6 Layout node attributes (general), with a sample of each shown at the right side. Image: Color Image: Color OK Cancel	Con	fluence nodes						
Down triangle	Sha	pe		Color	Siz	e	6	
Layout node attributes (general), with a sample of each shown at the right side.	Do	own triangle	\$		-	-0		V
Layout node attributes (general), with a sample of each shown at the right side.								
OK Cancel	Layou	t node attribut	es (general)	, with a sam	ple of each	shown at th	e right side	e.
						OK	Can	cel

I.4.3. Frame

As with the **Nodes** tab, this one refers to the system layout view – it like a canvas upon which the system layout plan-view is rendered. Specify the dimensions of the view to accommodate the coordinates of the nodes. The aspect ratio that is specified here will remain constant regardless of shifting or zooming the layout view. For example, if you find that some of the nodes are falling outside the *System Layout View*, you can change the frame dimensions to encompass the breadth and depth of the entire layout.

The area and aspect ratio (width divided by height) are calculated according to the parameter values in the **Origin**, **Width** and **Height** boxes. The coordinates of each corner of the frame are shown above and below the view area, where the lower-left corner is also the origin, always at (0.00, 0.00).

It is noted, however, that the absolute and relative locations of nodes in the system layout do not necessarily reflect the real distances or relative locations in the canal system – it is a schematic view showing connectivity among layout nodes. Thus, if you change the position of a node in the layout view, it will not (for example) automatically change the length of a reach.

Options	Nodes	Frame	Curves	Arrows	Notes	
Origin: x (m)	Wid	th (m)				
0.00	64	00.00				
Origin: y (m)	Heig	ght (m)	Ar	ea: 307	2 ha	
0.00	48	00.00	As	Aspect ratio: 1.333		
(0.00, 48	00.00)			(6400.00, 4)	800.00)	
(0.00, 0	0.00)	_	_	(6400.00,	0.00)	
Layout frame for drawing the (current) system layout nodes and connections.						
			l	ОК	Cance	əl

I.4.4. Curves

The *Graphs View* of the main window shows curves of simulation results, both during a simulation and after a simulation terminates, in which the abscissa is always elapsed time. In this tab, you can specify the default curve attributes for that view, and you can also reset the colors to the default values by clicking on the eponymous button, as seen in the figure below. There are four default curve thickness options and four line styles that you can choose from.

For reach flow rates or water depths, choose the upstream or downstream end of the reach. Do the same for regulators, which are found at either end of each reach. The index number on the slider refers to the number in the list of reaches, regulators, or turnouts for the selected branch in the system layout. For example, the first selected reach for results output in the *Graphs View* will have a red color, by default.

You can select an index number using the horizontal slider (see the figure below), or by clicking on a any of the curves in the *Default curve colors* box found at the upper right. Change the color of any of the 16 indices by clicking on the color bar below the slider – this will bring up a standard window for color

selection. There will always be at least one reach, and one regulator at each end of each reach.

Click on the **Reset Colors** button to apply the default set of 16 colors. The app defines a different set of colors for macOS light and dark modes, respectively, so the result of clicking on the button will depend on the mode that you are currently using in macOS (*Preferences, General, Appearance*).

In any case, you can choose the type and number of curves to display in the *Graphs View*, and you can override the default colors and other curve attributes for a given system layout. They are fully customizable. Additional modifications to curve attributes for the *Graphs View* of the main window are described in Chapter X.1.3.

Options	Nodes	Frame	Curves	Arrows	Notes	
Reach curves		Reach le	ocation	Default c	Default curve colors	
Line width Thin Line style Solid	•	Upstream ODownstream				
Regulator curve Line width Thin Line style Chain Dot	s ¢	Regulat Ups Dov	or location stream vnstream			
Turnout curves		Default	curve colors			
Line width Thin Line style Chain Dot	\$		number	1 1 1 1 1	1 1 1 1	
Default curve attrib	utes (genera	l) for simula	ation results.	R	eset Cold	
				ОК	Cano	

I.4.5. Arrows

You can optionally show arrowheads at the downstream end of any or all reach curves to indicate the general direction of flow through the system. The size of the arrowheads can be specified by choosing a length option, and the length-to-width ratio.

There are three display options: (1) do not show arrowheads [**None**]; (2) show arrowheads on selected reach curves [**Some reaches**]; or, (3) show arrowheads on all reach curves [**All reaches**]. If you edit the layout node options for a given node by double-clicking on a node in the layout (in the *Selection* mode), and you toggle the arrowhead display, then the display option on this tab will automatically change to **Some reaches**.

Arrowheads are only for display purposes, to indicate the stream-wise direction in a reach – they do not affect hydraulic simulations.

I.4.6. Notes

You can type in the name of the modeler(s) and notes about the current configuration data file to succinctly document the system layout and simulation scenario for future reference. These will also be specific to the current configuration data file. It is a good idea to include notes, especially when you work with different versions of a system layout and or different layouts.

Here, and throughout the app, you can enter names and text in any language found in Unicode, which includes practically every written language.

I.5. Using Data Files

The app works with four types of data files:

- 1. Layout configuration data;
- 2. Simulation results;
- 3. Text files; and,
- 4. CSV files.

The first two have custom binary formats and are only accessible through *mCanal* itself. The third, text files, can be both written to and read from by the model. Text files with scheduling (Chapter VII) data can be read and written to, as can files with cross-sectional coordinates, and pump and valve calibrations. Also, simulation results files can be formatted as separate text files by the app. Finally, simulation results can be written to CSV files, which are easily imported into spreadsheet applications for custom analysis and graphing.

The default file-name extensions are:

File type	Extension
Configuration data	mcld
Simulation results	mcsr
Text files	txt
Comma-separated values	csv

The simulation results files are only used temporarily during a simulation. Upon successful completion of a simulation their contents are outputted to one or more text or CSV files, and the *mcsr* binary files are automatically deleted. You can also choose to not output any results to files and only view dynamic results on the main window during a simulation.

Layout configuration data files contain information about the system layout, reach characteristics, structure types and dimensions, calibration data, operational parameters, and options. In short, configuration data files have all the information needed by the app to hydraulically define a canal system and to perform a specific hydraulic simulation.

When you select **File**, **New** from the main menu, you are loading the default configuration, which is for a single canal reach and relatively simple operating conditions. This is the starting point for all other configurations that you create in the app. The default configuration is a ready to run "demo," meaning that you can perform a simulation without entering any parameters or specifying any options. But in general, you will want to specify many parameters to define a particular system configuration and layout before you are ready to perform a simulation that reflects the hydraulic behavior of a proposed or existing canal network. By default, the simulation results and other files created by the app are saved on the computer in the same folder as the layout configuration file.

The use of text files to import data for scheduling graphs, channel cross sections, and others is described separately below.

I.5.1. Opening Configuration Data Files

You can open a configuration data file by selecting **File**, **Open** from the main menu, or by typing \mathbb{H} -**O** from the keyboard. The app also maintains a list of the most recently opened configuration data files as a submenu to the **File** menu item. You can select one of those files from the list to reopen it, assuming it has not been renamed, moved or deleted.

When a layout configuration file is successfully opened, the name of that file will be shown in the title bar (caption) of the main window.

I.5.2. Saving Configuration Data Files

Select File, Save or File, Save As... from the main menu to save the current configuration data to a file. You can also type **#-S** or shift-**#-S** to save a configuration data file. If you have started with the default configuration, the file name will be simply "mCanal," so when you save it you will be prompted for a different file name. But if you have already saved the configuration data, or if you have opened another configuration that was previously saved, you can select File, Save and it will be saved with the same name, overwriting the previous file contents.

If you have opened a configuration data file and want to retain the previous file contents, choose the **Save As...** menu command and use a different file name and or folder location.

Remember to periodically save the configuration data while making editing changes.

II. System Layout

The system layout view shows a plan view (looking down from the top) of the reaches and other nodes that define the simulated hydraulic system. It is like a schematic map of the system and is always at the left side of the app's main window. Reaches are contiguous physical lengths of open channel, demarcated by one or more regulator structures at each end, upstream and downstream. Reaches are displayed in the system layout window as Bezier curves. Each Bezier curve between two nodes in the system layout window represents a single reach. Nodes are displayed as symbols, which can be selected and modified as described below in Chapter II.2. The default layout has one reach, with one primary source node (*System Source*) and one reach node (*Main canal*), as seen in the figure below.



A rectangular layout region is displayed within the layout window. The aspect ratio of the layout region is maintained until you specifically change it, so that the ratio of height to width of the region remains the same for any and all resizing of the layout window. This means that there may be unused space to the sides of the layout region, or above and below, depending on the current size and aspect ratio of the layout view.

You can edit the layout through this window by moving nodes, adding and deleting nodes, and establishing node connections. Most of the layout editing can only be done through this view, and this is usually where you would begin to set up a new hydraulic system in the app. You will need a mouse, trackpad, or other pointing device to manipulate the data in the System Layout View because some of the editing features are unavailable through the keyboard. A reach node always defines the downstream end of a reach in the system layout, whereby a confluence node is a special type of reach that ends somewhere along the length of another reach. The upstream end of a reach may be another reach node, a source node, or a bifurcation node.

A *branch* is defined as one or more reaches connected in series, one after the other. Every system layout has at least one branch. Branches begin at source (primary and secondary) nodes, bifurcation nodes, and at the downstream end of split reaches (defined below). Branches end at reach terminus points (which have no downstream connection), confluence nodes, and split reaches (Chapter II.4).

A reach with no downstream connection is also referred to as a *terminus node* in the system layout, such as for some types of scheduling graphs (Chapter VII).

You can **control**-click or right-click over the System Layout View to open a popup menu with many different display options and commands.

II.1. Layout Node Types

There are five different types of nodes that can be used in the system layout. All system layouts will have one primary source node and at least one reach node. Branching layouts will have one or more bifurcation and or confluence nodes. Each bifurcation node will have one or more downstream reach nodes. Each confluence node will be associated with one or more secondary source nodes, or an upstream bifurcation in the case of a looping branch. There will always be one and only one primary source node, and there may also be zero or more secondary source nodes. All layout nodes are connected to one or more other nodes in the layout.

Node types are distinguished by the symbols that represent them on the layout view. You can edit these symbols in the **Preferences** sheet, as described in Chapter I.4.2.



One or more regulator structures will be found at every layout node, regardless of the node type. See Chapters IV and V for more information about regulators.

II.1.1. Source Nodes

For principal water sources, there are two sub-types: primary and secondary. As already indicated, there will always be one primary source node in the system layout, but there can be multiple secondary source nodes. Secondary source nodes can be deleted from the layout, but the primary source node cannot. Both types of source node indicate main entry points of water into the system layout, possibly representing a diversion dam, storage dam, groundwater well (or well field), lake, or other source of water. Water can also enter the system layout at locations downstream of source nodes, such as through *turnouts* (or "turn-ins," in this case), or from net seepage gain along a reach. Finally, water can enter as negative evaporation (direct precipitation) or through reverse flow at a terminus point.

II.1.2. Reach Nodes

In the app, a canal reach is a length of open channel (such as a canal), closed conduit (pressurized pipeline), or reservoir. A reach node in the system layout indicates the downstream end of the reach. Reaches can be connected in series along a branch, from upstream to downstream, or in parallel at bifurcation nodes.

The downstream end of a reach can also be *split*, meaning that two or more reaches are connected to the downstream end of the reach, at the same location. A split with two downstream reaches is very much like a bifurcation node (see below) which is located at the downstream end of the reach, with the exception that each of the two (or more) downstream reaches is the beginning of a new branch in the layout. That is, one way in which a branch ends is at a split reach node, as described above.

II.1.3. Bifurcation Nodes

Bifurcation nodes represent branches in a system layout where the flow of water can be divided, meaning that some of the water might continue along the reach and the rest of it enters the upstream end of another reach. Thus, you use bifurcation nodes to permit the simulation of water flowing out of one reach (at any location along the reach) and into another reach. A bifurcation node always demarcates the beginning of a new branch in the layout, and every branch in the layout has one or more reaches. Typically, lateral canals would begin a bifurcation nodes.

II.1.4. Confluence Nodes

A confluence node is a special kind of reach node whereby the downstream end of the reach connects to another reach at an intermediate point, such that any flow at this location enters another reach in the system layout. The branch containing the confluence node ends at that node. A confluence node can be positioned at any location along a reach. The upstream end of a branch ending in a confluence node is always either a bifurcation (defining a looping branch), a secondary source node, or a split reach.

II.1.5. Terminus Nodes

Terminus nodes are also a special kind of reach node and demarcate the downstream end of a branch – they have no downstream connection to another node. However, unlike confluence nodes, the flow (if

any) at a terminus node leaves the simulated system (instead of discharging back into another reach of the layout). Every system layout has at least one terminus node.

II.2. Layout Editing Modes

Success in editing the layout depends on knowing which node types are available and how to apply the various editing features. Familiarity with layout editing may require some trial-and-error at first, until you are comfortable with the features.

The cursor over the layout window will change to reflect the current editing mode, so you can always see at a glance which mode is in effect. In each of the editing modes described below, the corresponding cursor for the layout window is shown at the left.

II.2.1. Selecting and Editing Node Information



follows:

Click on **Select** from the popup menu, or type **S**. This is the default layout editing mode, whereby the cursor is a hand with a pointing index finger. If you double-click on a node, a sheet will open (Chapter II.6) to allow certain kinds of data editing for that node. You can also open other data editing sheets by double-clicking on a node in the layout view, as

Data Editing Sheet	Double-click with
Reach Configuration	Command
Structure Configuration	Shift
Flows and Depths	Shift-Command
Structure Settings	Ctrl-Command

In each of the above cases, the highlighted node will be selected when the data entry sheet is opened. These data-entry sheets are discussed in Chapters III, IV, and V. Flows, water depths and structure settings can be modified before and during simulations, and any of them can be defined through scheduling graphs, as explained in Chapter VII. Reach and structure configurations cannot be changed during a simulation.

II.2.2. Moving Nodes

With this mode, the cursor becomes a small cross-hair shape, and you can move individual nodes around the layout window or move all of the system layout nodes as a group. Select **Move** from the popup menu, or type **M**, position the cursor over a node, click the left mouse button, and drag the node to a new location. Finally, release the left mouse button to complete

the move. While dragging, you will see the layout dynamically change to accommodate the move.

You can then modify the reach curves as desired by moving the layout control points, as described below in Chapter II.5.

If you click the right mouse button before releasing the left mouse button during a move, the move is canceled. Also, if you press the **Esc** key during a move, the node move will be canceled.

Press and hold the **shift** key before clicking on any layout node to move the entire layout as a whole, while respecting the relative node positions. Thus, after such a move, the layout looks the same, but is in a different location in the layout window. This action can be useful to create more room for adding

new nodes to the layout.

You can directly move a bifurcation or confluence node by dragging it along the corresponding reach curve in the layout view. These two node types can also be moved by specifying a distance along the parent reach in the All Nodes tab of the Reach Configuration sheet.

Note that you can move nodes outside of the layout region, but not outside the confines of the layout view. Also, be aware that moving layout nodes changes the appearance of the system layout in this view, but it does not have any effect on the hydraulic calculations that take place during a simulation. That is because the layout view is a only a schematic of the system.

II.2.3. Inserting a Reach



Insert a reach between two existing layout nodes by selecting **Insert reach** from the popup menu, or by typing **R**. Click over an existing node to specify where the insertion will take place, drag the mouse to an adjacent node, and release the mouse button. The selected node will temporarily display a red circle around it. When you drag to an adjacent node to which

the operation can be completed, a green circle will appear around it — if there is no green circle, it means that you cannot insert a reach between those two nodes. When successful, a new reach node will be placed midway between the two selected nodes.



II.2.4. Appending a Reach



You can append a reach to the end of a reach or source (primary or secondary) by selecting **Append reach** from the popup menu, or by typing the letter **A**. Click over an existing layout node and drag the cursor away from the existing node, and then release the mouse button at the location where you want to place the downstream end of the new reach. A new reach is appended to the layout.

Click over an existing reach node to add a new reach on the downstream side. If the existing reach node is a branch terminus node, the new reach will be added in series. But if the existing reach is not a terminus node, the new reach will be split from the existing node (provided the maximum number of splits at the node is not exceeded). If you click over a source node (primary or secondary), the source will be split (if it is not already), and a new branch will begin from the source node.

II.2.5. Inserting a Bifurcation



Select **Insert bifurcation** from the popup menu, or type the letter **B**, to select this editing mode. Whenever you insert a bifurcation node, you will also insert a new reach node (and a new reach). The upstream end of the reach is at the bifurcation node, and the downstream end is at the reach node itself. To insert a bifurcation, click over an existing reach curve on

the layout to set the bifurcation position, and then drag away from the curve. When you release the mouse button, after having dragged some distance away from the reach curve, you will set the location of a new reach node.

After completing the operation, you can switch to the *Move* layout mode and change the position of the two new nodes, but the bifurcation node must necessarily be somewhere along the same reach that you initially clicked on. Of course, you can also change the positions of the upstream and downstream Bezier control points for the new reach, after the insertion.

If you click on a reach curve and release the mouse button without having dragged away at least a short distance, the operation will be canceled. You can also cancel by pressing the **Esc** key before releasing the mouse button.

II.2.6. Inserting a Confluence



Select **Insert confluence**, or type the letter \mathbf{C} , to choose this layout editing mode. Click over an existing node and release the mouse button, then move the cursor between the selected node and an adjacent node. During this movement, the cursor will follow the reach curve between the two nodes. Click and release again to set the location of the confluence node, then

stretch the cursor away and click a final time to set the location of a new upstream secondary source node.

A confluence node is actually a special type of reach node because it demarcates the downstream end of a reach. Whenever you insert a confluence node, you will also insert a secondary source node (and a new reach). This is the only way to insert a secondary source node into the layout.

II.2.6. Deleting a Node



You can remove nodes from the layout by selecting **Delete** from the popup menu, or by typing the letter **D**. Depending on your computer's operating system version, this cursor may appear as an arrow with an X. Click on the node to be deleted – if the deletion is allowed, you will see a red X drawn over the node. Then, if you click once more over the same node, it will

be deleted. Depending on the layout configuration, one other node might also be deleted along with it. For example, if you delete a bifurcation node, the downstream reach will also be deleted, and vice versa.

It is not possible to delete more than two nodes with a single click. That means that if a branch beginning at a bifurcation has multiple reaches, or if it has other downstream bifurcations, the deletion will not be allowed. Instead, you would have to delete the downstream reaches, one-by-one, until only one reach remained downstream of the bifurcation, which can then be deleted. In general, any operation that



would delete more than two layout nodes is not allowed, so you cannot delete a split node until you first delete enough of the downstream nodes, such that it is no longer split.

You can cancel the node deletion by clicking on the layout window anywhere except over the selected node, by pressing the **Esc** key, or by changing the layout editing mode.

The information (i.e. reach length, calibrations, and so on) associated with layout nodes will be lost after a node deletion. If you erase a non-terminus reach node, the remaining reach connections will be automatically "repaired" so that no reaches will be orphaned in the layout after the operation.

You cannot remove the primary source node, nor can you remove a reach node if it is the only reach node in the layout, or if it will remove all reach nodes from the layout. The minimum system layout (and the default layout) has a primary source node and a reach node.

II.3. Looping Branches

A looping branch begins at a bifurcation node and ends at a confluence node, thus looping back on some part of the simulated system, rather than ending at a branch terminus or a split reach. A looping branch may contain one or more reaches.

To create a looping branch, you must insert a bifurcation node into the system layout, then insert a confluence node. When you do this, a reach node will also be inserted at the downstream end of the reach which begins at the bifurcation node. A secondary source node will be inserted at the upstream end of the reach that ends at the confluence node. The following figure is an example of a layout with two reaches that are ready to be combined into a looping branch.



Finally, select **Move** from the layout popup menu (or type \mathbf{M}), then click on the reach node of the reach that begins at the bifurcation, and drag it over the top of the secondary source node at the upstream end of the confluence reach (or vice versa), and release the mouse button. If the reach node is over the secondary source node (or vice versa) in the layout view, you will be prompted to create a looping

branch. Click on the **Yes** button (next figure) to create the loop.

Note that, alternatively, you can also move the secondary source node over the reach node and release the left mouse button to create the same looping branch.

If you click on the **Yes** button, the recently-created secondary source node is eliminated, the terminus reach node of the downstream end of the branch that begins at the newly-created bifurcation node is eliminated, and the two branches are combined into a single branch. The single branch begins at the bifurcation node and ends at the confluence node, making a looping branch.



If you want to have more than one reach in the looping branch, append one or more reaches to the reach that starts at the bifurcation, and or insert one or more reaches into the reach connected to the secondary source node. Then, move the secondary source node over the new branch terminus, or vice versa, to create a looping branch with multiple reaches (see the figure below).

The number of reaches in the looping branch will be equal to the sum of the reaches in the branch that began at a bifurcation node, plus the number of reaches in the branch that began at the secondary source node, minus one. This is because one of the reach nodes (the terminus of the branch beginning at the bifurcation node) is eliminated from the layout. Thus, if you started with the arrangement shown in the example above, the loop would contain 1 + 1 - 1 = 1 reach, where *Another Reach* and *Secondary Source* have been eliminated from the layout as shown.

You cannot specify the reach drop (Chapter III.1.3) at the confluence node when the node is the downstream end of a looping branch. Instead, the model determines the drop value automatically at such a location, based on other drops and reach longitudinal bed slopes in the system layout.



II.4. Split Reaches

A *split* reach is a reach that has multiple connections to other reaches at its downstream end. It is like a bifurcation, except: (1) it could actually be a *trifurcation*; and, (2) it is always located at the downstream end of a reach (never at a midpoint, as opposed to a bifurcation node). Also, as noted above, a split reach signals the end of a branch, so that none of the downstream-connected reaches are considered a direct continuation of the upstream branch. Nevertheless, you could use one or more bifurcation nodes at the downstream end of a reach to effectively simulate a split reach, even though such an approach would perhaps be less elegant and less intuitive.

The downstream (DS) drop value at a split reach is ignored by mCanal – instead, the upstream (US) drop values for each of the downstream reaches (which are directly connected to the split reach) are used to determine elevation changes from the split reach to the upstream end of the others. This allows for the possibility of different bed invert elevation drops from the downstream end of the split reach to the upstream end of each connected reach.

The following figure shows *Main canal* as a split reach, with *Reach #2*, *Reach #3*, and *Reach #4* connected to the downstream end of Main canal. If you were to delete *Reach #3* and *Reach #4* from the system layout as shown in the figure, then *Main canal* would no longer be identified as a split reach.

Primary and secondary source nodes can also be split, meaning that the upstream ends of two or more reaches can coincide with a source node. However, a split source node is where two or more branches begin, and not where any branch ends. That is because no other nodes can exist upstream of a source node.



II.5. Layout Control Points

Control points are found at the upstream and downstream ends of each reach in the layout, and they can be used to change the shape of the reach curves, which are presented as Bezier curves. You can toggle the display of control points in the layout by typing \Re -C, or by right-clicking over the layout window and selecting Show Bezier points from the popup menu. Control points are connected to reach end points by straight dashed lines in gray color, as shown in the figure below, which is the same as the above figure after having typed \Re -C. When the layout editing mode is *Move*, you can click over a Bezier control point and drag it to a new location, just as for a layout node. You can also cancel a move by right-clicking, or pressing the Esc key, before releasing the left mouse button.

Moving control points changes the appearance of the system layout but does not affect the hydraulic calculations that take place during a simulation.

Notice that the shape of the reach curve changes as you drag a control point to a new location. If both control points fall on the (imaginary) line segment connecting the two endpoints, the reach curve will appear as a straight line.

When reaches are connected in series along a branch, the control points are collinear by default. In this case, collinear control points provide a continuous curvature in the reach curves that connect at a common endpoint. The line defined by two collinear control points will pass exactly through a reach node, and the control points will be equidistant from the reach node. If you move one of the control points, the other moves with it. In the above figure, the control points are collinear at the *Reach* #3 and *Reach* #4 nodes. Control points are not necessarily collinear at split reaches (*Main canal* in the figure above): they can be moved independently on the upstream and downstream sides of a node.



If a control point coincides with any other type of layout node, it will be hidden under that layout node and cannot be selected in the normal way. In such cases, hold down the **option** key before clicking on the node, and this will select the hidden control point, which can then be repositioned in the layout.

II.6. Node Name and Attributes

By default, the name of each node is displayed next to the node in the layout. In some cases, you may want to reposition a node name so that it doesn't overlap with another feature of the layout. There are several possible positions for a node name, as shown in the figure below. Node name positions are relative to the node itself. To change the node name location, change to the *Select* mode and double-click over the node to open an editing sheet, as seen below.



In this sheet, you can change the node name, select the position relative to the node location in the layout, and select the angle at which the name is displayed. You can also change the Bezier curve color, line thickness, and arrowhead display option if you double-clicked on a reach node. To change the position of the label relative to the node, click anywhere in one of the eight sectors shown at the right side of the window – the node name will move to reflect your choice. Changing the angle will also dynamically update in this sheet. The layout view will be updated to show the changes if you click on the \mathbf{OK} button.

The position of the node names will also apply to other node labeling options, such as flow rate, upstream and downstream water depths, elevations, and so on.

II.7. Undoing Layout Changes

Many system-layout editing changes can be canceled by pressing the **Esc** key before completing an operation. You can also undo a layout editing change by selecting **Edit**, **Undo** from the main menu, or by typing \Re -Z. The **Undo** menu command is disabled until an editing change is made, but multiple layout edits can be redone by repeatedly undoing, in order from the most recent to the initial one.

The undo feature only works for the most recent layout editing change, and only for the current editing session. You cannot undo multiple editing changes, unless you save the data file with a different name, whereby you can always go back to the original file with its layout information.

After an undo operation in the layout window, you can reverse it if desired, by selecting **Edit**, **Redo** from the main menu, or by typing **shift-#-Z**. Undo and redo only apply to the system layout window.

II.8. Zooming, Scrolling, and Rotating the Layout

None of the following three operations will affect the results of a hydraulic simulation – they are for display purposes, and to facilitate layout node-editing functions.

II.8.1. Zooming

When the system layout window is visible you can zoom in on features of the layout and scroll the window contents vertically and horizontally. You can zoom in on the layout by repeatedly typing \mathbb{H}^+ (command and plus), up to a maximum zoom value of 1,000%. Or, zoom out by repeatedly pressing the \mathbb{H}^- (command and minus) key combination, down to a minimum zoom of 10%. The keystrokes make zoom changes in increments of 10% each time they are pressed. You can also use the popup menu to zoom to 100%, to center the nodes in the view, or to center the layout view. Note that centering the nodes from the popup menu can have slightly different results depending on whether the Bezier control points are displayed.

II.8.2. Scrolling

You may want to scroll the contents of the layout window, especially at high-zoom values where the entire layout is not visible. Scrolling is accomplished by pressing any of the four arrow keys.

You can scroll left and right, up and down. Scrolling is limited to where the edges of the layout region advance to midway across the layout window, vertically and horizontally. You can reset scrolling by selecting the **Center view** command from the popup menu.

II.8.1. Rotating

The entire system layout display can be rotated, in increments of 10 degrees, by typing \mathbb{H} -**R** or \mathbb{H} -**L**. You can also select the respective popup menu commands. Rotations can be undone by typing \mathbb{H} -**Z**. As with moving layout nodes, rotations do not affect hydraulic simulations – they are only for display purposes so that the system layout appears as it would in an actual plan view.

III. Reach Configuration



After entering the system layout, the next data entry step is usually in the *Reach Data* sheet. Basic reach configuration data include sizing information about each reach, cross-sectional shape, and node locations on the system layout view. In this window, you can also set the elevation of the primary source node, from which all other elevations are calculated, and even switch the primary source node to a secondary source, if necessary.

When you hold down the \Re key and double-click on a given node in the system layout, the *Reach Data* sheet is displayed. If you clicked over a reach or confluence node, then the data for that node will be shown on the sheet. You can also open this sheet by selecting **Edit**, **Reach Data** from the main menu, or by clicking on the toolbar icon as shown above. As with other tabbed sheets in the app, it will open to the tab that was displayed when the sheet was last opened.

The *Reach Data* sheet has four tabs: *Reaches, Calibration, Cross Sections, and All Nodes.* Thus, in the last tab you have access to certain information about all layout nodes, not only the reach nodes. The figure below shows the **Reaches** tab of this sheet. The schematic side-view diagram shows relative dimensions according to the specified parameter values. This diagram updates automatically whenever you change a parameter, or when you select a different reach. The figure below is an example of a reach with a compound bed slope.



Select a reach name from the drop-down list at the upper left of any of the first three tabs, under *Reaches*. You can also select the next upstream reach (if any) in a branch by clicking in the left side of the schematic side-view diagram. Similarly, click in the right side to select the next downstream reach. The upstream and downstream gates are shown as black vertical lines, regardless of the actual regulator

types, in the diagram (this changes with *Dark Mode* in macOS 10.14 or later).

At the bottom edge of the schematic diagram you may see one or more small triangles: green triangles correspond to relative turnout locations, inverted blue triangles show confluence-node locations, and magenta triangles correspond to bifurcation nodes. The figure below shows examples of each of these symbols in the reach schematic diagram.



III.1. Reaches

This is the first of the four tabs and includes the reach type, as well as parameters for lengths, longitudinal bed slopes, and bed elevation changes at the upstream and downstream ends. The upper left area of the tab shows the selected reach name in a drop-down list. Note that if you change the reach selection here, it will also change for the other three tabs, and the data values will also change to correspond to the selected reach.

III.1.1. Reach Type

You can choose from three reach types (pipeline reaches are not currently enabled in the app). The default is "Open channel" in which the canal banks are parallel to the bed of the channel in the longitudinal (stream-wise) direction. The second type, "Level-top canal," is for reaches with a longitudinally-sloping bed, but level banks. This type of reach has an increasing depth in toward the downstream end if the bed slopes downhill in that direction (the usual case) — it is found in systems with some types of gate automation where the control point is at the upstream end of the reach and the flow rate can go to zero. That is, when the water surface is horizontal, there is no hydraulic gradient, and no flow. Of course, an "Open channel" reach with a zero bed slope is a special case of a level-top canal because the banks would also be level.

A "Reservoir" reach is handled through simple volume-balance calculations during a simulation, rather than the normal unsteady-flow calculations. This is appropriate when the reach is very short in length and the unsteady-flow equations would be difficult to solve. The reservoir reach type may be applied, for example, when there is a short section of reach between a bifurcation and a downstream flow measurement structure, like a broad-crested weir or flume. But perhaps more commonly, a reservoir reach really is an open reservoir, either in-line or off-line, possibly quite large in size, as described further in Chapter III.2.

III.1.2. Bed Elevations

The elevation values are for the bed of the selected reach, one for the upstream (US) end and the other for the downstream (DS) end. Usually, the DS elevation will be lower than the US elevation in a given reach. Elevation values are based on the elevation that you specify for a selected layout node,

which is explained below for the *All Nodes* tab of this sheet. In the *Reaches* tab, they are displayed for convenience but cannot be changed here. When you specify the canal-bed elevation at any given location, the app calculations the bed elevations for all other locations in the system layout.

III.1.3. Reach Drops

The upstream and downstream drop values correspond to abrupt changes in bed elevation (the lowest point in a channel cross section) from one reach to another, wherever reaches are connected in series. They are called "drops" because they are positive when the elevation decreases from the downstream end of a reach to the upstream end of the next reach in a branch, or negative when the elevation increases.

Positive or zero drop values are the most typical case. When two reaches are connected in series, and if the connection is not a split reach (Chapter II.4), the DS (downstream) drop of the upstream reach must be exactly the same as the US drop of the downstream reach. This mirroring across reach boundaries is handled automatically in the app.

At a split reach location, each of the downstream reaches can start at a different bed elevation, so you can specify individual US drop values for them. The DS drop value is ignored by the app at a split reach because at these locations the drop is only specified by the upstream value in each of the connecting downstream reaches. Thus, there is no mirroring of drop values at (the downstream end of) a split reach.

As mentioned above, reach drops are usually positive (or zero). Sometimes there are small negative drops from one reach to the next, but it is not common in canals because it would typically correspond to design and or construction mistakes. The following figure shows a reach with positive bed-elevation drops (ΔE_u and ΔE_d) at both the upstream and downstream ends.



However, it can make sense to have a rather large negative drop (increase in bed elevation) at the downstream end of a reach when the regulator structure(s) at that location are one or more pumps. The figure below shows an example of a reach with a positive upstream drop and a negative downstream "drop." A pump would be needed at the downstream end of a reach with such a large increase in bed elevation, otherwise the reach would overflow before water could spill by gravity into the next downstream reach.



III.1.4. Slopes and Lengths

You can partition a reach into one, two or three different lengths, each with its own longitudinal bed slope. Length means the horizontal distance along the reach, and when there is more than one length, they are added to arrive at the total reach length. A positive bed slope indicates that the bed elevation decreases in the downstream direction, which is the normal situation for canals. A negative slope value means and inverse slope — that is, the bed goes uphill in the stream-wise direction.

Pairs of slope and length values are activated when the corresponding box to the left is checked. You can check one of the boxes, any two boxes, or all three. When you check two or three boxes, the reach will have a compound slope with separate length segments unless all activated (checked) slope values are the same. If you uncheck all three boxes, the reach length will be zero and you will see an error message when the data are checked (Chapter IX.1). By default, only the top box is checked. Unchecking a box does not change the length or slope values that you entered, it only de-activates those values so that they are not applied in a simulation.



Avoid specifying muliple (2 or 3) reach length segments in a reach where one or more of the segments are only a small fraction of the total reach length. If a reach segment is less than 20% of the total reach length, a warming message will be issued for the reach. If any reach segment is less than 10% of the total reach length, and error message will result. This means that you should not attempt to use mCanal to simulate relatively short reach segments, such

as transitions at the US or DS end of a reach. Moreover, warnings or errors will be generated by the app if you specify multiple length segments with substantially different longitudinal bed slopes within a given reach.

Pipeline reaches do not use bed-slope values, so the corresponding edit boxes are disabled for this reach type. Instead, pipeline reaches rely on the specification of upstream and downstream drops to determine the elevation change from inlet to outlet.

III.2. Reservoir Reaches

Reaches that you specify as "reservoirs" are handled using simple volume balance calculations during a simulation, rather than applying the Saint-Venant equations, as is normally done in a reach. A reservoir reach is considered to be rectangular in shape, with specified length and width of the base (or floor). The app assumes a level bed for reservoir reaches, and the same linear inverse side slope (zero means vertical sides) on all four sides. The tops of the reservoir banks are also considered to be level.


Thus, a reservoir reach is like a level "tray" of water in which the volume of water in the reservoir depends on the depth, and vice versa, and on the dimensions of the reservoir.

Seepage and evaporation loss calculations are performed on reservoir reaches in a manner similar to that used for non-reservoir reaches. Seepage calculations are based on the wetted area of the reservoir, and evaporation loss calculations are based on the surface area of the reservoir.

A reservoir reach can be configured as "in-line" or "off-line." The distinction depends on how you set up the system layout. An "in-line" reservoir reach is connected in series along a branch, and an "off-line" reservoir is part (or all) of a looping branch. But in the system layout view, reservoir reaches look the same as all other reaches; it is whether or not you selected the *Reservoir reach* option that distinguishes between reservoirs and non-reservoir reaches.

You can specify the initial volume of water in a reservoir reach if regulators are "undefined," closed underflow gates, weirs with crest above upstream and downstream water levels, or pumps that are shut off. In other words, zero flow into and out of the reservoir. Otherwise, the initial volume (and initial water depth) must be calculated according to zero flow through the reservoir (based on adjacent non-reservoir reach depths). Also, keep in mind that if the reservoir reach capacity is "too small," it could overflow or underflow (go dry) within a single simulation time step.



The data check will produce a warning if the maximum reservoir volume (storage capacity) is less than that of the upstream and or downstream reaches. An error message will be issued during a simulation if a reservoir reach overflows.

The following reservoir geometric equations include the simplifying assumption of a level water surface, which is not always strictly true. These equations are used internally by the app and are given herein for convenience and completeness.

III.2.1. Reservoir Volume and Depth

If the inverse side slopes are less than 0.001 (m < 0.001, meaning vertical reservoir walls), the volume of water in the reservoir at any time is:

$$\Psi = A_b h \tag{III.1}$$

$$\Psi = \frac{A_b}{3} \left[\frac{\left(H+h\right)^3}{H^2} - H \right]$$
(III.2)

where,

$$H = \frac{\sqrt{A_b}}{2m} \tag{III.3}$$

and m is the inverse side slope of the reservoir tray (m = 0 corresponds to vertical walls). Finally, given the volume of water in the reservoir, the depth is calculated as:

$$h = \left[H^2 \left(\frac{3\Psi}{A_b} + H\right)\right]^{1/3} - H \tag{III.4}$$

III.2.2. Reservoir Surface Area

The surface area is used for determining losses due to evaporation. The water-surface area of the reservoir, A_s , is calculated as:

For m < 0.001:

$$A_s = A_b \tag{III.5}$$

For $m \ge 0.001$:

$$A_s = A_b \left(\frac{H+h}{H}\right)^2 \tag{III.6}$$

III.2.3. Reservoir Wetted Area

The wetted area of a reservoir is the total area of the submerged surfaces, including the reservoir floor and side slopes. This area is used for determining seepage loss rates during a simulation.

$$A_w = A_b + 2h(b_1 + b_2 + 2hm)\sqrt{m^2 + 1}$$
(III.7)

where A_w is the wetted area $(m^2 \text{ or } ft^2)$; and, all other values are as previously defined. It is noted that for zero water depth (h = 0), $A_w = A_b$.

III.3. Reach Calibrations

The **Calibration** tab is where you specify hydraulic-loss, seepage and evaporation parameters for reaches (open channels and pipelines). These parameters will generally have a significant impact on the hydraulic behavior during a simulation, and this is especially true of the hydraulic roughness value for an open-channel reach. There will always be *distributed* (see below) hydraulic losses, but local (or "minor") losses are irrelevant for many regulator structure types, in which the structure calibration already accounts for the local hydraulic losses.

Note that these so-called "losses" are actually energy conversions that result in a very slight temperature increase of the water (increased internal energy) and the noises produced by flowing water. These energy conversions can be explained by the first law of thermodynamics, in which energy is conserved. That is why they are referred to as hydraulic losses, rather than energy losses.

	Reaches	Calibra	ation	Cross Sections	All No	odes		
Reaches Split Reach		3 of 6	Seep	oage (mm/day)	5.40	Local	loss coeff	icients
Hydraulic roughness			Evap	oration (mm/day)	5.40	Opstr	ean	0.000
Use Chézy equ	ation quation		Тур	ical values for:	3.20	Dowr	nstream	0.000
Open-channel distribut	ted losses			Canals Pipes				
Manning n (roughne	78.500 ess) 0.01350		Can Con Bare	Canal material Concrete Bare rock		nning to 0.0	n)20)50	Chézy C 85 to 70 70 to 60
Pipe distributed losses Pipe roughness heig Water temperature	Pipe distributed losses Pipe roughness height (mm) 0.100 Water temperature (deg C)		Clay lining Masonry & brick Earthen Gravel & cobble Heavily vegetated			0 to 0.018 1 to 0.035 5 to 0.035 5 to 0.050 0 to 0.150		85 to 70 80 to 50 75 to 50 65 to 60 60 to 20
Reach hydraulic calibrati	10.0 on parameters		Actu shap	al values depend on t be, and channel size, a	the Reync and not o	olds nur nly on s	mber, cha surface ro	nnel oughness.
						(DK	Cancel

III.3.1. Distributed Hydraulic Losses

Hydraulic losses include distributed and local losses. You specify the distributed head loss in an openchannel canal reach, or a closed pipeline. The term "distributed" is used to distinguish from local head losses at structures and other boundaries in the system.

For open-channel reaches, you may choose either the Chézy or Manning equations, both of which are used to define uniform flow conditions in open channels. But these equations have traditionally been used to also define the hydraulic energy-loss gradient in non-uniform and unsteady open-channel flow, especially for one-dimensional approximations. Chézy equation:

$$Q = CA\sqrt{RS_o} \tag{III.8}$$

where Q is flow rate $(m^3/s \text{ or } cfs)$; C is the Chézy coefficient; A is cross-sectional area $(m^2 \text{ or } ft^2)$; R is hydraulic radius (m), equal to A/W_p , where W_p is wetted perimeter (m); and, S_o is the longitudinal bed slope.

Manning equation:

$$Q = \frac{C_m}{n} \frac{A^{5/3}}{W_p^{2/3}} \sqrt{S_o}$$
(III.9)

where n is the Manning roughness value; and, all other parameters are as defined above for the Chézy equation. For English units, with Q in cfs and linear dimensions in ft, the constant C_m is taken to be 1.49. For metric units, with Q in m^3/s and linear dimensions in m, C_m is 1.0.

When either the Chézy or Manning equations are applied in a simulation, the value of S_f is used instead of S_o , and it is calculated to provide an estimate of the distributed hydraulic-loss gradient along a reach. When S_o is used in those equations, it implies that steady, uniform flow is occurring, which can happen in a simulation, but *mCanal* never assumes the occurrence of this condition.

For the Chézy equation, you enter a Chézy C value, which is effectively "smoothness," because the higher the parameter value, the lower the losses. For the Manning equation, you enter a Manning n value, which is a "roughness," traditionally based on only the surface roughness of the channel sides and bed. Theoretically, the Chézy equation has important advantages over the Manning equation in that it not only accounts for surface roughness, but also factors in the Reynolds number. However, mCanal does not provide calibration equations (based on relative roughness or Reynolds number) for either Chézy C or Manning n — you must enter these values yourself.

These are some typically-used values of the Manning n for open channels (compiled from Chow 1959, Brater and King 1976, and French 1985):

Material	Min	Avg	Max
Asphalt	0.011	0.013	0.020
Brass & Glass	0.009	0.011	0.013
Cast Iron	0.010	0.013	0.016
Clay Lining	0.011	0.014	0.018
Concrete: Rough, Broken	0.015	0.017	0.020
Concrete: Smooth, New	0.010	0.012	0.014
Corrugated Metal	0.017	0.022	0.030
Earthen: Clean, Straight	0.016	0.022	0.030
Masonry and Bricks	0.011	0.014	0.017
Natural: Clean, Straight, Full	0.025	0.030	0.033
Natural: Some Weeds & Rocks	0.035	0.042	0.050
Natural: Many Weeds, Winding	0.075	0.113	0.150
Rock-Cut Channels	0.025	0.038	0.050
Rubble Masonry	0.018	0.025	0.030
Steel	0.010	0.014	0.017
Vitrified Clay	0.011	0.015	0.017
Wood	0.010	0.015	0.020
Wrought Iron	0.012	0.015	0.017

The next table provides some typical values of roughness height, ϵ , for different pipe and open-channel lining materials (Brater and King 1976, and Uni-Bell Plastic Pipe Association 1977):

Material & Condition	ϵ (mm)
Plastic (PVC, PE, others)	0.0015
Steel (enamel-coated)	0.005
Steel (welded, clean)	0.3 to 2.0
Steel (incrustated or tuberculated)	2.0 to 6.0
Very smooth, seamless concrete	0.2 to 0.3
Smooth concrete with joints between panels	0.4 to 0.6
Gunite with a smooth finish	0.5 to 1.5
Rough concrete surfaces	1.0 to 2.0
Very rough concrete surfaces	4.0 to 5.0
Untreated gunite	3.0 to 10.0

On this tab in the app, you can see some of the typical values for Chézy C, Manning n and pipe roughness height in the tables at the lower right area of the sheet.

III.3.2. Pipeline Reaches

These are closed conduits, assumed to be circular in cross section. The distributed head loss is calculated using the Darcy-Weisbach equation, and you must enter two values: (1) epsilon, ϵ ; and, (2) water temperature. Epsilon, ϵ , is the roughness height (mm or inches) value of the inside of the pipe and temperature is used to determine the kinematic viscosity of water (m^2/s) . The roughness height depends on the type of conduit or pipe material, and on the condition of the material, which is often dependent on the age of the pipeline, the water quality, and other factors.

For circular pipes flowing full, the Darcy-Weisbach equation is:

$$h_f = f \frac{L}{D} \frac{V^2}{2g} = \frac{8fLQ^2}{\pi^2 g D^5}$$
(III.10)

in which h_f is the so-called "friction loss" (m or ft); f is a friction factor, dependent on ϵ , D, and viscosity; L is the pipe length (m or ft); D is the inside diameter of the pipe (m or ft); V is the average velocity of flow along the pipe (m/s or ft/s), equal to Q/A; and, g is the ratio of weight to mass (9.81 m/s^2). For non-circular pipes, the expression 4R can be substituted for D, where R is the hydraulic radius:

$$R = \frac{A}{W_p} \tag{III.11}$$

where A is cross-sectional area $(m^2 \text{ or } ft^2)$; and, W_p is wetted perimeter. For circular cross sections, $R \equiv D/4$. The parameter f in the Darcy-Weisbach equation is defined in the app by the Swamee-Jain equation (Robertson, et al. 1998), which is for turbulent flow (it is noted that laminar flow is rarely encountered in pipeline reaches):

$$f = \frac{1}{4} \left[log_{10} \left(\frac{\epsilon}{3.75D} + \frac{5.74}{R_e^{0.9}} \right) \right]^{-2}$$
(III.12)

in which R_e is the Reynolds number:

$$R_e = \frac{VD}{\nu} \approx \frac{4Q(55, 120 + 2, 071T + 8.392T^2)}{\pi D}$$
(III.13)

where V = Q/A; ν is the kinematic viscosity of water; and, T is the water temperature (celsius). Kinematic viscosity is calculated based on the specified water temperature for the pipeline reach, and so Reynolds number can be closely approximated as a function of T. As stated above, 4R can be substituted for D in non-circular cross sections.

The app assumes that there is only one pipe diameter; if there are two or more diameters, you must figure out an equivalent length of pipe with a single diameter.

III.3.3. Local Hydraulic Losses

In the app, these are shown as **Local loss coefficients**, upstream and downstream, at the upper right area of the tab. Local hydraulic losses may occur at the upstream and downstream boundaries of a reach. The default local loss coefficient is zero for both ends of a reach (K_{ru} at the upstream end, and K_{rd} at the downstream end). If you specify a nonzero value, it will be multiplied by the velocity head at the corresponding reach end to determine an estimated hydraulic loss value.

The velocity head is calculated as:

velocity head
$$=$$
 $\frac{V^2}{2g} = \frac{Q^2}{2gA^2}$ (III.14)

Nonzero local loss coefficients for reaches are appropriate, for example, at the entrance to an inverted siphon or other cross-sectional converging or diverging transition. Divergence losses at the downstream end of a structure or boundary condition are usually much higher than those at the upstream (converging) end; thus, in general: $K_{ru} < K_{rd}$.

Local hydraulic losses can also be determined as an equivalent length of pipe, L_e , added to the physical pipe length for calculation purposes:

$$L_e = \frac{K_r D}{f} \tag{III.15}$$

where K_r is taken to be either K_{ru} or K_{rd} , as the case may be.

III.3.4. Seepage

When you start with the default system layout (selecting **File**, **New** from the main menu) the seepage value will be zero. You can enter a nonzero value to include seepage in a hydraulic simulation, where the value is always specified in mm/day, regardless of the system of units chosen in the Preferences sheet. During a simulation, the app will multiply the average wetted perimeter in a reach by the total reach length and multiply this product by the specified seepage rate in mm/day, to determine the volumetric flow-rate seepage value at any given time step. After units conversions, this will produce a value in m^3/s or cfs, assumed to be distributed equally along the reach length. This computational procedure helps

take into account the variability on seepage with different wetted areas (product of wetted perimeter and reach length), corresponding to different water depths, as often observed in the field. However, the volumetric seepage flow rate is modeled in the app as a linear function of wetted area, which may not reflect the true variability in seepage flow rate in some canals.

A positive seepage value will correspond to net seepage "loss," that is, seepage outflow from the canal reach. On the other hand, a negative seepage value will result in net seepage "gain," or inflow to the reach. In practice, a single reach may have loss in some sections and gains in others, but there will be an effective net value, positive or negative, over the entire reach. Seepage inflow to a canal occurs when the water table adjacent to the reach is above the water surface elevation, which can occur in a "cut" (excavated) section of canal, or where a canal traverses a sloping hillside and water enters through the uphill bank.

III.3.5. Evaporation

Evaporative losses from an open channel are usually very small, compared to the discharge in the channel, but nevertheless can be accounted for during a simulation to ensure that water volumes are correctly taken into account. In practice, unless you enter an unrealistically high value of evaporation, you will not usually notice a significant change in the hydraulic behavior of the reach during a simulation. An exception could be when the reach is very wide and shallow.

Like seepage rate, this value is always specified in units of mm/day and can be positive or negative. A negative evaporation value would correspond to a net inflow at the water surface, which would presumably be due to something like direct rainfall. But in most cases, evaporation is non-negative. The default value is zero, and the maximum value would not normally be more than about 15 mm/day, depending on the air temperature, relative humidity, wind speed, and other environmental factors that affect evaporative potential.

During a simulation, the evaporation value in mm/day is multiplied by the water surface area of the reach to arrive at a volumetric flow rate in m^3/s or cfs. The total surface area of the reach is calculated discretely, between *computational nodes* (see the box below) along the length of a reach, by multiplying the distance between two nodes by the average water surface width (sometimes called "top width") in that computational cell, for a given time step. Of course, the water surface width is a direct function of water depth, given a particular cross-section geometry.

Computational Nodes Chapter II presented five *layout node* types. *Computational nodes* are something entirely different, and in the app they are discrete locations in space (distance in the stream-wise direction) and elapsed simulation time. This is part of the four-point implicit solution method for the governing equations and the boundary conditions. You will not normally notice computational nodes except when viewing pool profile data in simulation results files (Chapter IX.3). Pool profiles show the computational node locations along a reach, and the dependent-variable values of discharge (flow rate) and water depth as a function of time.

Pipeline reaches cannot have evaporation losses, so you will only be able to edit this value for open-channel reaches.

III.4. Cross Sections

Each open-channel canal reach has its own cross-sectional shape and dimensions, though they can also be set to the same values for multiple reaches if desired. The cross-sectional shape determines the relationships between water depth and cross-sectional area, wetted perimeter, top width (water surface width), and depth to the area centroid. Knowledge of these relationships is an essential part of the calculations in a hydraulic simulation. Pipeline reach cross sections are assumed to be circular, so this tab is only for open-channel reaches.

You can select from four different cross-section shapes:

- 1. Circular
- 2. Trapezoidal
- 3. Coordinate section
- 4. Horseshoe

Click on the appropriate button to select the cross-sectional shape for a reach. The upper right area of the tab shows a graphical representation of the cross-sectional shape and its relative size. The four boxes (lower left) for dimensional parameters change captions depending on the selection of cross-sectional shape. Only the trapezoidal shape makes use of all four boxes.



Cross-sectional shapes can change from one reach to another, but within a reach there can only be one shape. However, you can modify dimensions at either end of a reach when the **Prismatic cross section** box is unchecked. For example, you can have a rectangular section gradually transition to a trapezoidal section from upstream to downstream along a reach. Or you can have the diameter of a circular section change linearly over the length of a reach. Specify the reach end by adjusting the **Reach location** horizontal slider to the left or right. The fully left position is at the upstream end of the reach, and the fully right position corresponds to the downstream end of the reach. Click and drag the slider to show a linear "figure blending" at various distances along the length of a reach. The figure blending is usually simple for all shapes except coordinate sections, which can undergo large transformations depending on the data you have entered at the upstream and downstream ends of the reach. The different shapes you see here for different distances along the reach are exactly how the app interprets the situation when performing a hydraulic simulation.

If you do not want a cross section to change with distance along a reach, check the **Prismatic cross** section box (the preferred method), or else manually ensure that the cross-sectional parameters are the same at the upstream and downstream ends of the reach. If you check the **Prismatic cross section** box, mCanal will use the cross-section parameters from the upstream end of the reach. In this case, whatever you enter for the upstream cross section, that is what will be applied all along the length of the reach, and the downstream section data are ignored.

The level-top reach type will override a non-prismatic cross section in terms of section depth. That means that when you specify a reach to be level-top, it does not make sense to also make that same reach non-prismatic with varying depth along the reach. In such a case, the two specifications will be conflicting, in general, and the level-top specification will take precedence. Also, the section depth in a level-top canal starts at the specified upstream depth and increases in the downstream direction (assuming the longitudinal bed slope is positive in the stream-wise direction) according to the length(s) and slope(s) along the reach.

You can use the vertical slider (as seen in the above figure) to change the water level in any cross section, whereby the depth, area, (wetted) perimeter and top-width values update accordingly. Moving the vertical slider does not change any of the configuration data – it is only to allow you to see how changes in water level affect those four variables for a given cross-sectional shape.

III.4.1. Trapezoidal Section

This includes rectangular, triangular and trapezoidal sections, which are the most common cross sections in lined canals. Also, many unlined (earthen) canal sections can be adequately approximated by a trapezoidal section without significantly altering the simulation results, especially when applying different left and right inverse side slopes to best characterize the geometry.



Note that the section is rectangular when the specified inverse side slope, m, is equal to zero on both the left and right banks. This is common for concrete flumes that cross drains or other depressions. The section is triangular when the base width, b, is zero. But if you specify a base width of zero and zero inverse side slopes, a null section with no area and zero flow capacity will result, so that an error message is generated when the data are checked.

Trapezoidal cross sections can be symmetrical or non-symmetrical, depending on whether the two (left and right, or vice versa) inverse side slopes are equal or not. The following equations are for nonsymmetrical trapezoids, but apply equally to symmetrical sections when $m_1 = m_2$.

Cross-sectional area:

$$A = h \left[b + \frac{h \left(m_1 + m_2 \right)}{2} \right] \tag{III.16}$$

Water surface width:

$$T = b + h(m_1 + m_2)$$
 (III.17)

Wetted perimeter:

$$W_p = b + h\left(\sqrt{m_1^2 + 1} + \sqrt{m_2^2 + 1}\right)$$
(III.18)

Side-slope length:

$$w = h\sqrt{m_1^2 + 1}$$
 or, $w = h\sqrt{m_2^2 + 1}$ (III.19)

Water-surface-to-centroid distance:

$$h_c = \frac{h^2}{2A} \left[b + \frac{h}{3} \left(m_1 + m_2 \right) \right]$$
(III.20)

Bed-to-centroid distance:

$$\bar{h} = \frac{h^2}{2A} \left[b + \frac{2h}{3} \left(m_1 + m_2 \right) \right]$$
(III.21)

where A is cross-sectional area $(m^2 \text{ or } ft^2)$; h is water depth (m or ft); b is base width (m or ft); m_1 and m_2 are the inverse side slopes; T is the width of the water surface (m or ft); W_p is wetted perimeter (m or ft); w is side-slope length (m or ft); h_c is the vertical distance from the water surface down to the area centroid (m or ft); and, \bar{h} is the vertical distance from the bottom of the section up to the area centroid (m or ft). Note that $h_c + \bar{h} = h$ for any cross-sectional shape.

III.4.2. Circular Section

Circular cross sections may be full or partially filled, so you need to specify the maximum depth of water in the app – that is, the depth threshold before the reach would overflow. Thus, a "circular" section may actually function in a simulation as a semicircle, as are some elevated pre-cast flumes, or it could be some other portion of a circle, open at the top. Pipeline reaches always have full circular cross-sections.

Cross-sectional area:

$$A = \frac{D^2}{8} \left(\beta - \sin\beta\right) \tag{III.22}$$

where

$$\beta = 2\cos^{-1}\left(1 - \frac{2h}{D}\right) \tag{III.23}$$



$$A = (h - r)\sqrt{2hr - h^2} + r^2 \left[\sin^{-1}\left(\frac{h - r}{r}\right) + \frac{\pi}{2}\right]$$
(III.24)

in which r = D/2.

Water surface width:

$$T = D\sin\left(\frac{\beta}{2}\right) = D\sqrt{1 - \left(1 - \frac{2h}{D}\right)^2}$$
(III.25)

Wetted perimeter:

$$W_p = \frac{\beta D}{2} \tag{III.26}$$

Water depth:

$$h = \frac{D}{2} \left(1 - \cos\frac{\beta}{2} \right) \tag{III.27}$$

Bed-to-centroid distance:

$$\bar{h} = \frac{D}{2} - \frac{2}{3A} \left(hD - h^2\right)^{3/2}$$
(III.28)

Water-surface-to-centroid distance:

$$h_c = h - \bar{h} \tag{III.29}$$

This graph shows the relationships of the above geometric equations for a circular cross section:



III.4.3. Horseshoe Section

This cross section is for the standard horseshoe as defined by the intersection of four circular arcs of radius r and r/2, as shown below.



The bold curves in the above figure indicate the shape of the standard horseshoe cross section. The top half of the cross section is a semi-circle of radius r/2 and the lower portion is a set of three circular arcs of radius r. The relatively flat floor, compared to a circular cross section of comparable size, allows for easier access by machinery during construction, and for post-construction inspection and maintenance.

Many tunnels for open-channel water flow, especially those designed by the USBR, have this cross-sectional shape. Note that it is only necessary to specify the radius, r, to define the size of a standard horseshoe cross section. See Annex D for the complete cross-sectional geometry equations that describe the standard horseshoe section.

III.4.4. Coordinate Section

This is the most generic cross-section in mCanal because it allows you to enter several pairs of x and y values to define the shape of the reach. The x values are along a horizontal axis, perpendicular to the stream-wise direction, while the y values are vertical.

You can enter the cross-sectional values in the table at the lower right of the sheet, which is activated for any reach in which you have specified a coordinate cross section. The table is like a simple spreadsheet with two columns. Every point in the cross section must have both x and y values, and you must have at least three pairs of data to define a cross section. Three pairs of data will define the trivial coordinate-based cross-section, that of a triangular shape (a triangular cross section can also be defined as a trapezoid with a zero base width). The app has a limit of 64 (x, y) coordinates per reach end.

All x and y values are relative to each other – the absolute values are unimportant. The app automatically determines the minimum y-value and the maximum depth. If the calculated or specified water depth exceeds this maximum depth at any time during a simulation, mCanal will report over-topping. Coordinate values are given in either in meters or feet, depending on the chosen system of units (Chapter I.4.1).

All coordinate section values (x and y) must be non-negative. So, make the y values at the left and right banks at least equal to the desired maximum depth of the cross section, with lower values at intermediate points in the section. If you enter any negative coordinate section values, mCanal will automatically shift all the values so that the minimum is zero.



It is required that the y-value of the first data pair be equal to the y-value of the last data pair (first and last rows of data), so that the left and right banks of the section are at the same elevation. Otherwise, the reach would overflow on one side before the other, and the "additional" information on the high side would never be applied during a simulation. Also, from the first row of data the y-values must remain the same or decrease – once the y-values begin to increase, they cannot again decrease. Otherwise, you could have multiple channels (see the two figures below) at low water depths, and this is not handled in the app.

Use one of the other cross-sectional shapes whenever possible because calculations tend to be slower when you have coordinate cross sections. As mentioned above, it is often adequate to approximate a cross section with a trapezoid (symmetrical or non-symmetrical), giving nearly the same simulation results as for a perhaps more precisely-defined coordinate section. This is because the hydraulic behavior of a canal reach is not nearly as sensitive to cross-sectional shape as it is to hydraulic roughness and longitudinal bed slope. This coordinate cross-section will generate an error when the data are checked (Chapter IX.1), and will prevent a simulation from running:



And, this is an example of the corresponding error message that you would see:

Show	Error 1 of
ErrorsWarnings	The coordinate cross section at the upstream end of Main canal has y-values that decrease, then increase, and subsequently decrease again.
0	Explanation and possible solutions
Error code: 1618	Change the data so that the y-values decrease from left to right, and once they start to increase up to the right side, don't let them decrease again. Otherwise, there could be multiple pools of water adjacent to each other in the reach.
Error and warning mess	sages, and suggestions for dealing with them.
Write to File	< Prev Next> OK

III.4.5. Coordinate Section Data Handling

Right-click (or **control**-click) over the graph view, which is labeled as *Channel* cross-section profile on the tab, to display a popup menu from which you can **Copy**, **Cut**, and **Paste** the (x, y) data that appear in the table. This allows you to reuse data, for example, from another cross section, then modify it as necessary, without having to manually reenter all of the values. For example, you can copy the coordinate cross-section data from one reach and paste it to another reach. You can also copy and paste from spreadsheet applications.



The popup menu is only available for the *Coordinates* cross section shape.

From the popup menu you can also read and write coordinate cross-section data, from and to plain text files. This is similar to reading and writing data for scheduling graphs (Chapter VII.5), and pump and valve data (Chapter VI.1.2-1.3). The data are written to the same folder as specified for the simulation results (Chapter IX.3) with a name like "Cross-Section Data (Main Canal).txt," where "Main Canal" is the reach name. The data will have three columns: (1) row number; (2) x-values; and, (3) y-values. In each row, the values from the first two columns are followed by a comma. Data for both the upstream and downstream ends of the reach will be written to the file. You can edit the text file of coordinate cross-section data, changing values, adding and deleting rows, as desired. Then, you can read the data back into the app by selecting **Read data...** from the popup menu. You can read any text file that was written as described in the previous paragraph, except that reading is only for the currently-display reach end (upstream or downstream). So, before reading a text file, ensure that it has only the values for the current reach end, otherwise you probably will not obtain the desired results. As indicated above, if the *Prismatic* box is checked, you will only be able to work with cross-sectional parameters at the upstream end of the reach.

The format of the data rows in the text file does not matter, as long as the values in each row are separated by commas, and there are at least two values per row. Spacing between values on a row is irrelevant. Rows without at least two values are ignored, as are those that are blank, or rows with only text (no numbers). For those rows with more than two columns (more than two numeric values), if any, the app takes the (x, y) data pair from the right-most two columns – this is why, in order to import data from a text file, you do not have to delete the row numbers from a file that was written as described above.

III.5. All Nodes

The list at the upper left of the *All Nodes* tab contains the names of all layout nodes, not just those corresponding to the downstream ends of reaches.

When you click on the **OK** button, the layout view will also be updated if node names are currently displayed in that view. You can also change a node name by double-clicking over the node in the layout view, when in the *Select* edit mode.

	Reaches	Calibratio	on Cr	oss Sections	All Nodes
System layout no	odes	2 of 9	Node co	oordinates	
Confluence		\$	X (m)		
					3672.89
Type: Confluer	ice		Y (m)		
Confluence node	e name	_			2968.05
Confluence					
change			Conflue	nce node locatio	n
Confluence US b	ped elevation (m)		Distan	e from US end	t (m)
	100	00.055	Distant		450.00
ohanga	Reach location:		Percen	t of reach leng	430.00
change			I croch	t of reaching	45.00%
	OUpstream				40.0078
	 Downstream 				
			Upstrea	m Node	
			Name:	Secondary So	urce
			Downst	ream Node	urce
			Name:	Terminus Nod	e
			Type:	Reach	
Names, elevations,	and locations for all	system layo	ut nodes		
					ОК

III.5.1. Node Names

In the *Reach node name* box you can change the name and then click on the **change** button below the box. This will update the list under *System layout nodes*. Enter a unique node name because duplicate names are not allowed. If the selected node is not a reach, the label changes to *Confluence node name*, and so on.

III.5.2. Bed Elevations

Modify the contents of this box to set the elevation of the selected reach node at the specified location (upstream or downstream end of the reach). Then, click on the **change** button below the box. Other elevations throughout the layout will be automatically calculated based on the elevation that you specify. This is accomplished by taking into account the layout, relative node connections, reach bed slopes, and reach drops. Alternatively, if you have selected a source or other node type, you can also specify the elevation there, and all other elevations will be updated automatically. Note that if you are only interested in water depths, and not water surface elevations, it is not necessary to specify a reach bed elevation.

The elevation of primary and secondary source nodes is necessary when you specify an upstream depth at any of them in the *Flows and Depths* sheet (Chapter IX.4.1). You can specify a depth upstream of a source node in order to have the model calculate the source flow rate based on one or more regulators at that location. In this case, the specified upstream depth at a source node is referenced to the elevation of the source node, as given in this tab. Thus, you can consider this elevation to be that of the bottom of the reservoir (for example) or other surface water supply, just upstream of the regulator(s) at the source node. The figure below shows an example of this.

Note that the editing of this value is only enabled when you have selected a primary or secondary source node in the list of nodes on this tab.



III.5.3. Node Locations

Node coordinates can be specified for all except bifurcations and confluence nodes. The coordinate values, X and Y, are referenced to the origin of the layout region, which is at the upper left corner of the layout view. You can change the coordinates of nodes here, instead of moving them in the layout window with the mouse, if you know the exact locations from a map, or if you want to specify relative distances between two or more nodes. The dimensions and aspect ratio of the layout region in the layout view can be modified in the *Preferences* sheet (Chapter I.4.3).

For bifurcations and confluence nodes, the X and Y coordinate cannot be directly specified, because they must fall along the path of the Bezier curve for the reach in which they are located. However, for these two node types you can specify the location in terms of distance along the reach, or as percent of the total reach length, measured from the upstream end. The percentage value must be between 0 and 100%.

When you click the **OK** button to accept changes to node coordinates or distance along a reach, the layout view will be redrawn to reflect the changes.

IV. Structure Configuration



After entering system-layout information, you will need to specify the number and types of control structures that will be included at different locations within the system. Regulator structures will be located at nodes in the layout, but turnouts can be positioned at any distance along a reach. Bifurcations and confluences can be located anywhere along a reach, like turnouts; but unlike turnouts, they are nodes in the layout and they also represent reach end points.

There are many different control structures to choose from, some of which are not really "structures." More precisely, these are all boundary conditions which are defined by mathematical relationships and are solved numerically along with the governing hydraulic equations during a simulation. All gates, weirs, valves, flumes, pumps, and other structures can be thought of as boundary conditions, as can other local hydraulic features that are included in the list of "structures" (sometimes referred to generically as "gates" in this Users' Guide).

Choose **Edit**, **Structures Data** from the main menu or click on the icon in the main toolbar. You can also hold down the **shift** key while double-clicking over a reach node (in the *Select* mode) in the layout view to open the *Structures Data* sheet. Note that you cannot change any of these parameters during a simulation.

	Regulators	Turnouts	Calibration)
Reaches		1 of 1	Structure categ	jory
Main canal		\$	Weirs	\$
Regulators		1 of 1	Structure type	
A generic weir		\$	Generic weir	\$
Regulator name change Reach location Upstream Ownstream	Valve type Null valve	+ – (pumps)		
Width (m)	Heigh	t (m)		
	1.000		1.000	n/a
Invert drop (m)				
	0.000		n/a	n/a
Names, types and di	mensions of regu	ılator bounda	ry conditions at re	ach ends.
Use as Default			ОК	Cancel

The two categories of structures are Regulators and Turnouts, corresponding to the first two tabs in this sheet. Regulators are "in-line" structures, located at the upstream and downstream ends of reaches. Regulators cannot be placed at intermediate distances along a reach. By default, a single structure is inserted at the upstream and downstream ends of each reach, but you can have multiple regulators in parallel at the upstream and downstream ends of a reach. Water that passes through a regulator does not leave the system, unless the regulator is at the downstream end of a terminus reach, at the end of a branch.

The photograph below shows two rectangular regulator gates in a canal, and a smaller gate at the left. In a simulation, the two regulators would demarcate the end of one reach and the beginning of another (because in this case it is seen that there is a downstream reach). The smaller gate is located at the downstream end of a reach, and in a hydraulic simulation it could be considered as either a turnout or a bifurcation, depending on whether the secondary canal is included in the system layout.

Every node in the system layout view has one regulator by default, and you can add regulators in parallel at any layout node. Another way of looking at this is that every reach has at least one regulator at its upstream end, and at least one regulator at its downstream end. The regulator(s) at a source node (primary or secondary) are specified as the regulator(s) at the upstream end of the reach(es) that are directly connected to the source node in the system layout.

As described in Chapter II, a system layout can also include bifurcation and confluence nodes. The regulator(s) at a bifurcation node



are specified as the regulator(s) at the upstream end of the reach that is directly connected to the bifurcation node in the system layout. That is, a bifurcation node marks the upstream end of a reach, and the beginning of a branch, so go to that reach to specify information about the regulators at a bifurcation node. A confluence node is actually the downstream end of a reach, so the regulator(s) at this node type are simply those found at the downstream end of that reach.

Turnouts are inserted individually along a reach and allow water to leave the reach, such as at a delivery point, and leave the system. After water leaves through a turnout during a simulation, mCanal is not able to keep track of where the water might go because it has left the system. Water can also enter a reach through a turnout (Chapter IV.2).

If you click the **Use as Default** button at the lower left of the sheet for either the *Regulator* and *Turnout* tabs, any new regulators or turnouts that you subsequently add will use the information shown at the time the button is clicked. This is convenient when adding multiple structures of the same type and or dimensions.

IV.1. Regulators

The *Regulators* list contains the names of all regulators at the current location (upstream or downstream) within the reach. By default, there is only one regulator at each location, but you can add others as necessary. Change the location (reach end) by clicking on the upstream or downstream button, which will update the list of regulator structures.

Add a regulator by entering a unique structure name and clicking on the + (plus) button. You can subsequently modify the name of a regulator by entering a unique name and clicking on the change button. Delete the currently selected regulator by clicking on the - (minus) button. Each end of a reach must have at least one regulator structure, so if there is only one regulator at a location (upstream or downstream), you will not be able to delete it.

You can scroll through the regulator types by clicking on the picture, or by selecting a Structure category and a Structure type from the drop-down lists. The name of the current structure type is given above the picture.

Note that the captions of the six edit boxes for parameters change according to the structure type. Only a few structure types need to use all six edit boxes to define the dimensions of the structure.

You do not need to specify regulator structure type(s) or other regulator information at the downstream end of a split reach because it will not be used by the model during a simulation. This is because the regulator information at the upstream end of each of the downstream connecting reaches is used at this location.

Also, when non-split reaches are connected in series (i.e. not connected through a bifurcation or confluence node), the regulator information at the upstream end of the downstream reach must be the same as the regulator information at the downstream end of the upstream reach, because they represent exactly the same physical location. The app will make sure that this is the case for all serial reach connections, so you can edit the regulator data at either "location" and you will obtain the same result. This is called "automatic mirroring," and it is done for scheduling graphs as well.



IV.2. Turnouts

By default, there are no turnouts in a reach, but you can insert up to a total of 32 in each reach. Add a turnout by entering a unique name and clicking the + (plus) button. The app performs onedimensional hydraulic calculations, so when you add a turnout it could be on either the left or right side of the reach. The model does not distinguish between left and right sides of a reach, but if you insert two turnouts into a reach and place them at exactly the same location (distance from the upstream end of the reach), the logical assumption is that in the real system, one turnout is on the left side and the other on the right. It would not make sense to place three or more turnouts at exactly the same location in a reach.

	Regulators	Turnouts	Calibration]
Reaches		1 of 1	Structure categ	gory
Main canal		\$	Gates	\$
Turnouts		1 of 2	Structure type	
L-29-5A		\$	Rectangular g	jate ᅌ
Turnout name change		+ -		
Location (m)	Valve type	e (pumps)		
560.00) Null valv	e 🗘		
Upstream		Downstream	Real and	And and a second
				And the second second
Width (m)	Heigh	nt (m)		
	1.000		1.000	n/a
Invert drop (m)				
().000		n/a	n/a
Names, types and dir	nensions of turr	nout (lateral-f	low) boundary con	ditions.
Use as Default			ОК	Cancel

As for regulators, you can change the name of a turnout by entering a unique name and clicking the change button. Delete the currently selected turnout by clicking the - (minus) button. You can change the type of a turnout at any time in the same way as for regulators. The name of the current structure type is given above the picture.

You can specify the location of a turnout along a reach by editing the *Location* value. Changing the value will result in an updated graphic below the edit box. This graphic is a schematic of all turnout locations (vertical bars) along the reach, where the triangle shows the relative location of the currently selected turnout. The schematic is scaled according to the reach length, from the upstream end to the downstream end (left to right).

IV.3. Invert Drops

The invert drop of a structure, ΔE_s , is not the same as the 'drop' as specified for reaches. An invert drop is the difference in elevation from where the bed of the reach would be, at the downstream end of the reach, and where the bottom of the structure is actually located. You can determine where the downstream end of the channel bed would be by extrapolating a straight line at the value of the bed slope for the last segment (if it has a compound slope) of the reach.

In many cases, the invert drop is zero because the bottom of a gate or other structure coincides with the bottom of the canal bed at the downstream end of a reach. But in some cases, there may be a short vertical transition that causes the bottom of the structure opening to be different than the canal bed. The invert drop value is positive when the structure invert is below the extrapolated channel bed at the downstream end of the reach, or negative if it is above, as seen in the two sample cases shown in the figure below. The left side of the figure is for a positive invert drop and a positive DS reach drop. The right side of the same figure has a negative invert drop and a positive reach drop.



Invert drops are often important parameters with regard to accurate hydraulic simulations, in large part because field measurements of longitudinal bed slopes in canals seldom give a perfectly uniform slope. Instead, local high spots and depressions are usually found along a canal reach, as in the figure below, such that it is necessary to use linear regression analysis to determine a representative longitudinal bed slope value based on measured elevations.

Thus, invert drops are local corrections to an approximating regression line, applied at the downstream end of a reach. Of course, you can use compound slopes along a reach, but even these may not adequately approximate the actual bed profile without the application of non-zero invert drop values.



Invert drops for structures are handled the same for turnouts as for regulators, but the downstream side of a turnout is not the beginning of another reach – it is outside the boundaries of the system being simulated. This is why all turnouts are considered external boundaries.

There is one difference in the interpretation of an invert drop for turnouts. That is, for turnouts it is not necessary to extrapolate the bed elevation to the "end of the reach." Rather, the invert drop for a turnout is referenced to the lowest point (or "invert") of the channel cross section in the reach at the turnout location.

IV.4. Structure Types

The same structure types are available in the app for both regulators and turnouts, even though some of the structures may not be practical for application as turnouts. Following is an introduction to each of the structure types, with comments about their application and dimensional parameters that you need to supply to define their physical size and (in some cases) shape. Calibration equations for each structure type are provided in Chapter V.

In many of the following structure types, the "height" parameter refers to the vertical range of structure adjustment. That is, the range of structure openings (or settings) is from zero to the specified "height." In mCanal, this applies to most weirs and underflow gates, of which there are a variety. Of course, some structure types are not adjustable. The opening of a structure can be changed by interrupting a simulation, or through a scheduling graph. But you can simulate a nonadjustable structure by simply not changing the setting manually. The "height" parameter is irrelevant when a structure is nonadjustable. But for adjustable structures, the opening cannot exceed the specified "height" value, either through manual or automatic control.

IV.4.1. Generic Weir

This is an adjustable, generic weir type for which you must supply calibration parameters. Enter the width, height, and invert drop. It may be presumed that this structure has a rectangular notch (opening), but this is not necessarily the case, because it depends on the calibration values that are used. However, if it is indeed a rectangular notch, the "width" parameter refers to the width (sometimes referred to as "length," L, in hydraulics books) of the crest at the notch. The width value may be greater than, equal to, or less than the width of the channel in which it is located. If the width is greater than the channel width, it means: (1) there is a local widening of the channel at the weir location; (2) the weir is oriented obliquely to the cross section; or, (3) it is some sort of "duckbill" weir. The figure below shows a plan view of duckbill and oblique weirs in a canal.



The height value refers to the vertical range of crest heights, from lowest to highest. If the crest is not adjustable, the height value can be zero. The minimum height of the crest, relative to the canal bed on the upstream side, is specified by the invert "drop" value of the structure, as shown in the figure below (a side view), which should be negative in the case of a weir structure.



This minimum crest height corresponds to a setting (G_o) of zero; if you specify a nonzero setting for the weir, the crest height will be that much higher than the minimum height. If you do not change the weir setting, it will function as a fixed-setting structure during a simulation.

IV.4.2. Cipoletti Weir



Enter the width, height, and invert drop. This type of weir is found in the technical literature for over 100 years, and is like the sharp-crested weir, except that the notch shape is trapezoidal and the theoretical calibration equation involves somewhat simpler equations (Flinn and Dyer 1894). In this case, the width value is the width of the crest, which is the narrowest part of the notch. Again, it is assumed that the crest is level, and that the face plate of the weir is vertical. The photograph at left shows such a weir with a sharp steel crest attached

to the upstream side of a concrete structure, operating under free-flow conditions.

IV.4.3. Rectangular Sharp-Crested Weir

This is an adjustable weir with a rectangular notch. Enter the width, height, and invert drop. The width means the "length" of the crest, L. The height means the vertical range of crest movement, and the invert drop is the minimum setting, just as defined above for the generic weir type.

This structure type uses calibration parameters from hydraulic laboratory data, so you only have to specify the structure dimensions to apply it in a simulation. Note that in this case the laboratory data were taken for rectangular channel sections and a negligibly small approach velocity.

It is assumed that there is no slope along the crest, that it is level. The calibration also assumes that the face plate of the weir is vertical.

IV.4.4. V-Notch Weir

V-notch, or triangular, weirs are more accurate than other sharp-crested weirs for flow measurement with small upstream water depths. Enter the notch angle (in degrees) and the invert drop. This weir is like

the Cipoletti, except that the notch shape is triangular, essentially making the lowest point of the weir crest a single point (or vertex). The calibration for this structure type is based on extensive laboratory studies. It is assumed that the weir plate is vertical, not leaning upstream or downstream.

IV.4.5. Broad-Crested Weir

This is a non-adjustable structure which, in practice, is used principally for flow measurement purposes, but can also be considered a boundary condition in *mCanal*. This structure type is also sometimes referred to as a "ramp flume," or "long-throated flume," and may have other names as well. Hydraulically, it is usually designed to operate under free-flow conditions in which the streamlines are mostly parallel and horizontal over the crest, where critical flow is assumed to occur. The app also assumes free-flow conditions over a broad-crested weir, which may not always be the case, especially if the downstream reach drop is zero or negative. The side view below shows a broad-crested weir with a negative invert "drop" (crest height), positive reach drop, and free-flow conditions.



In mCanal, the throat cross section of a broad-crested weir is limited to symmetrical trapezoidal shapes (which, of course, include rectangular sections). In this case, the word "symmetrical" simply means that the side slopes are exactly the same on the left and right sides of the throat section.

Enter the crest (or sill) width in the throat section. Also enter the crest height, where "height" is the elevation difference between the upstream channel bed and the crest of the broad-crested weir. The crest height must be a positive, nonzero value. Finally, specify the inverse side slope, which is zero for a rectangular throat cross-section.

The width and height of a broad-crested weir are usually related by the channel geometry when the upstream channel section shape is trapezoidal unless you want the section to be different at the throat of the broad-crested weir. For example, if the upstream channel has a trapezoidal cross-section, the throat section of the broad-crested weir will usually (in practice) have the same side slope. The next figure shows an end view of a broad-crested weir in which the upstream cross section is a symmetrical trapezoid, with the same side slopes in the upstream section and in the throat (note that, as in the previous figure, ΔE_s is negative because the crest is above the upstream channel invert).



Thus, if the upstream cross section is a symmetrical trapezoid and the side slopes are the same in the throat section of the broad-crested weir, the crest width would normally be determined as:

$$b_c = b_u + 2m_u z \tag{1}$$

where b_c is the crest width (m or ft); b_u is the upstream channel base width (m or ft); m_u is the upstream inverse side slope (equal to the inverse side slope in the throat, m_c); and, z is the crest height (m or ft), relative to the invert of the upstream channel at the structure location.

IV.4.6. Overshot Gate

Enter the width, length, and invert drop. This type of gate structure is like a weir in which the face plate is not vertical and fixed but hinged at the bottom. The width refers to the width of the face plate, also corresponding to the width of the rectangular notch of the weir (or gate). Even when this type of structure is used in a trapezoidal canal section, there is a transition to a rectangular section at the gate location, and another transition back to a trapezoid on the downstream side. The length, L, refers to the length of the face plate, from the hinges up to the crest. Note that this structure is sometimes referred to as a "leaf gate," or Oberman gate, although it behaves hydraulically as a weir.

IV.4.7. Rectangular Gate

Specify the width, height, and invert drop. This is a vertical sluice gate, which would be a simple slide gate for small sizes, but with more complex mechanical features for larger sizes. The opening of the gate is rectangular in shape, with a fixed width, G_w , and an adjustable opening, G_o . The height value specifies the vertical range of gate *opening* (not necessarily that of the gate leaf), from fully closed to fully open.

IV.4.8. Circular Gate

Enter the pipe diameter and invert drop. This is like a rectangular gate, except that the water passes through a circular pipe, while the gate leaf is flat (horizontal) on the bottom. The pipe diameter is the inside diameter of the pipe, which defines the maximum effective gate opening.

IV.4.9. Round Gate

Enter the pipe diameter, gate diameter, and invert drop. This is like a rectangular or circular gate, except that the water passes through a circular pipe, and the gate leaf at the end of the pipe is also circular. Thus, in the app it is called a "Round Gate." The gate leaf diameter must be greater than the inside diameter of the pipe, otherwise the gate would not seal when fully closed.

IV.4.10. Radial Gate

In the app, this structure is based on USBR design specifications and calibrations by Buyalski (1983). Enter the width, radius, pivot height, and invert drop. The width is of the gate opening and is considered to be fixed. The radius is from the pivot point to the upstream side of the gate leaf, and the pivot height is the vertical distance from the channel bottom at the gate leaf up to the pivot point. Radial gates also have rectangular openings, where the water passes through, but unlike rectangular gates, the gate leaf is curved at the top of the opening.

IV.4.11. Generic Constriction

This is a generic non-adjustable structure type which always operates under non-orifice flow regimes, either free or submerged. It includes any non-orifice structure that locally constricts the channel cross section, vertically and or horizontally.

IV.4.12. Cutthroat Flume



The Cutthroat flume has relatively simple geometry and is easier to build that many other measurement flumes. The original development of the Cutthroat flume included more than 20 standard sizes, as for the Parshall flume (Chapter IV.4.13), and similar calibration equations. However, in mCanal a generic, unified calibration (Temeepathanapongsa, et al. 2013) is applied, whereby it is not necessary to select a standard size – it is only necessary to respect certain dimensional limits and ratios.

Enter the width (W), length (L), and invert drop (ΔE_s) to define the size of the flume. It is recommended that the ratio of W/L be restricted as

follows: $1/9 \le W/L \le 4/9$, as per the original guidelines from Skogerboe, et al. (1967a). The largest standard size has a throat width of 4.00 ft.

IV.4.13. Parshall Flume



The app includes 18 standard sizes for Parshall flumes (Parshall 1945, 1953), from throat widths of $1 ft \leq W \leq 50 ft$. Each of the standard sizes has specific dimensions as seen below in the figures and tables for Parshall flumes. Therefore, you only need to select the throat width, W, which is unique for each standard size, and all of the other dimensions are thereby determined. The figures below show the form of a Parshall flume,

in which h_u and h_d indicate the locations where the upstream and downstream depths, respectively, are supposed to be measured. The floor of the inlet section is assumed to be level in the longitudinal and transverse directions (otherwise, special calibration adjustments must be applied, but they are not included in this app).



The next two tables show all of the dimensions of the 18 standard Parshall flume sizes that are included in the app (actually, there are 23 standard sizes, and those excluded here have W < 1 ft). The letters in the table headers correspond to the dimensions as shown above in the top and side views of a Parshall flume. The rightmost two columns indicate the minimum and maximum flow rates for each of the flume sizes – these limits are to control the accuracy of the calibrations, as seen in Chapter V.

W (ft)	Dimensions (ft)							Q (cfs)				
vv (10)	Α	В	С	D	E	F	G	Н	K	Х	Y	min	max
1.00	2.771	2.000	4.500	4.406	2.000	3.000	3.000	0.750	0.250	0.167	0.250	0.40	16.0
1.50	3.365	2.500	4.750	4.656	2.000	3.000	3.000	0.750	0.250	0.167	0.250	0.50	24.0
2.00	3.958	3.000	5.000	4.906	2.000	3.000	3.000	0.750	0.250	0.167	0.250	0.70	33.0
2.50	4.563	3.500	5.354	5.250	2.000	3.000	3.000	0.750	0.250	0.167	0.250	0.80	41.0
3.00	5.156	4.000	5.500	5.396	2.000	3.000	3.000	0.750	0.250	0.167	0.250	1.0	50.0
4.00	6.354	5.000	6.000	5.885	2.000	3.000	3.000	0.750	0.250	0.167	0.250	1.3	68.0
5.00	7.552	6.000	6.500	6.375	2.000	3.000	3.000	0.750	0.250	0.167	0.250	2.2	86.0
6.00	8.750	7.000	7.000	6.865	2.000	3.000	3.000	0.750	0.250	0.167	0.250	2.6	104
7.00	9.948	8.000	7.500	7.354	2.000	3.000	3.000	0.750	0.250	0.167	0.250	4.1	121
8.00	11.146	9.000	8.000	7.844	2.000	3.000	3.000	0.750	0.250	0.167	0.250	4.6	140
10.00	15.604	12.000	14.271	14.000	3.000	6.000	4.000	1.125	0.500	1.000	0.750	6.0	200
12.00	18.396	14.667	16.313	16.000	3.000	8.000	5.000	1.125	0.500	1.000	0.750	8.0	350
15.00	25.000	18.333	25.500	25.000	4.000	10.000	6.000	1.500	0.750	1.000	0.750	8.0	600
20.00	30.000	24.000	25.500	25.000	6.000	12.000	7.000	2.250	1.000	1.000	0.750	10	1000
25.00	35.000	29.333	25.500	25.000	6.000	13.000	7.000	2.250	1.000	1.000	0.750	15	1200
30.00	40.396	34.667	26.521	26.000	6.000	14.000	7.000	2.250	1.000	1.000	0.750	15	1500
40.00	50.792	45.333	27.542	27.000	6.000	16.000	7.000	2.250	1.000	1.000	0.750	20	2000
50.00	60.792	56.667	27.542	27.000	6.000	20.000	7.000	2.250	1.000	1.000	0.750	25	3000

Parshall flume dimensions in English units:

Parshall flume dimensions in metric units:

The standard Parshall flume sizes were developed in English units, but these are converted to metric units in the app for convenience. The app applies the calibration parameters (Chapter V.13) according to the system of units that you select in the **Preferences** sheet.

$\mathbf{W}(\mathbf{m})$	Dimensions (m)							Q (m	$^{3}/s)$				
vv (III)	Α	В	С	D	\mathbf{E}	F	G	H	K	X	Y	min	max
0.305	0.845	0.610	1.372	1.343	0.610	0.914	0.914	0.229	0.076	0.051	0.076	0.0113	0.453
0.457	1.026	0.762	1.448	1.419	0.610	0.914	0.914	0.229	0.076	0.051	0.076	0.0142	0.680
0.610	1.207	0.914	1.524	1.495	0.610	0.914	0.914	0.229	0.076	0.051	0.076	0.0198	0.934
0.762	1.391	1.067	1.632	1.600	0.610	0.914	0.914	0.229	0.076	0.051	0.076	0.0227	1.16
0.914	1.572	1.219	1.676	1.645	0.610	0.914	0.914	0.229	0.076	0.051	0.076	0.0283	1.42
1.219	1.937	1.524	1.829	1.794	0.610	0.914	0.914	0.229	0.076	0.051	0.076	0.0368	1.93
1.524	2.302	1.829	1.981	1.943	0.610	0.914	0.914	0.229	0.076	0.051	0.076	0.0623	2.44
1.829	2.667	2.134	2.134	2.092	0.610	0.914	0.914	0.229	0.076	0.051	0.076	0.0736	2.94
2.134	3.032	2.438	2.286	2.242	0.610	0.914	0.914	0.229	0.076	0.051	0.076	0.116	3.43
2.438	3.397	2.743	2.438	2.391	0.610	0.914	0.914	0.229	0.076	0.051	0.076	0.130	3.96
3.048	4.756	3.658	4.350	4.267	0.914	1.829	1.219	0.343	0.152	0.305	0.229	0.170	5.66
3.658	5.607	4.470	4.972	4.877	0.914	2.438	1.524	0.343	0.152	0.305	0.229	0.227	9.91
4.572	7.620	5.588	7.772	7.620	1.219	3.048	1.829	0.457	0.229	0.305	0.229	0.227	17.0
6.096	9.144	7.315	7.772	7.620	1.829	3.658	2.134	0.686	0.305	0.305	0.229	0.283	28.3
7.620	10.668	8.941	7.772	7.620	1.829	3.962	2.134	0.686	0.305	0.305	0.229	0.425	34.0
9.144	12.313	10.566	8.084	7.925	1.829	4.267	2.134	0.686	0.305	0.305	0.229	0.425	42.5
12.192	15.481	13.818	8.395	8.230	1.829	4.877	2.134	0.686	0.305	0.305	0.229	0.566	56.6
15.240	18.529	17.272	8.395	8.230	1.829	6.096	2.134	0.686	0.305	0.305	0.229	0.708	85.0

IV.4.14. Energy Balance

This is one of three *non-structure* boundary conditions – the other two are *Uniform Flow* and *Undefined*. There are no dimensions to be specified for this type of non-structure boundary condition, which is based on the specific-energy equation (derived from the Bernoulli equation) and specified upstream and downstream energy-loss coefficients. Nevertheless, you can specify a negative drop, ΔE_d , in the bed elevation at the location. Usually, for this boundary condition, you would let $\Delta E_d = 0$, but it could also be a negative value. Positive drop values at such a boundary are not allowed because they result in the square root of a negative number under certain operating conditions.

This boundary condition can sometimes be used at locations where a change in channel cross-section occurs, or other similar transition in the absence of any type of flow-control structure. An energy balance can only be used as a regulator, not as a turnout. Also, as a regulator, it can only be used at the downstream end of a non-split reach.

The energy-balance boundary condition is not allowed at the system source, nor as a turnout, confluence, or bifurcation structure. Also, multiple energy-balance boundary conditions at a single location are not permitted.

IV.4.15. Uniform Flow

There are no dimensions to be specified for this type of boundary. This is another non-structure boundary condition and is somewhat *theoretical* because it may be physically impractical for all but a few realistic conditions. The uniform-flow boundary condition is most valid at branch terminus points with a zero downstream drop value, but even under those circumstances it may be a significant idealization of true hydraulic conditions. The use of one or more uniform-flow boundary conditions at the downstream end of a split reach is perhaps even more prone to producing unrealistic simulation results. When you use a uniform-flow boundary condition, mCanal assumes that it is valid without performing any additional calculations to verify the assumption. A uniform-flow boundary can only be used as a regulator, not as a turnout.

IV.4.16. Undefined

There are no dimensions to be specified for this type of boundary, which is a *non-structure*. This boundary condition is used when you want to specify the flow rate or water depth at a location in the system without concern for the hydraulic behavior of a specific structure type. For example, you might have flow rate data for a particular reach and want to simulate the changes in water depth for the given flow rate(s).

IV.4.17. Centrifugal Pump



This structure type is only available when there are pump characteristic curve data available (Chapter VI.1), otherwise it will not appear as an option in the **Structures Data** sheet. There will be a list of pumps to choose from, each having the name of a characteristic curve data set.

Enter the pipe diameter, pipe length, invert drop, and outlet valve type. The pipe length is the total for both the suction side and discharge sides of the pump, and the diameter of the pipe is assumed to be the same on both sides. You can account for minor losses (e.g. entrance and exit losses) as equivalent length of pipe by entering a total pipe length slightly greater than the physical length.

Pumps include a valve in the outlet pipe, whereby valve operation affects the discharge through the pump; or, you can choose a "null" valve which will not cause any additional head loss. The invert "drop" for a centrifugal pump will usually be negative, meaning an increase in elevation, relative to the downstream bed elevation (Chapter IV.3).

IV.4.18. Valve



Valves are used at pump outlets, as mentioned just above, and in pipeline reaches. There are a few sample valve types in the default configuration layout, and you can modify these and add new ones (Chapter VI.2). Enter the invert drop and the valve type. The number of different valve types depends on the data that you enter or read in from text files; when you save a configuration file, all of the valve data are saved with it. Four sets of

sample valve calibration data are included in the default system layout. This structure type is only used in pipeline reaches, and for centrifugal pumps (as regulators or turnouts). Centrifugal pumps always include a valve in the discharge pipe, but you can specify a "null" valve at a pump, which effectively eliminates the valve.

V. Structure Calibration



This chapter follows directly from Chapter IV and pertains to the same *Structures Data* sheet. Herein, "structures" and "gates" refer to *boundary conditions* at upstream and downstream reach ends, and at turnouts in reaches. Boundary conditions include physical structures and certain non-structures, but each has one or more equations to define how they affect a hydraulic simulation, and the hydraulic behavior of a canal system, at point locations. Chapter IV introduced the types of boundary conditions, and Chapter V covers

the respective calibration equations, parameters and variables. This chapter pertains to the *Calibration* tab of the *Structures Data* sheet, but goes beyond that to describe the mathematical relationships between water depths and flow rates at boundaries.

To edit boundary-condition calibration parameters, where applicable, choose Edit, Structures Data from the main menu or click on the icon in the toolbar. You can also hold down the shift key while double-clicking over a reach node (in the *Select* mode) in the layout view to open the *Structures Data* sheet, and then click on the *Calibration* tab. These parameters cannot be changed during a simulation.

The *Calibration* tab is where you enter calibration values for individual regulator and turnout structures. This is different than the *Numerical Parameters* sheet (Chapter VIII), which has values that are applied to all reaches and or structures in a system layout.

	Regulators Turnout	ts Calibration
Reaches Main canal	1 of 1	Structure category: Weirs Structure type:
A generic weir		Transition submergence: 0.000
Structure kind	Orifice: free flow	Orifice: submerged flow
Regulators	Coefficient	Coefficient
Turnouts		n/a n/a
	Exponent	Exponent
Reach location		n/a n/a
Upstream	Non-orifice: free flow	w Non-orifice: submerged flow
 Downstream 	Coefficient	Coefficient
	1	1.800 n/a
	Exponent	Exponent
	1	1.500 n/a
Calibration parameter	s for regulator and turnout	t boundary conditions.
		OK Cancel



Calibration data are very important insofar as they often tend to make the difference between simulation results that accurately reflect the true hydraulic behavior of a specific system, and results that really correspond to some other system. That is, if you do not use appropriate calibration values, your simulation results will be for a system different than that for which you intend to model, thereby reducing the value of the simulation results and any conclusions

that might be drawn from them. For the simulation of an existing system, the best calibration parameter values are those that are derived directly from field measurements in that system. But if you mean to simulate a hypothetical system, perhaps based upon a new design or a proposed modification of an existing system, you will have to rely on established procedures and engineering judgment to determine appropriate calibration values.

The **Calibration** tab is where you enter calibration values for regulators and turnouts. Although these are referred to as "structures," they may be more generically called "boundary conditions," at least in terms of mathematical modeling, as discussed in Section IV. This is because some of the structure types are, in fact, not structures at all. For example, the uniform flow boundary condition is simply the application of a uniform flow equation (Chezy or Manning) at an upstream or downstream reach boundary. Boundary conditions are mathematical relationships (or a set of relationships), usually for flow rate as a function of depth or cross-sectional flow area, with one or more calibration parameters. Different structures types have different mathematical relationships that describe their hydraulic behavior.

For a given reach, you can select from any of the structures in the list to display the calibration parameters for that structure, and you can edit the values. The transition submergence, St, is found to the right of the list of structures. The value is automatically calculated based on non-orifice calibration parameter values, and it represents the submergence value at which the regime would change from free to submerged, and vice versa, as described below. Transition submergence applies only to non-orifice flow.

All reaches have at least one regulator at the upstream end, and one regulator at the downstream end. However, a reach might not have any turnouts. The calibration data are the same for regulators and turnouts but vary significantly from one structure type to another. For example, the calibration requirements for a weir are less than those for most gates (e.g. rectangular gate, circular gate, and others). Some structure types, such as V-notch weirs, require no user-supplied calibration data because their calibration is entirely based on the physical dimensions of the structure.

Calibration parameters for structures, in general, are divided into two main categories: (1) orifice flow; and, (2) non-orifice flow. Each of the two main categories are further divided into free and submerged flow. Some structure types can use the calibration parameters for all of these categories during a simulation (although that would perhaps constitute a range of hydraulic regimes that is rarely found in the field for a single structure), others use only some, and still others use none of them. For example, V-notch weir calibration parameters are generated internally based on dimensional criteria (which are from laboratory experiments) and you do not need to enter any data here, whereas rectangular weirs will use data from only the non-orifice main category. That is because weirs cannot operate under orifice conditions.



Note that you can perform field or laboratory calibration work yourself if you want to obtain the most representative values for the hydraulic simulation of a particular system. When simulating existing systems, it is always best to base calibrations on field measurements because they will provide more accurate modeling results. Or, you can use published procedures, based on laboratory and or field data analysis, for determining the calibration parameters

that are used in the app.

When calibration data are not required for a given structure type, the corresponding values will be disabled in this sheet.

Following are the calibration details of each of the structure types that were introduced in the previous section. These provide details of what calibration parameters (if any) you must supply to define the relationship between water depth(s) and flow rate through or over each of the structure types. The complete equations are also provided, which in some cases is complex, but it is important to know what the app is using to define these relationships because they provide the basis for boundary-conditions effects on simulation results. For some of the more general and some of the more complex calibrations, the details are found in some of the Annexes, as cited below, where appropriate.

V.1. Generic Weir

This structure type can accommodate many common weir shapes because the calibration values are user-specified. It can also be used to define the hydraulic behavior of non-sharp-crested weirs, duckbill weirs, and sudden drops in bed elevation, which can often be adequately field-calibrated using the same equation form:

$$Q = K_s C_f L \left(h_u + \Delta E_s - G_o \right)^{n_f} \tag{V.1}$$

where Q is flow rate $(m^3/s \text{ or } cfs)$; K_s is a submerged-flow coefficient (equals 1.0 if free flow); C_f is a calibration coefficient; L is the weir crest length (m or ft); hu is the upstream depth, referenced to the weir crest elevation (m or ft); ΔE_s is the upstream invert drop (m or ft), defining the minimum weir crest height; G_o is the weir crest setting; and, n_f is a calibration exponent. The K_s value is defined in Annex A.

See the figure below for a side-view explanation of some of the terms in the above equation for Q. The figure provides an example in which the upstream invert drop value is negative; in other cases, the invert drop might be positive or zero. Note that when the upstream invert drop is negative, it defines a minimum (i.e. $G_o = 0.0$) weir crest height relative to the upstream channel bed.



For rectangular weir notches and rectangular drops, the exponent, n_f , is theoretically equal to 3/2, but in practice it can vary from slightly below 3/2 to slightly above 3/2. For V-notch (triangular) weirs, n_f is theoretically equal to 5/2, and in practice is always very near this value. The figure above shows a side view of a sharp-crested weir with a negative upstream invert drop.

Rectangular weirs can be unsuppressed (b = L), or suppressed (b > L), where b is the base width of

a rectangular channel cross section. The term *suppressed* refers to the suppression of end-contraction effects, at the sides of the weir notch. The equations sometimes used to determine C_f are more complex for *unsuppressed* rectangular weirs than for suppressed weirs.

The submerged-flow coefficient, K_s , is determined according to the procedures outlined in Appendix A; however, it is noted that in all cases $0 \le K_s \le 1.0$. When there is no head differential across the weir, there is no flow and $K_s = 0$. When the flow regime is entirely free, $K_s = 1.0$, and when it is submerged, $K_s < 1.0$. The K_s coefficient is used in the same way for all weir structures to handle submerged-flow conditions.

The value of the upstream depth, h, should be measured upstream of the weir crest, at a distance of approximately four to six times the value of h for the maximum flow rate over the weir, or h_{max} . However, in the app, the water depth just upstream of the weir crest is used, being that the next calculated upstream value is usually much further away than four times h_{max} .

Below is a photograph of an inverse (facing upstream) duckbill weir operating under free-flow conditions in an irrigation canal in California.



If you want to use laboratory-derived calibrations for rectangular, sharp-crested weirs under free-flow conditions, assuming a negligible approach velocity (refer to V.3 below), which can be used to determine the values of C_f and n_f , as used in the app. Note that weir calibrations can be fairly complex, and the data from different researchers do not always agree, though the basic form of free-flow weir equations is well established in the technical literature. Corrections for approach velocity can also be made, but this is not currently included in *mCanal*. For a more thorough treatment of weir calibrations, refer to Shen (1981), Kindsvater and Carter (1957), Addison (1949), Villemonte (1947), and dozens of other publications on this subject.
V.2. Cipoletti Weir

The trapezoidal weir most often found in practice is the Cipoletti weir. This is a fully contracted weir in which the notch ends (sides) are not vertical, as they are for a rectangular weir. The effects of end contraction are compensated for by this trapezoidal notch shape, meaning that mathematical corrections for end contraction are unnecessary, and the equation is simpler than (for example), those of the rectangular weir described in Section V.3.

The side slopes of the notch are designed to correct for end contraction (as manifested in a rectangular weir), splayed out at angle of 14° with the vertical, or nearly 1 horizontal to 4 vertical (tan 14° = 0.2493, or almost 1/4). The sloping sides provide the advantage of having a stable discharge coefficient. The general form of the discharge equation is also analogous to that of Eq. V.4, but simpler, and where n_f is equal to 3/2.

$$Q = K_s C L \left(h_u + \Delta E_s - G_o \right)^{1.5} \tag{V.2}$$

where K_s is a coefficient for submerged flow (Annex A); C is a constant coefficient; L is the length of the weir crest (m or ft); and, h is the depth of water on the upstream side, measured from the crest elevation (m or ft). The discharge equation by Addison (1949) is:

$$Q = K_s \left[0.63 \left(\frac{2}{3} \right) \sqrt{2g} \right] Lh^{1.5} = K_s C_{cip} Lh^{1.5}$$
(V.3)

with the addition of K_s , as defined in Annex A, where $h = h_u + \Delta E_s - G_o$.



The units for L and h are m for Q in m^3/s , with $C_{cip} = 1.86$. The units for L and h are feet for Q in cfs, with $_{Ccip} = 3.37$. Note that Eq. (V.3) is of the same form as a rectangular sharp-crested weir, but it is simpler than that for unsuppressed rectangular and triangular (V-notch) sharp-crested weirs because the coefficient is a simple constant. This is the equation used in *mCanal* for free flow past a Cipoletti weir.

V.3. Rectangular Weir

Laboratory calibration data are applied for this structure, so you do not need to enter any calibration parameters. The general form of the discharge equation is analogous to that of the generic sharp-crested weir as described above, where n_f is equal to 3/2. In metric units, the discharge equation is:

$$Q = K_s C_e L_e h_e^{n_f} \tag{V.4}$$

where Q is in m^3/s ; K_s is a coefficient for submerged flow (Annex A); C_e is the discharge coefficient $(m^{1/2}/s)$; L_e is the effective crest length (m); h_e is the effective head (m), referenced to the crest elevation; and, n_f is defined to be equal to 3/2.

The effective crest length is:

$$L_e = L + K_L \tag{V.5}$$

where L is the measured crest length (m); and, K_L is a small calibration adjustment on L(m), as defined below.

The effective head is:

$$h_e = h_u + \Delta E_s - G_o + K_H = h_u - P + K_H \tag{V.6}$$

where h_u is the upstream depth (m), referenced from the bottom of the channel on the upstream side of the weir; ΔE_s is the upstream invert drop of the structure (m); G_o is the vertical crest "setting" (m); K_H is a constant, equal to 0.000914 m. When ΔE_s is zero, G_o is equivalent to P, where $P = G_o - \Delta E_s$.

The discharge coefficient is defined as:

$$C_e = \alpha \left(\frac{h_u - P}{P}\right) + \beta \tag{V.7}$$

where h_u is the upstream depth (m); $P = G_o - \Delta E_u$; and,

$$\beta = 1.724 + 0.04789 \left(\frac{L}{T}\right) \tag{V.8}$$

where T is the water surface width (m); and,

$$\alpha = \frac{-0.00470432 + 0.030365 \left(\frac{L}{T}\right)}{1 - 1.76542 \left(\frac{L}{T}\right) + 0.879917 \left(\frac{L}{T}\right)^2} \tag{V.9}$$

Note that the flow rate past the weir is zero when both the upstream and downstream water surface elevations are below that of the weir crest, or when the upstream and downstream water surface elevations are equal. The value of K_L is: For $0 \le L/T \le 0.35$:

$$K_L = 0.002298 + 0.00048 \left(\frac{L}{T}\right) \tag{V.10}$$

For $0.35 < L/T \le 1.0$:

$$K_L = -0.10609 \left(\frac{L}{T}\right)^4 + 0.1922 \left(\frac{L}{T}\right)^3 - 0.11417 \left(\frac{L}{T}\right)^2 + 0.028182 \left(\frac{L}{T}\right) - 0.00006$$
(V.11)



where K_L is in m.

For English units, first convert (multiply by 0.3048) lengths $(L, T, G_o, h_u, P, \text{ and } \Delta E_s)$ to meters, apply Eqs. (V.4) to (V.11), and then multiply the resulting flow rate (Q) by 35.31 to obtain units of cubic feet per second (cfs).

For the above weir calibration, the ratio h_u/P is limited as follows: $0 \le hu/P \le 2.5$. The ratio L/T is limited to: $0 \le L/T \le 1.0$, where L/T = 1.0 corresponds to a suppressed weir in a rectangular upstream channel. In a rectangular channel, T = b, where b is the base width of the section. Recall that the parameter ΔE_u is a "drop," and for weirs it is usually a negative value, defining the minimum value of P. Note that for L/T = 1 (suppressed end contractions): $\alpha = 0.224$, $\beta = 1.772$; and, $K_L = 0.000062 \ m$.

V.4. V-Notch Weir

The boundary condition for V-notch, or triangular, weirs uses a standard laboratory-developed calibration. The only specified parameter is the angle of the triangular opening, θ , which is limited to the range 20° to 100°. Calibration values are calculated, not specified, meaning that if you wish to use a custom calibration for this type of boundary condition, you should use the generic weir boundary condition (Section V.1). Also, the standard calibration for the sharp-crested V-notch weir assumes a negligible approach velocity.



The discharge equation is as follows:

$$Q = \frac{8}{15}\sqrt{2g} C_e \tan\left(\frac{\theta}{2}\right) h_e^{2.5} \tag{V.12}$$

where Q is the discharge; θ is the angle of the opening ($20^{\circ} \leq \theta \leq 100^{\circ}$), in degrees; and, h_e is the effective head. For h_e in m, Q is in m^3/s ; for h_e in ft, Q is in cfs. The calibration may have significant error for opening angles outside of the specified range.

The coefficient C_e is a function of θ and can be approximated as:

$$C_e = 0.02286\theta^2 - 0.05734\theta + 0.6115 \tag{V.13}$$

where θ is in radians.

The effective head is defined as $h_e = h_u + \Delta E_s - G_o + K_h$, where h_u is the measured upstream head relative to the lowest point (vertex) of the weir opening, and K_h is approximated as:

$$K_h \approx 1.395\theta^2 - 4.296\theta + 4.135 \tag{V.14}$$

where θ is in radians; and, K_h is in mm. The value of K_h is always less than 3 mm, so the difference between h_u and h_e is negligible, except for very small upstream depths. The above approximations for C_e and K_h are accurate representations of published laboratory data (Shen 1981).

Including the K_s term for submerged weir flow (Annex A), the flow rate function is expressed as:

$$Q = \frac{8K_s}{15}\sqrt{2g} C_e \tan\left(\frac{\theta}{2}\right) h_e^{2.5} \tag{V.15}$$

Based on the above equations, the weir setting (if adjustable, because in practice V-notch weirs are usually fixed-height structures) can be calculated as follows:

$$G_o = h_u + \Delta E_s + K_h - \left[\frac{15Q}{8K_s\sqrt{2g} C_e \tan\left(\frac{\theta}{2}\right)}\right]^{0.4}$$
(V.16)

V.5. Broad-Crested Weir

The procedures for complete theoretical calibration of broad-crested weirs is somewhat complex (mostly due to estimations of hydraulic losses), but if free-flow conditions can be assumed, a simpler energybalance calibration will usually yield a very accurate calibration. In the app, it is assumed that the hydraulic energy loss from the upstream channel section to the location of critical flow over the weir sill is negligible, which is often an acceptable premise, but presumes that critical flow does, in fact, occur — that the regime is free flow. When the downstream water surface elevation is above the sill elevation, it might not be free flow — you should check the simulation results for this condition to determine whether the free-flow assumption is correct, or possibly not.

Using an energy-balance approach from upstream to the critical flow section, and assuming free-flow conditions across the structure, the following equations apply for a broad-crested weir as a flow-control structure in a hydraulic simulation:



Specific-energy balance:

$$h_u - h_c - z + \frac{Q^2}{2g} \left(\frac{1}{A_u^2} - \frac{1}{A_c^2} \right) = 0$$
 (V.17)

where Q is the free-flow discharge $(m^3/s \text{ or cfs})$; z is the sill height (m or ft) relative to the upstream channel bed; A_c is the cross-sectional area $(m^2 \text{ or } ft^2)$ corresponding to critical depth, h_c ; A_u is the cross-sectional area corresponding to upstream depth, h_u ; and,

$$h_c = \frac{-b_c + \sqrt{b_c^2 + 4m_c A_c}}{2m_c}$$
(V.18)

where b_c is the base width of the sill (*m* or ft); and, m_c is the inverse side slope, also at the sill section. The cross-sectional area at the (presumed) location of critical flow over the sill is:

$$A_{c} = \left[\frac{Q^{2} \left(b_{c} + 2m_{c}h_{c}\right)}{g}\right]^{1/3}$$
(V.19)

The above equation for A_c is from the expression $F_r^2 = 1.0$ (critical flow), where F_r is the Froude number. The subscript c refers to the critical-flow section over the sill and the subscript u refers to the upstream section. The dependent variables here are Q and A_u . A_c and h_c are intermediate calculation values and are not of interest in the present context. Note that the critical-flow section is assumed to be a symmetrical trapezoid with m_c as the inverse side slope and b_c as the width of the sill.

V.6. Overshot Gate

So-called "overshot" gates, also known as "leaf" gates, are weirs with a hinged base and adjustable angle setting. In one version, steel cables on either side of the gate leaf are attached to a shaft above and upstream of the gate, and the shaft rotates by electric motor to change the setting. At large values of the angle setting the gate behaves like a weir, and at lower angles it approximates a free overfall (but this distinction is blurred when it is recognized that these two conditions can be calibrated using the same basic equation form).

These gates are manufactured by the Armtec company, Aqua Systems 2000 (*Langemann*), Rubicon, and others, and are easily automated. The calibration equations presented below for overshot gates are based on the data and analysis reported by Wahlin & Replogle (1996). The photograph below is an installed overshot (*Langemann*) gate in an irrigation canal in Utah.



The representation of overshot gates herein is limited to rectangular gate leafs in rectangular channel cross sections, whereby the specified leaf width is assumed to be the width of the cross section, at least in the immediate vicinity of the gate. This means that weir end contractions are suppressed.

The equation for both free and submerged flow is:

$$Q = K_s C_a C_e C_f G_w \left(\frac{2\sqrt{2g}}{3}\right) \left(h_u + \Delta E_s - G_o\right)^{n_f}$$
(V.20)

where Q is the discharge $(m^3/s \text{ or cfs})$; K_s is a coefficient for submerged flow (Annex A); G_w is the gate width (m or ft); G_o is the vertical distance to the gate lip (m or ft); h_u is the upstream depth (m or ft), taken to be equal to the effective head; and, ΔE_s is the upstream invert drop (m or ft). For h_u in m, Q is in m^3/s ; for h_u in ft, Q is in cfs.

The following figure shows an example side-view of an overshot gate with $\Delta E_s = 0.0$.



The exponent on the head term, n_f , should be near 1.5 for a rectangular channel, and in fact, Wahlin & Replogle (1996) consider it to be a constant at $n_f = 1.5$, so in the absence of custom calibration data, n_f should be set equal to exactly 1.5. The coefficient C_f is a non-orifice, free-flow coefficient and is included to allow for custom calibrations ("tweaking" the calibration based on field or laboratory data); in all other cases C_f should be set equal to exactly 1.0.

The coefficient C_e is approximated as:

$$C_e = 0.075 \left(\frac{h_u + \Delta E_s - G_o}{G_o}\right) + 0.602$$
 (V.21)

where G_o is the height of the gate sill with respect to the gate hinge elevation (*m* or *ft*); and, θ is the angle of the opening ($10^\circ \le \theta \le 65^\circ$), measured from the horizontal on the downstream side. The calibration may have significant error for opening angles outside of the specified range, and in the app, θ can take values from 1° to 90° . The value of G_o can be calculated directly based on the angle of the gate opening and the length of the gate leaf:

$$G_o = L\sin\theta \tag{V.22}$$

where L is the length of the gate leaf (m or ft), from the hinge up to the upper edge. Alternatively, solve for the angle θ :

$$\theta = \sin^{-1} \left(\frac{G_o}{L} \right) \tag{V.23}$$

The coefficient C_a is a function of the angle setting, θ , and described by a polynomial:

$$C_a = 1.0333 + 0.003848\theta - 0.000045\theta^2 \tag{V.24}$$

for θ in degrees.

The submerged-flow coefficient, K_s , is taken as defined by Villemonte (1947), but with custom calibration parameters for this structure type:

$$K_s = C_1 \left[1 - \left(\frac{h_d + \Delta E_s - G_o - \Delta E_d}{h_u + \Delta E_s - G_o} \right)^{n_f} \right]^{C_2}$$
(V.25)

where ΔE_d is the reach drop (Chapter III.1.3), and

$$C_1 = 1.0666 - 0.00111\theta \tag{V.26}$$

for $\theta < 60^{\circ}$; otherwise $C_1 = 1.0$ when $\theta \ge 60^{\circ}$. The exponent is defined as:

$$C_2 = 0.1525 + 0.006077\theta - 0.000045\theta^2 \tag{V.27}$$

Of course, when $h_d + \Delta E_s - G_o - \Delta E_d \leq 0$, the regime is free flow, whereby the coefficient K_s must be equal to 1.0. Also, the equation for K_s is for regulators — it takes a slightly different form for turnouts.

The structure setting, θ , can be calculated indirectly (by iteration), knowing the upstream and downstream depths, and the desired flow rate. It cannot be directly calculated because the coefficient C_a is a function of the opening, as is the flow regime.

The figure below shows an example side-view of an overshot gate operating under submerged-flow conditions, in this case with a negative upstream invert drop, ΔE_s , and a positive downstream reach drop, ΔE_d . Notice that the downstream water surface is above the gate lip. In general, ΔE_s and ΔE_d can be zero, negative, or positive, depending on the details of the installation.



V.7. Rectangular Gate

This structure type includes vertical sluice gates (slide gates) with a flat, horizontal lower gate edge, and a rectangular gate opening. Like other orifice, or "underflow" structures, rectangular gates can operate under free- or submerged-orifice flow regimes, as well as free or submerged non-orifice regimes. The app assumes that all water moving past the gate is underflow, meaning that no water spills over the top of the gate during a simulation.

V.7.1. Free Orifice Flow

Free, or modular orifice flow is assumed to prevail when the downstream momentum function corresponding to $0.61G_o$, where G_o is the vertical gate opening, is less than or equal to the momentum function value using the downstream depth (Annex B). Under these conditions the rating equation is:

$$Q_f = C_{df} \left(h_u + \Delta E_s \right) \sqrt{2g \left(h_u + \Delta E_s - C_c G_o \right)} \tag{V.28}$$

where C_{df} is the discharge coefficient (defined below); h_u is the upstream depth (*m* or ft); ΔE_s is the upstream elevation drop (*m* or ft); and, G_o is the vertical gate opening (*m* or ft).

The value of ΔE_s is positive if the gate sill is below the upstream bed elevation. This elevation drop is included in the equations to allow for short transitions on the upstream side of a gate structure, in which the gate invert is not at the same elevation as the canal bed if the bed slope were extrapolated downstream to the gate location.

The discharge coefficient is defined as:

$$C_{df} = \xi_{1f} \left(\frac{A_o}{h_u + \Delta E_s} \right)^{\xi_{2f}} \tag{V.29}$$

where A_o is the area of the gate opening $(m^2 \text{ or } ft^2)$, equal to the product of G_o and G_w , where G_w is the width of the gate opening (m or ft). If the orifice free-flow exponent, ξ_{2f} , is equal to 1.0, the term $h_u + \Delta E_s$ cancels and the discharge coefficient becomes a constant, equal to the free-orifice coefficient, ξ_{1f} .

In the **Calibrations** tab of the **Structures Data** sheet you must enter the values for the orifice freeflow coefficient, ξ_{1f} , and the orifice free-flow exponent, ξ_{2f} . If you do not know what exponent value to use, then set it equal to 1.0; in any case, that value should be near 1.0, but might be slightly lower or slightly greater than 1.0. The exact value should be determined from field or laboratory calibrations. The coefficient should be in the following range of values for free-orifice flow: $0.60 \leq \xi_{1f} \leq 0.90$. Higher values of ξ_{1f} tend to correspond to higher approach velocities in the flow upstream of the gate.

V.7.2. Submerged Orifice Flow

For submerged-flow conditions, the flow rate, Q_s , is proportional to the square root of the difference in upstream and downstream water surface elevations:

$$Q_s = C_{ds} \left(h_d + \Delta E_s - \Delta E_d \right) \sqrt{2g \left(h_u + \Delta E_d - h_d \right)}$$
(V.30)

where C_{ds} is the discharge coefficient; and ΔE_d is the reach drop (*m* or *ft*), described in Chapter III.1.3. In most cases, ΔE_d is either zero, or it is a positive value, indicating a drop in the bed elevation.



The submerged-orifice discharge coefficient is a function of the downstream depth, the bed elevation change, and the gate opening:

$$C_{ds} = \xi_{1s} \left(\frac{A_o}{h_d + \Delta E_s - \Delta E_d} \right)^{\xi_{2s}} \tag{V.31}$$

where A_o is the area of the gate opening $(m^2 \text{ or } ft^2)$, equal to the product of G_o and G_w , where G_w is the width of the gate opening (m or ft). As is the case for the free-flow equation, if the submerged-orifice exponent, ξ_{2s} , is equal to unity, the submerged-orifice coefficient becomes a constant, equal to ξ_{1s} .

In the **Calibration** sheet, you must enter the values for the submerged-orifice coefficient, ξ_{1s} , and the submerged-orifice exponent, ξ_{2s} . As for the free-flow equation, if you do not know what exponent to use, then set it equal to 1.0; nonetheless, that value should be near 1.0, but might be slightly lower or slightly greater than 1.0. The exact value is best determined from field or laboratory calibrations. The coefficient should be in the following range of values for submerged-orifice flow: $0.60 \leq \xi_{1s} \leq 0.90$. As for free flow, higher values of ξ_{1s} tend to correspond to higher approach velocities in the flow upstream of the gate.

The calibration of non-orifice flow conditions for this structure type is handled as per Cutthroat flume calibrations (Chapter V.12.2).

V.8. Circular Gate

In the app, a circular gate is the same as a rectangular gate in all respects except that the gate opening is circular in shape — but the gate leaf itself is rectangular, and the width of the gate leaf exceeds the diameter of the circular opening (otherwise it could not seal when closed). As with rectangular gates, circular gates can potentially operate under free or submerged orifice flow regimes, as well as free or submerged non-orifice regimes. Also, it is assumed that all water moving past the gate is underflow, meaning that no water spills over the top of the gate during a simulation.

The area of the gate opening, A_o , is calculated as:

$$A_o = \frac{D_o^2}{8} \left(\beta - \sin\beta\right) \tag{V.32}$$

where,

$$\beta = 2 \cos^{-1} \left(1 - \frac{2G_o}{D_o} \right) \tag{V.33}$$

in which A_o is the area of the gate opening $(m^2 \text{ or } ft^2)$; D_o is the diameter of the circular opening (m or ft); β is in radians (not degrees); and, G_o is the vertical gate opening (m or ft). Note that effectively, $0 \leq G_o \leq D_o$.

In the end-view figure below, the shaded area is an example of the opening (below the bottom edge of the gate leaf), through which water passes.



The free-flow equations are identical to Eqs. (V.28) and (V.29), and the submerged-flow equations are the same as Eqs. (V.30) and (V.31). Only the calculation of A_o is different.

V.9. Round Gate



Herein, a round gate is the same as a circular gate in all respects except that the gate leaf is circular in shape, and the diameter of the gate leaf exceeds that of the circular opening so that it will seal when closed. As with rectangular and circular gates, round gates can potentially operate under free- or submerged-orifice flow regimes, as well as free or submerged non-orifice regimes. Also, the app assumes that all water moving past the gate is underflow, meaning that no water spills over the top of the gate during a simulation. The photograph at left shows a (damaged) *Waterman* round gate in a canal.

The area of the gate opening, A_o , involves somewhat more complex geometry than that of the circular gate, and is calculated as:

$$A_o = r_o^2 \left(\beta - \cos\beta\sin\beta\right) - r_g^2 \left(\alpha - \cos\alpha\sin\alpha\right) \tag{V.34}$$

where,

$$\alpha = \cos^{-1}\left(\frac{y_1 - y_2}{r_g}\right) \tag{V.35}$$

$$\beta = \cos^{-1} \left(-\frac{y_2}{r_0} \right) \tag{V.36}$$

where r_g is the gate diameter (*m* or *ft*); r_o is the opening (pipe) diameter (*m* or *ft*), where $r_g > r_o$; and, y_1 and y_2 are as shown in the figure below, in which the shaded area is an example of the opening (below the bottom edge of the circular gate leaf), through which water passes. In equation form, y_1 and y_2 are defined as:

$$y_1 = G_o + r_g - r_o$$
 (V.37)

$$y_2 = \frac{r_o^2 - r_g^2 + y_1^2}{2y_1} \tag{V.38}$$



The free-flow equations are identical to Eqs. (V.28) and (V.29), and the submerged-flow equations are the same as Eqs. (V.30) and (V.31). Only the calculation of A_o is different.

V.10. Radial Gate

This structure type includes radial (or "Tainter") gates as calibrated by the USBR (Buyalski 1983). Also see Clemmens, et al. (2003) and Wahl (2005). The calibration of the gates follows the specifications in the USBR REC-ERC-83-9 technical publication, which gives calibration equations for free- and submerged-orifice flow, and corrections for the type of gate lip seal. In the app, coefficient C_{df} (free flow) and C_{ds} (submerged flow) are included to allow you to tweak the calibration, because the equations given by Buyalski (ibid) depend only on the dimensions and some installation details of the gates and cannot be adjusted according to other data that might become available. If you do not have custom calibration data for a particular radial gate, leave C_{df} and C_{ds} at their default value of 1.0. To use a custom-calibrated radial gate in mCanal, select the *Rectangular Gate* structure type and specify the calibration parameters for the radial gate.





The USBR calibration work was exclusively for orifice flow under radial gates, but *mCanal* considers the possibility of non-orifice flow, as with other underflow gate types. Buyalski (ibid) did not include ΔE_s or ΔE_d terms in the calibration work, but they are included in this app to generalize the application of this structure type in simulations. Thus, when $\Delta E_s \neq 0$ and $\Delta E_d \neq 0$, the calibration procedure given below could have significant error.

Three gate lip seal designs are included in the calibrations: (1) hard-rubber bar; (2) music note; and, (3) sharp edge. The gate lip seal is the bottom of the gate leaf, which rests on the bottom of the channel when the gate is closed. The discharge coefficients need no adjustment for the hard-rubber bar gate lip, which is the most common among USBR radial gate designs but do have correction factors for the other two lip seal types. These are given below for free and submerged orifice flow. In the app, you must select one of the three gate lip seal designs for each radial gate, so the appropriate calibration correction can be applied during a simulation. The following figure shows side views of the three USBR gate-lip designs.



The gate radius divided by the pinion height should be within this range: $1.2 \leq G_r/P \leq 1.7$. And, the upstream water depth divided by the pinion height should be less than or equal to 1.6; that is, $h_u/P \leq 1.6$. If these and other limits are observed, the accuracy of the calculated flow rate from Buyalski's equations are reported to be within 1% of the true flow rate.

Free, or modular, orifice flow is assumed to prevail when the downstream momentum function corre-

sponding to $C_c G_o$, where G_o is the vertical gate opening, is less than or equal to the momentum function value using the downstream depth. Under these conditions the rating equation is:

$$Q_f = C_{fcda} C_{df} G_o G_w \sqrt{2gh_u} \tag{V.39}$$

where C_{fcda} is the free-flow discharge coefficient; C_{df} is a user-defined coefficient; G_o is the vertical gate opening (m or ft); G_w is the width of the gate opening (m or ft); and, h_u is the upstream water depth (m or ft). Both C_{fcda} and C_{df} are dimensionless. Unless custom calibration data are available, set C_{df} equal to 1.0. The orifice free-flow exponent is ignored for this structure type.

 C_{fcda} is determined according to a set of conic equations as defined by Buyalski (1983) from a reported analysis of over 2,000 data points. These involve eccentricity, directrix, and focal distances, as explained in Annex C, along with the corresponding equation adjustment for submerged-flow conditions.

V.11. Generic Constriction

The generic channel constriction boundary condition is indeed very general because it includes most any local lateral and or vertical reduction in flow cross-sectional area. These include open-channel measurement flumes. Channel constrictions are non-gated, non-adjustable structures. The flow regime can be free non-orifice or submerged non-orifice. The free non-orifice equation is very similar to the free-flow equation for weirs, but the submerged-flow equation differs, as seen below. The flow rate for free flow is:

$$Q_{fn} = C_f W \left(h_u + \Delta E_s \right)^{n_f} \tag{V.40}$$

where Q_{fn} is the free non-orifice discharge $(m^3/s \text{ or cfs})$; W is the constriction width (m or ft); and, C_f and n_f are calibration parameters. The constriction width is usually taken as the width of the throat section — the narrowest part of the cross-section — assuming the throat is rectangular in shape.

The value of ΔE_u for constrictions is usually less than or equal to zero, meaning that the floor of the constriction is at or above the upstream channel invert elevation. However, ΔE_d is usually greater than or equal to zero, especially for flow measurement flumes. The flow rate equation for submerged flow is:

$$Q_{sn} = \frac{C_s W \left(h_u + \Delta E_d - h_d\right)^{n_f}}{\left(-C_2 - \log_{10} S\right)^{n_s}}$$
(V.41)

in which Q_{sn} is the submerged non-orifice discharge; n_f is as previously defined; and, C_s and n_s are calibration parameters. The value of C_2 is taken to be 0.0044 for Parshall flumes, and zero otherwise. The submergence is:

$$S = \frac{h_d + \Delta E_s - \Delta E_d}{h_u + \Delta E_s} \tag{V.42}$$

Note that Q_{sn} cannot be calculated for S = 0 because of a mathematical singularity; for S = 0 the flow regime is free non-orifice anyway. S is also undefined for $h_u = \Delta E_s$, whereby the discharge is zero because the floor of the constriction coincides with the upstream water surface elevation.

V.12. Cutthroat Flume

Cutthroat flumes (Skogerboe, et al. 1967a) originally had 24 standard sizes, in the same general range as Parshall flumes, but more recently the calibration has been generalized (Temeepattanapongsa, et al. 2013a, b) to include any size flume within specified limits, provided that certain dimensional ratios are respected. You are not required to enter any calibration data.



The Cutthroat flume calibration equations, as applied in the app, are based on Temeepattanapongsa, et al. (2013a, b). In using the calibrations described below, the ratio of W/L should be between 1/9 and 4/9, as in the original specifications by the flume developers, but it can be any value within this range. The converging and diverging wall slopes (1:3 and 1:6, respectively) should be as shown in the figure above.

V.12.1. Free Flow

Under free-flow conditions, the discharge, Q_f , is calculated as a function of the upstream water depth, y_u , and the W and L dimensions of the flume. In the following, all linear dimensions are in meters and flow rate is in cubic meters per second. For English units, the app makes the appropriate conversions to feet and cubic feet per second.

$$Q_f = C_f (h_u)^{n_f} \tag{V.43}$$

where,

$$C_f = 0.036 + 2.058 \, W^{0.979} \tag{V.44}$$

$$n_f = 1.514 \, L^{0.021} \, W^{-0.027} \tag{V.45}$$

Free-flow conditions are assured when the downstream water surface elevation is below the elevation of the flume floor. In this case, it is best to use the free-flow equations because they are somewhat more accurate. In other cases, the app uses the unified calibration equations, which accommodate both free and submerged flow through Cutthroat flumes.

V.12.2. Free and Submerged Flow

The procedure for the unified calibration (free and submerged flow) is based on specified or calculated values of W, L, h_u , and h_d , and a series of algebraic equations leading to flow rate, Q. The calculated flow rate could be for free flow, submerged flow, or transitional flow — the distinction is unnecessary herein. These equations are also for units of m and m^3/s , where h_d is the downstream water depth.

First, calculate the values of a, b, and c, each of which is based on the W and L dimensions:

$$a = -0.071 + \frac{0.026}{L} + 0.202 \left(\frac{W}{L}\right) \tag{V.46}$$

$$b = \frac{-0.093 + 0.019 \ln (L) + 0.227 \left(\frac{W}{L}\right) - 0.295 \left(\frac{W}{L}\right)^2}{1 + 0.579 \ln (L) + 0.659 \left(\frac{W}{L}\right)}$$
(V.47)

$$c = 1.004 + \frac{6.22(10)^{-3}}{L} - 0.034 \left(\frac{W}{L}\right)$$
(V.48)

where $\ln(L)$ is the natural logarithm of L.

Next, solve for h_{uf} , where S is the submergence, equal to h_d/h_u :

$$\frac{1}{h_{uf}} = a + b \frac{S}{\ln\left(S\right)} + \frac{c}{h_u} \tag{V.49}$$

Finally, the flow rate is based on h_{uf} :

$$Q = C_f \left(h_{uf}\right)^{n_f} \tag{V.50}$$

The coefficient and exponent values are defined by Eqs. (V.44) and (V.45) for free flow. Based on these equations, it is seen that there is actually a smooth transition between free and submerged flow through a Cutthroat flume, not an abrupt transition as implied by the Parshall flume equations. Also, laboratory research has shown that there is little or no hysteresis effect from free to submerged and vice versa.

The unified calibration is applied to non-orifice flow at gate structures (rectangular, circular and round).

V.13. Parshall Flume

You do not need to enter any calibration data for Parshall flumes because each of the standard sizes has accompanying parameters to define the hydraulic behavior for both free and submerged flow. These parameters include the transition submergence, S_t , which is the threshold ratio of h_d/h_u at which the regime changes from free to submerged, and vice versa. The transition submergence assumes no hysteresis effects, but in practice this is not usually an issue since most flumes operate under either free or submerged flow, only rarely changing from one regime to the other. The photo below is of a Parshall flume with stilling wells and upstream depth recorder in Taiwan.



Parshall flumes are designed for flow measurement in open channels, but they can be applied as boundary conditions in the app because (as measurement structures) they have well-defined relationships between depth(s) and flow rate. This type of structure only operates under non-orifice flow regimes: free non-orifice (fn) or submerged non-orifice (sn). The respective relationships among water depths and discharge are:

$$Q_{fn} = C_f W \left(h_u + \Delta E_s \right)^{n_f} \tag{V.51}$$

which is similar to the free-flow equation for weirs; and,

$$Q_{sn} = \frac{C_s W \left(h_u + \Delta E_d - h_d\right)^{n_f}}{\left(-C_2 - \log_{10} S\right)^{n_s}}$$
(V.52)

where W is the throat width, unique for each standard flume size, and the narrowest part of the rectangular lateral constriction in the flume; $S = h_d/h_u$; and, ΔE_s and ΔE_d have the same units as h_u and h_d (m or ft).

The value of ΔE_s is usually less than or equal to zero, meaning that the floor of the flume is at or above the upstream channel invert elevation. However, ΔE_d is usually greater than or equal to zero, especially for flow measurement flumes like the Parshall.

These are	the	Parshall	flume	$\operatorname{calibration}$	parameters	$_{\mathrm{in}}$	English	units,	for	water	depths	in	feet	and
discharge	in cu	bic feet p	er seco	ond:										

W (ft)	C_{f}	C_s	C_2	n_{f}	ns	$\mathbf{S_t}$
1.00	4.000	3.110	0.0044	1.520	1.080	0.620
1.50	4.000	2.947	0.0044	1.540	1.115	0.640
2.00	4.000	2.970	0.0044	1.550	1.140	0.660
2.50	4.000	2.888	0.0044	1.555	1.150	0.670
3.00	4.000	2.867	0.0044	1.560	1.160	0.680
4.00	4.000	2.775	0.0044	1.570	1.185	0.700
5.00	4.000	2.710	0.0044	1.580	1.205	0.720
6.00	4.000	2.642	0.0044	1.590	1.230	0.740
7.00	4.000	2.593	0.0044	1.600	1.250	0.760
8.00	4.000	2.550	0.0044	1.600	1.260	0.780
10.00	4.013	2.479	0.0044	1.590	1.275	0.800
12.00	3.958	2.445	0.0044	1.590	1.275	0.800
15.00	3.904	2.411	0.0044	1.590	1.275	0.800
20.00	3.850	2.378	0.0044	1.590	1.275	0.800
25.00	3.818	2.358	0.0044	1.590	1.275	0.800
30.00	3.796	2.345	0.0044	1.590	1.275	0.800
40.00	3.769	2.328	0.0044	1.590	1.275	0.800
50.00	3.753	2.318	0.0044	1.590	1.275	0.800

In metric units, the calibration parameters are as follows, for water depths in meters and discharge in cubic meters per second (transition submergence, S_t , exponents n_f and n_s , and the C_2 constant are all dimensionless, so the values do not change based on the system of units):

W (m)	$\mathbf{C}_{\mathbf{f}}$	C_s	C_2	$\mathbf{n_{f}}$	n _s	$\mathbf{S_t}$
0.305	2.261	1.758	0.0044	1.520	1.080	0.620
0.457	2.316	1.706	0.0044	1.540	1.115	0.640
0.610	2.344	1.740	0.0044	1.550	1.140	0.660
0.762	2.357	1.702	0.0044	1.555	1.150	0.670
0.914	2.372	1.700	0.0044	1.560	1.160	0.680
1.219	2.400	1.665	0.0044	1.570	1.185	0.700
1.524	2.429	1.645	0.0044	1.580	1.205	0.720
1.829	2.458	1.623	0.0044	1.590	1.230	0.740
2.134	2.487	1.612	0.0044	1.600	1.250	0.760
2.438	2.487	1.585	0.0044	1.600	1.260	0.780
3.048	2.466	1.523	0.0044	1.590	1.275	0.800
3.658	2.432	1.502	0.0044	1.590	1.275	0.800
4.572	2.399	1.482	0.0044	1.590	1.275	0.800
6.096	2.365	1.461	0.0044	1.590	1.275	0.800
7.620	2.346	1.449	0.0044	1.590	1.275	0.800
9.144	2.332	1.441	0.0044	1.590	1.275	0.800
12.192	2.315	1.430	0.0044	1.590	1.275	0.800
15.240	2.306	1.424	0.0044	1.590	1.275	0.800

As for the generic constriction, Q_{sn} cannot be calculated for S = 0 due to a mathematical singularity; for S = 0 the flow regime will be free non-orifice anyway. Moreover, S is undefined for $h_u = \Delta E_s$, whereby the discharge is zero because in this case the floor of the constriction coincides with the upstream water surface elevation.

V.14. Energy Balance

As described in Chapter IV.4.14, this boundary condition represents a *non-structure* location at the downstream end of a canal reach (and the upstream end of another reach), somewhat like a uniform-flow boundary, except in this case the specific-energy equation is applied. Any of the following can change abruptly at an energy-balance location in the app: cross-sectional channel shape or size, longitudinal bed slope, hydraulic roughness, or seepage loss rate. Thus, this boundary condition could involve only a change in the roughness, for example, without any difference in the section shape or size. You can use this boundary condition to effectively divide a single reach into multiple reaches with different roughness values, side slopes, and others.

The energy-balance boundary condition applies only to submerged-flow, non-orifice conditions, and the bed elevation drop at this location cannot be positive. That is, $\Delta E_d \leq 0$. Typically, $\Delta E_d = 0$.

The specific energy is balanced at the boundary as follows:

$$\left(h_u + \Delta E_d + \frac{V_u^2}{2g}\right) - K_{ru}\frac{V_u^2}{2g} = \left(h_d + \frac{V_d^2}{2g}\right) - K_{rd}\frac{V_d^2}{2g}$$
(V.53)

where h_u and h_d are the upstream and downstream water depths, respectively (*m* or *ft*); K_{ru} and K_{rd} are the upstream and downstream local loss coefficients, respectively; V_u and V_d are the mean upstream and downstream flow velocities, respectively (*m*/*s* or fps); and, ΔE_d is the drop in bed elevation (*m* or *ft*).

Substituting Q = VA, and solving for flow rate, Q, at the boundary:

$$Q = \sqrt{2g\left(h_u + \Delta E_d - h_d\right)\left(\frac{1 + K_{rd}}{A_d^2} + \frac{K_{ru} - 1}{A_u^2}\right)^{-1}}$$
(V.54)

where A_u and A_d are the upstream and downstream cross-sectional areas (of water), respectively $(m^2 \text{ or } ft^2)$. This equation for Q defines the relationship between flow rate and upstream and downstream water depths.

The local loss coefficient at the downstream end of the reach where this boundary condition is located is taken as K_{ru} in Eq. (V.54) because it is the upstream side of the boundary. Analogously, the upstream local loss coefficient in the next downstream reach is taken as K_{rd} . However, in practice, you would not need to specify separate values for these two coefficients — in the typical case, you would specify one of them and leave the other one as zero. Note also that the majority of the hydraulic head loss occurs on the downstream side of the boundary.



This boundary condition can cause difficulties with the numerical convergence of the solution to the system of equations during a simulation, especially when sudden changes in flow rate or water depth occur. It is recommended to check the *Simultaneous solution* box in the *Preferences* sheet, *Options* tab (Chapter I.4.1), whenever you have one or more energybalance boundary conditions in a system layout. This will help stabilize the solution at each

time step.

V.15. Uniform Flow

This non-structure boundary condition is handled according to the selected equation for calculating energy loss in each canal reach. This one is very simple to set up: there are no structure dimensions or calibration parameters to enter because the channel cross section and reach calibration parameters are used at uniform-flow boundaries. The choices for hydraulic energy loss (and uniform flow) are the Manning equation or the Chézy equation.

The uniform-flow boundary condition is generally a gross idealization of the true flow conditions at the downstream end of a reach, or at a bifurcation, because uniform flow is not usually observed during unsteady flow conditions in open channels. Also, the app bases uniform-flow boundary condition calculations on the upstream channel section and water depth, with no regard to the downstream depth. Thus, it is assumed that the uniform-flow condition is satisfied at that location in the system without performing any checks to ensure validity. This means that such boundary conditions are probably more valid at the ends of canal branches (terminus points) than at intermediate locations in the system.

Finally, hydraulic disturbances downstream of a uniform-flow boundary condition will not be transmitted upstream during a simulation, possibly causing unrealistic modeling results. This is because of the fact that the downstream water depth does not factor into the calculations.

V.15.1. Manning Equation

The Manning equation can be used to characterize uniform-flow conditions as follows:

$$Q = \frac{C_{me}\sqrt{S_o}}{n} \frac{A^{5/3}}{W_p^{2/3}} \tag{V.55}$$

where Q is the flow rate $(m^3/s \text{ or cfs})$; S_o is the longitudinal bed slope (m/m); n is the Manning roughness factor; A is the upstream cross-sectional area $(m^2 \text{ or } ft^2)$; and, W_p is the wetted perimeter (m or ft). The coefficient C_{me} is 1.0 for metric units, and 1.49 for English units. The area and wetted perimeter are defined according to the channel cross-section geometry.

V.15.2. Chézy Equation

The Chézy equation can also be used to characterize uniform-flow conditions:

$$Q = CA\sqrt{RS_o} \tag{V.56}$$

where C is the Chézy coefficient; and, R is the hydraulic radius (m or ft), equal to cross-sectional area divided by wetted perimeter (A/W_p) . Chézy C is approximately related to Manning n as follows:

$$C = \frac{R^{1/6}}{n} \tag{V.57}$$

This relationship between Chézy and Manning *roughness* factors is approximate because it does not account for the Reynolds number, nor the roughness height of the cross-section surface. In general, the Chézy equation more accurately represents uniform-flow conditions, compared to the Manning equation.

V.16. Undefined

This boundary condition has no calibration — it is a non-structure boundary. It can be used when a depth-discharge relationship is not needed at the upstream or downstream end of a reach. In such cases, either the flow rate is specified, or the water depth is specified at a reach end (or at a turnout structure), thereby eliminating the need for a separate boundary condition equation in the hydraulic simulation. You cannot combine this boundary condition with any other structures at a reach end or a turnout, and if you use it in a simulation, you must specify either the flow rate or water depth — this can be done through a specified value, or through a scheduling graph (Chapter VII). Note that you cannot specify both flow rate and water depth at any given location in an open channel — one or the other (or both) must be calculated.

V.17. Centrifugal Pump

Centrifugal (and turbine) pumps are hydraulically defined by a *characteristic curve* relating total dynamic head, TDH, to flow rate, Q_p . The pump characteristic curve is the same for a given pump design, size, and operating speed, and is independent of the system in which the pump is placed, wear and tear notwithstanding. On the other hand, a *system curve* represents the relationship between TDH and Q_p for the pumping installation, independent of the pump characteristics. That is, it depends on the static lift, hydraulic losses, and outlet pressure, but not on the pump. The system curve is parabolic (or approximately so), given that hydraulic losses are approximately proportional to the square of the flow rate.

The TDH and corresponding Q_p for a operating pump is referred to as the operating point. This point must be on the pump characteristic curve and must also be on the system curve; so, the operating point is necessarily the intersection between the two curves. The app attempts to ensure that there is only one possible operating point for a given TDH value, whereby TDH decreases monotonically with increasing flow rate along the pump characteristic curve; this is not always the case in practice with commercially-available centrifugal pumps.

In the app, pumps always include a valve in the outlet pipe, but you can choose a *null valve* (one of the default valve types) to simulate cases in which no valve is present. Thus, a pump is, essentially, taken to be a compound "structure" which includes both the pump and a valve.

V.17.1. Characteristic Curve

A look-up table with user-supplied values of TDH and Q_p is used to define the characteristic curve, and the app linearly interpolates to find TDH, given Q_p (or vice versa). It is assumed that the pump speed is constant, meaning that the characteristic curve does not change shape nor position with any of the operational conditions. However, you can edit the look-up table values during a simulation if desired. The app requires at least three points on the curve, with a maximum of 512 points per pump. Each point has a pair of flow rate, Q_p , and TDH values.

The app automatically sorts the characteristic curve data according to flow rate. During a cross check the app will examine the data for each pump curve and will issue warnings if the flow rate for any point is equal to the flow rate of the previous point. A warning will also be given by the app, after sorting the data by Q_p , if the TDH increases with an increase in flow rate for any of the adjacent curve points in the lookup table.

V.17.2. System Curve

The system curve is defined by calculating the TDH as the sum of the static lift, pipe hydraulic losses, valve loss, and velocity head. Thus,

$$TDH = h_z + h_f + h_v + \frac{V_p^2}{2g}$$
(V.58)

where h_z is the static lift (*m* or *ft*); h_f represents the pipe friction losses (*m* or *ft*); h_v is the valve's hydraulic loss (*m* or *ft*), if any; and, $V_p^2/2g$ is the velocity head. The static lift is defined to be the difference in upstream and downstream water surface elevations when the pipe outlet is submerged, or the difference in the upstream water surface elevation and the center of the outlet elevation for free discharge. These two cases are illustrated in the figures below.

A pump with a **submerged** outlet pipe:



In the figures, ΔE_s is the invert drop as specified for a pump in the *Structures Data* sheet (Section IV). Also, in these two figures, ΔE_s is <u>negative</u> because the pipe outlet is higher than the downstream bed elevation, so they are the opposite of a drop. The downstream reach drop, ΔE_d , is also <u>negative</u> in each of these two figures. This is the typical configuration for pump installations in canals.

Note that for pumps, ΔE_s (*invert drop*) refers to an elevation change on the downstream side of the structure, which is different than how ΔE_s is applied for other structure types.

The pump outlet is submerged when $h_d + \Delta E_s > 0$, and in these cases the static lift is:

$$h_z = h_d - \Delta E_d - h_u \tag{V.59}$$

A pump with an **unsubmerged** outlet pipe:



where ΔE_d is the downstream reach "drop," as specified in the *Reaches* tab of the **Reach Data** sheet (Chapter III).

The pump outlet is unsubmerged when $h_d + \Delta E_s \leq 0$, and in these cases the static lift is:

$$h_z = -\Delta E_s - \Delta E_d - h_u \tag{V.60}$$

where h_u and h_d are, of course, always non-negative.

V.17.3. Pipe Hydraulic Losses

The pipe hydraulic head loss is defined by the Darcy-Weisbach equation (Finnemore and Franzini 2002):

$$h_f = f \frac{L}{D} \frac{V_p^2}{2g} = \frac{8fLQ_p^2}{\pi^2 g D^5}$$
(V.61)

where f is the (so-called) friction factor; L is the pipe length (m or ft); D is the pipe inside diameter (m or ft); V_p is the mean flow velocity in the pipe (m/s or fps); and, Q_p is the flow rate through the pump (m^3/s or cfs). You can take entrance, elbow and other localized losses into account by adding an appropriate extension to the true pipe length. Thus, the length, L, that you enter may be greater than the physical pipe length.

The Darcy-Weisbach friction factor, f, is defined in the app by the Swamee-Jain equation, as seen in Eq. (III.10). Kinematic viscosity is calculated based on the specified water temperature for the reach in which the pump is located. Also, it is assumed that there is only one pipe diameter; if there are two or more diameters, you must figure out an equivalent length of pipe with a single diameter.

V.17.4. Valve Hydraulic Losses

The hydraulic effect of a valve can be simulated in the pump outlet pipe. In the app, all pumps have a valve somewhere in the outlet pipe, but you can choose a *null* valve type for those cases in which a valve is not desired. Valve head loss is defined according to the valve type and percent valve opening (Tullis 1984) as follows:

$$h_v = \frac{V_p^2 \left(1 - C_d^2\right)}{2gC_d^2} = \frac{8Q_p^2 \left(1 - C_d^2\right)}{\pi^2 g D^4 C_d^2} = \frac{8Q_p^2}{\pi^2 g D^4} \left(\frac{1}{C_d^2} - 1\right)$$
(V.62)

where h_v is the hydraulic energy loss past the valve (*m* or *ft*); and, C_d is a function of the gate opening. The value of C_d is determined by linear interpolation in tables of empirical data, given the percent opening.

Alternatively, the app obtains the required valve opening by interpolating with a given value of C_d , which would be determined according to a required valve head loss. This is done to simulate pump valve automation with a given flow rate requirement (remember, the app assumes the pump speed is constant).

For specified water levels and valve opening, the operating point is determined by the intersection of the pump characteristic curve and the system curve. In this case, the value of C_d is determined by direct interpolation in the valve lookup table for the known opening.

V.17.5. Defining the System Curve

The system curve can be defined as a polynomial function of TDH and Q_p as follows:

$$TDH = h_z + C_v Q_p^2 \tag{V.63}$$

in which,

$$C_v = \frac{8}{\pi^2 g D^4} \left(\frac{fL}{D} + \frac{1 - C_d^2}{C_d^2} + 1 \right)$$
(V.64)

where f is a function of Q_p (except for fully turbulent flow), and all other values are known. Equation (V.64) combines the head losses from Eqs. (V.61) and (V.62), plus the velocity head, divided by Q_p^2 .

In using a look-up table of Q_p versus TDH, a straight-line pump characteristic curve segment is defined as:

$$TDH = \alpha Q_p + \beta \tag{V.65}$$

The parameters α and β are determined by fitting a straight line through two adjacent points on the pump curve, as illustrated in the figure below. Thus, the operating point (Q_p , TDH) is the intersection between a straight-line segment of the pump curve and the parabola defining the system curve.

Note that in the following figure, the system curve begins at the origin $(Q_p = 0, \text{ and TDH} = 0)$, which is representative of a case in which there is no static lift. However, pump installations usually include a non-zero static lift, whereby the system curve is shifted vertically upward by h_z . This shifting of the curve is taken into account in *mCanal* calculations by inclusion of the h_z term in Eq. (V.63).



The app tries each straight-line segment of the pump characteristic curve until the intersection falls between the two adjacent flow rates on the pump curve, indicating a feasible operating point. If the static lift is greater than or equal to the pump curve TDH at zero flow (the *shut-off head*), the app computes a discharge of zero.

The intersection of the parabola that defines the system curve, and the straight line which passes through two adjacent points on the pump characteristic curve, is located directly by combining Eqs. (V.63) - (V.65):

$$Q_p = \frac{\alpha + \sqrt{\alpha^2 + 4C_v(\beta - h_z)}}{2C_v} \tag{V.66}$$

where the values of α and β are determined by trial-and-error to find a valid intersection between the system curve and appropriate pump characteristic curve segment.

Once Q_p is calculated from Eq. (V.66), the TDH is determined directly from Eq. (V.65). However, Cv is a function of f, which is a function of the Reynolds number, which in turn is a function of Q_p . Thus, the calculation of Q_p is necessarily iterative.



During a simulation, it is assumed that each pump is running continuously. Thus, the only mechanism to stop the flow through a pump is to make the valve setting equal to zero, wherein the shut-off head will be produced, corresponding to zero flow. However, flow through a pump will also stop if the upstream water depth becomes too low, at which point the app will assume that the pump has begun entraining too much air to continue pumping water. This occurs

when a pre-set threshold depth for pumps is encountered during a simulation, or when the calculated depth at the upstream end of a pump goes to zero during the iterative numerical solution process at

any given time step. If the calculated upstream water depth goes to zero, the threshold depth for which pumping ceases will be automatically increased to the depth from the previous time step (provided that it is greater than the current threshold depth); this is done to help avoid numerical instabilities and to make the simulations more realistic.

V.18. Valve

Valves are used at pump outlets and as regulator structures in pipeline reaches. A valve is hydraulically defined by a lookup table of percent valve opening versus discharge coefficient, C_d . These data are usually taken from measurements in hydraulic laboratories, and published data exist for several valve types and sizes. Calibrations depend not only on valve type, but on the brand and model, and in general, on the size. In the app, multiple valve types can be defined for application in a hydraulic simulation.

The coefficient of discharge is defined following typical hydraulic engineering convention:

$$C_d = \frac{V}{\sqrt{2g\Delta h + V^2}} \tag{V.67}$$

where C_d is the coefficient of discharge (dimensionless); V is average velocity (m/s or fps) at the valve location; and, Δh is the hydraulic head loss (m or ft), or head differential, across the valve. In the app, the average velocity, V, is defined as:

$$V = \frac{Q}{A_{pipe}} = \frac{4Q}{\pi D^2} \tag{V.68}$$

where Q is the flow rate in the pipe $(m^3/s \text{ or cfs})$ at the valve location; A_{pipe} is the pipe cross-sectional area $(m^2 \text{ or } ft^2)$, equal to $\pi D^2/4$; and, D is the pipe inside diameter (m or ft). Thus, only pipes of circular cross section are considered herein.

The value of C_d is assumed to be a function of percent valve opening. In reality, C_d can vary according to the Reynolds number, but this effect is usually small and is not considered in the app. The value of C_d can vary significantly with pipe diameter, types of connections or flanges, and other factors. This is why it is important to understand the conditions in which the C_d data were measured, and the limitations of their application.

Of course, percent valve opening can vary from 0 to 100%, whereas C_d values are restricted as follows:

$$0 \le C_d < 0 \tag{V.69}$$

When $C_d = 0$, the flow rate is defined as zero (which is logical, and also avoids a mathematical singularity). In fact, for all valve types and sizes, C_d must be equal to zero when the valve is completely closed, that is, when the percent valve opening is zero. This induces the shut-off head when the valve is at the pump outlet. The maximum C_d value can be nearly 1.0 for some valve types, but must always be less than 1.0. Except perhaps in very unusual cases, the maximum C_d value will occur at 100% valve opening. But the shape of the curve of C_d versus percent valve opening can vary significantly from one valve type to another.

Rearranging Eq. (V.67), two alternate forms can be obtained:

$$\Delta h = \left(\frac{1}{C_d^2} - 1\right) \frac{V^2}{2g} \tag{V.70}$$

and,

$$Q = A_{pipe} \sqrt{\frac{2g\Delta h}{\left(\frac{1}{C_d^2} - 1\right)}} \tag{V.71}$$

In practice, $\Delta h = h_f$, where h_f is the hydraulic head loss across the valve. It is noted that when $C_d^2 = 1/2$:

$$Q = A_{pipe} \sqrt{2g\Delta h} \tag{V.72}$$

which is what would be obtained for ideal flow through an orifice, based on the Bernoulli equation (Finnemore and Franzini 2002).

In the app, a centrifugal pump will have one value at the pump outlet. A pipeline reach may have one value at the upstream end, with C_{dUS} , and another value (possibly of a different type) at the downstream end, with C_{dDS} . During a simulation, linear interpolation is used in the lookup tables to determine the C_d value for a given value type and percent opening.

VI. Pump and Valve Calibration



As indicated in Chapter V, the calibration requirements for centrifugal pumps are much different than those of most other structure types. Pumps and valves both use look-use tables in the app to define their hydraulic behavior, and the data for both structure types can be edited from the same sheet.

Note that there may be a mixture of pumps and other structure types among the turnouts in a reach, but if any of the regulators at a location (upstream or downstream end of a

reach) are pumps, all of the regulators at that location must be pumps (otherwise an error message will be generated).

A layout configuration file can store data for up to 128 pumps and 128 valves. The tables of calibration data for either can have up to 512 pairs of values.

VI.1. Pump Calibrations

For pump data, click on the **Pump characteristic curves** button under *Type of data* at the upper left of the sheet. A pump characteristic curve defines the calibration of the pump; that is, the hydraulic behavior of the pump. This is largely analogous to the calibration of any other structure type, except that instead of defining the relationship between water depth and flow rate, it defines how a pump's flow rate reacts to different total hydraulic heads, referred to as total dynamic head, or TDH, values. Total dynamic head can be thought of as the resistance the pump faces in moving water within a piped system, both upstream and downstream of the pump location. When the resistance is lower, the flow rate will be higher.



When you create one or more pump characteristic curves, pumps will appear as as one of the boundarycondition options in the *Structures Data* sheet. However, there are no sample pump curves in the default system layout, so until you define a characteristic curve, you won't see pumps as an option for regulators and turnouts. The figure above shows some sample characteristic curve data on the sheet, including a graph view at the left. Note that the highest point on the curve for the example data is for zero flow: the TDH at this point is referred to as the *shut-off* head, which is the TDH when the pump is operating, but no flow is passing through it. This will be the case when the outlet valve is fully closed.

Most centrifugal pumps develop the maximum head at or very near zero flow, meaning the *shut-off* head is the highest point on the pump characteristic curve. In fact, this is usually a desirable feature of a pump curve because it means the pump performance will be stable. The alternative is maximum TDH at a non-zero discharge, which makes it possible to have two different operating points for the same head.

In many practical situations, the static lift will be the majority of the total dynamic head, but pipe hydraulic ("friction") losses may also be significant, especially when the pipe is long and or of small diameter. When pumping from one open-channel reach to another, and the pipe outlet is submerged, the static lift is simply the difference in water surface elevations. You specify the relationship between flow rate and TDH in a pump through a look-up table with pairs of Q_p and TDH values. These data are available from pump manufacturers, and sometimes from independent test facilities.

The app performs parabolic interpolations within the look-up table, so you usually will not need to enter more than 10 to 15 data pairs to have an acceptable approximation of the pump characteristic curve. Each pump look-up table must have at least three Q_p -TDH data pairs, and the lowest Q value must be zero (with a corresponding TDH that defines the *shut-off* head). TDH must not increase with an increase in Q_p .

The app automatically sorts the data by flow rate, Q_p , from low to high, and eliminates any entries with the same Q_p value.

The last two rows of data points (with the largest two Q_p values) will define the final straight-line segment on a graph of TDH versus Q_p , and this segment will be extrapolated beyond the point defined by the last data pair (i.e. to an even higher Q_p value) during a simulation, if necessary. Also, if the first data pair does not have $Q_p = 0$, the segment will be extrapolated to a flow rate of zero during a simulation, if necessary.

VI.1.1. Entering and Editing Data

There are two sets of +/- buttons on this sheet: one is for data sets and the other is for table rows. The name of the currently-selected pump (if any) is seen in the box just below the *Pump data set* list. To create a new pump characteristic curve, enter a unique pump data set name, and then click the + (plus) button for **Data sets** at the upper right . Next, type in data pairs for TDH and Q_p in the table - click on the + (plus) button (the one for *Table rows*) below the table to add more rows, if necessary. Each non-blank row of the table must contain a pair of values.

To delete rows from the table, first select one or more of them, and then click the - (minus) button below the table, or press the **delete** key on your keyboard. You can select multiple rows in the table, individually with the \Re key and in blocks with the **shift** key.

The **Table rows** - (minus) button does not do anything unless one or more rows in the table have been selected. You can can also click on **Select all rows** or **Unselect all rows** from the popup menu.

To delete an entire characteristic curve, select one from the *Pump data set* list at the upper right and click the - (minus) button for **Data sets**, above the table. The - (minus) button is disabled if there are no pump data sets. Note that deleting all rows of data in the table will not delete the pump curve – you must click on the - (minus) button for **Data sets** to accomplish this.

You can change the name of an existing pump curve by entering a unique name in the box below the list and clicking the **change** button.

VI.1.2. Writing Data to a Text File

Right-click over the graph view to see a popup menu (seen at right). It is the same list of menu commands for both pumps and valves, from which you can copy and paste data, and read from and write to text files. These operations are described in more detail below.

You can write the data from a pump characteristic curve look-up table to a text file, and you can input data from text files into a look-up table. This can facilitate the transfer of data between mCanal and other applications. For example, you might have data in a text file and you want to have the app read it without having to retype all the values into the grid component. Or, you may wish to output a look-up table from the app to a text file so that it can be imported into a spreadsheet application.

Right-click over the curve view and select **Write data** from the popup menu. The information will be written to a file in the same folder as the simulation results data (Chapter IX.3.1) with a file name something like this: "Pump Data (index 3).txt". If a file with the same name in that folder already exists, it will be overwritten, so you might want to rename the file to something more specific. The outputted text file will look something like the sample shown in the table below when opened into a text editor or similar app, in which the three columns of numbers are separated by commas.

mCanal: Pump Data
Layout file: /users/HSimpson/Documents/mCanal
Columns: Row, Flow rate, TDH
Pump name: Centrifugal ACME (key: 1)
1, 0.000, 10.000
2, 0.020, 9.800
3, 0.040, 9.400
4, 0.060, 8.800
5, 0.080, 7.700
6, 0.100, 6.000
7, 0.110, 4.700
8, 0.120, 3.000
End of Listing

Cut Copy Paste Write data Write all data Read data...

Select all rows Unselect all rows The *index* (in the text file name) and *key* values are internally managed by the app to keep track of the different data sets, even if you change the pump name, add new pump data sets, or delete others. This is because you may already be using one or more sets of pump curve data at locations within a system layout, and so they will still be recognized by the app, regardless of the name given to the pump data.

To write all of the pump-curve data sets (if you have more than one) to a single text file, select **Write all data** from the popup menu.

VI.1.3. Reading Data from a Text File

When you read from a text file, you import data from that file into a pump characteristic curve look-up table. Right-click over the graph view and select **Read data** from the popup menu. A window will come up and allow you to select a text file from the available drives and other storage media to which your computer has access. You can choose from files that do not have the "txt" filename extension, but they must be plain text files.

The data in a text file for a pump look-up table must be organized into at least two columns: the left column has the flow rates and the right column has the corresponding total dynamic head values. Thus, each row of data should have two numeric values, separated from each other by a comma on each row. The end of each row must have a *return* and or *line-feed* character to demarcate the end of the row and beginning of the next one - if it looks like the table below when opened in a text editor, it will have one or both characters at the end of each row, and that is fine.

0.000,	10.000
0.020,	9.800
0.040,	9.400
0.060,	8.800
0.080,	7.700
0.100,	6.000
0.110,	4.700
0.120,	3.000

You can also import data from a text file that was previously created from mCanal using the Write data popup menu item. If there are more than three comma-separated values in the file, the app will ignore all be the right-most two values – this means that you can read an outputted text file from the app, even though the first column has row numbers (they are ignored when reading).

VI.1.4. Copying and Pasting Data Sets

If you only want to copy a look-up table from one pump to another, it is more convenient to use the **Copy** and **Paste** commands from the popup menu. You can also use **Cut** if you want to move all of the table data to another pump by subsequently pasting it, but this operation also deletes the pump curve from which the data were cut.

Copying and pasting also works between Apple Numbers, or Microsoft Excel, and mCanal. However, if you copy from Numbers or Excel to a pump curve, be sure to copy only two adjacent columns of data (numeric values), otherwise the operation may not produce the desired results.

Copying and pasting operations are exactly the same for valve calibration data in this sheet.

VI.2. Valve Calibrations

For valve data, click on the Valve calibrations button under Type of data at the upper left of the sheet.

To add a new valve type, enter a unique valve name and click the "+" (plus) button. To delete a valve type, select it from the list and click the "-" (minus) button. You can change the name of a valve type by entering the new name and clicking the **change** button.



The ordinate range on the graph view for valve calibrations is always $0 \leq C_d \leq 1$, and the abscissa scale always goes from 0 to 100%.



When entering value C_d data, you must supply a full range of openings, from a minimum of 0% to a maximum of 100%. Valve C_d values usually increase monotonically with increasing valve opening. When this is not the case for a particular valve, you should be skeptical of the calibration and or data. Note that if you delete valve types from the list, you may have to go back and check the valve subtypes (Chapter IV) in the app for layout configuration files that contain valve structures. This should be done for both regulators and turnouts.

You can also read value C_d values from a text file or write the data to a text file, just as described above for pump data sets. Range-checking is automatically performed by the app so that $0.0 \leq C_d \leq$ 0.9999.

VII. Scheduling Operations



You can program the operation of control structures (regulators and turnouts) and specify time-dependent flow rates and water depths through scheduling graphs. All of the different kinds of scheduling graphs are based on two-column tables in which the left column (abscissa values on the graph view) is always elapsed time during a simulation. Thus, when the right column (graph ordinate) is flow rate, it is a hydrograph, which is a plot of flow rate versus time. You can use time graphs to specify how flow rates, depths, and or adjustable control-

structure settings change with elapsed time during a simulation. In summary, through the use of these graphs, you can pre-program certain operational characteristics.

Select **Parameters, Scheduling** from the main menu, or click on the corresponding toolbar icon, to display the scheduling sheet. The figure below shows an example of the sheet with flow rates for a turnout.



Recall that "setting" usually refers to the vertical opening of an adjustable control structure. When multiple changes are desired during a simulation, and they are known ahead of time, it is more convenient to use scheduling graphs than to interrupt the simulation manually (or through the application of simulation pause times) and type in those changes.

Simulations always begin at time zero, and the times that you enter in scheduling graphs must be based on this starting point.

When you save a configuration data file, all of the scheduling data are stored along with layout, dimensional, calibration, and other information (Chapter I.5).

The *Scheduling category* list allows you to select from various data categories, depending on the node type selected under the *Node name* list. Bifurcation nodes never have scheduling graphs – instead, apply scheduling data to the reach at the downstream side of the bifurcation. Confluence nodes are a special kind of Reach node and have all the same scheduling graph options as reaches.

Nevertheless, Confluence nodes are never identified as Terminus nodes because the downstream water depth is calculated, not specified. The scheduling-graph options under the **Scheduling category** list are as follows:

Reach nodes:

- Reach depth (US or DS)
- Reach flow rate (US or DS)
- Regulator setting (US or DS)
- Regulator flow rate (US or DS)
- Turnout setting
- Turnout flow rate
- Turnout downstream depth
- Terminus downstream depth

Source nodes:

- Source flow rate
- Source upstream water depth

Flow rates will be in m^3/s or cfs, and water depths in m or ft. But structure settings can be in m or ft, opening angle (degrees), or percentage opening, depending on the type of boundary condition.

The parameter "Turnout downstream depth" (for reach and confluence nodes) refers to the specified depth of water on the downstream side of a turnout, which is outside of the simulated hydraulic system. This depth is relative to the bottom of the channel at the turnout location. See Chapter V.6 for further explanation of this parameter. "Terminus downstream depth" is analogous to "Turnout downstream depth," except at the end of a branch with no downstream layout nodes – this pertains to one or more regulator structures at the downstream end of a terminus reach, to specify the depth at the downstream side of the structure(s). Turnout downstream depth is only provided as a scheduling-graph option for those reach nodes that are identified as terminus nodes.

For scheduling graphs, primary and secondary source nodes both have the same two options: flow rate and upstream depth. Upstream depth at a source node needs to be specified because it is outside of the simulated system – otherwise, the flow rate needs to be specified. The same applies to turnouts, except that the downstream depth is specified because for turnouts, the downstream depth is what is outside of the simulated system.

The table at the upper right of the sheet is where you enter and edit data values for a scheduling graph. This component functions similar to a spreadsheet app, with multiple rows but only two columns. The left column has time values, and the right column contains flow rates, depths, or structure settings. A label above the right column indicates the type of value and units. Note that you cannot do anything in the grid component if you have selected a category and parameter that do not exist for the current layout (e.g. if you select turnout flow rate when the selected reach has no turnouts).

When you have entered some data, you can see a plot of the values in the graph view at the lower part of the sheet. Time will be on the abscissa and the flow rate, depth, or setting will be on the ordinate, and the data are always arranged in chronological order before the plot is drawn.
Right-click over the graph view to display a popup menu (shown on the next page) with various options: showing or hiding symbols in the graph view, changing line and symbol colors, changing line thickness, and others. Symbols are optionally shown at each data point, but are always connected by straight lines, which can be either orthogonal (*staircase*) or sloping.

The graph-view ordinate is automatically scaled to match the data range in the right column of the table, but if all of the ordinate values are the same, the plot will not be displayed. The abscissa is also scaled by the app, according to the time values that you specify in the left column of the table. Abscissa scaling is automatically set to seconds, minutes, hours, or days.

Line-width options include *Thin*, *Medium*, *Thick*, or *Heavy*. You can change line and symbol colors from the same popup menu, and you can show or hide the symbols that correspond to the data pairs in the table. You can also export the data to a text file to use graphing and data analysis features in other applications, such as spreadsheets, as discussed below in "Using Text Files." If there is more than one scheduling graph, the **Prev** and **Next** buttons will be enabled, and you can click them to scroll through the graphs.



VII.1. Data Validation

When a configuration data file is read into mCanal the scheduling values (if any) are checked for consistency and validity to eliminate potential problems. For example, any graph with no valid data points is ignored upon reading the file, and any scheduling graph with an invalid system layout node reference is ignored. Thus, if you delete a node from the system layout, all scheduling graphs that are referenced to that node will be ignored upon reading that file, and when you save the file, those scheduling data will be lost.

As indicated above, the data are also arranged in chronological order by the model – this is done automatically. When sorting the data, the app will also eliminate any data pairs with duplicate time values from the table for a given scheduling graph.

VII.2. Entering Data

Going back to the table, there are many features for entering, editing, displaying, and applying the scheduling data. Each of the two data columns has a number of cells that can contain data values. When there are more than a few rows of data, the table values can be vertically scrolled.

As you scroll through the values in the table, a vertical light-gray line can be optionally (**Show graph highlight** from the popup menu) displayed at the corresponding point in the graph view. Times are always shown as days, hours, minutes, and seconds, in the left column of the table, and they are always non-negative. However, the values in the right column can be negative for some kinds of scheduling graphs.

Each scheduling graph can have either sloping or staircase lines that connect the data points in the table. The figure above gives an example of a staircase scheduling graph, and the figure below shows the same data with sloping lines. This option changes the way that the data are displayed in the graph on the sheet, but more importantly, it affects the way the scheduling values are interpreted (based on elapsed time) when applied in a simulation.



If you choose the *Sloping lines* option, mCanal linearly interpolates during a simulation to determine an ordinate value for a specific elapsed time. But if you select the *Staircase lines* option, ordinate values change abruptly after the specified time value, in a staircase fashion, as seen in the graph view, where all line segments are either horizontal or vertical.

The rows in the grid component will automatically extend downward as you enter data. You can use the arrow keys to move to a cell in the grid component, or you can click over a cell to select it. Then you can type in a data value and press the **Tab** key, the up or down arrow keys, or the **Enter** key. If you have a list of values to enter, you may find it convenient to type in each value and press the **Tab** key to move down through the list. Or, if you want to enter all of the time values first, just press the down arrow key after each entry. Edit the value in a cell by clicking on the cell, then clicking once again to enter the edit mode. After editing, press the **Tab** key, or the up or down arrow keys.

You can select all of the table rows or unselect all rows from items at the bottom of the popup menu. You can select/unselect individual rows in the table by holding down the \Re key while clicking, or blocks of rows by holding down the **shift** key and clicking. Add new rows of data by clicking on the + (plus) button, and delete selected row(s), if any, by clicking on the - (minus) button. You can also delete selected rows by pressing the **delete** key. If you delete all rows of data in the table, the currently-displayed scheduling graph will also be removed.

Finally, from the popup menu you can choose the **Sort data** item to arrange the data pairs chronologically in the grid component. This is only necessary if you have not entered time values chronologically in the left column. Usually, mCanal will automatically arrange the data.

The only way to undo scheduling-graph editing changes is to click on the **Cancel** button at the bottom of the window, but remember that this action will undo all changes since the scheduling sheet was last displayed.

VII.3. Graph Mirroring

The contents of some scheduling graphs will be automatically mirrored when you enter data. These kinds of graphs include reach flow rates, regulator settings, and regulator flow rates. Mirroring occurs only at serial reach connections, which means reach interfaces along a given branch in the system layout. Thus, this applies only to branches with two or more reaches. Mirroring does not take place at the downstream end of a split reach.

Regulator structures at serial reach connections are shared by two reaches, except at the downstream end of a reach with a split (Chapter II.4). That means that the downstream regulator(s) at a serial reach connection are exactly the same regulator(s) which are found at the upstream end of the next downstream reach, so regulator flow rates and settings at one side of the connection must be the same as those on the other side. This is automatically ensured by the app. You can verify this by specifying regulator data at such a reach connection, then moving to the next upstream or downstream reach to find the same scheduling data values.

The other kind of scheduling graph which is subject to mirroring is reach flow rate. The flow rate leaving the downstream end of one reach must be exactly the same as the flow rate entering the next downstream reach if it is a serial reach connection (assuming the upstream reach does not end in a split).

Graph mirroring will involve the creation of a new graph if it does not already exist at a serial reach connection, or will modify the existing data set. All mirroring actions are taken care of by the app. See Chapter IV.1 about automatic mirroring for regulator configuration data.

VII.4. Copying and Pasting Data

You can copy and paste scheduling data to and from other applications, and among graphs within the scheduling sheet. For example, you can copy scheduling data from one reach to another, or from one structure to another. These commands are available through the popup menu over the graph view. **Copy** is enabled on the menu when there are one or more data points in the table. **Paste** is enabled on the menu when text data are available on the macOS global pasteboard.

Copying means taking the data currently loaded (or entered) in the grid component and duplicating the values, in a text format, on the pasteboard. The data are copied in successive rows of text, with two data values on each row (time and ordinate values), separated by a tab character. Pasting means taking text data from the macOS global pasteboard and replacing any information already found in the table. In both cases, the time values are integers, interpreted as elapsed seconds.

You do not need to highlight the data values before copying or cutting to the pasteboard; all of the currently displayed data values will be copied to the pasteboard. When you use the **Copy** command, the data remain in the pasteboard until they are replaced by another cut or copy command from mCanal or any other application.

Scheduling data that are copied will not contain any information about formatting, with the exception of the line connection type (*sloping* or *staircase*). The line connection type is included by appending the word "sloping" or "staircase" to the end of the data, and it will be applied when pasting the data to another scheduling graph. You will see this below the last row of data if you paste scheduling data into a spreadsheet application.

If you copy data from a spreadsheet application, the data must be in two columns (time on the left and flow rate, depth, or setting on the right) and one or more rows. Include the word "sloping" or "staircase" just below the last row of data – otherwise, set the option in the app after pasting the data. When you copy the data from the spreadsheet to the pasteboard, the pairs of data on each row will be separated by a tab character, and the data can be directly pasted into the table of the scheduling sheet. If you copy data from a word processor or text editor to the pasteboard, and will be pasting into a scheduling graph, be sure to place a tab character after the time value on each row of data. Otherwise, the data will be misinterpreted when pasted into the table.

VII.5. Using Text Files

You can write the data from scheduling graphs to text files and you can input data from text files into a scheduling graph. This can facilitate the transfer of data between mCanal and other applications. For example, you might have some data in a text file and you want to have the app read it without your having to retype all the values into the scheduling graphs data entry window. Or, you may wish to output a scheduling graph from mCanal to a text file.

VII.5.1. Writing to Text Files

Right-click over the graph view and select **Write data** from the popup menu. The filename extension will be "Scheduling data (index 2).txt," or similar, and it will be written to the same folder as specified for simulation results. If a file with the same name in that folder already exists, those data will be overwritten – to avoid this, be sure to change the file name or move it to a different folder immediately after writing the data. The outputted text file will look something like the example shown below when opened into a text editor or word processor. Columns of values are separated by commas.

mCanal: Scheduling Data Layout file: /users/Someone/Documents/mCanal

Data category: Reach flow rate Reach location: downstream Layout node type: reach Layout node name: Main canal

Row	Time (s)	Ordinate (m)
1,	0,	0.000
2,	1800,	0.300
3,	4800,	0.571
4,	6180,	1.204
5,	8100,	1.250
6,	14100,	1.480
7,	18900,	1.090
8,	21780,	0.875
9,	23580,	0.505
10,	28740,	0.292

End of Listing

You will not necessarily know what kind of scheduling data they are by simply looking at the text file unless you recognize the values in the two data columns. Similarly, the units are not specified for the values in the right column – this depends on the units option selected in the *Preferences* sheet. The left column always contains elapsed time values (in seconds), and the right column has data values for flow rate, water depth, or structure setting.

Choose the **Write all data** popup menu command to send all scheduling graphs to a single text file with the name "All scheduling data.txt." Each set of data will be preceded by descriptive header information.

VII.5.2. Writing Metadata to a Text File

There can be many scheduling graphs in a configuration file, so you can create a list of scheduling header information to a text file. The list will show metadata for all scheduling graphs in a given configuration data file, even some of which might not visible or accessible. A scheduling graph can become hidden when you make structure (including regulators and turnouts) and or reach deletions to configuration data after having entered scheduling graph data. For example, if you add a turnout structure to a reach, define a scheduling graph (flow rate or setting) for that turnout, then delete the structure, the scheduling graph will still be in the configuration data file, but will not be accessible. However, if you were to subsequently add one or more turnouts to the same reach, the hydrograph will again be accessible.

Right-click over the grid component of the window and select **Write metadata** from the popup menu. This will create a file called "Scheduling metadata.txt" in the same folder as for the results output files. The following shows an example of the contents of a list of scheduling graph headers:

mCanal: Scheduling Metadata Layout file: /users/BSimpson/Documents/mCanal

Number	of scheduling graphs: 3	

Num	Kind	Rows	Lines	Node name	Reach end	Gate
0	Source flow rate	12	sloping	Primary Source	n/a	0
1	Reach flow rate	18	staircase	Reach 1L	Downstream	-1
2	Regulator flow rate	8	staircase	Reach 2R	Upstream	2
3	Turnout setting	25	sloping	Reach 2R	n/a	4

End of Listing

VII.5.3. Reading from Text Files

When you read from a text file, you import data from that file into a scheduling graph. Right-click over the graph view and select **Read data...** from the popup menu. Another window will come up and allow you to select a text file. The data will be checked for duplicate time values and then they will be sorted chronologically before being loaded into the table and displayed in the graph view.

As mentioned above, the time values must be provided in seconds in the text file. But when they are displayed in the app, they will be shown as days, hours, minutes and seconds. Following are the contents of a sample scheduling-data text file, with time (s) in the left column, and setting (or other) in the right column:

0,	0.1
60,	0.12
133,	0.145
207,	0.107
483,	0.95
592,	0.9
750,	0.1

These are the results in the app interface after reading the data, in which the time values are automatically formatted as minutes and seconds (in this example, days and hours are zeroes in all rows):

0:00:00:00	0.1
0:00:01:00	0.12
0:00:02:13	0.145
0:00:03:27	0.107
0:00:08:03	0.95
0:00:09:52	0.9
0:00:12:30	0.1

The data in text files for scheduling graphs must be organized into two columns: the left column has the time values (in seconds) and the right column has the flow rates, depths, or settings. Thus, each row of data must have (at least) two numerical values, separated from each other by a comma. If there are more than two columns of data values per row, on any given row, the app will ignore all values except those of the right-most two columns. For this reason, you can read an outputted scheduling-data text file, because the first column with row numbers will be ignored.

VII.6. Using Scheduling Data in a Simulation

One or more different scheduling graphs can be used during a hydraulic simulation. When a scheduling graph is used, the current elapsed time determines the entry location into the scheduling graph. This entry point will not necessarily coincide with a particular point on the graph, so linear interpolation is performed between two adjacent points when the line connection option is "sloping." If the line connection is "staircase," no interpolation is performed – in this case, the ordinate value of the most recent scheduling graph data value (equal or prior to the current elapsed time) is used.

If the elapsed simulation time exceeds the largest abscissa value in the table, the ordinate value of the last data point in the graph is applied. For example, if the last data point has a time value of five hours and an ordinate (flow rate, depth, or setting) value of 2.000, the scheduling graph will return that same value of 2.000 for elapsed simulation times of five hours or greater. Similarly, if the elapsed simulation time is less than the smallest time value in the graph, the first ordinate value will be used until the elapsed time exceeds that smallest value. The figure below shows a sample hydrograph with sloping line connections that starts at one hour and ends at five hours, with an interpolated value of $4.5 \text{ m}^3/\text{s}$ at an elapsed time of 2.5 hours.



You can disable the application of individual scheduling graphs during a simulation without deleting the corresponding data values. This is accomplished by unchecking the appropriate *Use graph* box in the *Flows and Depths* or *Gate Operations* sheets (Chapter VIII.4). Unchecking any of those boxes does not affect the scheduling graph itself – it only tells the app to ignore the graph during a simulation.

VIII. Numerical Parameters



Select Edit, Numerical Parameters from the main menu to display this sheet. There are many calibration and other numerical parameters that can be specified prior to a hydraulic simulation. The parameters in this window are all *global* in scope, meaning that they are for the entire system layout, not for individual layout nodes, reaches, or structures. Calibration data for individual elements of a layout are specified in the *Reach Calibration* and *Structure Calibration* sheets (Chapters III.3 and V).



It is not recommendable to change these numerical parameter values without a thorough knowledge of how the change(s) might affect a hydraulic simulation, and what the change(s) might mean in terms of the numerical solution of the governing mathematical equations. That being said, tweaking the values in this window can mean the difference between numerical convergence and divergence of the solution to the hydraulic equations, and between success

and failure of a simulation. Thus, a knowledgeable user can modify these values to achieve a successful simulation for specific conditions when the simulation might otherwise fail with the default numerical parameter values.

The Numerical Parameters sheet has three tabs, each of which is described below.

VIII.1. General

VIII.1.1. Time Step

You can specify the time step used in a simulation by entering a value in minutes and seconds. For example, to use a time step of 5 minutes, enter a value of "5:00." Or, to use a time step of 30 seconds, enter a value of "0:30." Larger time steps may be appropriate when only slight and infrequent operational changes (e.g. flow rates, structure settings, and others) occur during a simulation, but when frequent and large changes occur, a smaller time step is justified.

General Limits	Transitions				
Time step (min:s)	Gradually-varied flow profiles				
h:00	Euler method				
Spatial weighting					
0.600 Calculated	Runga-Kutta 4th & 5th order				
Temporal weighting	Linear system solution method				
0.600	Caussian elimination				
Water density (kg/m3)	Gaussian einmation				
1,000.000	Gauss Jordan				
	LU decomposition				
General numerical parameter values for hydraulic calculations.					
Use Defaults	OK Cancel				

VIII.1.2. Spatial Weighting

The symbol ϕ is used for spatial weighting in the implicit numerical solution of the governing hydraulic equations. It is part of a type of four-point implicit solution method and is used to help account for the spatial (in the direction of flow) nonlinearities that exist in unsteady flow. It is also used to balance numerical accuracy with numerical stability. The typical range is $0.50 < \phi < 0.65$.

When the **Calculated** box is checked, the app will determine the value of ϕ for each computational cell (Chapter III.3.5) in a pool, at each time step of a simulation, according to the following equation:

$$\phi = 0.5 + \frac{(WSE_{us} - WSE_{ds})}{5\Delta x} \tag{VIII.1}$$

where WSE is the water surface elevation (m or ft); the subscripts "us" and "ds" refer to the upstream and downstream ends, respectively, of a given computational cell (Chapter III.3.5); and, Δx is the length of the computational cell (m or ft). A computational cell is defined by four computational nodes in the x-t plane.

Note that in Eq. (VIII.1), ϕ is equal to 0.5 whenever the slope of the water surface is zero, corresponding to zero flow. This is especially important when starting a simulation from a zero-flow, steady-state condition. Also note that the calculated value of ϕ is not likely to exceed 0.70, even in regions of very high hydraulic gradients, such as near the advancing front of a pool over a dry channel bed. Finally, observe that, according to Eq. (VIII.1), the calculated value of ϕ will necessarily be less than 0.5 when reverse flow exists, that is, when $WSE_{us} < WSE_{ds}$.



You should consider leaving the "Calculated" box in the checked state unless you are doing research and or comparing model results to another hydraulic model and need to specify a fixed value of ϕ during a simulation. But, when the box is unchecked and the value of ϕ is not set at 0.5, the steady-state startup condition will be erroneously determined to be unstable by the model when the flow rate is zero and the reach is full of water (at full supply level).

VIII.1.3. Temporal Weighting

The symbol θ is used for temporal weighting in the implicit numerical solution of the governing hydraulic equations. It is also part of a type of four-point implicit solution method and is used to help account for the temporal (time) nonlinearities that exist in unsteady flow. It is also used to balance numerical accuracy with numerical stability. Typical ranges are $0.55 < \theta < 0.65$.

VIII.1.4. Water Density

The water density value is used in mCanal to calculate the wave speed in pipeline reaches. The value in metric units for plain water is approximately 1,000 kg/m^3 , and in English units this value is taken to be weight per unit volume. Thus, in English units it is approximately 62.4 lbs/ft^3 .

VIII.1.5. Gradually-Varied Flow

The ordinary differential equation (ODE) solver can be: (1) *Euler method*; or, (2) *Runge-Kutta*. These are two common methods used to solve ODEs in general, and provide sufficiently accurate solutions to the ODE, which is used in the app to determine steady, non-uniform flow profiles in open channels.

The *Euler method* is the simplest and the fastest, but perhaps the least accurate. Note that for small layouts of only one or two reaches, the speed of the ODE solver will not be relevant: both methods will produce very fast results.

The Runge-Kutta method uses a pair of 4^{th} - and 5^{th} -order methods. It is very accurate for the purposes in *mCanal*, typically generating ODE solutions to within an accuracy of a fraction of a millimeter of water depth in an open channel.

VIII.1.6. Linear System Solution

The matrix solution method is *Gaussian elimination* by default, but you can choose one of the other options if desired. Gaussian elimination is the fastest and most computationally efficient of the methods included in the app, and it uses the least amount of memory. Nevertheless, this method can lead to numerical divergence of the mathematical solution in rare cases. Thus, you can also select from the *Gauss-Jordan* and *Lower-Upper (LU) Decomposition* methods for the solution of the linear system of equations that is created at each iteration of the Newton-Raphson method (Press, et al. 1992), in each pool of each reach at each time step.

The *Gauss-Jordan* method uses full pivoting and can provide a more accurate matrix solution than Gaussian elimination. But it is usually the slowest of the three methods included herein, and it becomes exponentially slower with increasing number of computational nodes. In most cases, application of the *Gauss-Jordan* method will not be necessary. *LU Decomposition* uses partial pivoting and is usually faster than *Gauss-Jordan* elimination, but not as fast as *Gaussian elimination*.

These solution options are relevant only to pool-by-pool calculations. For simultaneous pool solutions (Chapter I.4.1), a special implementation of *Gaussian elimination* is applied, and is not affected by the choice made herein.

VIII.2. Limits

This tab contains a group of numerical values that affect the mathematical solution to the equations involved in a hydraulic simulation. You can edit each value within pre-defined parameter ranges.

VIII.2.1. Convergence

Area tolerance is used as part of the definition for numerical convergence of the equations in a reach at a time step during a simulation. This is a coefficient multiplied by the calculated flow cross-sectional area at a computational node during Newton-Raphson iterations. The product of coefficient and area is the largest absolute value of the corresponding function vector in the augmented *Jacobian matrix* (see below) that will trigger numerical convergence for the node.

Theoretically, all function vectors are equal to zero at convergence, but in practice this is usually not possible – thus, the application of tolerance values. Newton-Raphson iterations continue until convergence criteria are met at all function vector elements, or until a specified maximum number of iterations has occurred at a given time step. **Jacobian Matrix** This is a square matrix populated with first-order partial derivatives of the hydraulic equations with respect to the dependent variables. It is a banded matrix, used as part of a method to essentially convert a system of non-linear equations to linear equations. It is an integral component of the four-point implicit solution to the set of equations at each simulation time step. It is an augmented matrix because it includes the column vector of adjustments to the dependent-variable values.

In the app, dependent variables include flow rates and crosssectional areas (convertible to depths, as defined in Chapter III.4) at each computational node. The independent variables are distance along a reach and elapsed simulation time.

General Lim	its Transitions
Convergence: maximum absolute deviations	Inter-step iteration tolerances
Distance (m)	Maximum iterations
0.005	3
Flow rate (m3/s)	Flow rate tolerance (m3/s)
1.000E-05	1.000E-03
Cross-sectional area (m2)	Cross-sectional area tolerance (m2)
1.000E-06	1.000E-03
Numerical convergence limits and tolerances.	
Use Defaults	OK Cancel

VIII.2.2. Inter-step Iterations

The app updates hydraulic conditions on a reach-by-reach basis at each time step during a simulation. With some structure types, this approach is inadequate because a careful balance is required across reaches to avoid numerical divergence. Thus, there may be "inter-step iterations" which repeatedly update hydraulic conditions for each reach in the system layout, one-by-one, during a single time step, until flow rates at reach ends cease to change (for that time step). This parameter lets you specify the maximum number of such iterations before the simulation proceeds to the next time step. This parameter is ignored unless the system layout includes more than one reach.

The inter-step tolerance affects inter-step iterations. The inter-step flow-rate tolerance is the maximum allowable difference in flow rates at a reach end (upstream or downstream), from one inter-step iteration to another. When the actual flow rate difference at all reach ends is less than or equal to the specified tolerance value, inter-step convergence is attained in a system layout, and the simulation advances to the next time step. This parameter is ignored unless the system layout includes more than one reach. The inter-step area tolerance is based on calculated cross-sectional areas and is applied as described above for flow-rate tolerance.

VIII.3. Transitions

This tab applies to underflow gates and to centrifugal pumps. Underflow gates can operate under four different hydraulic regimes, including free and submerged orifice flow, and free and submerged non-orifice flow. The threshold between orifice and non-orifice flow occurs approximately when upstream water surface coincides with the bottom of the gate. However, this is not exactly correct, so you can specify transition parameters that affect both the threshold and the transitional range. A transitional range can be used to smooth the change between some of the possible regimes, providing additional robustness and stability to hydraulic simulations. See Annex E for more details.

The two pump parameters are for defining the threshold upstream water depth at which the pump "breaks suction" by entraining so much air that it causes water to stop flowing through the pump.

	General Lim	nits Transitions			
Orifice-flow contraction co	pefficient	Minimum pump upstream depth			
	0.610	Absolute minimum (m)			
Orifice free-submerged tra	ansition range	0.000			
	0.050	Fraction of reach depth			
Orifice-nonorifice transitio	on range	0.000			
	0.050				
Hydraulic seal coefficient		Below minimum upstream depth, pumps are			
	0.900	assumed to break suction and stop pumping.			
Use momentum function for free-submerged orifice flow Transitional flow parameters at underflow and constriction structures, and for pumping.					
Use Defaults		OK Cancel			

VIII.3.1. Orifice-flow contraction coefficient

This is the contraction coefficient (C_c) on the downstream side of an underflow gate, which defines the ratio of the minimum depth of the high-velocity jet through the gate to the vertical opening of the gate. A theoretical value for C_c can be derived from hydrodynamics of what is referred to as "potential flow", resulting in $C_c = \pi/(\pi + 2) = 0.611$. Although this exact value is often used in hydraulic equations, the true value can vary significantly in practice.

VIII.3.2. Orifice free-submerged transition range

This is a parameter (χ_o) used to determine the extent of the transitional flow rate between free and submerged orifice flow. If it is equal to zero, the transition range collapses and a simulation regime change will occur instantly when the threshold is reached. The threshold can be based on a comparison between the product of the contraction coefficient, C_c , and the vertical gate opening, G_o , and the calculated downstream depth. If the calculated downstream depth exceeds $C_c G_o$, the regime is submerged orifice; otherwise, it is free orifice.

On the other hand, the momentum function can be applied to distinguish between free and submerged orifice flow.

In the app, there is no non-orifice free-submerged transition range because the calibration equations for Cuthroat flumes are used for non-orifice flow at underflow gates. Those equations already have a smooth transition and they cover both free and submerged orifice flow. Applying the Cuthroat flume equations can be appropriate because they model channel constrictions, in general, for non-orifice regimes, and underflow gates behave hydraulically as channel constrictions in such conditions.

VIII.3.3. Orifice-nonorifice transition range

This is a parameter (χ_{on}) used to determine the extent of the transitional flow rate between orifice and non-orifice flow. If it is equal to zero, there is no transition range and a regime change will occur instantly during a simulation when C_o multiplied by the upstream depth equals the vertical gate opening.

VIII.3.4. Hydraulic seal coefficient

This is a coefficient to determine the transition point between orifice and non-orifice flow regimes. It is multiplied by the depth of water on the upstream side of an underflow gate structure and compared with the vertical opening. If the product of C_o and upstream depth is less than the vertical gate opening, insufficient upstream hydraulic seal exists, and the regime at the gate is determined to be non-orifice; otherwise, it is considered to be orifice flow.

VIII.3.5. Use momentum function

When this box is checked, the momentum function is used to define the threshold between free and submerged orifice flow. This is the default option. When the box is unchecked, a simple ratio between the calculated downstream water depth and the depth at the *vena contracta* is applied. The definitions for the momentum function and free-submerged orifice transitions are provided in Annexes B & E.

VIII.3.6. Absolute minimum

This is the absolute minimum depth (m or ft) at the upstream side of a centrifugal pump, below which the app assumes that water will stop flowing through the pump. It is a threshold used to approximate the point at which a pump would break suction due to entrainment of air. During a hydraulic simulation, mCanal may automatically increase the effective absolute minimum if the calculated upstream depth goes to zero during solution of the equations at a pump.

VIII.3.7. Fraction of reach depth

The specified fraction is multiplied by the depth of the canal reach at a pump location to determine the minimum depth at which pumping can occur. However, if this depth is less than the absolute minimum value, the absolute minimum is used as the threshold value instead.

VIII.4. Resetting to Defaults

Click on the **Use Defaults** button reset numerical parameters to their default values. The button only affects the parameters for the currently displayed tab. Clicking the button for the *General* tab resets those parameters, and then switching to the *Limits* tab and clicking the button again will reset those other parameters. And the same for the *Transitions* tab. When you select **File**, **New** from the main menu, all numerical parameters and options will be at their respective default values.

IX. Performing Hydraulic Simulations

Performing hydraulic simulations is what most of the previous chapters of this Users' Guide lead up to. Of course, setting up the system layout, inserting structures and calibration values, creating scheduling graphs, specifying numerical parameters, determining startup and termination conditions, and others are important and necessary preludes to running a meaningful simulation.

After what may seem to be a long series of data entry and option sheets, you are finally ready to begin a simulation and see some modeling results. There are four round toolbar icons to control simulation status, as seen below.

Click on the green toolbar icon to start or resume a simulation. If no simulation is in progress, clicking the green icon will cause a new simulation to begin, but if a simulation is paused, this will cause it to continue. Click on the blue icon to advance a simulation by one time step (after which it will enter the pause status), or the yellow one to pause a simulation that is running. Or, click on the red icon to terminate a simulation when it is either running or paused.

You can also use the commands under the **Simulate** main-menu item, or the following key combinations:



Depending on the current simulation status, one or more of the **Simulation** menu items and toolbar icons will be disabled. Of course, you can also set simulation pauses to stop at predetermined elapsed times, as discussed in Chapter IX.2.3.

During a simulation (running or paused), several menu items and toolbar icons will be disabled, and the **OK** button will be disabled in most of the data-editing sheets (so that parameter values cannot be modified until a simulation terminates). These items correspond to system layout editing functions, and others that could signify simulated infrastructure or configuration changes. The reason these are disabled during a simulation is that such changes might invalidate a simulation and cause it to terminate unsuccessfully. For example, you cannot abruptly change a channel cross section or longitudinal bed slope, and you cannot change the types, sizes, or numbers of control structures — those kinds of actions would mean that you are suddenly simulating the hydraulic response of a different canal system.

You can view the system layout during a simulation, but you cannot make any changes to it until the simulation terminates. However, you can change the layout node labels; for example, you can change from flow rate to depths, and so on. You can also change specified control structure settings and flow rates at any time during a simulation, as well as scheduling graphs.

If you move the dividers to resize the views (Chapter I.3) in the main window during a simulation, the simulation will pause and the status indicator at the right side of the toolbar will turn yellow. To resume, type \Re -1 or \Re -2 (or click the green or blue button on the toolbar).

IX.1. Data Check

In the **Preferences** sheet (Chapter I.4), you can choose to show warnings about the parameters to be used in a hydraulic simulation. When you start a simulation, the app will cross-check for errors in the parameters, and if any are found, they will be displayed in a window and the simulation will not begin. Note that you cannot avoid or turn off error checks — they will always be performed before a simulation. If you have chosen to show warnings and or hints, and if any are found, they will also be displayed before the start of a simulation. But if there are no errors, you will have the option of running the simulation in spite of warnings (if any), or of canceling and going back to check on the cause(s) of the messages.

The figure below shows an example of the *Data Check* sheet when there are one or more warnings or errors to show. If there are no warnings or errors, another sheet appears, but only if you manually request the data check – if it is an automatic check that is performed before beginning a simulation, then the absence of warnings or errors will be indicated by the simulating running.

Show	Error 1 of 3				
ErrorsWarnings	Radial gate radius divided by pinion height is greater than 2.0 for regulator Radial-1A at the upstream end of reach Main canal.				
0	Explanation and possible solutions				
Error code: 2212	A calibration limitation for the types of radial gates included herein is that the ratio of gate radius to pinion height be less than or equal to 2.0. Decrease the gate radius and or increase the pinion height to satisfy this criterion.				
Error and warning mess	ages, and suggestions for dealing with them.				
Write to File	< Prev Next> OK				

Before running a simulation, and even before completing your data entry tasks in preparation of a simulation, you can request a data check. This is done by selecting **Simulate**, **Data Check** from the main menu.

IX.2. Simulation Modes



Open the **Simulation Modes** sheet by selecting **Simulate**, **Modes** from the main menu, or by clicking on the gears icon. This is where you specify how a simulation begins (*Start mode*), under what conditions it will terminate normally (*Termination mode*), and whether you want it to pause at pre-set elapsed times, as seen below.

Start mode	Simulation pause times			
 Steady-state condition Continue previous simulation Start with all reaches empty 	1 of 1 2:06:00:00 ♀ 2:06:00:00 + - Ignore all			
Termination mode				
• Fixed duration	Stability tolerance			
	0.0010			
Hydraulic stability	Delay (days:hrs:min:sec) 0:00:05:00			
Stability with duration limit				
·	Duration (days:hrs:min:sec)			
	0:05:00:00			
Hydraulic simulation start and termination modes, pause times, and parameters.				
	OK Cancel			

IX.2.1. Start Mode

There are three simulation startup options, as described below.

1. Steady State With this option, *mCanal* will attempt to generate steady-state, gradually-varied flow profiles for each of the open-channel reaches before starting a simulation. You can control the results of the generation of startup flow profiles through values specified in the *Numerical Parameters* sheet, described in Chapter VIII, and by various specifications in the *Operational Settings* sheet. In branching networks, the steady-state start-up condition can require several iterations to balance water depths and flow rates, and the calculations can be complex, such that in certain situations error messages may be generated by the app.

2. Continue a Simulation After having performed a simulation with a successful termination, and having saved the file, you can continue it in a subsequent simulation. This can be done either immediately or by re-opening the file at a later time. The continuation process can be repeated indefinitely. But if you run multiple simulations, only the ending status of the last simulation will be available for continuation. You can get around this restriction by saving the configuration data file with different names after each simulation that you would like to subsequently continue.

If you select **File**, **New** from the main menu, or have not loaded a configuration data file, there will be no simulation status data, and, in such cases, you cannot choose this startup option. Also, if you modify the system layout, add or delete any structures (regulators or turnouts), or other "infrastructure" changes to the configuration data, this startup option will be disabled – that is to prevent the continuation of a simulation from one configuration into a different configuration, potentially causing a physically-infeasible situation.

3. Start Empty The default option is to start with all reaches empty. This means that there are no pools in any reach and the simulation will begin with water advancing in one or more reaches. If the system layout is such that there are several reaches connected in series, along a single branch, the most likely scenario is that water will enter the upstream-most reach, advance to the end, spill into the next reach, and so on, until all of the reaches have water, or until the simulation terminates. However, it could also be that water enters through one or more turnouts at midpoints along the reaches, causing multiple

pools of water to advance simultaneously. A trivial situation would be when all reaches are empty and no water enters the system, so that the simulation runs without anything significant happening. There are many possibilities.

IX.2.2. Termination Mode

There are three options for successful termination of a hydraulic simulation. First, you can specify a fixed duration in days:hours:minutes:seconds. Second, you can have the simulation terminate upon satisfying certain hydraulic stability criteria. And third, you can specify hydraulic stability criteria and limit the simulation to a maximum value of elapsed time. The third option means that the simulation will terminate upon meeting the hydraulic stability criteria, unless those criteria are not met by a specified elapsed time, at which point the simulation will terminate regardless of the hydraulic stability index.

When you choose the first option, *Fixed Duration*, the two *Stability* parameters are disabled and irrelevant. But for the other two options you need to specify the stability tolerance and the stability delay values. *Stability Tolerance* is the minimum absolute (plus or minus) deviation from zero for the app to terminate the simulation based on hydraulic stability criteria. This criterion must be met during two consecutive time steps of a simulation before mCanal will terminate the simulation, because a single occurrence may mean that the simulation is merely passing through a hydraulically stable condition (that is, the hydraulic stability index is going from positive to negative, or vice versa).

The *Delay* parameter is the minimum elapsed time after starting a simulation before the hydraulic stability criterion is applied. This delay is sometimes necessary because a simulation might start at a hydraulically stable condition, which would cause the termination criterion to be met before even beginning the simulation. For example, the simulation might start under a stable condition, but after five minutes of simulation time you change a structure setting or source flow rate, causing a hydraulic transient, or in other words, temporarily disturbing the hydraulic stability.

Hydraulic Stability Index This is a measure of the hydraulic stability of the system layout as defined at the time of a simulation. The index is equal to zero when the simulation is at, or is passing through, a stable hydraulic condition, meaning that all water depths and flow rates are unchanging with time (even though the flow rate and water depth are likely to be changing as a function of distance along the reaches). When the index is a positive nonzero value, there is more flow entering the system than leaving, and vice versa for negative values. The index can vary from a minimum of -1.0 to a maximum of +1.0.

The *Duration* parameter is disabled when you choose the second option, *Hydraulic Stability*, because in that case the duration value is not applied.

IX.2.3. Pause Times

Pause times are specific elapsed times at which a simulation will be suspended to allow you to check on its status or modify certain parameter values before continuing. Alternatively, you can pause a simulation by clicking on the "yellow-dot" icon, selecting **Simulate**, **Pause** from the main menu, or by pressing \Re -3 on the keyboard. However, in these spontaneous pause methods, it may be difficult to

stop the simulation exactly at the desired elapsed time, unless the simulation is running slowly.

Of course, if a pause is set for a time that equals or exceeds the elapsed time when the simulation terminates, it will not be triggered.

View all of the pause times by scrolling within the drop-down list box at the upper right of the sheet. When you start with a new system layout, there will be no pause times. You can add one by entering a time value in the edit box below the drop-down list, and then clicking the + (plus) button. You can repeat this action to add up to a total of 32 values. Enter new pause times in "days : hours : minutes : seconds," as you would for most any other temporal values in the app. You can substitute any non-numeric character for the colon to separate the values and obtain the same results.

You can modify an existing pause time, delete an existing one, and add a new one. Delete a pause time by selecting it in the drop-down list and clicking on the - (minus) button. Finally, you can check the *Ignore all* box to disable all pause times without the need to delete them.

IX.3. Results Output

Before beginning a simulation, be sure to specify the desired results output options by selecting **File**, **Write Results...** from the main menu. This is how you determine what kind of information is saved during a simulation, and which text and or CSV files are produced when a simulation terminates. Note that you cannot open this sheet during a simulation, so you need to make these specifications before beginning a simulation.

Check 1+200		Choose Path
Check 2+350 Check 4+000		Selected reaches: 5 of 5
Lat R-880	Select All	
		Select None
Output period	Include in results	Output as
✓ Full simulation time Start time	 Summary data Source data Reach data 	✓ Plain text ○ Comma separated
0:00:00:00 End time 0:05:00:00	 Regulator data Upstream ✓ Downstream 	Depths or elevations • Water depths • Water surface elevation
Time-step multiplier	Turnout data	Water surface elevation

IX.3.1. Results Path

This is where you specify the drive and folder where you want the results files to be saved. Results file names will be as specified in the path, whereby the default name is that of the layout configuration file. But, if you like, you can give the results files a different name than that of the configuration file. The file extensions will be TXT for text ("Plain text") and CSV for comma-separated values.

Click on the *Results path* label, or on the **Choose Path** button, to open a sheet from which you can change the drive, folder, and file name. The file name that you specify will have "(Reach)", "(Regul)", and so on, automatically appended to indicate the type of data that each output file contains.

This same folder is where other text files will be sent, such as layout parameters, scheduling data, pump and valve calibrations, and others that you can create from various main- and popup-menu commands.

IX.3.2. Reach Selection

A list of all reaches in the current system layout is shown here — the above figure shows six reaches as an example layout. Reaches that are selected for results output will have a green check mark to the left of the reach name, and those that will not be included have a red X. You can toggle the check mark on any row in the list by double-clicking on the reach name. You can also click on the **Select All** or **Select None** buttons.

Only those selected (with a green check mark) reaches in the list will be included in one or more simulation results output files. If you do not select any of the reaches, a warning message will be seen on the sheet, and there will be no results output following the close of the simulation; that is, you will see dynamic results in the *Branch View* and *Graphs View* of the main window during the simulation, but there will be no text or CSV file output.

The *Selected reaches* counter confirms the number of currently-selected reaches.

IX.3.3. Output Period

Commonly, the entire simulation time period is included in the output files. But you can also specify a certain sub-period for output by unchecking *Full simulation time* and entering the start and end times for results to be outputted. As with other time values in the app, these are specified as days : hours : minutes : seconds. You can also specify a time-step multiplier, which is a coefficient on the simulation time step, Δt . By default, the multiplier is equal to 1, meaning that the results are outputted each and every time step. If you set it equal to 2, results are outputted every other time step, and so on for higher values of the multiplier.

IX.3.4. Data Output Categories

The six main data output categories that can be included in results files are: (1) Summary data; (2) Source data; (3) Reach data; (4) Regulator data; (5) Turnout data; and, (6) Pool data. Check the boxes of all the categories that you want to include in the output files. For regulator data, you can check one or both of the *Upstream* and *Downstream* boxes, pertaining to the upstream and downstream ends of the selected reaches.

If you check the *Turnout* or *Regulator* boxes, data for all turnouts and or regulators in each of the selected reaches will be outputted to file(s) during a simulation. That is, you cannot select only some of the turnouts or regulators in a reach – it is all or nothing. If you check the *Turnout data* box and there are no turnouts in any selected reach of the current system layout, no new results file will be created for turnout data during the simulation.

If you do not check any of the options boxes, there will be no results output from the simulation, even if one or more reaches have been selected above in this sheet. In some cases, there could be a delay after a simulation finishes, especially if the elapsed time is large, while the text and or CSV files are created.

IX.3.5. File Types

One or two output file formats can be selected. Check *Plain text* and or *Comma-separated* as desired for the output files. If you do not check either of them, you will see a warning message.

IX.3.6. Depths or Elevations

By default, water depths are shown in the output files for reach, regulator, turnout, and source data. But you can choose to show water surface elevations instead. Water surface elevations are relative to the source and reach bed elevations.

IX.4. Operational Settings



Open the **Operational Settings** sheet by selecting **Simulate**, **Operational Settings** from the main menu, or by clicking on the toolbar icon. This is where you specify operational settings and modes before a simulation begins, and also during a simulation, if desired. While a simulation is in progress, you can change any of the operational parameters included in this

sheet, as often as needed.

Many of the values specified in this window can also be supplied through scheduling graphs (Chapter VII). If a particular value has an associated scheduling graph, the **Use graph** box will automatically be enabled at the right of the text box. If it is enabled and has a check mark, it means that the scheduling graph will be applied to the corresponding node, reach, or structure; otherwise, the scheduling graph (if one exists) is ignored during a simulation.

You can toggle the check mark by clicking on the box – the text will be a light gray color, so you can easily see when it is disabled. When a **Use graph** box is disabled, it does not matter if the box is checked or not.

There are four tabs in this sheet, each of which is discussed in the following subsections.

IX.4.1. Sources

There is a list of source nodes at the upper left corner of the sheet. Every system layout in the app will have one primary source node and may optionally have several secondary source nodes. All of these will be found in this list. Select a source node from the list (if there is more than one item) by clicking on the name of the source node.

Source Operational Mode

Each source node will have one of two operational modes during a simulation. The default is *Specified* flow rate, but you can change this to *Calculated flow rate* for one or more source nodes, individually, in the system layout. When the mode is *Specified flow rate*, you tell the app what the flow rate is at this location, corresponding to inflow to one or more reaches directly connected to the source node, either through a single value or a scheduling graph. In this case, the app ignores the regulator structure type(s) at the upstream end of the reach directly connected to the source, and it ignores the upstream depth at the source node. Thus, *mCanal* accepts the specified flow rate at this boundary without attempting to verify it or otherwise calculate its value.

	Sources	Reaches	Regu	lators	Turnouts	
Source nodes		1	of 1	Source	operational m	ode
HT Reservoir	-		\diamond	o s	pecified flow	/ rate
US water dept	:h (m)			Oc	alculated flo	w rate
	1.500	🗌 Use gra	ph			
Flow rate (m3/	/s)					
	0.120	🗹 Use gra	ph			
Primary and sec	ondary water	source opera	itional s	ettings an	id modes.	
					ОК	Cancel

If you have a split source node, meaning that there are two or more reaches directly connected to the source node, you cannot use the *Specified flow* mode at the source because the app will not know how to divide that flow between the upstream ends of the respective reaches.

If you choose the *Calculated flow* operational mode, the app needs to know what regulator structure types are located at the upstream end of each reach that is directly connected to the source node, their calibration values, the elevation upstream of the source node, and the depth upstream of the source node. With this information, the app calculates the flow rate at the source node during each time step of a simulation.

US Water Depth

This is a specified upstream water depth, located just upstream of the beginning of the reach(es) connected to the selected source node (primary or secondary). This depth could be that in a reservoir, river, lake, canal, or other surface water source, but it is outside the boundaries of the system layout with regard to the hydraulic simulation. The depth is referenced to the specified elevation of the source node (Chapter III.3.2).

The depth at the source node is not used unless the source mode is *Calculated flow* (see above). However, if the mode is *Calculated flow*, *mCanal* will use the specified upstream depth, the regulator type(s) and calibration(s) at the source node, and the calculated upstream depth in the reach directly connected to the source node to calculate the flow rate at the source node. Note that the upstream water depth in that reach is actually the downstream depth at the regulator(s), and that there may be multiple reaches directly connected to a source node. The regulator structures at a source node are specified by the regulators at the upstream end of all reaches directly connected to the source node — you do not specify regulator type(s) and dimensions at a source node itself, but the effect is the same.

If you have scheduling graph data for water depth at a source node, you can check the *Use graph* box to apply it during a simulation. Otherwise, the value entered here will be taken as the upstream depth at a source node, referenced to the source elevation, and will remain constant until you change it again.

Flow Rate

This is a specified flow rate at a source node, entering a reach directly connected to the node. If the source node is split, two or more reaches are directly connected to the source node, and you cannot use the *Specified flow* operational mode at the source. In such a case, this discharge value will be ignored during a simulation.

If you have a scheduling graph for flow rate at a source node, you can check the *Use graph* box to apply it during a simulation. Otherwise, the value you enter here will be taken as the flow rate at a source node and will remain constant until you change it (either before or during a simulation).

IX.4.2. Reaches

A list of all reaches in the current system layout is found at the upper left of this window. You can select one reach at a time in the list, then view or edit the corresponding data values in this tab sheet. You can also specify the reach end, upstream or downstream by clicking over the word "Upstream" or "Downstream" at the lower right of the tab.

	Sources	Reaches	Regu	lators	Turnouts	
Reach nodes		5	of 5	Reach	operational mo	ode
Lat R-1590			\Diamond	O N	lormal	
Water depth	(m)			Os	pecified dep	th
	0.900	Use gra	ph	Os	pecified flow	rate
Flow rate (m3	3/s)					
	0.100	Use gra	ph	Reach	location	
Terminus DS	depth (m)			0		
	0.000	Use gra	ph	Οu	lpstream	
Full supply de	epth (m)			0 C	ownstream	
	0.850					
Reach operatio	nal settings a	nd modes.				
					OK	Cancel

Reach Operational Mode

There are three possible operational modes for reach ends (upstream and downstream): (1) Normal; (2) Specified depth; and, (3) Specified flow rate. The Normal mode is the default, whereby both water depth and flow rate are calculated by the app at a reach end. The Specified depth mode allows you to set the depth at a reach end, and the Specified flow rate mode lets you set the discharge at a reach end. With these last two modes, the regulator structure type(s) and calibration(s) at the specified reach end are ignored by the app during a simulation.

For pipeline reaches, the mode must be *Normal*, the flow rate is calculated at both reach ends, and the other parameters in this tab are ignored. Also, for open-channel reaches connected in series to a pipeline reach, the mode must be *Normal* at these connection locations. That is, the downstream reach mode for an open-channel reach at the upstream end of a pipeline must be *Normal*, whereby the flow rate is calculated according to the pipeline equations, as described in Chapter III.5, and the water depth at

the same reach end is calculated according to the solution of the governing hydraulic equations. This same restriction applies to the upstream reach mode when the upstream connection is to a pipeline reach.



The *Specified flow rate* reach mode is unavailable at the downstream end of split reaches. This is because flow rate at such locations is determined based on the regulators at the upstream end of the connecting reaches, or on specified discharge values at the upstream ends of one or more of those reaches.

Water Depth

If the operational mode at a reach end is *Specified depth*, the app will use the water depth entered here when performing a simulation; otherwise, the depth value is ignored. Note that specifying depth at a reach end can easily lead to unfeasible hydraulic conditions, possibly causing a simulation to terminate unsuccessfully. The *Specified depth* operational mode must be used carefully.

If you have scheduling graph data for depth at a reach end, you can check the *Use graph* box to apply it during a simulation. Otherwise, the value you enter in this tab will be taken as the depth at a reach end, referenced to the bed elevation, and will remain constant until you change it again.

Flow Rate

If the operational mode at a reach end is *Specified flow rate*, the app will use the flow rate entered here when performing a simulation; otherwise, the value is ignored. If you have scheduling data for flow rate at a reach end, you can check the *Use graph* box to apply it during a simulation. Otherwise, the value you enter in this tab sheet will be taken as the flow rate at a reach end and will remain constant until you change it again. You should be careful to not specify a nonzero flow rate at the downstream end of a reach when no pool exists at that location (such a situation is physically impossible).

If you specify a flow rate at the downstream end of a reach which is the furthest downstream reach in a branch, the flow rate for the corresponding branch terminus will be automatically updated, thereby maintaining consistency (Chapter VIII.4.2).

Terminus Downstream Depth

The downstream depth (*DS water depth*) at a terminus location (which is the downstream end of the last reach in a branch) can be specified to determine the calculated flow rate through or over the regulator at that location. The depth is referenced to the bed elevation at the downstream end of the reach. If the downstream depth is zero or negative, the flow regime at those regulator(s) will be "free," otherwise they might be submerged, in which case a head differential would be applied to the calculation of the flow rate, depending on the structure type and calibration.

If you have scheduling graph data for depth at a terminus location, you can check the *Use graph* box to apply it during a simulation. Otherwise, the value you enter in this tab will be taken as the downstream depth at a terminus node, referenced to the specified elevation of the downstream end of the reach, and will remain constant until you change it again.

Reaches that do not correspond to a terminus node will show "n/a" in the text box.

Full Supply Depth

In operational terms, this is a kind of "target" depth of water in an open-channel reach. However, the *Full supply depth* is used only when the app is generating a steady-state start-up condition in an open-channel reach with: (1) zero flow; or, (2) a specified downstream flow rate. The value entered here must be equal to or less than the maximum depth of the cross section, but preferably less, in order to provide some amount of *free board* (from the water surface to the top of the canal lining or canal bank, as the case may be).

The *Full supply depth* value is applied to the downstream end of the reach if the longitudinal bed slope is positive, otherwise the value is applied to the upstream end of the reach. If the downstream reach mode is that of a specified non-zero flow rate, the app calculates the Froude number (Chow 1959) at the downstream end of the reach. If the calculated Froude number is greater than 0.99, supercritical flow would be expected, and an error message is issued.

IX.4.3. Regulators

Here you see a single reach name at the upper left corner of the tab, which you can change by rightclicking over the name and selecting another reach from popup menu (assuming there is more than one reach in the current system layout). Below the reach name is a list of regulators in that reach, at the specified reach end (upstream or downstream). You can toggle the reach end by clicking over the word "Upstream" or "Downstream" at the lower right area of the tab. The upstream and downstream ends of every reach will have at least one regulator structure.

You can see the name of the structure type just below the list of regulators. This corresponds to the selected item in the list of regulators.

	Sources	Reaches	Regu	lators	Turnouts	
Reach nodes		5	of 5	Regula	tor operational	mode
Lat R-1590			\Diamond	o s	pecified sett	ing
Regulators		1	of 1	A	utomatic set	ting
Check R-15	90 spill		\$	Os	pecified flow	rate
Type: Round	gate			Oc	alculated flo	w rate
Setting (m)						
	0.000	🗸 Use gra	ph	Reach	location	
Flow rate (m3	3/s)			Ou	lpstream	
	0.000	Use gra	ph	OD	ownstream	
Regulator setting, flow rate and operational mode.						
					ОК	Cancel

Operational Mode

Each regulator structure can have one of four operational modes: (1) Specified setting; (2) Automatic setting; (3) Specified flow rate; and, (4) Calculated flow rate. Specified setting is the default mode. If you choose Automatic setting, then you must also choose an automation method. For non-adjustable structures (e.g. flumes) without a specified flow rate, you can select the Calculated flow rate mode. In fact, that mode is only applicable to non-adjustable structures. The same rules apply to turnouts.

Mode	Applicability
Specified setting	Only for adjustable structures in which the flow rate
	is calculated.
Automatic setting	Only for adjustable structures that can be auto-
	mated, and where flow rate is calculated.
Specified flow rate	This mode can be used with any structure type, and
	must be used for the <i>Undefined</i> structure type.
Calculated flow rate	Only for non-adjustable structures, but not for the
	Undefined structure type.

Setting

If the operational mode for a regulator is *Specified setting*, during a simulation the app will use the setting value you specify here. For many regulator types, the setting refers to the vertical opening of a gate or weir. But some regulators are not adjustable (for example, constrictions, flumes, uniform-flow boundaries, and others), or are not even "structures," and for these the setting value is meaningless. Of course, you can effectively make any regulator nonadjustable by simply maintaining its setting value constant while a simulation is running.

- If it is an underflow gate structure, the setting is the vertical opening under the gate leaf at the center of the gate.
- If it is a weir, the setting is the height of the crest above the minimum crest position, as determined by the "invert drop" value (Chapter IV.3).
- If it is a centrifugal pump, the setting is the valve opening, in percent, downstream of the pump outlet. A setting of zero means no flow through the pump; a non-zero setting means there might be flow, depending on the upstream and downstream conditions, and the valve calibration.

If you have a scheduling graph with regulator setting values, you can check the *Use graph* box next to the setting, so that the scheduling graph supplies the changes in regulator setting during a simulation. Otherwise, if you do not check that box, of if it is disabled, the app will use the setting entered in this tab and it will stay the same until you change it again.

Flow Rate

You can specify the flow rate at a regulator. If there is only one regulator at that location, it will have the same effect as specifying a flow rate at the corresponding reach end, upstream or downstream. If you do both of these things, the regulator will take precedence over the reach.

If you have a scheduling graph with regulator flow rates, you can check the *Use graph* box next to the flow rate value, so that the scheduling graph supplies the changes in regulator discharge during a simulation. Otherwise, if you do not check that box, of if it is disabled, the app will use the flow rate entered on this tab and it will stay the same until you change it again.

Automation Method

This version of mCanal does not include local automation features, but will be found in a future version of the app.

IX.4.4. Turnouts

This tab is similar to the *Regulators* tab, but for turnout structures. The location of the selected turnout in the list is shown to the right of the list, referenced to the upstream end of the reach in which the turnout is located.

Turnouts are a kind of modeling interface between the simulated hydraulic system and the unknown. That is, turnouts are external boundaries where water can leave or enter a reach in the simulated system. When flow rate is calculated at a turnout, the downstream depth can be relevant, but it is defined as a parameter outside of the simulated system.

When water leaves a reach through a turnout, mCanal does not know where the water goes — only that it leaves the system. Conversely, when water enters a reach through a turnout, mCanal does not know where the water came from.

	Sources	Reaches	Regu	ulators	Turnouts	
Reach nodes		1	of 5	Turnou	ut operational r	node
Check 1+20	00		\Diamond	O S	Specified set	ting
Turnouts		1	of 1		utomatic set	ting
Farmer B			\Diamond	Os	Specified flow	v rate
Type: Round	gate			O c	Calculated flo	w rate
Setting (m)						
	0.000	Use gra	ph			
Flow rate (m3	3/s)			DS dep	th (m)	
	0.000	Use gra	ph		0.000	Use graph
Turnout setting, flow rate, downstream depth and operational mode.						
				(ОК	Cancel

Operational Mode

The same four operational modes as described above for regulators also apply to turnout structures.

Setting

The setting must be specified only when the mode is *Manual setting*, otherwise it is ignored during a simulation. Note that some turnout structures are not adjustable (e.g. constrictions, flumes, uniform-flow boundaries, and others), and for these the setting value is meaningless. But you can effectively make any turnout nonadjustable by simply maintaining its setting value constant while a simulation is running.

- If it is an underflow gate structure, the setting is the vertical opening under the gate leaf.
- If it is a weir, the setting is the height of the crest above the minimum crest position, as determined by the *invert drop* value (Chapter IV.3).
- If it is a centrifugal pump, the setting is the valve opening, in percent, downstream of the pump outlet. A setting of zero means no flow through the pump; a non-zero setting means there might be flow, depending on the upstream and downstream conditions, and the valve calibration.

If you have a scheduling graph with turnout setting values, you can check the *Use graph* box next to the setting, so that the scheduling graph supplies the changes in turnout setting during a simulation. Otherwise, if you do not check that box, of if it is disabled, the app will use the setting entered in this tab sheet and it will stay the same until you change it again.

Flow Rate

If the mode for a turnout is Automatic setting or Specified flow rate, you must enter a flow rate here. If you choose the Specified flow rate mode, mCanal will ignore the turnout type and setting during a simulation and assume that the specified flow rate is achievable. However, if the mode is Automatic setting, the app will try to match your specified flow rate by making incremental setting adjustments during a simulation.

If you enter a positive flow rate, flow leaves the reach and exits the simulated system through the turnout. If you enter a negative flow rate, water enters from outside the simulated system and goes into the reach through the turnout at that location.

If you have a scheduling graph with turnout flow rates, you can check the *Use graph* box next to the discharge value, so that the scheduling graph supplies the changes in turnout discharge during a simulation. When the box is not checked, or if it is disabled, mCanal will use the flow rate entered in this tab and it will stay the same until you change it again.

DS Depth

To more realistically simulate the operation of turnout structures, you can specify the downstream (assuming water flows out from the reach, through the turnout) depth, thereby allowing the app to determine whether the regime is free or submerged. You can do the same thing for turnouts that allow water to enter the canal reach, but in this case, it would be the depth on the upstream side of the turnout, outside the simulated system.

If you have a scheduling graph of downstream depths for a turnout, you can check the **Use graph** box. Note that you can enter turnout depth data even if the structure type is **Undefined**, but in this case, it will not matter what depth data you specify because mCanal will ignore it in a simulation.

Specify the water depth downstream of each turnout, relative to the canal bed of the turnout's location along the reach. That means that the depths upstream and downstream of the turnout are referenced from the same elevation datum. As explained above, the specification of downstream depth is necessary because the downstream side of a turnout structure is outside of the simulated system and cannot be calculated by applying the governing equations.

The water depth downstream of a turnout will only be relevant when it would cause a submerged-flow regime at the structure. For example, if the turnout is a weir and the downstream water surface remains below the lowest point of the weir crest (or sill), the flow rate over the weir will be determined by the (calculated) upstream depth alone.

X. Simulation Results

Results can be displayed dynamically on the main window during a hydraulic simulation, and they can also be outputted to a set of text and or CSV files when a simulation finishes. The most important results are the flow rate and water depth variables at locations within the system layout. But results can also include parameters like structure settings, specified flow rates and water depths, status indicators, hydraulic stability index, and many others. Most results are shown as a function of both elapsed time and location within the system layout.

X.1. Dynamic Results

Live results are automatically displayed in the branch and graphs views, which are at the upper right and lower right, respectively, of the main window. Some results can optionally be displayed in the system layout view. Also, the status of a simulation is shown dynamically on the right side of the toolbar, as seen below:



In the toolbar, the status will be shown as *Idle* (white), *Running* (green), *Paused* (yellow), or *Finished* (red). The *Idle* status is only seen when the app is open, but no simulations have taken place yet. Elapsed time is shown in days:hours:minutes:seconds. The dial meter depicts the hydraulic stability index, where three zones are:

- 1. **Red**: System is emptying
- 2. Green: System is relatively stable
- 3. **Blue**: System is filling

When the dial points straight to the left, some water is leaving the simulated system and no water is entering. When it points horizontally to the right, some water is entering the system, but no water is leaving. And, when it points straight up (as in the figure above), the sum of all inflows equals the sum of all outflows and the hydraulic stability index is zero (Chapter IX.2.2). However, even when the hydraulic stability index is zero, some internal instabilities (flow rates and water depths) can be manifested, at least temporarily, especially for a canal network containing many reaches.

The following sections describe the dynamic simulation results that can be displayed on the three views of the main window. The horizontal and vertical separators for the three views can be dragged to resize them at any time. The system layout view can optionally show simulation results, while the other two views will always show results during and after a simulation.

X.1.1. System Layout View

Certain node labeling options for the system layout view will show live values during a simulation. These are available for source and reach nodes, and they include:

- Flow rate
- Pool count

- Cumulative volume through a node
- Upstream and downstream reach water depths

When any of these node-labeling options are selected before or during a simulation, the results will update each time step to show the current values. Other display aspects of the system layout view will remain unchanged.

In the current version of the app, the pool count will always be a number from 0 to 2, in which a value of 0 means that the reach is completely empty. Usually, if there are two pools in a reach, one will soon disappear¹ or the pools will merge together to form a single pool. Or, an advancing pool merges with a static pool² at the downstream end of a reach (with a positive longitudinal bed slope) Thus, during a simulation, two pools in the same reach is typically an ephemeral situation.

The following figure shows an example of a two-reach layout in which flow rates are displayed during a simulation at the primary source node, and at the downstream end of each reach. If there are multiple regulators in parallel at any of these locations, the displayed flow rate is the sum of the discharges through each regulator.



X.1.2. Branch View

As described in Chapter I.3, the *Branch View* is located in the upper right area of the main window, showing a schematic side view of a branch with the stream-wise direction from left to right. If the system layout has more than one branch, you can right-click anywhere over the view to select one of the branches from a popup menu. The currently-selected branch has a check mark, and the items in the list are the names of the furthest upstream reach in each branch. When you change the selected

 $^{^{1}}$ For example, a pool draining away completely at the downstream end of a reach due to outflow from turnouts, regulators, and or seepage losses.

²Zero flow rates and a horizontal water surface.

branch, the displays immediately update for both the Branch and Graphs views.

The *Branch View* shows the water surface profiles in each reach of the selected branch. The profiles are displayed as elevations relative to the bottom of the view, and as depths relative to the channel beds (shown as brown lines). Triangle indicators at the downstream end of each reach show the settings of most potentially-adjustable regulator structures. When the horizontal side of the triangle is on top, it represents an overflow structure, like a weir. On the other hand, when the horizontal side is on the bottom, it represents and underflow structure, such as a gate. These setting indicators are not displayed for non-structure boundary conditions, or for pumps or valves.

In the figure below, an overflow structure is seen at the downstream end of *Upper Reach*, while an underflow structure is seen at the downstream end of *Lower Reach*. In this example, water is flowing from the first reach into the upstream end of second one because the downstream water surface in *Upper Reach* is above the overflow structure setting indicator. However, in this case the water level at the upstream end of *Lower Reach* is also above the weir crest, despite the elevation drop at the reach interface, so the weir is operating under submerged-flow conditions. Note that the reach names are displayed under each reach in this example of a two-reach branch.



The dotted lines in the view show the tops of the canals; when the water surface exceeds any part of these lines, overflow conditions are assumed to occur. The vertical line near the center of the view indicates the boundary (interface) between two reaches in series.

X.1.3. Graphs View

The *Graphs View* is at the lower right of the main window. You can choose from different types of results curves from a popup menu. The options include reach flow rates, water depths, regulator flows, turnout flows, cumulative volumes, and hydraulic stability index. The *Full ordinate range* option is only for hydraulic stability; when it is checked, the ordinate scale will be from -1 to +1 (the maximum possible range), regardless of the range of hydraulic stability values during a simulation. You can also hide upstream or downstream curves, but if you hide both, no curves will be shown, and you will see the message "All curves are hidden" in the lower left corner of the graphing area.

An example of reach flow rates for two reaches is shown below. *Reach* has both US and DS curves, but *Terminus* 2 has only one. This is because the two reaches are connected in series, so the upstream flow rate in *Terminus* 2 is exactly the same as the downstream flow rate in *Reach*. Vertical spikes in two of the curves are due to sudden regulator setting changes. The blue dotted curve is a horizontal line, indicating that the inflow to *Reach* is constant. Both the vertical and horizontal axes are automatically scaled to accommodate the range of values in the curves; whenever either axis needs to be re-sized, all of the curves will be redrawn, otherwise the curves will only be extended at the right side each time step. When a simulation finishes, the time scale (abscissa)



will be re-drawn, if necessary, so that it goes from zero up to the time the simulation ended.

The legend appends (US) and or (DS) to reach and regulator names. When a simulation is paused or finished, click on the legend and drag anywhere within the graphing region. If you bring it into a corner, it will remain there even if you resize the view or the main window. Otherwise, it will float. You can hide the legend from a command on the popup menu, as seen above.

You can modify curve attributes by right-clicking on the window and choosing **Display preferences...** from the popup menu. This brings up a sheet as seen in the example below, where you can select the color, line thickness and style of each curve. A sample curve is shown on this sheet for the selected item so that you can see what it will look like on the *Graphs View*.

Curves are constructed from straight-line segments. Click on the **All Heavy** button to change the line width to "Heavy" for a group of items. The name of the button changes according to the currently-selected line width (just to the left of the button). Groups of items are defined as upstream or down-stream reaches, upstream or downstream regulators, or all turnouts. Similarly, click on the **All Solid** (or whatever is currently displayed in the drop-down list to the left of the button) to change the line style to "Solid." These two buttons will be disabled under certain circumstances, such as when *Turnouts* is chosen and the selected reach has no turnouts.





You can also reset all line colors to the defaults, as defined on the *Curves* tab of the *Preferences* sheet, by clicking on the **Reset Colors** button. Whenever you click on any of the above three buttons, a confirmation message will appear for clarification.

Nothing that you do in this sheet will affect curve attributes for other branches, if there is more than one branch in the layout. To change attributes for another branch, right-click over the *Branch View* and choose it from the popup menu. All of these attributes are saved with the layout configuration file, so they are specific to a particular file.

The branch number is shown at the upper left of the sheet; in this case it is *Branch 1*, which is selected from the popup menu in the *Branch View*, as explained above. Note that there is a green check mark to the left of *Check 1+200*, *Check 2+350* and *Check 4+000* in the example above. This means that all three of the reaches in this branch are selected for curve drawing in the view. You can double-click over a reach name to toggle between the green check mark and a red **X**. When a red **X** is shown, the data for that reach will not be drawn. You can also click on the **Select All** or **Select None** buttons, and then double-click on any of the reach names that you want to include or exclude.

If you click just once on any reach name, the names of the regulators for that reach will appear in the list below the reach names. Use the *Reach location box* at the lower right to select *Upstream* or *Downstream* regulators. Then, after selecting the reach, you can click on one of the regulator names (there will also be at least one for the downstream end, and one for the upstream end) to select regulator curve attributes. In this case, the "Reach curve attributes" label will change to say "Regulator curve attributes." Click once on any reach name in the top list to revert back to reach curve attributes.

Turnout curve attributes work the same way as for regulators, except that a reach could have zero turnouts and the upstream or downstream reach end selection becomes irrelevant. To specify turnout curve attributes, select **Turnouts** from the *Structure kind* box, click once on a turnout name (if any) in the list, and then the label will say "Turnout curve attributes." For regulators and turnouts, there are also **Select All** or **Select None** buttons to the right of the list box, and you can use these in the same way as the analogous buttons for the reach list box. You can double-click on individual structure names to toggle between the green check mark and the red **X**.

X.2. Tabular Results

As described in Chapter IX.3, before a simulation begins, you can select the kinds of results to write tables of data to files. All of these data are written to one or more binary files after each time step. The binary format is more efficient that plain text in terms of file size and speed of access. When a simulation ends, the data are extracted and reorganized according to reach, regulator, and so on, and outputted to text and or CSV files according to your specifications. Then, the binary files are automatically deleted. The text and or CSV files are saved in the folder (path) that you specify, also as explained in Chapter IX.3.1.

For plain text files, elapsed time is shown in days, hours, minutes, and seconds (each separated by a colon). On the other hand, for CSV files, elapsed time is shown in seconds. This is because CSV files are meant to be imported into a spreadsheet application, and data processing and visualization in a spreadsheet is easier without the day-hour-minute-second formatting. Most other data are the same in both output file formats, but CSV formatting is simpler.

These tabular data show the evolution of hydraulic changes and status, both in terms of variables (calculated) and parameters (specified). Text files can be convenient to open in a word processor or text editing app, while CSV files are preferred for import into a spreadsheet app. The tables in these files all have header information to define each column and its units. Rows are elapsed time.

Note that pool profile data can be very lengthy because these results show the flow rate and water depth at every computational node, every time step, for every pool.

X.2.1. Summary Data

The *Summary* data include general information about a simulation for each time step. These include whether one or more reaches are over-topping (yes or no), total current stored volume of water in the system, net inflow rate, volume balance error, and other calculated values. The net inflow rate could be positive or negative, or zero, in any given time step.

The volume balance error, defined as a percentage, indicates the cumulative difference in stored volume versus cumulative net volume of inflow, multiplied by the size of the time step, and divided by an average stored volume:

$$error (\%) = 100 \left[\frac{\Delta t \Sigma (Q_{in} - Q_{out}) - \Delta V}{\frac{1}{2} (V_{current} + V_{initial})} \right]$$
(X.1)

where Δt is the time step size (s); Q_{in} and Q_{out} are the total system inflow and outflow $(m^3/s \text{ or } cfs)$, respectively, for a given elapsed time during a simulation. The summation symbol means that the cumulative net system inflow is calculated at each time step. The change in stored volume is $\Delta V = V_{current} - V_{initial}$, which is the increase in stored volume of water since the beginning of the simulation. Of course, the increase could be negative, corresponding to a decrease in volume. The denominator is the average of the current and initial total stored volumes to avoid division by zero when one or the other is zero at some point in a simulation.

Theoretically, the cumulative net inflow volume should be equal to the change in volume of stored water in the simulated system at all time steps. That is, the numerator of Eq. (X.1) should be equal to zero. However, due to numerical approximations, the volume balance will not, in general, be perfect.

If both $\mathcal{V}_{current}$ and $\mathcal{V}_{initial}$ are zero at any elapsed time, the percent error is defined to be zero.

But this is a special case in which all the reaches were (for all practical purposes) dry at the start of the simulation, and either no water entered during the simulation, or at some point the whole system de-watered.

Column headers in the text and CSV simulation results files are defined as follows:

- Any Over Topping If one or more reaches is over-topping its banks, the value is "Yes". Otherwise, it is "No."
- Stability Index This is the hydraulic stability index as described in Chapter IX.2.2. It can range from -1 to +1, where a value of zero means that $Q_{in} = Q_{out}$.
- Initial Vol The total volume of water stored in the reaches, throughout the system, at the beginning of the simulation. This value will remain constant for all elapsed times of a given simulation.
- **Current Vol** This is the total volume of water stored in the reaches, throughout the system, at the current elapsed time.
- Inflow The current total system inflow rate (always non-negative).
- **Outflow** The current total system outflow rate (always non-negative).
- Cumulatv Net Inflow The cumulative net inflow is a volume of water, which is defined above as: $\Delta t \Sigma (Q_{in} Q_{out})$.
- Error The current difference in stored and cumulative inflow volumes, which is also defined above: $\Delta t \Sigma (Q_{in} - Q_{out}) - \Delta \Psi$. Ideally, this value would always be zero.
- Vol Bal Error This is the volume balance error expressed as a percentage, defined by Eq. (X.1).

X.2.2. Source Data

Both primary and secondary source nodes can be split (Chapter II.4), meaning that that the upstream ends of two or more reaches can take water from a single source node. Thus, in these results outputs, all branches (that is, one or more) will be listed, and there will be a column to show the flow rate entering each of them. Branch names are the same as their furthest upstream reach.

The flow rate at a source can be *specified* or *calculated* – those are the two mode options. If a source node is split, the flow must be determined based on the inflow conditions to the upstream end of each reach that is connected to the source node. Source nodes themselves do not have regulator structures. Source nodes that are not split can have either mode (Chapter IX.4.1) – the flow rate can be specified or calculated. You can specify the flow rate in the *Sources* tab of the *Flows and Depths* sheet (Chapter IX.4), or you can apply a scheduling graph (Chapter VII). But if the flow rate is calculated, it will use the specified depth at the source node, the calibrations and settings (if adjustable) of the regulator(s) at the upstream end of the reach (which is connected to the source node), and the upstream depth in that reach.

These are the column headers:

- Flow Rate As described above, the flow rate at a source can be *specified* or *calculated*.
- **Depth** This is the specified water depth at the source node. It is used when the source operational mode is *Calculated flow rate*; otherwise, it is ignored. The depth is used in the calculations of discharge through one or more regulators in parallel at the upstream end of a reach, which is connected to the source node. That is, it becomes the upstream depth at the regulator(s).
- \bullet WS Elev You can choose to display water surface elevations instead of water depths.
- Branch 1 This is the flow rate entering a branch that begins at the source node. That means it is the flow rate entering the upstream end of a reach. If the source is split, there will be multiple

such columns (e.g. "Branch 2," and so on).

• **Total** The total flow rate leaving the source node is shown here. If the source is not split, this column will show the same values as the previous column, for each elapsed time.

X.2.3. Reach Data

Reach data results files include the following columns:

- # of Pools This is the number of pools of water in the reach.
- US Flow The upstream discharge entering the reach.
- **US Depth** The water depth at the upstream end of the reach, or **US Elev** for the corresponding water surface elevation.
- **DS Flow** The downstream discharge exiting the reach.
- **DS Depth** The water depth at the upstream end of the reach, or **DS Elev** for the corresponding water surface elevation.
- Seepage Seepage loss rate, which is positive for seepage outflow, and negative for seepage inflow.
- Evap Surface evaporation rate positive for losses, and negative for gains (e.g. direct rainfall).
- Over-Topping This is "Yes" if it is over-topping anywhere along the reach, or else "No."
- **Storage** The estimated volume of water stored within all pools in the reach.
- **US Froude** The Froude number at the upstream end of the reach.
- **DS Froude** The Froude number at the downstream end of the reach.
- US VelH The velocity head $(V^2/2g)$ at the upstream end of the reach.
- **DS VelH** The velocity head at the downstream end of the reach.

X.2.4. Regulator Data

When you check the *Regulator data* box, you can choose to include upstream and or downstream structures in the tabular results. Note that if you choose both the *Upstream* and *Downstream* boxes, redundant output will occur for branches containing multiple reaches – this is because, for reaches in series, the downstream regulators of one reach will be the same as the upstream regulators in another reach. See Chapter IV.1 regarding the automatic mirroring of regulator parameters for (non-split) reaches in series. Split reaches are described in Chapter II.4.

The following describe the column headers:

- **Opening** The is the opening, or setting of an adjustable structure, or "n/a" for a non-structure boundary condition.
- **US Depth** The water depth at the upstream side of the regulator, or **US Elev** for the water surface elevation.
- **DS Depth** The water depth at the downstream side of the regulator, or **DS Elev** for the water surface elevation.
- **Subm** This is the submergence value, equal to downstream depth divided by upstream depth, also accounting for the specified drop value.
- Flow The current discharge *through* (e.g. underflow gates) or *over* (e.g. weirs) the regulator.
- Gate Mode The gate mode can be: (1) setOpen; (2) setFlow; or, (3) auto. These correspond to the choice of *Specified setting*, *Specified flow rate*, and *Automatic setting* in the *Regulators* tab of the *Structure Operations* sheet (Chapter IX.5.1).
- Flow Regime This is the hydraulic regime under which the regulator is operating, and includes the following possible values: valve, orifice, nonOrifice, transition, freeOrifice, submOrifice,
freeNonOrif, submNonOrif, reverseFlow. Most of these correspond to free or submerged, and orifice or non-orifice. The regime is "none" for non-structure boundary conditions.

• **Issues** For some structure types, there can be issues that arise from conditions outside of those for which the calibration is valid. For example, when certain ratios are out of range with radial gates. The existence of any issue is a warning that the simulation results are not as accurate as they normally would be.

X.2.5. Turnout Data

Turnout results are similar to those of regulators, and these are the column header descriptions:

- **Opening** The is the opening, or setting of an adjustable structure, or "n/a" for a non-structure boundary condition.
- **US Depth** The water depth at the upstream side of the turnout, almost always calculated as part of the solution to the governing equations in the reach. Optionally, it will be **US Elev** for the water surface elevation.
- **DS Depth** The water depth at the downstream side of the turnout. This depth is specified, not calculated. Optionally, it will be **DS Elev** for the water surface elevation.
- **Subm** This is the submergence value, equal to downstream depth divided by upstream depth, also accounting for the specified drop value.
- Flow The current discharge *through* (e.g. underflow gates) or *over* (e.g. weirs) the turnout. A negative flow rate indicates that water is entering the reach through the turnout.
- Gate Mode The gate mode can be: (1) setOpen; (2) setFlow; or, (3) auto. These correspond to the choice of *Specified setting*, *Specified flow rate*, and *Automatic setting* in the *Turnouts* tab of the *Structure Operations* sheet (Chapter IX.5.2).
- Flow Regime This is the hydraulic regime under which the turnout is operating, and includes the following possible values: valve, orifice, nonOrifice, transition, freeOrifice, submOrifice, freeNonOrif, submNonOrif, reverseFlow. Most of these correspond to free or submerged, and orifice or non-orifice. The regime is "none" for non-structure turnout boundary conditions.
- **Issues** For some structure types, there can be issues that arise from conditions outside of those for which the calibration is valid. For example, when certain ratios are out of range with radial gates. The existence of any issue is a warning that the simulation results are not as accurate as they normally would be.

X.2.6. Pool Data

These results output files contain the details of calculations for every computational node in a pool, including the locations (distances along the reach) of each computational node. Unlike the other results output files, each row presents data for a particular computational node, rather than elapsed time during a simulation. However, elapsed time is shown for groups of computational nodes. For each elapsed time, the estimated volume of water stored in the pool is displayed.

As with source, reach, regulator and turnout results files, you can choose to show either water depths or water surface elevations.

- **Node Index** The index number of the computational node. Note that the total number of nodes can change with time when a pool is advancing and or receding.
- Location The distance of the computational node from the upstream end of the reach.
- Flow The current discharge at a given computational node.

- **Depth** The current water depth at a given computational node.
- WS Elev The current water surface elevation at a given computational node.
- Area The current cross-sectional area of water at a given computational node.
- Froude Number The Froude number at a given computational node.

X.3. Text Files of Parameters

Besides creating simulation results files, you can also output much of the configuration data to text files, which include parameters that you entered or specified through the app interface. From the main menu, you can select **File**, **Write Parameters** to create plain text files of layout configuration parameters. There are three file types: (1) Node Data; (2) Regulator Data; and, (3) Turnout Data. These files are sent to the same folder as specified in the Results Output sheet (Chapter IX.3). For regulator and turnout parameters, there are many columns of values, some of which only apply to certain types of boundary conditions – those that do not apply have a dash instead of a value.



Note that you cannot import parameters from these files back into mCanal – you must edit the parameters through the appropriate data entry sheets in the app. However, text files of scheduling data (Chapter VII.5) and pump and valve calibrations (Chapter VI.1.2-1.3) can be both created, edited, and imported back into the app.

Node data include some internal values, such as array indices and node IDs, as well as coordinates of the nodes (x, y), and of the Bezier control points (bx, by).

Annex A Weir Submerged Flow

According to Villemonte (1947), submerged flow across sharp-crested weirs (including rectangular, V-notch, and Cipoletti weirs) can be represented as a function of the upstream and downstream water depths, and the free-flow discharge:

$$Q_s = K_s Q_f \tag{A.1}$$

where Qs is the submerged-flow discharge $(m^3/s \text{ or cfs})$; Q_f is the free-flow discharge; and, K_s is a calculated coefficient ($0 \le K_s \le 1$). In the app, it is assumed that free-flow conditions exist across a weir when $|K_s| > 0.999$, otherwise the regime is submerged-flow. Villemonte's coefficient is defined as:

$$K_s = \left[1 - \left(\frac{y_d}{y_u}\right)^{n_f}\right]^{0.385} \tag{A.2}$$

where y_d is the downstream water depth, referenced from the elevation of the lowest part of the weir crest; y_u is the upstream water depth, referenced from the same datum; and, n_f is the exponent of the free-flow equation ($n_f \approx 1.5$ for rectangular weirs, or 2.5 for triangular weirs, with intermediate values for other notch shapes).

For regulator structures, the two water depth values are defined as:

$$y_u = h_u + \Delta E_s - G_o \tag{A.3}$$

$$y_d = h_d + \Delta E_s - G_o - \Delta E_d \tag{A.4}$$

Refer to the figure below for a an example side view of a submerged weir, and a definition of the terms in Eqs. (A.3) and (A.4). In general, ΔE_s and ΔE_d can be positive, negative, or zero. The weir setting, Go, is a constant for some weirs, but can vary within a specified operational range for adjustable weirs. Equations (A.3) and (A.4) also apply to turnout structures. However, in the model, ΔE_d is always equal to zero at turnouts. Note that K_s cannot be greater than 1.0 because that would imply that Q_s could be larger than Q_f for the same value of h_u , which does not make sense. Also note that submergence, S, is defined as y_d/y_u .

The figure below shows an example of a weir (side view) under submerged-flow conditions, in which ΔE_s and ΔE_d are both positive. It is clearly submerged flow because the downstream water surface elevation is above that of the lowest point on the weir crest. Note that the downstream reach refers to ΔE_d as ΔE_u – unless the upstream reach is a split reach, they refer to exactly the same value.



A.1. Interpolating Polynomial

The occurrence of an infinite slope at $y_d/y_u = 1$ tends to cause numerical instability in a hydraulic simulation involving submerged-weir flow for abscissa (y_d/y_u) values near 1.0. An approximation of the relationship is used in the app to flatten the curve in the vicinity of $y_d/y_u = 1$, providing better numerical stability for unsteady-flow conditions. In fact, the uses a polynomial to interpolate in the range $0.70 \leq y_d/y_u \leq 1.00$, where the curve slope is steepest. The polynomial is defined as follows:

- 1. The polynomial passes through the point (1.0, 0.0);
- 2. The polynomial passes through the point (x_p, K_s) , where x_p is a specified abscissa value (e.g. 0.70) and K_s is the ordinate value at x_p ;
- 3. The slope of the polynomial at (1.0, 0.0) is equal to zero; and,
- 4. The slope of the polynomial at (x_p, K_s) is equal to the slope of Villemonte's curve at the same location.

The above four conditions are sufficient to uniquely define a 3rd-degree polynomial of the form:

$$K_s = ax^3 + bx^2 + cx + d \tag{A.5}$$

where x is the value of the abscissa (y_d/y_u) ; and, the coefficients are defined as:

$$a = \frac{2\xi_1 \left(1 - x_p\right) / \xi_2 - \xi_4}{3 \left(x_p^2 - 1\right) + 2\xi_3 \left(1 - x_p\right) / \xi_2}$$
(A.6)

$$b = \frac{\xi_1 - a \,\xi_3}{\xi_2} \tag{A.7}$$

$$c = -3a - 2b \tag{A.8}$$

$$d = 2a + b \tag{A.9}$$

where,

$$\xi_1 = \left(1 - x_p^{n_f}\right)^{0.385} \tag{A.10}$$

$$\xi_2 = x_p^2 - 2x_p + 1 \tag{A.11}$$

$$\xi_3 = x_p^3 - 3x_p + 2 \tag{A.12}$$

$$\xi_4 = \frac{0.385 n_f x_p^{n_f - 1}}{\left(1 - x_p^{n_f}\right)^{0.615}} \tag{A.13}$$

The next figure gives a graphical representation of the K_s function for two typical values of n_f . Note that for $y_u = y_d$, the flow rate will be zero (because $K_s = 0$, and because there is a zero head differential).



With the approximating polynomial, the flow rate is still zero for $y_d = y_u$, as must be the case. But the slope of the polynomial is zero at $y_u/y_d = x = 1.0$ (instead of infinity), which helps considerably with convergence of the hydraulic equations in a transient simulation.

Using a sample value of $x_p = 0.7$ may be considered reasonable because the published experimental results published by Villemonte are in the $0.0 \le y_d/y_u \le 0.8$ range. The complete composite curve with $x_p = 0.7$ appears as shown in the figure below (using $n_f = 1.5$ for the Villemonte segment).



In this graph, the dashed curve represents the extrapolation of Villemonte's function to $y_d/y_u = 1.0$, and the interpolating polynomial is defined as:

$$K_s = 43.5356 \left(\frac{y_d}{y_u}\right)^3 - 109.6313 \left(\frac{y_d}{y_u}\right)^2 + 88.6558 \left(\frac{y_d}{y_u}\right) - 22.5601$$
(A.14)

which is for $0.7 \leq y_d/y_u \leq 1.0$, with $n_f = 1.5$. This is only one example. The equation would have different coefficients for other ranges of y_d/y_u , or other values of n_f . Note also that the K_s values in the y_u/y_d half (right side) of the graph are actually negative, whereas K_s is positive on the left half of the graph.

A.2. Reverse-Flow Polynomial

When h_d exceeds h_u , back (reverse) flow occurs across the weir. In this case, the abscissa of the curve is defined as h_u/h_d , and the flow rate is given a negative value. And,

$$K_s = -2\left(\frac{y_u}{y_d}\right)^3 + 3\left(\frac{y_u}{y_d}\right)^2 - 1 \tag{A.15}$$

where $0 \le y_u / y_d \le 1$.

Scoresby (1997) gives a similar but more complete representation of K_s . However, the results are nearly the same as those given by Villemonte.

Annex B Momentum Function

In the app, orifice gates include rectangular, circular, round, and radial gates, all of which may operate under orifice or non-orifice flow conditions. These may also be referred to as "underflow gates." For the other three types of orifice gates in mCanal, the momentum function is optionally applied to determine the flow regime for orifice flow: free or submerged. This is only done for regulators, not for turnouts. For non-orifice flow over weir crests, a different approach (Appendix A) is used to distinguish between free- and submerged-flow regimes.

As a quick check, if the water surface on the upstream side of the gate is above the bottom of the gate, then an orifice flow regime is probably (but not necessarily) occurring. In this case, the *momentum* function can be used to determine whether the orifice flow is free or submerged.

In the figure below, h_u is the depth at the upstream face of the gate, h_{vc} is the depth of the relatively high-velocity jet just downstream of the gate (depth at the *vena-contracta* section), relative to the bed of the upstream reach; h_{d1} is the depth at the downstream face of the gate, also relative to the upstream reach bed; and, h_{d2} is the downstream depth, a short distance downstream of the *vena contracta* location. Note that h_{d2} is referenced to the bed of the downstream reach.



Above the *vena contracta*, downstream of the gate, turbulent eddies are assumed to result in approximately zero average flow in the downstream direction. This assumption is also applied to the water downstream of the gate, below the upstream bed elevation.

Depth h_{d1} is calculated based on an assumption of zero hydraulic energy loss, as explained below. Depth h_{d2} is determined separately, based on downstream conditions, assuming subcritical flow. If the regime is supercritical downstream of the gate, then it is clear that the gate will operate under free-orifice conditions, obviating the need for the momentum function.

In the above figure, the gate delineates the boundary between two reaches, and ΔE is known as ΔE_d to the upstream reach, or ΔE_u to the downstream reach. Also, the above figure is an example in which ΔE has a positive value because it represents a drop in bed elevation in the downstream direction.

If the value of the momentum function corresponding to the *vena contracta* is greater than that corresponding to h_{d2} , free orifice flow will occur; if not, submerged orifice flow occurs across the gate,

meaning that a hydraulic jump has effectively moved upstream to the downstream face of the gate, drowning out the high-velocity jet at the *vena contracta*, as shown in the above figure. For any cross section, the momentum function is defined as follows:

$$M = h_c A + \frac{Q^2}{gA} \tag{B.1}$$

where A is the cross-sectional area (m^2) , which is a function of water depth and cross-sectional shape; h_c is the depth (m) from the water surface down to the centroid of the area; Q is discharge (m^3/s) ; and, g is the ratio of weight-to-mass (9.81 m/s^2).

The orifice-flow regime is determined as follows:

Free-Orifice Flow

If the downstream regime is supercritical, or if $h_{d2} \leq h_{vc} + \Delta E$, then the gate regime is free orifice flow and there is no need for the momentum function. Otherwise, free orifice flow occurs when:

$$M_{\rm vc} \ge M_{d2} \tag{B.2}$$

where the subscript "vc" refers to vena contracta.

Submerged-Orifice Flow

$$M_{\rm vc} < M_{d2} \tag{B.3}$$

The terms are based on Eq. (B.1) at different downstream locations, as per the illustration above, and are defined as follows:

$$M_{\rm vc} = (h_c A)_{d1} + \frac{Q^2}{gA_{vc}}$$
(B.4)

in which $(h_c A)_{d1}$ corresponds to depth h_{d1} ; and, A_{vc} corresponds to h_{vc} .

$$M_{d2} = (h_c A)_{d2} + \frac{Q^2}{gA_{d2}}$$
(B.5)

in which the depth h_c and both area terms are based on h_{d2} .

The depth just downstream of the gate is based on a specific-energy balance:

$$h_{d1} = h_u + \frac{Q^2}{2g} \left(\frac{1}{A_u^2} - \frac{1}{A_{vc}^2} \right)$$
(B.6)

As indicated above, the depth at the *vena contracta* is estimated as $h_{vc} = C_c G_o$, where C_c is the user-specified contraction coefficient, which usually about 0.61, from the theoretical ratio $\pi/(\pi + 2)$; and, G_o is the vertical gate opening at the centerline of the gate (m), as seen above. Note that the area A_{vc} is equal to the cross-sectional area corresponding to a depth of $h_{vc} + \Delta E$ minus the area for a depth of ΔE .

If the option to apply the momentum function is not chosen by the user, the app uses the simple ratio defined in Annex E.2.2 to distinguish between free and submerged orifice flow.

Annex C USBR Radial-Gate Calibration

The following calibration equations are based on Buyalski (1983).

C.1. Free Orifice Flow

The free-orifice flow equation for radial gates is:

$$Q_f = C_{fcda} C_{df} G_o G_w \sqrt{2gh_u} \tag{C.1}$$

where Q_f is discharge $(m^3/s \text{ or cfs})$; G_o is vertical gate opening (m or ft); G_w is width of the gate opening (m or ft); h_u is the upstream water depth (m or ft); and, g is the ratio of weight-to-mass (taken to be $9.81m/s^2$ or $32.2ft/s^2$). C_{fcda} is defined below. Unless you have custom calibration data for a radial gate, let $C_{df} = 1.0$.

C.1.1. Eccentricity

$$AFE = \sqrt{0.00212 \left[1.0 + 31.2 \left(\frac{G_r}{P} - 1.6 \right)^2 \right]} + 0.901$$
(C.2)

$$BFE = \sqrt{0.00212 \left[1.0 + 187.7 \left(\frac{G_r}{P} - 1.635 \right)^2 \right]} - 0.079$$
(C.3)

$$FE = AFE - BFE\left(\frac{G_o}{P}\right) \tag{C.4}$$

where G_r is the radial-gate radius (*m* or *ft*); and, *P* is the pinion height (*m* or *ft*).

C.1.2. Directrix

$$AFD = 0.788 - \sqrt{0.04 \left[1.0 + 89.2 \left(\frac{G_r}{P} - 1.619 \right)^2 \right]}$$
(C.5)

$$BFD = 0.0534 \left(\frac{G_r}{P}\right) + 0.0457 \tag{C.6}$$

$$FD = 0.472 - \sqrt{BFD\left[1.0 - \left(\frac{G_o}{P} - AFD\right)^2\right]}$$
(C.7)

C.1.3. Focal Distances

For $(G_o/P) \le 0.277$:

$$FX_1 = 1.94 \left(\frac{G_o}{P}\right) - 0.377 \tag{C.8}$$

For $(G_o/P) > 0.277$:

$$FX_1 = 0.18 \left(\frac{G_o}{P}\right) + 0.111 \tag{C.9}$$

and,

$$FY_1 = 0.309 - 0.192 \left(\frac{G_o}{P}\right)$$
 (C.10)

$$FXV = \frac{h_u}{P} - FX_1 \tag{C.11}$$

Finally, the free-flow coefficient is:

$$C_{fcda} = C_{correct} \left(FY_1 + \sqrt{FE^2 \left(FD + FXV\right)^2 - FXV^2} \right)$$
(C.12)

For the hard-rubber-bar gate-lip seal design, $C_{correct} = 1$. The correction on C_{fcda} for the music note gate-lip seal design is:

$$C_{correct} = 0.125 \left(\frac{G_o}{P}\right) + 0.91 \tag{C.13}$$

The correction on C_{fcda} for the **sharp edge** gate-lip seal design is:

$$C_{correct} = 0.11 \left(\frac{G_o}{P}\right) + 0.935 \tag{C.14}$$

The above corrections were chosen from the linear options proposed by Buyalski (1983).

C.2. Submerged Orifice Flow

The submerged-orifice rating equation is:

$$Q_s = C_{scda} C_{ds} G_o G_w \sqrt{2gh_u} \tag{C.15}$$

where C_{scda} is the submerged-flow discharge coefficient; C_{ds} is a user-defined coefficient; and, all other terms are as previously defined. Both C_{scda} and C_{ds} are dimensionless. Unless you have custom calibration data for a particular radial gate, let $C_{ds} = 1.0$. Otherwise, enter a value for Orifice submerged flow coefficient in the **Structures Data** sheet, **Calibration** tab. The Orifice submerged flow exponent value is ignored in the app for radial gates. Note that the square-root term in Eq. (C.15) does not include the downstream water depth, h_d , but it is included in the definition of C_{scda} , as defined by the following series of conic equations.

C.2.1. Directrix

$$ADA = \left[11.98\left(\frac{G_r}{P}\right) - 26.7\right]^{-1} \tag{C.16}$$

$$ADB = 0.62 - 0.276 \left(\frac{P}{G_r}\right) \tag{C.17}$$

$$AD = \left[ADA\left(\frac{G_o}{P}\right) + ADB\right]^{-1} \tag{C.18}$$

$$BD = \left(\frac{G_o}{P}\right) \left[0.025 \left(\frac{G_r}{P}\right) - 2.711\right] + 0.071 - 0.033 \left(\frac{G_r}{P}\right) \tag{C.19}$$

$$D = \left[AD\left(\frac{h_d}{P}\right) + BD\right]^{-1.429} \tag{C.20}$$

C.2.2. Eccentricity

$$AEA = 0.06 - 0.019 \left(\frac{G_r}{P}\right) \tag{C.21}$$

$$AEB = 0.996 + 0.0052 \left(\frac{G_r}{P}\right)$$
 (C.22)

$$AE = \left[AEA\left(\frac{G_o}{P}\right) + AEB\right]^{-1} \tag{C.23}$$

$$BEK = 0.32 - 0.293 \left(\frac{G_r}{P}\right) \tag{C.24}$$

$$BE = BEK + \sqrt{0.255 \left[1 + 1.429 \left(\frac{G_o}{P} - 0.44 \right)^2 \right]}$$
(C.25)

$$ER = AE(D) + BE \tag{C.26}$$

$$E = \sqrt{\ln \frac{ER}{D}} \tag{C.27}$$

C.2.3. Vector V_1

$$V_1 = \frac{E(D)}{1+E} \tag{C.28}$$

C.2.4. Focal Distance

$$AF = \left(\frac{G_o}{P}\right) \left[0.038 - 0.158 \left(\frac{P}{G_r}\right)\right] + 0.29 - 0.115 \left(\frac{G_r}{P}\right) \tag{C.29}$$

$$BF = \left(\frac{P}{G_o}\right) \left[0.0445 \left(\frac{P}{G_r}\right) - 0.321\right] + 0.155 - 0.092 \left(\frac{P}{G_r}\right) \tag{C.30}$$

$$FY = BF - \frac{AF(h_d)}{P} \tag{C.31}$$

If $FY \leq 0$ then let FY = FX = 0. Otherwise, calculate FX:

$$FX = \sqrt{V_1^2 + FY^2} - V_1 \tag{C.32}$$

Then, calculate VX:

$$VX = \frac{h_u}{P} - V_1 - \frac{h_d}{P} - FX \tag{C.33}$$

The correction on C_{scda} for the **music note** gate-lip seal design is:

$$C_{correct} = 0.39 \left(\frac{G_o}{P}\right) + 0.85 \tag{C.34}$$

The correction on C_{scda} for the ${\bf sharp}\ {\bf edge}$ gate-lip seal design is:

$$C_{correct} = 0.11 \left(\frac{G_o}{P}\right) + 0.9 \tag{C.35}$$

For the hard-rubber-bar gate-lip seal design, $C_{correct} = 1$. The submerged-flow coefficient is:

$$C_{scda} = C_{correct} \left(FY + \sqrt{E^2 (D + VX)^2 - VX^2} \right)$$
(C.36)

Annex D Horseshoe Cross-Section Geometry

Standard horseshoe cross sections have been designed and built by the U.S. Bureau of Reclamation and others, and are found in many tunnels for water conveyance in the USA and several other countries. The horseshoe cross section can be considered a variation of a circular section whereby the lower half of the tunnel height is widened to facilitate the passage of machinery and equipment during tunnel construction, and later for inspection and maintenance purposes. The standard horseshoe cross section is defined by the intersection of four circles: three offset circles of radius r, and one of radius r/2, as shown in the figure below, where a shape resembling a "horseshoe" might be discerned from the bold curves in the center region. Some of the geometric relationships can be derived from the equations for the intersecting circles, while others are obtained through integration and differentiation of those equations. Various different forms of the mathematical relationships can be derived, but those presented below are believed to be the simplest.



Davis and Sorensen (1969) presented tables of approximate non-dimensional values for some of the geometric relationships of standard horseshoe cross sections. Hu (1973, 1980) presented non-dimensional equations for cross-sectional area, top width, and wetted perimeter for standard horseshoe cross sections, omitting the depth to area centroid, and the partial derivative of centroid depth with respect to water depth. Merkley (2004) published additional equation forms.

The equations for cross-sectional area, wetted perimeter, top width, and depth to area centroid are given below as functions of the depth of water in standard horseshoe sections. The depth can be divided into three logical segments as defined by the intersections of the four circles, where $h_1 + h_2 + h_3 = r$. This is illustrated in the figure below.



Determine h_1 by solving for the intersection of two of the larger circles:

$$h_1 = r \left[1 - \left(\frac{1 + \sqrt{7}}{4} \right) \right] \tag{D.1}$$

then,

$$h_2 = \frac{r}{2} - h_1 \tag{D.2}$$

and,

$$h_3 = \frac{r}{2} \tag{D.3}$$

Separate definitions for each geometric relationship are given below for each of the three depth segments, where water depth, h, is measured from the center of the lower curve.

D.1. Water Surface Width

The water surface width, T, inside the cross section is a function of the depth, h. The surface width is zero at h = 0, and reaches a maximum of T = r when h = r/2.

For $0 < h \leq h_1$:

$$T = 2r\sqrt{1 - \left(1 - \frac{h}{r}\right)^2} \tag{D.4}$$

$$T = 2\sqrt{r^2 - \left(h - \frac{r}{2}\right)^2} - r$$
 (D.5)

For r/2 < h < r:

For $h_1 < h \leq r/2$:

$$T = r\sqrt{1 - \left(1 - \frac{2h}{r}\right)^2} \tag{D.6}$$

D.2. Cross-Sectional Area

The cross-sectional area is the area occupied by water, in a direction perpendicular to the main (streamwise) direction of flow.

For $0 < h \leq h_1$:

$$A = (h - r)\sqrt{h(2r - h)} + r^2 \left[\sin^{-1} \left(\frac{h - r}{r} \right) + \frac{\pi}{2} \right]$$
(D.7)

For $h_1 < h \leq r/2$:

$$A = r^{2} \left[C_{2} + \sin^{-1} \left(\frac{2h - r}{2r} \right) \right] - \left(h - \frac{r}{2} \right) \left(r - \sqrt{r^{2} - \left(h - \frac{r}{2} \right)^{2}} \right) + A_{1}$$
(D.8)

where A_1 is the area corresponding to $h = h_1$ from Eq. (D.7). The constant C_2 is defined as:

$$C_2 = \frac{C_1}{2} \left(1 - \sqrt{1 - \frac{C_1^2}{4}} \right) - \sin^{-1} \left(\frac{C_1}{2} \right)$$
(D.9)

where,

$$C_1 = 1 - \left(\frac{1+\sqrt{7}}{2}\right)$$
 (D.10)

For $r/2 < h \leq r$:

$$A = \left(h - \frac{r}{2}\right)\sqrt{h(r-h)} + \frac{r^2}{4}\sin^{-1}\left(\frac{2h-r}{r}\right) + A_2$$
(D.11)

where A_2 is the area corresponding to h = r/2, which is a simplified version of Eq. (D.8):

$$A_2 = r^2 C_2 + A_1 \tag{D.12}$$

D.3. Depth to the Area Centroid

The depth to the area centroid, \bar{h} , is measured from the bottom of the channel section up to the location of the centroid. The depth from the water surface down to the area centroid is required for momentum-function calculations, and can be defined as $h_c = h - \bar{h}$.

For $0 \le h \le h_1$, the moment of area with respect to x is:

$$M_x = -\frac{2}{3} \left[h \left(2r - h \right) \right]^{3/2} \tag{D.13}$$

and the depth to the area centroid is:

$$\bar{h} = \frac{M_x}{A_x} + r \tag{D.14}$$

where A_x is as calculated by Eq. (D.7).

For $h_1 < h \leq r/2$, the moment of area with respect to x is:

$$M_x = r^3 C_3 - \frac{r}{2} \left(h - \frac{r}{2} \right)^2 - \frac{2}{3} \left[r^2 - \left(h - \frac{r}{2} \right)^2 \right]^{3/2}$$
(D.15)

in which,

$$C_3 = \frac{C_1^2}{8} + \frac{2}{3} \left(1 - \frac{C_1^2}{4} \right)^{3/2} \tag{D.16}$$

where C_3 is a constant; and, C_1 is as defined above. The value of M_x will be negative because it is calculated based on coordinate origins at h = r/2, so the depth to the area centroid for a given water depth, h, must be shifted upward by the amount r/2:

$$\bar{h}_x = \frac{r}{2} + \frac{M_x}{A_x} \tag{D.17}$$

which will be a positive value, where A_x is the cross-sectional area from h_1 up to some depth h:

$$A_{x} = r^{2} \left[C_{2} + \sin^{-1} \left(\frac{2h - r}{2r} \right) \right] - \left(h - \frac{r}{2} \right) \left(r - \sqrt{r^{2} - \left(h - \frac{r}{2} \right)^{2}} \right)$$
(D.18)

which is Eq. (D.8) less the A_1 term, and C_2 is as defined above. The composite value of \bar{h} must account for the calculations up to $h = h_1$, so for depths from h_1 to r/2, the following area-weighted relationship is used to obtain the depth to the area centroid:

$$\bar{h} = \frac{A_x \left(\frac{r}{2} + \frac{M_x}{A_x}\right) + A_1 \bar{h}_1}{A_x + A_1} \tag{D.19}$$

where A_1 and \bar{h}_1 are the values corresponding to $h = h_1$. See Eqs. (D.7) and (D.14).

For $r/2 < h \leq r$, the moment of area with respect to x is:

$$M_x = \frac{r^3}{12} - \frac{2}{3} \left(h(r-h) \right)^{3/2}$$
(D.20)

The cross-sectional area from h = r/2 up to depth h is:

$$A_x = \left(h - \frac{r}{2}\right)\sqrt{h(r-h)} + \frac{r^2}{4}\sin^{-1}\left(\frac{2h-r}{r}\right)$$
(D.21)

which is Eq. (D.11) without the A_2 term. The composite value of \bar{h} must account for the calculations up to h = r/2, so for depths from r/2 to r, the following area-weighted relationship is used to obtain the exact depth to the area centroid:

$$\bar{h} = \frac{A_x \left(\frac{r}{2} + \frac{M_x}{A_x}\right) + A_2 \bar{h}_2}{A_x + A_2} \tag{D.22}$$

where A_2 and \bar{h}_2 are the values corresponding to h = r/2. See Eqs. (D.8) and (D.19).

D.4. Wetted Perimeter

For $0 \le h \le h_1$:

$$W_p = 2r\cos^{-1}\left(1 - \frac{h}{r}\right) \tag{D.23}$$

For $h_1 < h \leq r/2$:

$$W_p = 2r \left[\cos^{-1} \left(\frac{r - 2h}{2r} \right) - \cos^{-1} \left(\frac{-C_1}{2} \right) \right] + W_{p1}$$
(D.24)

where C_1 is as defined above in Eq. (D.10); and, W_{p1} is the wetted perimeter corresponding to $h = h_1$, from Eq. (D.23). The $\cos^{-1}(-C_1/2)$ term is a constant.

For $r/2 < h \leq r$:

$$W_p = r \left[\cos^{-1} \left(1 - \frac{2h}{r} \right) - \frac{\pi}{2} \right] + W_{p2}$$
 (D.25)

where W_{p2} is the wetted perimeter corresponding to h = r/2, from Eq. (D.24).

Annex E

Transitional Flow at Underflow Gates

In general, four different flow regimes can occur at underflow gate structures. Each of the regimes has a unique calibration equation, and flow can transition from any regime to the other. In practice, many canal gate structures and channel constrictions (e.g. flumes) operate mostly under a single flow regime, but simulations must allow for the possibility that the regime might change at a structure. The four regimes are defined as:

- 1. Free orifice (FO)
- 2. Submerged orifice (SO)
- 3. Free nonorifice (FN)
- 4. Submerged nonorifice (SN)

	Free	Submerged
Orifice	FO	SO
Non-orifice	FN	SN

Changes between some of the flow regimes usually involve flow rate discontinuities which can case numerical instability during a hydraulic simulation. This annex deals with a transitional flow algorithm which is applied in the app with the intent on reducing the numerical problems associated with flow regime changes at underflow gate structures.

The following figures illustrate side views of the four flow regimes through a gate structure, with the flow moving from left to right, where G_o is vertical gate opening, h_u is upstream water depth, and h_d is downstream water depth.

Free Orifice Flow



Submerged Orifice Flow







Submerged Non-Orifice Flow



Structures such as rectangular sluice gates, radial gates, and circular gates can potentially accommodate any of the above four flow regimes. Non-gated channel constrictions, such as flow measurement flumes, can have both free and submerged non-orifice flow, but not orifice flow. Weirs also have free and or submerged non-orifice flow, but the transition from free to submerged, and vice versa, can be handled differently.

E.1. Transitional Flow Algorithm

The algorithm for creating a smooth transition between flow regimes involves the application of a 3^{rd} -degree polynomial interpolation within a specified parameter range, centered about the mathematically precise transition value which is made to correspond to a magnitude of unity. That is, the threshold between one regime and another is defined by an algebraic expression, and this threshold is artificially diffused toward each of the two regimes (whichever they might be) in order to reduce the typical discharge discontinuity manifested at the exact threshold value. In some cases this numerical diffusion is physically justified, and in others it is not. However, the objective of the approach to dealing with transitional flow for the afore-mentioned structures is to provide acceptable numerical stability during unsteady flow simulations.

The polynomial defines a weighting factor, W, which is applied to the respective equations for each of the two pairs of flow regimes. The W value is different for each pair of flow regimes, but in all cases, $0 \le W \le 1$. At the limits, when W = 0 or W = 1, there is no transition effect — it is all one regime or the other. The figure below shows a sample transitional flow curve in which the abscissa ("ratio") refers to the threshold expression for regime transition.



Transition between flow regimes only occurs between $(1 - \chi)$ and $(1 + \chi)$, where the parameter χ is user-specified, but limited to $0.0 < \chi < 0.5$. When $\chi = 0$, the transition from one regime to the other is abrupt, generally associated with a discharge discontinuity. The 3^{rd} -degree polynomial is defined as:

$$W = ax^3 + bx^2 + cx + d \tag{E.1}$$

where x is the ratio value on the abscissa of the above figure. Defining $L = (1 - \chi)$ and $R = (1 + \chi)$, the coefficients are:

$$a = \left[R^3 - L^3 + 3\left(R^2 L - L^2 R \right) + \frac{3\left(R^2 - L^2 \right)^2}{2(L - R)} \right]^{-1}$$
(E.2)

$$b = \frac{3a(R^2 - L^2)}{2(L - R)}$$
(E.3)

$$c = \frac{a(R^3 - L^3) + b(R^2 - L^2) - 1}{L - R}$$
(E.4)

$$d = -aL^3 - bL^2 - cL \tag{E.5}$$

The above polynomial has zero slope at x = L and x = R, and it passes through W = 0 at x = L, and through W = 1 at x = R. By symmetry, the curve passes through W = 0.5 at x = 1, where an unweighted average of two equations results.

E.2. Orifice Free – Submerged Transition

This transition can generally be abrupt (in reality it usually is) and not cause problems with a hydraulic simulation. But a transitional range through χ_o , which is referred to as the "orifice free – submerged transition range" can, nevertheless, be specified in the app, as described in Section VIII.3.2 of this Users' Guide.

With orifice flow, the threshold expression is defined in the app in either of two different ways, depending on a user-specified option (Chapter VIII.3.5). From a technical standpoint, the momentum function is generally preferred; the other option is somewhat simpler.

E.2.1. Applying the momentum function (Annex B):

$$\Omega_o = \frac{M_{d2}}{M_{vc}} = 1 \tag{E.6}$$

E.2.2. Not applying the momentum function:

$$\Omega_o = \frac{h_{d2}}{C_c G_o + \Delta E} = 1 \tag{E.7}$$

where h_{d2} is the calculated downstream depth (relative to the gate location) as indicated in Annex B. Note that, in either case, the water depth at the *vena contracta* is defined as $h_{vc} = C_c G_o$.

When the ratio is greater than unity, the regime is free orifice, and when less than unity, it is submerged orifice. But, when applying a transition range, the regime is strictly free orifice for:

$$\Omega_o < 1 - \chi_o \qquad (W_o = 0 \text{ and } F_o = F_{fo}) \tag{E.8}$$

And, it is strictly submerged orifice when:

$$\Omega_o > 1 + \chi_o \qquad (W_o = 1 \text{ and } F_o = F_{so}) \tag{E.9}$$

If the ratio is between the above limits, it is considered to be transitional flow between free and submerged orifice, and the following weighting factor is applied:

$$W_o = a \,\Omega_o^3 + b \,\Omega_o^2 + c \,\Omega_o + d \tag{E.10}$$

In the transition zone, the function defining the relationship between water depths and flow rate for and underflow gate is defined as:

$$F_o = W_o F_{so} + (1 - W_o) F_{fo} = 0$$
(E.11)

because in the app all structure functions are defined to be equal to zero, and where F_{so} and F_{fo} are the submerged- and free-orifice equations, respectively.

E.3. Orifice – Non-orifice Transition

The threshold expression between orifice and non-orifice flow is defined as:

$$\frac{C_o \left(h_u + \Delta E_s\right)}{G_o} = 1 \tag{E.12}$$

where C_o is a user-specified coefficient ($0.8 \le C_o \le 0.99$); G_o is the vertical gate opening; h_u is upstream water depth; and, ΔE_s is the upstream invert elevation drop. The value of ΔE_s is negative when the invert of the gate opening is above the upstream canal bed. But perhaps in most cases, ΔE_s will be equal to zero.

This expression relates to the upstream "hydraulic seal" at the gate structure. When $h_u + \Delta E_s = G_o$, there is no hydraulic seal and non-orifice flow prevails. In Section VIII.3.3, χ_{on} is called the "orifice – non-orifice transition range," and in Section VIII.3.4, C_o is referred to as the "hydraulic seal coefficient." The regime is taken to be strictly non-orifice for:

$$\frac{C_o \left(h_u + \Delta E_s\right)}{G_o} < 1 - \chi_{on} \qquad (W_{on} = 0) \tag{E.13}$$

and strictly orifice for:

$$\frac{C_o \left(h_u + \Delta E_s\right)}{G_o} > 1 + \chi_{on} \qquad (W_{on} = 1)$$
(E.14)

otherwise the regime is transitional between orifice and non-orifice flow, and the following weighting factor is applied:

$$W_{on} = a \left[\frac{C_o \left(h_u + \Delta E_s \right)}{G_o} \right]^3 + b \left[\frac{C_o \left(h_u + \Delta E_s \right)}{G_o} \right]^2 + c \left[\frac{C_o \left(h_u + \Delta E_s \right)}{G_o} \right] x + d$$
(E.15)

where W_{on} is the orifice – non-orifice weighting factor $(0 \le W_{on} \le 1)$; and a, b, c, and d are as defined in Eqs. (E.2) – (E.5).

If the regime is strictly non-orifice (i.e. $W_{on} = 0$), then it is unnecessary to deal with the free- and submerged-flow orifice equations. Likewise, if the regime is strictly orifice (i.e. $W_{on} = 1$), it is irrelevant to apply the non-orifice equations. However, if the ratio is between the above limits, it is transitional flow and the gate function is defined as a weighted average of the two:

$$F_{on} = W_{on}F_o + (1 - W_{on})F_{no} = 0$$
(E.16)

where F_o and F_{no} are the orifice and non-orifice gate functions, respectively.

Annex F

References

- Addison, H., 1949. Hydraulic Measurements. Chapman and Hall, London.
- Brater, E.F., and H.W. King. 1976. Handbook of hydraulics. 6th edition. McGraw-Hill Book Company, New York, N.Y. 583 pp.
- Buyalski, C.P. 1983. Discharge algorithms for canal radial gates. Technical Report REC-ERC-83-9. U.S. Bureau of Reclamation, Denver, CO. 232 pp.
- Chow, V.T. 1959. Open-channel hydraulics. McGraw-Hill Book Company, New York, N.Y. 680 pp.
- Clemmens, A.J., Strelkoff, T.S., and Replogle, J.A. 2003. Calibration of Submerged Radial Gates. ASCE J. Hydr. Engrg. Vol. 129, No. 9. September. pp. 680-687.
- Finnemore, E.J., and Franzini, J.B. 2002. Fluid Mechanics with Engineering Applications. 10th edition. McGraw-Hill Book Company, New York, N.Y. 790 pp.
- Flinn, A.D., and C.W.D. Dyer. 1894. The Cipoletti trapezoidal weir. Trans. ASCE, Vol. 32.
- French, R.H. 1985. Open-channel hydraulics. McGraw-Hill Book Company, New York, N.Y. 705 pp.
- Kindsvater, C.E., and Carter, R.W.C. 1957. Discharge characteristics of rectangular thin-plate weirs. ASCE J. Hydr. Div. 83 (HY6): 1-36.
- Larson, R., and Edwards, B. 2017. Calculus. 11th edition. Cengage Learning, Boston, MA.
- Merkley, G.P. 2004. Standard horseshoe cross section geometry. Agric. Water Manag., Elsevier (www.elsevier.com/locate/agwat). 71(2005): 61-70.
- Parshall, R.L. 1945. Improving the Distribution of Water to Farmers by Use of the Parshall Measuring Flume. USDA Soil Conservation Service, in cooperation with the Colorado Agric. Exp. Station, Colorado State Univ., Fort Collins, CO.
- Parshall, R.L. 1953. Parshall Flumes of Large Size. USDA Soil Conservation Service, in cooperation with the Colorado Agric. Exp. Station, Colorado State Univ., Fort Collins, CO.
- Press, W.H., S. A. Teukolsky, W. T. Vetterling, and B. P. Flannery. 1992. Numerical recipes in C: the art of scientific computing. 2nd Ed. Cambridge Univ. Press, Cambridge, U.K. 994 pp.
- Robertson, J.A., Cassidy, J.J., Chaudry, M.H. 1998. Hydraulic Engineering, 2nd Ed. John Wiley and Sons, New York, N.Y.
- Shen, J. 1981. Discharge characteristics of triangular-notch thin-plate weirs. Water Supply Paper 1617-B. U.S. Geological Survey.
- Skogerboe, G.V., Hyatt, M.L. and Eggleston, K.O. 1967a. Design and calibration of submerged open channel flow measuring structures, Part 1: Submerged flow. Utah Water Research Laboratory. Logan, Utah.

- Skogerboe, G.V., M.L. Hyatt, R.K. Anderson, and Eggleston, K.O. 1967. Cutthroat flumes. Utah Water Research Laboratory Report WG31-4. 37 pp.
- Temeepattanapongsa, S., Merkley, G.P., Barfuss, S., and Smith, B. 2013. Generic unified rating for Cutthroat flumes. J. Irrig. Sci. 32(1): 29-40.
- Temeepattanapongsa, S., Merkley, G.P., Barfuss, S., and Smith, B. 2013. Generic free-flow rating for Cutthroat flumes. ASCE J. Hydr. Engrg. 139(7): 727-735. Torres, A. 2006. Transition Submergence and Hysteresis Effects in Three-Foot Cutthroat Flumes. M.S. Thesis, BIE Department, Utah State Univ., Logan, UT.
- Tullis, J.P. 1989. Hydraulics of Pipelines: Pumps, Valves, Cavitation, Transients. John Wiley & Sons, Inc. New York, N.Y.
- Uni-Bell Plastic Pipe Association. 1977. Handbook of PVC pipe: design and construction. Uni-Bell Plastic Pipe Association, Dallas, TX.
- Villemonte, J.R. 1947. Submerged-weir discharge studies. Engrg. News Record, p. 866.
- Wahl, T.L. 2005. Refined Energy Correction for Calibration of Submerged Radial Gates. ASCE J. Hydr. Engrg. Vol. 131, No. 6. June. pp. 457-466.
- Wahlin, B., and Replogle, J. 1996. Discharge Characteristics of Overshot Gates. Proceedings of the North American Water and Environment Congress & Destructive Water. ASCE, Civil Engineering Database.

Bibliography

- Abt, S.R., and Staker, K.J. 1990. Rating correction for lateral settlement of Parshall Flumes. J. Irrig. and Drain. Engrg., ASCE, 116(6):797-803.
- Abt, S.R., and Staker, K.J. 1992. Closure to 'Discussion of Rating correction for lateral settlement of Parshall flumes. J. Irrig. and Drain. Engrg., ASCE, 118(2): 339-340.
- Abt, S.R., Cook, C., Staker, K.J., and Johns, D.D. (1992). Small Parshall flume rating correction. J. Hydr. Engrg., ASCE, 118(5): 198-803.
- Abt, S.R., Florentin, C.B., Genovez, A., and Ruth, B.C. 1995. Settlement and Submergence Adjustments for Parshall Flume. ASCE J. Irrig. and Drain. Engrg., 121(5): 317-321.
- Abt, S.R., Genovez, A., and Florentin, B. 1994. Correction for Settlement in Submerged Parshall Flumes. J. Irrig. and Drain. Engrg., ASCE, 120(3): 676-682.
- Abt, S.R., Ruth, B.C., Mitchell, C.M., Lipscomb, C.M. 1997. Condition Assessment of Parshall Flumes in Colorado, Part 2. Water Operation and Maintenance Bulletin No. 180, U.S. Government Printing Office, Washington, DC.
- Abt, S.R., Ruth, B.C., Skowron, E.M. 1998. Rating Adjustment for Settlement of Cutthroat Flumes. J. Irrig. and Drain. Engrg., ASCE, 124(6): 311-315.

- Abt, S.R., Thompson, K., and Staker, K. 1989. Discharge correction for longitudinal settlement of Parshall flumes. Trans., ASAE, 32(5): 1541-1544.
- Ackers, P., White, W. R., Perkins, J.A., and A.J.M. Harrison. 1978. Weirs and flumes for flow measurement. John Wiley and Sons, New York, N.Y.
- Alminagorta, O., and Merkley, G.P. 2009. Transitional Flow between Orifice and Nonorifice Regimes at a Rectangular Sluice Gate. J. Irrig. and Drain. Engrg., ASCE, 135(3): 1-6.
- Blaisdell, F.W. 1994. Results of Parshall Flume Tests. ASCE J. Irrig. and Drain. Engrg., 120(2): 278-291.
- Bos, M.G., J.A. Replogle, and A.J. Clemmens. 1984. Flow Measurement Flumes for Open Channel Systems. John Wiley & Sons, New York, NY. 321 pp.
- Chittaladakorn, S., and Merkley, G.P. 2005. Classifier System for Rule-Based Operation of Canal Gates. J. Water Res. Planning and Manag., ASCE. 131(1): 3-13.
- Chun, S.J., and Merkley, G.P. 2008. ODE solution to the characteristic form of the Saint-Venant equations. J. Irrig. Sci. 26(3): 213-222. March.
- Clemmens, A.J., Strelkoff, T.S., and Replogle, J.A. 2003. Calibration of submerged radial gates. J. Hydr. Engrg., ASCE. 129(9): 680-687.
- Clemmens, A.J., T.L. Wahl, M.G. Bos, and J.A. Replogle. 2001. Water Measurement with Flumes and Weirs. ILRI Publication 58. International Institute for Land Reclamation and Improvement, Wageningen, The Netherlands. 382 pp.
- Cone, V.M. 1917. The Venturi Flume. J. of Ag. Research, 9(4): 115-123.
- Daugherty, R.L., and J. B. Franzini. 1977. Fluid mechanics with engineering applications. 7th edition. McGraw-Hill Book Company, New York, N.Y. 564 pp.
- Davis, C.V. and K.E. Sorensen (eds.). 1969. Handbook of applied hydraulics. McGraw-Hill Book Company, New York, N.Y.
- Davis, R.W., and Deutsch, S. 1980. A numerical-experimental study of Parshall flumes. J. Hydr. Res., 18(2): 135-151.
- Fulford, J.M. 2002. Comparison of Price Meters to Marsh-McBirney and Swoffer Meters. WRD Instrument News, March.
- Genovez, A., Abt, S., Florentin, B., and Garton, A. 1993. Correction for Settlement of Parshall Flume. ASCE J. Irrig. and Drain. Engrg., 119(6): 1081-1091.
- Gichuki, F.N., Walker, W.R., and Merkley, G.P. 1990. Transient Hydraulic Model for Simulating Canal-Network Operation, J. Irrig. and Drain. Engrg., ASCE, 116(1): 67-82.
- Hernández, J.E., Merkley, G.P. 2011. Canal Structure Automation Rules Using an Accuracy-Based Learning Classifier System, a Genetic Algorithm, and a Hydraulic Simulation Model. I: Design. J. Irrig. and Drain. Engrg., ASCE,137(1): 1-11.
- Hernández, J.E., Merkley, G.P. 2011. Canal Structure Automation Rules Using an Accuracy-Based Learning Classifier System, a Genetic Algorithm, and a Hydraulic Simulation Model. II: Results. J. Irrig. and Drain. Engrg., ASCE,137(1): 12-16.

- Holley, F.M., and Merkley, G.P. 1993. Unique Problems in Modeling Irrigation Canals, J. Irrig. and Drain. Engrg., ASCE, 119(4): July-August.
- Hu, W.W. 1973. Hydraulic elements for USBR standard horseshoe tunnel. J. of the Transportation Engrg. Div., ASCE, 99(4): 973-980.
- Hu, W.W. 1980. Water surface profile for horseshoe tunnel. Transportation Engrg. Journal, ASCE, 106(2): 133-139.
- Israelson, O.W. and V.E. Hansen. 1962. Irrigation principles and practices. John Wiley, 3rd Ed., pp. 140-145.
- Jaafar, H., and Merkley, G.P. 2010. High-Resolution Method for Modeling Hydraulic Regime Changes at Canal Gate Structures. J. Irrig. and Drain. Engrg., ASCE, 136(12): 795-808.
- Kra, E.Y., and Merkley, G.P. 2004. Mathematical Modeling of Open-Channel Velocity Profiles for Float Method Calibration. J. Agric. Water Manag., Elsevier B.V., 70: 229-244.
- Kraatz D.B. and Mahajan I.K. 1975. Small hydraulic structures. Food and Agriculture Organization of the United Nations, Rome, Italy.
- Kruse, E.G. 1992. Discussion of "Rating correction for lateral settlement of Parshall flumes." J. Irrig. and Drain. Engrg., ASCE, 118(2):337-339.
- Labye, Y., M.A, Olsen, A. Galand, and N. Tsiourtis. 1988. Design and optimization of irrigation distribution networks. FAO Irrigation and Drainage Paper 44, Rome, Italy. 247 pp.
- Lin, C.H., Yen, J.F., and Tsai, C.T. 2002. Influence of sluice gate contraction coefficient on distinguishing condition. J. Irrig. and Drain. Engrg., ASCE, 128(4): 249-252.
- Lindeburg, M.R. 1999. Civil engineering reference manual. 7th Ed. Professional Publications, Inc., Belmont, CA.
- Lozano, D., Mateos, L., Merkley, G.P., and Clemmens, A.J. 2009. Field Calibration of Submerged Sluice Gates in Irrigation Canals. J. Irrig. and Drain. Engrg., ASCE,135(6): 763-772.
- Marjang, N., and Merkley, G.P. 2009. Velocity profile modeling in rectangular and compound openchannel cross sections. Irrig. Science. 27:471-484.
- Marjang, N., and Merkley, G.P. 2009. Surface velocity coefficients for application of the float method in rectangular and compound open channels. Irrig. Science. 27:457-470.
- Merkley, G.P., and Rogers, D.C. 1993. Description and Evaluation of Program CANAL, J. Irrig. and Drain. Engrg., ASCE, 119(4): July-August.
- Merkley, G.P., and Walker, W.R. 1991. Centralized Scheduling Logic for Canal Operation, J. Irrig. and Drain. Engrg., ASCE, 117(3): May-June.
- Merkley, G.P., Walker, W.R., and Skogerboe, G.V. 1990. Field Installation of a Model for Transient Simulation of Canal Flow. Irrig. and Drain. Systems, Kluwer Academic Publishers, Vol. 4, pp. 77-90.
- Miller, R.W. 1996. Flow measurement engineering handbook. 3rd Ed. McGraw-Hill Book Co., New York, N.Y.

- Nguyen, C.C. 2000. Hydraulic analysis of unsteady flow in open channels with changing flow regimes at a gate structure. PhD dissertation, Utah State Univ., Logan, UT.
- Parshall, R.L. 1926. The Improved Venturi Flume. ASCE Trans. 89: 841-851.
- Parshall, R.L. 1936. The Parshall Measuring Flume. Bull. No. 423, Agric. Experiment Station, Colorado Agric. Coll., Fort Collins, CO.
- Parshall, R.L. 1950. Measuring water in irrigation channels with Parshall flumes and small weirs. U.S. Department of Agriculture, SCS Circular No. 843.
- Peck, H. 1988. Submerged flows in Parshall flumes. Proc., Nat. Conf. of Hydr. Div., ASCE, New York, NY.
- Pongput, K., and Merkley, G.P. 1997. Comparison and Calibration of Canal Gate Automation Algorithms, J. Irrig. and Drain. Engrg., ASCE, 123(3): 222-225.
- Robinson A. R. and A.R. Chamberlain. 1960. Trapezoidal flumes for open-channel flow measurement. ASAE Transactions, vol.3, No.2. Trans. of American Society of Agricultural Engineers, St. Joseph, Michigan.
- Robinson, A.R. (1965). Simplified flow corrections for Parshall flumes under submerged conditions. Civil Engrg., ASCE, 25(9): 75.
- Robinson, A.R. 1957. Parshall measuring flumes of small sizes. Agricultural Experiment Station, Colorado State University, Technical Bulletin 61.
- Rogers, D.C. and, Merkley, G.P. 1993. Description and Evaluation of Program USM, J. Irrig. and Drain. Engrg., ASCE, 119(4): July-August.
- Schlichting, H. 2003. Boundary Layer Theory. 8th Edition. Springer-Verlag, Berlin. 800 pp. (originally published by McGraw-Hill in 1965).
- Scoresby, P. 1997. Unpublished M.S. thesis, Utah State Univ., Logan, UT.
- Skogerboe, G.V., Hyatt, M.L., Anderson, R.K., and Eggleston, K.O. 1967a. Design and calibration of submerged open channel flow measurement structures, Part 3: Cutthroat flumes. Utah Water Research Laboratory, Logan, Utah.
- Skogerboe, G.V., Hyatt, M.L., England, J. D., and J. R. Johnson. 1967c. Design and calibration of submerged open-channel flow measuring structures Part 2: Parshall flumes. Utah Water Research Laboratory. Logan, Utah.
- Skogerboe, G.V., Hyatt, M.L., England, J.D., and J. R. Johnson. 1965a. Submerged Parshall flumes of small size. Report PR-WR6-1. Utah Water Research Laboratory, Logan, Utah.
- Skogerboe, G.V., Hyatt, M.L., England, J.D., and J. R. Johnson. 1965c. Measuring water with Parshall flumes. Utah Water Research Laboratory, Logan, Utah.
- Skogerboe, G.V., Hyatt, M.L., England, J.D., and Johnson, J.R. 1965b. Submergence in a two-foot Parshall flume. Report PR-WR6-2. Utah Water Research Laboratory, Logan, Utah.
- Skogerboe, G.V., Hyatt, M.L., and Eggleston, K.O. 1967b. Design & Calibration of Submerged Open Channel Flow Measurement Structures, Part 2: Parshall Flumes. Utah Water Research Laboratory, Utah State Univ., Logan, UT.

- Skogerboe, G.V., L. Ren, and D. Yang. 1993. Cutthroat flume discharge ratings, size selection and installation. Int'l Irrig. Center Report, Utah State Univ., Logan, UT. 110 pp.
- Torres, A.F., and Merkley, G.P. 2008. Cutthroat Measurement Flume Calibration for Free and Submerged Flow Using a Single Equation. J. Irrig. and Drain. Engrg., ASCE, 134(4): 521-526.
- Transient Hydraulic Modeling for Improved Canal System Operation. G.P. Merkley, W.R. Walker, & F.N. Gichuki. 1990. Agric. Water Mngmt., Elsevier's Agric. Science J., 18:181-194.
- USBR. 1963. Linings for irrigation canals. U.S. Government Printing Office, Washington, D.C. 149 pp.
- USBR. 1974. Design of small canal structures. U.S. Government Printing Office, Washington, D.C. 435 pp.
- USBR. 1994. Water operation and maintenance. USBR technical bulletin No. 170, Denver Federal Center, Denver, CO.
- USBR. 1997. Water Measurement Manual. U.S. Bureau of Reclamation, Denver, CO. (also available from Water Resources Publications, LLC, www.wrpllc.com)
- Walker, J.F., and Jenkins, N. 1997. Wind Energy Technology. John Wiley & Sons.
- Weber, R.C., Merkley, G.P., Skogerboe, G.V., and Torres, A.F. 2006. Improved Calibration of Cutthroat Flumes. J. Irrig. Sci. DOI 10.1007/s00271-006-0052-x.
- Working Group on Small Hydraulic Structures. 1978. Discharge Measurement Structures, 2nd Ed. International Institute for Land Reclamation and Improvement/ILRI, Wageningen, Netherlands.
- Wright J.S. and B. Taheri. 1991. Correction to Parshall flume calibrations at low discharges. ASCE J. Irrig. and Drain. Engrg. 117(5).
- Wright J.S., Tullis, B.P., and T.M. Tamara. 1994. Recalibration of Parshall flumes at low discharges. J. Irrigation and Drainage Engineering, vol. 120, No. 2, ASCE.
- Wright, S.J., and Taheri, B. 1990. Limitations to standard Parshall flume calibrations. Proc., Nat. Conf. on Hydr. Engrg., San Diego, Calif., pp. 915-920.
- Wright, S.J., and Taheri, B. 1991. Correction to Parshall Flume Calibrations at Low Discharges. ASCE J. Irrig. and Drain. Engrg., 117(5): 800-804.
- Wright, S.J., Tullis, B.P., and Long, T.M. (1994). Recalibration of Parshall flumes at low discharges. J. Irrig. and Drain. Engrg., ASCE, 120(2), pp. 348-362.