VISUAL HYDRAULICS
USER MANUAL

Innovative Hydraulics, Version 4.2

Treatment Plant Hydraulic Analysis Software

Innovative Hydraulics
915 Valley View Avenue
Pittsburgh, PA 15202
(412) 334-1637
support@innovativehydraulics.net

© Innovative Hydraulics, 2003-2015. All rights reserved.
Copyright: © 2003-2015, Innovative Hydraulics; all rights reserved. No part of this manual or the associated software may be copied, illegally stored, transmitted, or translated into any computer language, in any form or by any means, without express written consent from Innovative Hydraulics.

Disclaimer: Visual Hydraulics (Version 4.2) has been thoroughly tested by Innovative Hydraulics in an attempt to produce an error free application. However, no representation or warranties are made, express or implied, in regards to the Visual Hydraulics software or the contents of the user manual. In no event shall Innovative Hydraulics be held liable for any damages, loss of data, loss of property, or loss of profits arising out of the use of this software. This product was created to be a tool to aid the designer and should not be used as a substitute for proper engineering practice.
# Table of Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>I. Introduction</strong></td>
<td></td>
</tr>
<tr>
<td>a. System requirements</td>
<td>5</td>
</tr>
<tr>
<td>b. Program installation</td>
<td>5</td>
</tr>
<tr>
<td>c. Visual Hydraulics capabilities</td>
<td>10</td>
</tr>
<tr>
<td>d. Program help</td>
<td>10</td>
</tr>
<tr>
<td><strong>II. Program features</strong></td>
<td></td>
</tr>
<tr>
<td>a. The main screen</td>
<td>15</td>
</tr>
<tr>
<td>b. Drawing screen/drag/drop functions</td>
<td>16</td>
</tr>
<tr>
<td>c. Program toolbar</td>
<td>25</td>
</tr>
<tr>
<td>d. Program menus</td>
<td>26</td>
</tr>
<tr>
<td>e. The design forms</td>
<td>28</td>
</tr>
<tr>
<td>f. Program options</td>
<td>31</td>
</tr>
<tr>
<td>g. Program modes</td>
<td>39</td>
</tr>
<tr>
<td><strong>III. Beginning a new hydraulic profile</strong></td>
<td></td>
</tr>
<tr>
<td>a. Initiating the hydraulic profile</td>
<td>40</td>
</tr>
<tr>
<td>b. Setting the starting water elevation and units</td>
<td>40</td>
</tr>
<tr>
<td>c. Specifying the system flows</td>
<td>44</td>
</tr>
<tr>
<td>d. Adding a new section to the profile</td>
<td>46</td>
</tr>
<tr>
<td>e. Overview of the hydraulic sections available</td>
<td>51</td>
</tr>
<tr>
<td>f. Flow split and flow combination elements</td>
<td>75</td>
</tr>
<tr>
<td>g. Parallel train considerations</td>
<td>82</td>
</tr>
<tr>
<td><strong>IV. Using the individual section mode</strong></td>
<td>83</td>
</tr>
<tr>
<td><strong>V. Altering an existing profile</strong></td>
<td></td>
</tr>
<tr>
<td>a. Editing an existing section</td>
<td>87</td>
</tr>
<tr>
<td>b. Inserting a new section between two existing sections</td>
<td>89</td>
</tr>
<tr>
<td>c. Deleting a section</td>
<td>92</td>
</tr>
<tr>
<td>d. Changing flow conditions</td>
<td>92</td>
</tr>
<tr>
<td>e. Specifying a no flow scenario</td>
<td>95</td>
</tr>
<tr>
<td>f. Using the flow splits to take units/trains off-line</td>
<td>96</td>
</tr>
<tr>
<td>g. Changing the section description</td>
<td>103</td>
</tr>
<tr>
<td>h. Using the undo/redo commands</td>
<td>103</td>
</tr>
</tbody>
</table>
### VI. Summary data
- a. The summary forms 104
- b. Generating the summary equations 107
- c. The head loss graph 109
- d. Operating conditions 111
- e. The summary reports 112
- f. Generating the water surface elevation graph 118
- g. Saving the main screen diagram 119

### VII. Design options
- a. Hydraulic profile flow management 121
- b. Starting a separate flow path 123

### VIII. Specialized design tools
- a. Critical elevations 126
- b. Using the special loss feature 127
- c. The pump analysis tool 135
- d. The flow split analysis tool 140
- e. The manifold/diffuser tool 148

### IX. Dealing with Unexpected Errors
- a. “Disconnected” element 153

### X. Appendix 156
PART I - INTRODUCTION

System Requirements

In order to install Visual Hydraulics, the target computer must meet the following requirements:

- Running Windows® 95 or higher or Windows® NT 3.51 or higher
- Operating at a screen resolution of 800 x 600 or higher.
- Hard drive with at least 25 MB of free space
- 25 MB RAM
- Mouse (or other pointing device)
- Printer (in order to obtain hard copy results of any analysis)
- Microsoft® Internet Explorer version 3 or higher (for viewing help files)

Program Installation

In order to use Visual Hydraulics it must be installed on the target computer’s hard drive. This is done by extracting the program and its associated files from the Visual Hydraulics software CD. The following procedure should be followed to install Visual Hydraulics on your computer directly from the software CD:

- First, it’s recommended that you close out of all other applications before attempting to install the program.
- Insert the Visual Hydraulics CD into your computer.
- The install procedure should be initiated once the CD has been inserted into the CD ROM drive. If it does not, you have to initiate the install process yourself – hit the ‘Start’ button at the bottom of the screen, and choose ‘Run’. The following box will appear:

![Run Dialog Box]

- In the “Open:” box type the letter of the drive containing the program CD followed by \setup.exe (as shown above). Then hit the “OK” button.
- This will initiate the Visual Hydraulics setup program. The setup program contains an installation procedure that creates a directory on your hard drive where all the required files are stored. The installation program will also create a shortcut icon to the program on your desktop. The default location for this directory is in the ‘Program files’ folder on your hard drive. The initial installation screen appears as follows:

![Initial Installation Screen]
If you exit out of the setup process at any time (such as hitting Cancel), the following warning message will appear:

If setup is exited before the installation is complete, the setup program may be run at another time to complete the installation. The program will not run correctly if the installation program is not completed.

Choose “Next” on the initial setup screen, and this forwards the user to User and Product Information screen. This screen is shown below:
This screen requires the Name of the Licensee, Company that purchased the Software, and Product ID Number that was provided with the CD packet. A valid Product ID number must be entered onto this screen or the setup program will not install Visual Hydraulics. The letters in the Product ID Number are case sensitive. All product IDs should be in capital letters. If an invalid Product ID Number is entered in the setup screen, the following message will appear:

Once the appropriate information is provided and a valid Product ID Number is entered, the installation program will display the End User License Agreement, which is a standard agreement for any typical software installation. The choice is given for accepting or not accepting the agreement. If the agreement is not accepted, the install program will not continue. The license agreement form is shown as follows:
Upon accepting the agreement, you will be forwarded to the next step, which allows you to specify the start menu folder the program will be located in:

Unless you want to change the name of the folder, hit next. The next step allows you to specify whether or not a desktop icon is created on your computer:
Keeping this box checked will create an icon on your desktop that provides a direct link to the program.

The install program will now have all the information it needs to install Visual Hydraulics on your machine. Clicking “Install” on the following form will run the installation process, and Visual Hydraulics will then be installed on the target computer.
Visual Hydraulics was designed as a state of the art modeling tool that will help engineers and designers evaluate full treatment plant hydraulic profiles as well as individual hydraulic units. The program follows the traditional method for analyzing water surface profiles: a downstream control point is selected, and the hydraulic profile is then determined upstream of that control point.

Visual Hydraulics covers most of the hydraulic features typically encountered in a treatment plant application. A user may analyze the following features: full flow pipe, open channel (rectangular, trapezoidal, triangular, circular), orifice/baffles, gates, weirs (v-notch, rectangular, sharp crested, Cipolletti, contracted, and broad crested), flumes (Parshall, rectangular, trapezoidal, cutthroat), pumps, flow control valves, racks/screening devices, tank launders, filter media, contraction/enlargements, manholes, tee connections, and Venturi meters. The program also offers the ability to analyze flow splits and flow combinations. Submerged weir and flume analysis is also addressed.

A flow split analysis tool and manifold/diffuser tool are also included with the program. The flow split feature allows the designer to determine how flow will be distributed between varying flow paths. Unequal flow splitting is very common in treatment plant applications and may have a significant impact on the hydraulics of that system. The manifold/diffuser tool allows the designer to analyze a pipe system with discharge orifices (ports) along its entire length. These are common in items such as trickling filters and outfalls.

One of the most useful features included in the program is the special loss feature. This feature was designed into the program because head loss equations do not exist for everything that may be encountered in a plant. Many pieces of equipment have head losses that are manufacturer specific, for example. This feature actually allows the user to design his/her own head loss equation for anything in a plant. This is done using regression equations. The program is provided with a set of data points (flow and head loss), and then the program will perform a regression analysis on those points to come up with a best fit head loss equation.

Program Help

Extensive help is provided throughout the program. The help files that are shipped with Visual Hydraulics are in the standard Windows help format. Program help may be accessed in a variety of ways. On the main screen, either the toolbar or main menu may be used:
By selecting the “Help topics” option, the main help file will be displayed:

![Help Topics Screen](image)

The contents tab provides a general grouping of the topics available in the help file. Double clicking on any of the topics will bring up the individual topics under that section:
In order to view all the help topics available in the help file, simply access the “Index” tab on top of the form, and the topics will be displayed in alphabetical order:

Double clicking on any topic or highlighting a topic and selecting the “Display” button will access that particular help file:
Many of the help topics will have links to other topics. Links are always underlined and usually follow the content of the topic. They provide a quick means of jumping easily to other related subjects:

In addition to the help provided on the main screen, help options are provided throughout the program. Many of the forms have a “Help” button, which provides a direct link to the help topic related to the section of the program that is currently being used:
In this case, the program would link the designer directly to the help file for analyzing a pipe or conduit:

**Pipe**

Pipe or conduit losses are the most common type of losses that occur in a system or treatment plant. The total loss through a conduit flowing full is a two-step process. The first type of loss associated with conduit flow is due to friction (major losses). The second type of loss is due to pipe fittings, which alter the direction or area the flow passes through (minor losses). The total loss through a pipe is the sum of both of these types of losses. Additional information on the losses associated with pipes is available below:

- Required input
- Friction losses
- Friction resistance factors
- Fitting (minor) losses
- Adding fittings
- Solids in flow
- Pipe age factor
PART II - PROGRAM FEATURES

The Main Screen

The program’s main screen is displayed when the program is opened:

The large open space represents a drawing board for the hydraulic profile, on which hydraulic elements can be dropped and moved around to form the representative process train that will comprise the hydraulic profile. The top left portion of the screen has buttons for adding new sections to the hydraulic profile. Below these buttons are boxes that display the current flow conditions being analyzed for the hydraulic profile. The top of the screen contains the program menu and toolbar. Each feature of the main screen will now be discussed in detail.
Program Drawing Screen/Drag/Drop Functions

This newest version of Visual Hydraulics is different than the earlier versions in that it provides a much more user friendly, flexible user interface for working with a hydraulic profile. Drawing options are provided that allow the user to add items, drop them onto the drawing board, and manipulate their location using drag/drop operations. This is much different than the tabular format the earlier versions of Visual Hydraulics used to display the hydraulic profile elements and water surface elevations. For example, hydraulic elements are displayed in graphical format as follows:

Selecting an element

Visual Hydraulics will look to work with the hydraulic element that has been selected by the user. Selecting the appropriate element is important, because it lets the software know which element is active. In the example above, the selected element (“Outfall pipe”) is highlighted with four red dots surrounding the element itself. If the user were to choose to add a new section to the hydraulic profile, Visual Hydraulics will add it downstream of the active element.

Moving an element

Elements may be moved anywhere on the screen simply by holding down the mouse and dragging the element to the desired position:
Copying an element

Any element that has been added to the hydraulic profile can be copied at any time, with the exception of the starting water surface elevation. The element that is active is the element that will be copied. There are a variety of ways to copy an element: the user can select copy from the drop down menus, select copy from the toolbar, or right click on the element to be copied and select copy from the drop down menu that is provided.

Pasting an element

Once the active element has been copied, it can be pasted anywhere in the hydraulic profile that the user wishes. It can be pasted upstream of the most upstream existing element or in between two existing elements. One thing to note, the software will paste the copied section upstream of the currently active section, so make sure that the correct section is selected. The following process is followed when pasting an element that has been copied – from the example above, if the pipe section is copied and then paste is chosen, the software provides a message making sure that the paste location is correct:

If the operation is confirmed, the user is then asked to provide a description for the pasted element:
The user is provided with the name of the section that was copied. This is for convenience, but this name MUST be changed to something different. Two sections of the hydraulic profile cannot have the exact same name. If the name is slightly modified to “Outfall pipe section from chlorine tanks”, then the mouse will become the add element icon, and the user simply needs to click the location where the section will be added:

The other paste option would be to actually paste the new element between the two existing elements. The paste operation is always performed upstream of the selected element, so in this case the “Starting water surface elevation” element would have to be selected before using the paste operation. When this is done, the following confirmation is provided:
If the user confirms this paste operation, the pasted section is then inserted in between the two existing elements:

When an element has been copied and pasted, the software automatically recalculates the hydraulic profile to account for the new element.

**Deleting an element**

It may be desirable at some point to delete an element, if there was an error or something in the hydraulic profile changes. Deleting is fairly simple, but there are a few rules that need to be considered in terms of what can be deleted and when. For the most part, any hydraulic element that is not a flow combination or split element can be deleted at any time by simply selecting the element and then deleting it using either the menus, toolbar, or right clicking the element to be deleted and using the drop down menu. In the example above, if it was an error to insert the element titled “Outfall pipe section from chlorine contact tanks” before the “Outfall pipe” section, delete is chosen and the following message is received confirming the delete:
The element is then deleted and removed from the hydraulic profile, and the profile is re-evaluated to account for the element being removed:

As was mentioned previously, there are specific rules for deleting the flow combination and flow split elements from a hydraulic profile. The rule is fairly simple, if there is nothing upstream of the element, it can be deleted. If elements exist upstream of the flow combination or split, those upstream elements must first be deleted before the flow combination or split can be deleted.
In the previous diagram, if the user selects and attempts to delete the “Outfall flow combination” element, the following message will be received:

![Delete flow combination element dialog box]

The flow combination can be deleted, however, if there are no upstream elements dependent on it:

![Sample hydraulic profile]

Moving the entire screen through dragging

Visual hydraulics has a feature that allows you to “grab” the entire screen and drag it in any direction that is desired. This is useful when the profile gets lengthy and the user wishes to move from one part of the hydraulic profile to another quickly. To grab and drag the screen, simply hold down the mouse on any area of the screen – this will bring up the screen drag icon:
Continuing to hold the mouse down, the entire screen can now be dragged in any direction. If the screen is dragged up and to the left, the entire screen moves until the mouse is released:
Main screen arrows

At the bottom and left side of the hydraulic profile drawing area are arrows that also allow the user to scroll up and down and left and right around the screen:

Hydraulic profile scenarios

One of the new features of Version 4.2 is the ability for the user to model up to 5 different hydraulic profile scenarios as part of one file. Previous versions did not have this capability. This allows the user to try different options and save those changes within one file, greatly increasing the flexibility of the software. As was shown previously, the current scenario is tracked on the main screen in this location:
To add a new scenario to the existing hydraulic profile, the “Scenario – Add New Scenario” option is chosen from the main menu:

This will bring up the following message:

As can be seen from the message, the new scenario is created from the scenario that is currently active, in this case scenario 1. Selecting “OK” then creates the new scenario, and scenario 2 can be modified separately from scenario 1. The scenarios can be switched between at any time simply by selecting the desired scenario from the main screen:

A scenario can also be deleted any time by selecting the “Scenario – Delete Scenario” option from the main menu.
Program toolbar

Both the toolbar and main menu may be used interchangeably to access most of the features provided in the program. The function of each toolbar button is as follows:

- Instructs the program to create a new hydraulic profile.
- Opens an existing hydraulic profile.
- Saves the current hydraulic profile.
- Runs the help files for the program.
- Copies the selected section.
- Pastes the most recently copied section to a chosen point in the hydraulic profile.
- Zooms in on the current screen.
- Zooms out on the current screen.
- Accesses the program’s project properties.
- Edits an existing section of the hydraulic profile.
- Deletes an existing section of the hydraulic profile.
- Undo the last action performed on the hydraulic profile.
- Redo the last action performed on the hydraulic profile.
- Accesses an element’s properties
- Performs a calculation summary on the selected section of the hydraulic profile.
- Accesses the program’s flow options form.
- Generates a head or head loss graph for the selected section of the hydraulic profile.
- Accesses the program’s units on-line options form.
- Summarizes the operating condition of the selected section.
- Provides a link to the summary report menu.
Allows the user to save the main screen diagram (.jpg format)

Program menus

The main screen’s menu is very similar in functionality to the toolbar. All of the commands that can be performed by using the toolbar can also be performed by using the main menu. In addition, the menu has other options that are not available on the toolbar. The menu items are as follows:

The “File” menu item contains options for creating a new profile, opening an existing profile, saving the current profile, printing the current profile, and exiting the program.

The “Profile” menu item contains the options for working with the existing hydraulic profile.

The “Summaries” menu item contains all the options for summarizing the current hydraulic profile. This includes the individual calculation summaries for each section, generating the head / head loss graph, the operating condition, accessing the program’s summary reports, and creating an Excel generated water surface elevation graph for the current hydraulic profile.
The “Mode” menu item sets the current operating mode for the program, either to analyze a full hydraulic profile or an individual section. The full profile and individual section modes are discussed later in this section, and Part IV covers the individual section mode features and options in more detail.

The “Options” menu item contains all of the options for features that may be specific to a certain hydraulic profile. This includes the project properties (flow characteristics, element hydraulic factors, program defaults, etc.), the flows to be analyzed for the hydraulic profile, the flow split distribution analysis tool, the manifold/diffuser tool, notes that the designer may choose to make for any section, and the ability for a user to specify a critical elevation for any section of the profile. In addition, the “Calculator” item instructs the program to run the standard Windows® calculator. The final item allows the user to convert files created in previous versions (prior to Version 4.1) of Visual Hydraulics to the most recent version. The project options are discussed in more detail later in this section.

The final menu item is the “Help” option, which provides a link to all of the program’s help files. The ‘About Visual Hydraulics’ item simply pulls up a summary form that contains information about the program and its copyrights. A link is also provided which will connect the user to the Innovative Hydraulics website. The help files were described in more detail earlier in this manual.
The profile section design forms

As is common with most standard Windows programs, Visual Hydraulics uses forms that allow the designer to supply the program with the necessary information the program requires for performing its various hydraulic analysis capabilities. The design forms for adding a new section to the hydraulic profile (i.e. pipe, channel, orifice, weir, etc) follow a common format. For example, the design form for an orifice/baffle/gate(s) is shown following:

This particular design form contains two tabs (tabs are located at the top of the form). One tabbed section contains the general information required for analysis of the baffles, in this case the gate shape, size of each gate, invert of the gate(s), and number total number of gates. The other tab, which is common to almost all of the section’s design forms, is the “Flows” tab. This section of the form appears as follows:
This section of the form contains the information required on the flow through that section. This flow tab appears on every section design form. Detailed information on working with the hydraulic profile flows and using this section of the form is provided in the next section of the manual.

It should be noted that not all of the tabs for each hydraulic section are the same. Some sections, such as pipes and channels, have more information required and therefore have additional tabs. For example, the pipe section form has four tabbed sections:
This design form has two additional tabs, one for specifying the pipe’s fittings (if any) and an additional tab for pipe specialties (such as conveying flows with solids and specifying pipe age).

In order to analyze a section, the program must have enough data from the designer to perform that analysis. If the designer fails to provide the required data, the program will provide a message similar to the following:

![Non-numeric value]

Errors such as this one have been provided throughout the program to catch design errors, unreasonable data, and simple typos as well.
Program options

Project Properties

The project properties form controls all of the optional or “changeable” features that exist within the software. This includes items such as general hydraulic profile characteristics as well as characteristics specific to individual hydraulic elements. There is also a default options tab within the project properties form that allows the users to set up certain defaults for hydraulic elements. This form is shown below:

The first tab (General Options) contains the options that apply to the overall hydraulic profile and appearance of the data presented within the software. This includes the system of measurement (English or SI), flow units to be used for hydraulic profile flows, and the flow temperature (which can affect the hydraulics of pipes). Also included on this tab are options for changing the appearance of the display, including the number of decimals that will be displayed for water surface elevations, whether or not the section description for each hydraulic element is displayed, and the ability to lock hydraulic elements to a grid.

The default system of measurement is the English system and the default units of flow are million gallons per day (MGD). The default flow temperature is 60 °Fahrenheit (15.5 °Celsius). Any of these values may be changed at any time. If the system of measurement is changed in the middle of a design, the entire current hydraulic profile will be converted to the new system. If the temperature is changed in the middle of the design, the temperature change will only affect pipe sections that are added to the hydraulic profile from the time of the change on. Previously added pipe section analyses will reflect the old temperature.
If the system of measure is changed, such as from English to SI, the flow units will also change. The default flow units for the SI system are cubic meters per second (cms), but liters per second (lps) are also available.

The second tab of the Project Options form provides the user with the ability to set default parameters for pipes and open channels, as well as provide more detailed flow descriptions for the flow scenarios. That tab is shown as follows:

As can be seen, this tab allows the user to specify default roughness values for the three friction loss methods that are available for pipes. This can also be done for the Manning’s “n” value for open channels. The flow descriptions section allows the user to provide a more detailed description for any of the flow scenarios. In the example above, the user has chosen to rename the “Return I” flow condition to “RAS flow”. If default friction parameters are active for either pipes or open channels, that default value will be applied to the design form. When the user opens a pipe, for example, the default friction value will automatically be displayed on the form.

The final option on this tab is the “Apply defaults to existing hydraulic profile” option. Selecting this option will instruct the software to apply any changes made to the entire existing hydraulic profile. If this option is not selected, any changes will only apply to the hydraulic profile from the current point moving forward. So in this case, if the existing hydraulic profile has multiple pipe sections with a Hazen-Williams “C” value of 110, and the user chooses to apply the defaults to the existing hydraulic profile, all of those “C” values will be changed from 110 to 120. If this option is not selected, the default value of 120 will only be applied from the current point forward.

The final tab available on the Project Options form is the Element Options tab. This tab has all of the specific hydraulic characteristics used by the software for a variety of the hydraulic
elements. This includes hydraulic characteristics for pipes, open channels, channel transitions, orifices/gates, rectangular weirs, bar racks/screens, filters, tee connections and Venturi meters. This tab is shown as follows:

The left side of the tab contains the selected element type, and the list on the right side of the form contains the hydraulic characteristics associated with that hydraulic element that are stored within the software. In the case of pipes, that would include all of the fittings and fitting K values. To choose another hydraulic element’s characteristics, the user would simply choose another element from the drop down box:
For rectangular weirs, the weir “C” coefficient for each weir is displayed. Innovative Hydraulics recognizes the fact that hydraulics is not an exact science, and that varying loss coefficients exist for almost every hydraulic feature available. Because of this, flexibility is key: in order to meet the needs of all users, it is imperative that the user be able to control the hydraulic coefficients that are assigned to the various hydraulic elements that exist within the software. For this reason, the hydraulic elements options tab was instituted into Version 4.2.

Going back to the “Pipe” element, it is important to note that not only can the user change a K value for any fitting, the user can also add a fitting or delete a fitting from the software’s fitting database at any time. In looking at the form when the “Pipe” element is active, it can be seen that “Add fitting” and “Delete fitting” options are available:

These options can be used to control the fittings that reside within the software’s fitting database. To add a new fitting and K value that does not exist in the current database, the user would simple select the “Add fitting” option, and the following form is displayed:
Upon selecting the “Add fitting” button on the form, the fitting will then be added to the Pipe fitting database:

Any fitting can easily be removed by highlighting the desired fitting and selecting the “Delete fitting” option:

In addition to adding or delete fittings, the hydraulic coefficient for any hydraulic element that is stored within the software’s database can be changed at any time. This is true of pipe fittings and any other element’s hydraulic characteristics. For example, for a rectangular weir, the user may wish to change the “C” value for the weir coefficient. This can be done by highlighting the feature to change, and then providing a new value in the box provided:
Selecting the “Apply” button will then change the “C” value for the rectangular weir from 3 to 3.15:

The software saves default values for all characteristics of all hydraulic elements, so the user may easily revert back to that default value at any time. Simply highlight the desired hydraulic characteristic and then select the “Restore Default” option:
In the case of the previous example, this would change the “C” value for the rectangular weir back from 3.15 to 3:

**System Flows**

The system flows option will bring up the system flows summary form:
This form allows the designer to add flow conditions, remove flow conditions, or change flow conditions. In addition to the forward flow through the treatment plant, up to three different return flow scenarios may be specified. The process of working with the system design flows is discussed in detail in Part III, Beginning a new hydraulic profile.

Add/Check Note

The add/check note option provides the designer with the option of making notes about any section that has been added to the hydraulic profile. This may be useful if a design is in progress and saved for further analysis, and the designer wishes to make a note of what was being analyzed previously:

![Add Note Window](image)

Calculator

The final item under the “Options” menu is the Windows® calculator. This option instructs the program to run the standard Windows® calculator:

![Calculator Window](image)
If the Windows® calculator is not available for whatever reason, the program will simply provide a message saying the calculator could not be accessed.

**Program modes**

Visual Hydraulics has two operational modes – full profile mode and individual section mode. Full profile mode is the default mode. The full profile mode is used when the designer is analyzing a variety of hydraulic features that comprise part of or all of a hydraulic profile. Individual mode was placed in the program to allow the designer to analyze one hydraulic feature separately and individually.

As was mentioned previously, changing between full profile mode and individual section mode is accomplished by using the “Mode” option on the main menu:

![Mode menu](image)

Individual section mode may be used at any time, even while in the middle of a hydraulic profile. This may be useful for “testing” the hydraulic characteristics of a feature before actually adding it to the profile. Individual section mode is discussed in detail in Part IV of this manual.
PART III – BEGINNING A NEW HYDRAULIC PROFILE

Initiating a new hydraulic profile

This section of the manual is dedicated to analyzing a new hydraulic profile. To initiate a new hydraulic profile, either the “File – New Profile” menu items may be used or the new profile option on the toolbar:

This will bring up the New Project information form, which is shown below. This form requires a description of the new project, the unit of measure (English or SI), units of flow to be used, and the starting water surface elevation for the profile. The starting water surface elevation is the point from which the new hydraulic profile will be based, so a fairly well established point of reference should be chosen. Either a known measured value or a control point (such as a weir) is often chosen as the downstream starting water surface elevation.

If a weir is to be used as the start of the hydraulic profile, Visual Hydraulics provides a built-in weir calculator tool, which is accessed by using the "Determine starting elevation from weir" button located at the bottom left of the form:
This brings up the weir types form, from which the user selects the type of weir to be used in the starting water surface elevation calculation:

![Weir Types Form](image1)

In this case, if a V notch weir is chosen, the V notch weir design form is then displayed:

![V-notch Weir Form](image2)

For the V notch weir, the necessary weir characteristics are provided, including the angle of the V notch, invert of each notch, and number of notches comprising the entire weir. On the second tab, "Flows", the flow value over the weir is also entered:

![Flows Tab](image3)
All of the required information has been provided for the weir, and the user simply needs to click the "Finished/Update Profile" button the General tab:

The program will calculate the water surface elevation over the weir and provide this value on the new project form:
The program also provides the option of determining a starting water surface elevation from a free drop. Free drops are very common at outfalls. If the “Determine from drop” option is chosen the following form appears:

![Calculate Starting Elevation from Drop]

The user simply supplies the drop and flow data, and the depth of flow going over the drop is then determined by selecting the “Calculate Upstream Depth”. The elevation over the drop is then placed on the new project form:

![Starting water surface elevation]

Determining the starting elevation from a weir or drop is of course not required, an elevation can be entered if known.

Once the new project information has been provided and the “OK” button is chosen to continue, the program will proceed to the next step, which is specifying the design flows that will be used for the new profile.

Visual Hydraulics provides the designer with the option of specifying forward flow conditions and up to three different return flow conditions. Return flows are very common in treatment plant applications, especially biological treatment where a certain percentage of the organisms are returned to the biological process in order to maintain proper treatment. In addition, waste flows are often returned to the head of the plant. These return flows can have a significant impact on the hydraulics of the plant and should always be carefully analyzed.
The plant’s flow conditions are entered into the program using the following form, which is displayed to the designer once the new profile conditions have been established:

To add a new flow, the appropriate flow condition must first be chosen – either a new forward flow, return I flow, return II flow, or return III flow. The flow value is then entered in the box next to the “Add” button, and the “Add” button is selected. This will instruct the program to add the new flow value to the appropriate flow condition. For example, to add a flow of 50 mgd as the first forward flow condition, the form would appear as follows:

Notice that when a flow value has been entered into the box the “Add” button becomes active. Upon selecting the “Add” button the program will confirm the flow addition and place the new flow in the appropriate flow group:
This is how flows are added to the program for the profile. Each flow scenario may have up to 20 different flow conditions. Note that only a forward flow value is required to work with a hydraulic profile. Return flow conditions are optional and may not apply to many hydraulic profile analyses. Once all of the flow conditions to be analyzed have been specified for the new profile, the “Finished” button should be chosen. The program will then display the main screen for the designer:
As can be seen, the first point in the hydraulic profile table is displayed as the starting water surface elevation. The current active flow conditions are displayed on the left side of the main screen. The active flow conditions are the conditions that have been selected on the flows form:

At this point, new sections can be added to the hydraulic profile. Consider the following example: A new pipe is to be analyzed upstream of the control point. The pipe is 60” concrete, 500 feet in length. To add this new pipe, select the new pipe button:

This brings up the types of losses form:
The pipe option would be selected (if it hasn’t been automatically selected) and a description of the new section must also be provided. The program will not continue on unless a description is specified for the new section. Once this is done, the pipe design form is displayed:

The “General” tab is shown first. A circular pipe is chosen, and the appropriate diameter and length is entered into the boxes provided. For this pipe, the Hazen-Williams method is chosen for the friction analysis, and a “C” value of 120 is specified. Next is the Fittings tab:
For this example, one flush entrance, three 90 degree elbows, and one exit are included for this pipe analysis. The next tab is the Flows tab:

The flows tab is common to every type of element except the step/drop (which has no flow), the flow combination element, and the flow split element. A detailed breakdown of the flow tab will describe in detail exactly how flows through each section are handled.

This section contains the forward flow through the section being analyzed.

This section is where the return flows are included, if applicable. To include a return flow, the check box is checked and the return flow value is entered into the box provided.
For this example, let’s assume there are two parallel pipes conveying the flow, with each pipe receiving half of the total flow. If two pipes conveyed the flow, the flow tab would appear as follows:

![Flow Tab Example]

For 2 pipes in this case, half the total forward flow is entered and half of the total return I flow is entered, and the total flow is $2 \times 37.5 = 75$ mgd. It should be noted that if multiple units are specified that the flow split between them is assumed to be equal.

There are no specialties in this case (see detailed breakdown of analyzing a pipe section later in the part), so all the necessary information has been provided. Clicking on the “Finished/Update Profile” button on the “General” tab will instruct the program to analyze the pipe section and add it to the hydraulic profile. When this is done, the main screen then shows a “cross hairs” mouse icon, which designates that the user must select the location for the new element to be placed in the drawing area:
Cross hairs indicate the element has been analyzed and is ready to be placed on the drawing screen.

Once the user chooses the location of the new element by clicking on the drawing screen, the element is added to the hydraulic profile:
The new pipe section has been added to the profile. The loss in the pipe totaled 0.55 feet. Additional sections are added in a similar fashion. This is how a new hydraulic profile is constructed.

**Overview of the hydraulic features available**

Visual Hydraulics offers the designer the ability to analyze most of the features in a plant that will create hydraulic restrictions and generate head losses. This section breaks down each feature available for analysis and the input data that is required for each from the designer.

**Pipe/conduit**

Pipes are the most common head loss generators in treatment plant applications. Because of this, many options have been provided for their analysis. The pipe design form appears as follows:

![Pipe design form](image)

The first tab is the general information tab. The designer must provide the pipe/conduit shape, size of the pipe (including width if rectangular), length of the pipe, and friction loss coefficient. There are three methods available from the program for determining the friction loss – the Darcy-Weisbach equation, Hazen-Williams equation, and Manning’s equation.

The second tab is the fittings tab. This section of the form allows the designer to specify all the fittings that exist for the pipe section being analyzed:
All of the fittings currently available in the program are listed in the drop down box at the bottom left of the form:

To add a new fitting to the pipe section, select the fitting from the drop down box and specify the number of these fittings in the “number of fittings” box, then hit the “Add Fitting” button. The new fitting will be placed in the fitting storage list:
As fittings are added to the storage list, the program keeps a running tabulation of the total K value of all the fittings currently specified for the pipe section. This value is on the right side of the form:

![Image](image.png)

Also shown above is the button that provides a link to the program’s fitting and K value database, where the designer may change K values of fittings, add new fittings, or remove fittings from the database if needed. See the detailed discussion on using this database in Part II – Program options for more information. To remove a fitting that has been added to the fitting storage area, simply highlight the fitting to be removed from the list and select the “Delete selected fitting” button:

![Image](image.png)

It should be noted that fittings are optional, although most pipe sections will at least have some sort of entrance and exit fitting loss.

The third tab is the flows tab, which was discussed in detail earlier in this section. Again, this tabbed page is common to all hydraulic features except the elevation step/drop. The last tab on the pipe design form is the specialties tab. This tab allows the designer to account for two unique items – the additional friction head loss caused by the conveyance of flows with high amounts of solids and the affect of age on a pipe. This tabbed section is shown as follows:
This tabbed page is broken into two sections, the top section being for the pipe age and the bottom section for the analysis of flows with solids. For the pipe age section, the designer must provide the pipe diameter and approximate age of the pipe. When the “Calculate Multiplying Factor” button is selected, the program will display this multiplying factor based on the pipe diameter and pipe age. For the flows with solids section, the designer must provide the type of solids in the flow and an estimate of the percentage of solids in the flow.

If pipe age and/or flow with solids are to be included in the overall pipe head loss, the program multiplies the calculated friction loss in the pipe by the appropriate multiplying factor(s). In order for the program to include these multiplying factors, the designer MUST check the box provided on the form that instructs the multiplying factor to be included:
If this box is not checked, the calculated multiplying factor will not be considered.

As a note about the pipe age and solids multiplying factors, it should be considered that these factors are very conservative and should be used for estimating purposes only. Because of the variability of flow types, diameters, field conditions, and pipe materials, it is impossible to accurately predict the affects of pipe age and solids flow on pipe head loss.

**Open Channel**

Open channels are also extremely common in treatment plant applications. The open channel design form is as follows:
The open channel design form has three tabs, the diagram/shape tab, general information tab, and the flows tab. The first piece of information required is the shape of the open channel, which may be rectangular, trapezoidal, triangular, or circular. Once a shape has been chosen, the general tab contains the majority of information required from the designer:

![Open channel design form](image)

The designer must supply the Manning’s “n” value of the channel, downstream invert of the channel, length of the channel, bottom width (if applicable), bottom slope, and side slope (if applicable). There is also a box provided for the channel height, and this information is optional. The channel height can be used to determine if the channel depth is near the top of the channel or overflowing, if desired. If a triangular channel is specified, a bottom width is not required. If a rectangular or circular channel is being specified, a side slope is not required. Note that for the bottom slope that the slope is positive when the channel slopes from upstream to downstream and negative if the channel slopes from downstream to upstream.

A channel bend may also be included in the channel analysis. Flow going around bends can create turbulence and result in additional head loss. To specify an open channel bend, the type of channel bend should be selected from the options provided:
Note that the K value for the current channel bend that has been selected is displayed in the box to the right. Similar to the pipe fitting database, K values can also be changed at any time by accessing the project properties form and selecting the “Open Channel” element on the “Element Options” tab. This will bring up a list of all of the channel bends and their respective K values:

If a different K value is desired for any particular channel bend, the user simply has to highlight that bend in the list, provide a K value for the bend in the box provided, and select the “Apply” option.

It should be noted that the change in K value will only apply to channels that are specified after the change has been made. If a channel with the same bend has already been added to the hydraulic profile before the K value is changed, the software will use the old K value for that particular channel. In that case the channel that was added previously should be edited to make sure that the updated K value is used.

One other important point needs to be made about the calculation of the head loss from an open channel bend. The software calculates this head loss based on the channel velocity at the bend, and this velocity will be the **downstream** velocity in the channel. Consider the following scenario:

In this case, channel “1” should be the channel section that has the channel bend specified, since the bend is on the downstream end of channel “1”. Channel “2” would have no channel bend.

The hydraulics of open channels can be quite complex. A variety of flow conditions can exist based on the natural flow in the channel and the downstream control, which plays a big part in open channel analysis in treatment plant applications. The head loss theory should be consulted in the appendix for a detailed discussion on the evaluation of open channel hydraulics.
Open Channel Transition – Contraction or Expansion

Version 4.2 offers an addition to the open channel analysis, the channel transition element. It is not uncommon to encounter a channel that changes width, and this change in width is typically done as a transition rather than abruptly. The channel transition form is as follows:

![Channel Transition Diagram]

This form contains three tabs, the transition shape tab, general information tab and the flows tab. Required input is the type of transition (channel expansion or contraction), invert of the channel transition, and size of the channel upstream and downstream of the transition. Based on the type of transition chosen, the software will assign a “K” value to the transition, which is then multiplied by the velocity head through the transition to determine the head loss. The “K” value assigned to the transition type is shown on the General tab:

![General Tab]

Innovative Hydraulics
Orifice/Baffle/Gate(s)

This option allows the designer to determine the head loss that will occur when flow passes through any type of opening or orifice(s). Common examples are baffles, settling ports, flow diffusers, and gates. The loss is a function of the velocity of the flow passing through the opening, the orifice shape (usually circular or rectangular), and the level of submergence. The design form is as follows:

This form contains two tabs, the general information tab and the flows tab. Required input is the shape of the opening(s), size of each opening (assumed to be the same), invert of the openings, and total number of openings used to convey the flow. Again, the flow is assumed to be equally conveyed through each opening. The head loss through the opening and the downstream water elevation are used to determine if the flow through the opening(s) will be submerged or unsubmerged. While submerged analysis is fairly straightforward, openings that are unsubmerged behave similarly to weirs and require a fairly complex solution process.
Weirs

Weirs are flow control devices that are used to measure flow, equalize flow distribution, or establish set water levels in a treatment plant. They are very common at tank outlets and flow splits. They are also good points for starting hydraulic profiles because the water elevation at a weir is easily determined and fairly accurate.

Visual Hydraulics offers the designer the ability to analyze 7 different types of weirs – triangular (v-notch), rectangular, sharp crested, Cipolletti (trapezoidal), contracted, broad crested, and rectangular notched weirs. When a weir is chosen for analysis, the following form is displayed:

![Image of weir types]

The designer chooses the appropriate type of weir and is then taken to the design form for that weir. For example, for a v-notch weir, the design form is as follows:

![Image of v-notch weir design form]
For a v-notch weir, the required input is the angle of the v-notch (must be greater than 0 and less than 180 degrees), the invert elevation of the notches, and the total number of notches. The notch inverts are all assumed equal and therefore the flow over each notch is assumed equal. A total notches calculator is provided for the designer as well. To use the total notches calculator, the entire length of the weir and spacing between notch inverts must be known, and the program will determine the number of notches comprising the entire weir.

**Submerged weirs**

Submergence of a weir occurs when the water elevation downstream of a weir rises to a level that is higher than the weir invert. Submerged weirs behave differently than weirs operating under free flow conditions, and this is dependent on the degree of submergence. Submerged weirs are not usually recommended because they may signify a hydraulically overloaded unit. This is not always the case, however. Broad crested weirs are often designed to operate in the submerged condition. For the most part, good weir design ensures that there is a comfortable distance (freeboard) between the downstream water level and the invert of the weir, so that the weir flows “freely”.

Special methods and equations have been developed for the analysis of submerged weirs and are used by Visual Hydraulics if a submerged weir condition is encountered. This methodology is discussed in detail in the head loss theory section of the appendix.

If a submerged weir condition is calculated by the program, the designer will be alerted by the software, which provides a flag at the corner of the icon that represents that element:
In this case, the “Turbine pumps effluent splitter weir” is flagged as operating outside of the normal range, which indicates submergence for a weir. Two types of submergence can occur, partial submergence and full submergence. Partial submergence occurs when the downstream water elevation is higher than the invert of the weir but not higher than the head over the weir under free flow conditions. Full submergence occurs when the weir is “drowned out”, or the downstream water elevation is higher than the head over the weir under free flow conditions.

**Elevation Step or Drop / Constant Water Elevation**

Situations may be encountered while analyzing a profile that requires a change in water surface elevation that cannot be accounted for by any of the features available. To provide additional flexibility when analyzing a hydraulic profile, an option has been provided that will allow the water level to be raised or dropped from the previously calculated point as much as desired. The user may also set a constant water level value, which will remain unchanged when the hydraulic profile is analyzed. The process is very simple – a step/drop is specified, and the designer enters the amount the profile should be raised or lowered:

![Elevation change - Screw pump drop to suction box](image)

In this case, an adjustment is being made for an uncommon piece of equipment, the stilling plate. The profile would be adjusted up 0.5 feet to account for the loss. To drop the hydraulic profile, a negative value would be used.

Additionally, the user could select the second option of providing a constant water elevation and that elevation would be set at a constant value within the hydraulic profile. It should be noted that the form does require a flow value in order to account for flows when streams are taken off-line or placed on-line.
Filter

Filters are primarily used as secondary/tertiary treatment for removal of fine particles in the flow. Head loss occurs as flow passes through the filter’s porous media. Equations have been developed for the calculation of this head loss, which is dependent on a number of factors, including the shape of the filter media, porosity of the filter media, depth of the filter media, and velocity of flow through the media.

Designer input includes the shape of the media, depth of the media, surface area of the filter, average particle diameter, and media porosity. Common filter media porosities are as follows:

- Spherical = 0.30-0.40
- Average sand = 0.35-0.45
- Rounded sand = 0.30-0.40
- Coal/angular sand = 0.40-0.50

The filter design form is shown as follows:

Another media option shape is provided in the filter media drop down box. There is an option titled “other”, if the user wishes to use a media type not provided by Visual Hydraulics. If the “other” option is chosen, the user has to provide to the software the media shape factor and a description of the media.
Bar Rack / Screen

Bar racks or screens are placed at the head of treatment plants and are preliminary devices that are used to remove large debris such as sticks, rags, cans, bottles, and other large objects from the influent plant flow. The bar rack/screen design form is shown as follows:

This element was further advanced as part of Version 4.2 to provide additional options for the user. The bar rack/screen option now offers two different methodologies for analysis. Because both are so widely used, Visual Hydraulics includes both options. The Kirschmer theory is a little more in depth and detailed than the Bernoulli theory, and includes a few more factors – namely the bar shape and the vertical angle of the rack. Both theories require the invert of the rack, rack width, widths of the bars and spacing between the bars, and a blockage allowance, if any. The width of the bars must be less than the spacing between the bars. The analysis of a bar rack involves determining the depth downstream of the rack, which in turn can be used to determine the velocity of flow through the bars. The head loss is based on the difference in velocity from the flow downstream of the rack and the velocity of flow through the bars. The downstream elevation is calculated by subtracting the invert of the rack from the previous water surface elevation. If a rack invert is provided that is actually higher than the previous water surface elevation, the program will assume that the flow exits the rack and the water elevation then drops off. In this case, the water elevation downstream of the rack or screen will be equal to the natural flow depth in the channel the rack is placed in, which will be critical flow.

Junction (Contraction/enlargement, Manhole, or Tee connection)

Junctions occur when two or more pipes combine or separate, and are extremely common in treatment plant applications. Visual Hydraulics offers the capability of modeling most of the
junctions that are typically encountered, including contractions/enlargements, tee connections, and manholes. The junction design form is shown as follows:

**Contractions/Enlargements**

Contractions or enlargements involve transitions from one pipe size to another, which in turn results in a head loss. This head loss is dependent on how fast the pipe transitions from one size to another (angle of transition) and the velocities in the two different pipe sizes. To analyze a transition, the designer must provide the type of transition to be analyzed (contraction or enlargement), sizes of both pipes before and after the transition, and angle of the transition:
If the angle of transition is unknown, it may be calculated by using the “Calculate angle of transition” button provided on the form.

**Tee Connections**

Tee connections occur when two pipes combine to form a single pipe. There are three distinct flow paths that can occur as flow passes through a tee, and these will greatly affect how head losses are determined. These flow paths are determined by how the flow enters and leaves the tee, and are illustrated in the following diagrams:

![Branch to Line Flow](image1)
![Line to Branch Flow](image2)
![Run of Tee Flow](image3)

Head losses for tees are determined by multiplying the velocity head through the pipe by a tee “K value”, which is determined from the size of the pipes comprising the tee as well as the configuration of flow through the tee. The head loss for a run of tee, for example, is less than the head loss for a branch to line tee flow path. The theory behind the head losses for tee connections is discussed in more detail in the appendix.

Visual Hydraulics offers two options for analyzing tee connections – a general, basic analysis that is based on the type of tee connection only and a more detailed analysis that analyzes tee pipe diameters, angles of connection, and flow distribution throughout the tee. It is up to the user to determine which analysis suits the situation being examined.

**Basic Tee Connection:**

For the basic tee connection, only the configuration of the tee is used in determining the tee’s “K” value. The basic tee connection is more frequently used when flow is only originating from one direction. In other words, flow is not being combined or divided at the tee connection, it is simply passing through the tee. If a tee connection does have flows being divided or combined, it is recommended that the combining/dividing tee option be used to analyze the tee.

The “K” values for basic tee connections are stored in the software’s options database, which can be accessed at any time:
The design form for a tee connection is shown as follows:

As can be seen, the user simply provides the configuration of flow through the tee as sizes of the pipes comprising the tee, and the program determines the appropriate K value and uses this to calculate the head loss through the tee connection.

**Combining/Dividing Tee Connection**

The combining/dividing tee connection option should be used for junctions where flow paths are combined or split at the tee connection. For combining/dividing tees, the K value is dependent on the type of connection, angle of the tee, ratio of flow in to flow out of the tee, and area ratio of the tee diameters along the flow path. Extensive research has been performed by Miller, and that research has been resulted in numerous graphs that represent the varying scenarios that could occur at the tee connections. The data from these graphs has been incorporated into Visual Hydraulics, and that data is used to determine the K value for the tee and subsequent head loss.

Consider the following example for a combining tee:

Main diameter – 24 inches
Branch diameter (bypass line) – 12 inches
Flow in main line – 4 MGD
Flow in bypass line – 1 MGD
Tee angle of connection – 45 degrees

Using the data form for the combining tee element, as is shown below, the data can be provided for the tee that combines the flow from the effluent line and the bypass line:
The user selects the type of tee, provides the diameters of the connecting pipes, as well as the flow splits between the pipes and tee angle. Once this information is provided, the software will use this data to determine the appropriate K from the numerous graphs published by Miller. This K value is multiplied by the appropriate velocity head in the pipe (downstream of tee for combining tee and upstream of tee for dividing tee). A sample of the Miller graphs is shown as follows:
The head loss in the combining tee is then provided in Visual Hydraulics as part of the hydraulic profile. A more extensive discussion of the theory behind combining/dividing tees is provided in the appendix.

**Manholes**

The final type of junction that may be modeled with the program is a manhole. Manholes are used for a variety of reasons: as a combining point for two or more pipes, as an access point for a pipe, for allowing sharp bends or turns in a pipe direction, or for a change in grade of the pipe. Because flow exits a pipe, enters a manhole, and enters another pipe to exit a manhole, head losses can be encountered in manholes. The factors that determine how significant those head losses are include the number of pipes entering a manhole and the size of the pipes entering and leaving the manhole. If only two pipes exist in a manhole (one entering and one leaving), the angle of the influent and effluent pipes relative to each other are also a factor. The manhole design form is shown as follows:

For a manhole the user provides the configuration of the manhole and size of the pipes going in and out of the manhole. As is indicated in the note at the bottom of the form, for manholes with more than one pipe entering the manhole, the pipe that is along the path of the hydraulic profile being analyzed should be the pipe that is used as “Dout”. Again, the angle between the influent and outlet pipe for the manhole is only required if there is only one pipe entering the manhole.
Flumes

Flumes are open channel flow measuring devices that are very common in treatment plant applications. Flumes operate by restricting the flow through the throat area to create a flow condition that is easily correlated to a head value. This program offers the designer the ability to analyze four types of flumes – Parshall, cutthroat, rectangular, and trapezoidal. Parshall flumes are probably the most common because of their relatively simple construction, low maintenance, and reliability. The Parshall flume design form is shown as follows:

In addition to being fairly easy to construct, Parshall flumes are quite simple to analyze hydraulically. The head flowing through the flow is dependent only on the width of the throat and the flow through the flume. The designer simply needs to provide the throat width and flume invert for a Parshall flume to be analyzed.

Submerged flumes

Flumes most commonly operate in an “unsubmerged” condition, or a condition where there are no downstream affects on the flow through the flume. However, in some designs or very high flows, “submerged” conditions may result, meaning that the downstream level of flow actually affects the flow through the flume. These conditions make it very difficult to accurately assess the hydraulic conditions of the flume and are usually not recommended for normal operation, unless the flumes are simply being used as flow splitting devices. If the program calculates a condition where the previous water surface elevation is actually higher than the invert of the flume, the following procedure is used: the downstream controlling head is calculated by subtracting the flume invert from the previous water surface elevation. Then the head is
calculated for the provided flow operating under a normal or “unsubmerged” condition. The head through the flume is taken as the greater of the two calculated head conditions.

Tank Launderers

Tank launders and troughs are special open channel applications. Almost all launders are rectangular in shape and have either zero or a very minimal slope from their high point to the launder outlet. The hydraulic “worst case scenario” point for a launder is established as the point in the launder that is furthest away from the launder outlet.

In order to analyze a tank launder or trough, the following input is provided by the designer: the length of the launder (* note: for a tank perimeter launder or trough, this is the distance from the launder outlet to the furthest upstream point, which should be HALF of the entire launder perimeter length), the width of the launder, slope of the launder (if any), and launder invert elevation at its point of discharge. The launder main design form is shown as follows:

![Laundry characteristics diagram]

The depth downstream of the launder at the launder outlet is very important in determining the final or upstream depth in the launder. If the outlet is unsubmerged and flow free falls from the launder into the outlet box or pipe, then the critical depth is used as the downstream starting depth in the launder. If the launder outlet is submerged and the submerged depth is greater than the critical depth in the launder, the submerged depth is used as the starting depth.
Special Loss and Pump

It’s highly unlikely that every type of loss to be analyzed will be available in this program. Treatment plants often have specialized applications or equipment that do not have standardized equations or methods for determining the losses encountered while they are in operation.

The special loss portion of the program is the most advanced feature. Because of this, an entire section has been devoted to using this option in the manual. See Part VII of the manual for an in depth look at the special loss feature.

Pump analysis follows a very similar methodology to the special loss feature. Pump analysis is also included in Part VII of the manual.

Venturi Meter

Venturi meters in are “in-line” pipe meters that restrict the flow as it passes through the meter, creating a pressure drop. This pressure drop can then be correlated to a flow value depending on the amount of drop experienced by the meter. Because the Venturi meter involves a reduction in pipe size to achieve this pressure drop, head losses are fairly common.

The Venturi meter design form is as follows:

For the program to analyze a Venturi meter the designer simply has to provide the size of the inlet diameter and throat diameter of the meter.
Flow Control Valve

Flow control valves are very common in applications where the flow through a particular treatment section is to be limited or where a desired head loss needs to be generated to maintain a certain upstream water elevation. Flow control valves are more common in water treatment applications than wastewater applications. The calculation of the head loss through a flow control valve is quite simple and is based upon the flow through the valve and the valve’s “flow coefficient”, which is manufacturer specific. Refer to the theory in the appendix for a discussion of the head loss equation associated with flow control valves.

The flow control valve is actually considered a special loss in the program because of its unique application. Therefore, to add a flow control valve to the hydraulic profile, the special loss button first needs to be selected:

![Special Loss Button]

This will bring up the types of special losses available in the program:

![Special Loss Types]

When the “Control Valve” option has been selected, the user will be taken to the main screen for the flow control valve:
The flow control valve requires two pieces of information to determine the head loss through the valve, the flow conveyed through the valve and the valve’s Cv or flow coefficient. The Cv coefficient relates pressure drop across the valve to the amount of flow being conveyed by the valve. The higher the Cv value, the smaller the pressure drop (or head loss) for a particular flow. Cv values are manufacturer specific and should be available for almost any flow control valve. As a note, the Cv values are assumed to be based on gpm and psi for the English system of measure and l/s and kPa for the SI system of measure. These are the standard Cv units for both systems of measure, respectively.

In the control valve form provided above, a Cv value of 2500 is entered at a flow of 5 MGD (half the total flow). If this valve is then added to a hydraulic profile, it would appear as follows:

In this case the head loss was calculated to be 4.45 feet for the flow control valve specified.
The flow split and flow combination elements

There are two other elements that have yet to be mentioned and are important enough to deserve their own section of the user manual. One of the benefits of Visual Hydraulics 4.2 is that it lets the user analyze any number of parallel trains at once, and even take elements or entire treatment trains off-line to see how the hydraulic profile is affected. Flow diversions and combinations are handled in VH 4.2 with the flow split and flow combination elements:

The purpose of these elements is as simple as their description. The flow split element is used whenever the flow is to be split into one or more trains. The flow combination element is used to combine flows from multiple trains into one train. To examine these elements, consider the following hydraulic profile that has just been started:
It’s important to remember that Visual Hydraulics works from downstream to upstream, following the traditional methodology of calculating hydraulic profiles. This can be slightly counterintuitive when thinking about flow splits and combinations, because when you are working upstream with a hydraulic profile you will most likely encounter a flow combination before a flow split. Consider the previous diagram as an example. The most recent element added to the profile is the outfall pipe to stream. Upstream of that, let’s assume that two final clarifier effluent pipes discharge into a junction box before entering the final outfall pipe. Therefore, a flow combination is required upstream of the outfall pipe. When the flow combination element is selected, the user is requested for a description:

The user provides a description of the flow combination element to be placed upstream of the outfall pipe. Similar to adding any other new element, the user then must click on the screen to indicate the location the new element should be placed:
The flow combination element has now been added to the profile, and separate trains can be branched off from this flow combination element. As was stated previously, assume there are two final clarifier effluent pipes upstream of the outfall pipe, so the first can be added to the hydraulic profile:

Once the first effluent pipe has been added, the second parallel effluent pipe can be added. Remember to first select the flow combination element BEFORE adding the second pipe, since Visual Hydraulics adds elements upstream of the selected element. The easiest way to add the second pipe would be to just copy and paste the first pipe. When the second pipe is added, the profile appears as follows:
Note that the water elevations for the two effluent pipes are the same, since the effluent pipes are identical. Addition of new elements upstream of both trains can now proceed, with the hydraulics of both streams being calculated independently.

To analyze a flow split element, consider the same hydraulic profile that has progressed upstream through the final clarifiers to the final clarifier influent pipe. Upstream of the final clarifier influent pipes is a flow distribution structure that splits the flows between the two final clarifiers. In this case, assume the distribution structure consists of two weirs that divide the flows evenly between the two clarifiers. The following diagram depicts the hydraulic profile just downstream of the flow split, before that element has been added:

To add the new flow split element upstream of the two final clarifier influent pipes, the flow element option is selected, and the software first requests a description for the new flow split:

Once the description is provided, the flow split element design form is displayed:
There are two tabs contained within the flow split element design form. The first tab displays the available downstream elements that will contribute to the flow split. Note that any element that does not have an upstream dependency will be considered as a possible contributor to the flow split, and the user has to select which downstream elements apply. In this case, both of the final clarifier influent pipes are downstream of the flow split, so both boxes can be checked:

Once the downstream elements have been checked, the “Characteristics” tab can be selected, and will appear as follows:
The preceding diagram illustrates the various sections of the flow split element design form. Note that the user may choose a weir split or a user defined split. If a weir split (most common) is chosen, the user will then also provide the characteristics of the weir (type of weir, weir invert, and total weir length), and the associated water level through the flow split will be calculated by Visual Hydraulics and will be based on the head over the weir. If the user defined split is chosen, the user must provide an estimate of the head loss through the flow split device:
In this case, assume that a weir is used to split the flow, and the weir is a sharp crested weir with a total length of 8 feet (4 feet per division). The information is then provided as follows:

![Flow Split Element](image)

The required information for the flow split element has now been provided, and once the user clicks on the screen to specify the location of the flow split element, it will be added to the hydraulic profile:

![Visual Hydraulics - VH Plant Upgrade](image)

The water elevation for the flow split in this case (726.52) is the elevation of the head over the weir.
Parallel train considerations

Using the flow combination and flow split elements provided with Visual Hydraulics, the user can analyze any number of process trains that may occur. There is no limit to how many different trains can be split and combined. Additionally, the split trains do not have to be equal. Once a treatment train is split from another, the hydraulics of each train are independent until they are combined at some point upstream (if at all). Take a look at the following hydraulic profile, which has two slightly different treatment trains and slightly variable hydraulics:

![Hydraulic profile diagram]

In the hydraulic profile above, two separate flow trains exist. Part of the flow goes through its normal path all the way through the chlorine contact tanks and out the outfall. The remaining flow bypasses the chlorine contact tanks until it reaches the chlorine tank discharge point, where it is recombined before exiting to the outfall. In this case, the hydraulics of the parallel trains are different and independent of each other between the flow combination and flow split elements. The water elevation in any flow split element upstream of multiple trains will be dependent on the highest water elevation just downstream of the flow split, in this case that elevation would be at the final clarifier effluent pipe, which has a higher water elevation than the final clarifier bypass pipe. Therefore analysis of the flow split device will consider the downstream elevation of the final clarifier effluent pipe as the downstream controlling elevation.
PART IV – USING THE INDIVIDUAL SECTION MODE

The individual section mode allows the designer to analyze any hydraulic feature at any time, independent of the current hydraulic profile. The default mode of operation is the full profile mode, and the individual section mode can be set by using the ‘Mode’ option on the main menu:

Use of the individual section mode was covered briefly in Part II – Program features. Again, when the individual section mode is selected, many of the program options are “locked out”, including the hydraulic profile table and hydraulic profile diagram. In addition, menu items applicable only to working with a full hydraulic profile will be disabled as well.

To analyze a section individually, the section toolbar is used. For example, to analyze a sharp crested weir separately from the hydraulic profile, the weir option is chosen:

Similar to working in full profile mode, the types of losses form is then displayed:
The weir option is selected by default, and a description of the weir to be analyzed must be provided as well. This brings up the types of weirs form:

A sharp crested weir is chosen, which brings up the sharp crested weir design form:

The designer provides the necessary weir characteristics, in this case the weir invert and weir length on the “General” tab. The “Flows” tab is much different for the individual section mode – only one flow box is provided:
Once all the required information has been provided, the “Finished/Update Profile” button is used to initiate the analysis:

When an analysis is performed on an individual section, the program automatically summarizes the results and places them on the summary form for the type of loss being analyzed. In this case, the summary form for the sharp crested weir is then displayed:
Similar to performing a calculation summary when in full profile mode, the summary forms contain both a diagram and general information summary tab. Also located at the bottom of the form is an “Edit Section” button, which will allow the designer to make changes and reanalyze that section if desired.

The summary report of an individual section may also be generated. At the bottom of the form is another button, the “Generate Summary” option. This will instruct the program to create a report of that section:

This allows the designer to print a hard copy of the summary results if desired.

Once finished with the individual section mode, the main menu is used to switch back to full profile mode and reactivate the current hydraulic profile:
PART V – ALTERING AN EXISTING PROFILE

Editing an existing section

Once a section has been added to a profile, it may be necessary to go back and change some of its features. Coming up with a system that is hydraulically sound is not a cut and dry solution, it’s a trial and error method that requires multiple runs to make it work. And when a section is changed and this change affects its hydraulic properties, that in turn may affect many or all of the sections upstream of that point.

One of the main advantages of a computer generated model is being able to quickly make changes and have the model determine the affect of those changes.

To edit a section that has been added to the profile, there are a variety of options available. The user may double click on the element that is to be modified, the user may select the edit option from the toolbar, or the user may use any of the drop down menus available that provide the edit option. The diagrams below demonstrate these options:

Option I – double click element

Option II – toolbar

Option III – main menu

Option IV – right click element for drop down menu

As an example, consider a portion of a hydraulic profile, which is shown following:
If the user wants to edit the section titled “24-inch filter effluent flume pipe 1”, one of the methods to edit the section discussed previously can be used. When the section is selected for editing, the design form for that element is displayed with all of the element’s data for the user:

![Design form for pipe characteristics](image)

At this point the designer can change any values on the design form, erase fittings, add fittings, change flows, etc. Assume the length is increased from 17 feet to 170 feet:

![Updated design form with new length](image)

When all the desired changes have been made, the “Finished / Update Profile” button is selected, and the program will recalculate the hydraulics of that section as well as the entire hydraulic profile upstream of the section being edited:

![Updated hydraulic profile](image)
With the increased length, the pipe head loss jumps from 0.05 feet to 0.34 feet. If the rest of the profile is examined upstream, this change also affects all the sections upstream of the profile. If changes are made to a section, it is important to check and see how the changes will affect sections upstream of the profile.

**Inserting a new section between two existing sections**

It isn’t uncommon to encounter a situation where a profile is analyzed and a new loss is introduced into the system or overlooked during design. Because of this, the program offers the option of inserting a new section between any two existing sections of the hydraulic profile. When the new section has been inserted, the entire profile is re-analyzed to account for the new section.

To insert a new section in between two existing sections, the point of insertion must first be provided by selecting the appropriate element. The section to be inserted will be placed AFTER the selected element. In the diagram below, the element titled “Final clarifier loss” is selected:

![Diagram showing element selection](image)

In this case the new section to be inserted will be placed between the sections titled “Final clarifier loss” and “Final clarifier influent pipe”. The user would then choose to add the new section in a similar fashion to adding any new section of the hydraulic profile. Assume that the user wishes to insert a head loss for the final clarifier influent ports. The orifice/baffle element is then selected:

![Orifice/baffle element selection](image)

To confirm the correct insertion point, the program will then provide the following message:
From this point on inserting a new section is like adding a new section to the profile, which is essentially what is being done. First the section type form is displayed:

For this example, influent ports in the final clarifier will be analyzed. The orifice/baffle feature would be used in this case:
On the general information tab of the form, 6 inch by 6 inch inlet ports are specified, and there are 18 total ports. The flow information is also required, and once this information has been provided, the profile can then be updated to include the new section. Similar to adding any new section to the hydraulic profile, the user has to graphically specify the insertion point, in this case the point is chosen somewhere between the two elements the new element is being inserted in:

Space is created for the element to be inserted

The new element is inserted, and the entire hydraulic profile is reanalyzed

The port loss is 0.75 feet. It does create an issue upstream, however. Two sections upstream of the newly specified ports is the final clarifier influent splitter weir. With the addition of the port loss, the water level just downstream of the splitter weir at design flow is now actually slightly higher than invert of the weir (which is set at 723.65). The program catches this and reports it with an alert:
The designer has various options to correct this situation. The invert of the weir is currently set at 723.65, so the weir is submerged a few inches under high flow conditions. The weir can be slightly raised, or the influent ports in the final clarifier can be made slightly larger (say 8” x 8” instead of 6” x 6”). In this case, the weir is used primarily for flow splitting, therefore the slight submergence under high flow conditions should not have a dramatic affect on the operation of the plant.

Deleting an existing section

Any section of the hydraulic profile may be deleted at any time. The process is very simple – the section to be deleted is highlighted:

In this case the section titled “Final clarifier influent ports” is to be removed. A section may be deleted in one of two ways, either by using the toolbar or the drop down menus:

Once a section is deleted, the program re-analyzes the entire profile.

Changing flow conditions

Flow conditions are set at the outset of the analysis of a new profile, but flow conditions may be added or removed at any time. In addition, many hydraulic profiles are analyzed at a variety of flow scenarios, not just one.

The flow menu may be accessed by either using the toolbar or the drop down menus:
This will bring up the flow form for the active design:

Flows may now be added or removed following the procedures outlined in Part III, Beginning a new hydraulic profile. If multiple flow conditions have been specified for the profile, a new flow condition may be selected, and in turn the entire profile can be analyzed at the new flow condition.

For this example, a new flow scenario is chosen – 10 mgd for the forward flow and 0.5 mgd for the return I flow condition:
When the desired flow condition(s) have been chosen, the “Finished” button is selected. When new flow conditions have been selected for analysis, the profile will have to be reanalyzed at these new flows. The program provides the following message:

The active flow conditions will now be 10 mgd for the forward flow and 0.5 mgd for the return I flow. The active flow conditions are displayed on the left side of the main screen:
Specifying a no flow scenario

It may be desired during an analysis to disable a flow condition, so that it is not considered. A common example would be an activated sludge plant under high flow conditions due to excessive rainfall. Because the majority of the flow being conveyed through the plant will be rainwater, the plant will most likely operate with no return flow because rainwater will simply be recycled back through the plant.

To include a no flow scenario for any of the flow conditions, a flow value of “0” should be added to the appropriate flow condition using the flow form:

Here, the 0 flow value is to be added to the return I flows. The procedure is the same as adding any flow condition, the 0 value is provided in the box and the “Add” button is chosen:
The program updates the flow database and displays the 0 flow value in the appropriate flow list on the form. Again, the active flows are the ones highlighted and displayed in the blue boxes on the form, so make sure the appropriate flows are selected before reanalyzing the profile. If the 0 flow condition is chosen, in this case for the return I flow condition, the main screen simply shows no active flow for this condition:

Using the flow splits to take units/trains off-line

Most treatment plants have redundancy when it comes to treatment units, meaning two or more units accepting the flow through that particular section of the treatment process. Additionally, some treatment trains may operate in parallel but have completely different hydraulic characteristics. Visual Hydraulics provides the option of taking any treatment train off-line at any time. This is not uncommon at treatment plants, as trains are kept off-line due to low flows, optimizing operations, or maintenance. Treatment trains are taken off-line at the flow split element where they are split into multiple trains.

Treatment trains can be put on or taken off-line using the “units on-line” option available on the toolbar or the drop down menus:

Note that taking units off or putting units on-line can only be accomplished at the flow split device. The flow split device MUST be selected for this option to be available, otherwise the software will provide the user with the following message:
Consider the following abbreviated hydraulic profile, which was shown as part of an earlier example:

In this case, there are two separate trains, and either train can be taken off-line (or put back on-line) at the flow split element titled “Chlorine bypass chamber”. So if this element is selected and the “Units on-line” option is chosen, the following form is displayed:

The flow split element is currently splitting the flow between two trains, with the elements just downstream of the flow split being displayed in the “Downstream Element” box for the user. The form also provides the user with the flow through the train, as well as whether the train is on or off-line. For this example, assume the bypass train is to be taken off line. The user would simply highlight the “Final clarifier bypass pipe” (as shown) and select the “Take off-line” button:
The following confirmation message is then provided:

As the confirmation message states, the software will take all units off-line downstream of the flow split element until a flow combination is reached. If the selection is confirmed, the train is taken off-line and the flow is redistributed through the remaining on-line train(s). If more than one train remains on-line, the flow from the train taken off-line will be redistributed through the trains remaining on-line BASED ON THE CURRENT FLOWS THROUGH EACH TRAIN. In other words, if two trains remain on-line and one of those trains has more flow than the other, the additional flow from the train taken off-line will be redistributed accordingly. In this example, both trains were accepting 5 MGD of flow. Therefore, when the bypass train is taken off-line, all of the flow is rerouted through the remaining train:

Additionally, the bypass train is marked as “Off-line”, and the remaining train remains “On-line”. The hydraulic profile can now be updated by selecting the “Update Profile” button:
Innovative Hydraulics

The bypass line has been taken off-line, the flow re-routed through the remaining train, and the profile reanalyzed. In this case, the chlorine tank effluent weir and chlorine bypass chamber weir are showing alerts, and if the alerts are checked it would be found that both weirs would be partially submerged under the higher flow conditions. Putting the bypass train back on-line would restore the hydraulic profile to its previous condition:

Handling return flows when trains are taken off-line or put on-line

Another topic that deserves extra attention is the case where return flows exist in the hydraulic profile and streams are taken off-line or put on-line. Unlike forward flows, which are transmitted throughout all of the elements in the profile, it is likely that return flows are only transmitted through part of the hydraulic profile. This makes them very complex to handle when it comes to redistributing return flows among treatment trains.

No matter what condition is encountered, Visual Hydraulics will always redistribute forward flows when streams are taken off-line. Visual Hydraulics will also redistribute return flows if the element downstream of the flow split element where the train is being taken off-line (or put on-line) has return flows associated with it. If the software encounters a situation where return flows exist “midstream” along a train being taken off-line or put on-line, those flows will not be redistributed to the remaining on-line streams. There are too many complex scenarios that can
occur for redistribution when this occurs, and the software was not designed to try and “predict” what the user may have in mind for redistributing these return flows. Consider the following scenarios, which explain this in more detail.

In the diagram above, Qf is forward flow (5 MGD total, all trains), and Qr is return flow (1 MGD total, all trains). The flow is divided equally between the two trains. In this case, the flow split occurs at the “Influent flow split” element. Gates 1 and 2 are directly downstream of the split and have the return flow associated with them. Because of this, if one of the trains is taken offline, the forward flow and return flow will be distributed accordingly:
As can be seen, both the forward flow and return flows are redistributed to the remaining on-line train. Note, however, that no return flow is distributed to “Effluent weir 2”, since no return flow was specified for that element originally. If no return flow exists for an element, no return flow will be redistributed to that element when trains are taken off or put back on-line.

Now consider this scenario, slightly modified from the previous scenario:

In this case, the flow for the elements downstream of the flow split element where the train is being taken off-line is “0”. For this scenario, Visual Hydraulics will redistribute the forward flows, but will not redistribute the return flows. If one of the trains is taken off-line in this case, the user will receive the following message:

When the stream is taken off-line and the flows redistributed to the remaining on-line stream, the software will not redistribute the return flow and the breakdown of flows will be:
In this case the user will need to go in and adjust the return flows as intended to insure that the appropriate flow redistribution occurs.

Looking at this sample hydraulic profile, it seems easy to see how the return flow should be distributed even if the return flow is “0” at the two gates. However, this is a very simplistic profile. Much more complex flow splits and combinations of streams make flow redistribution extremely complex when trains are taken off or put on-line. For example, consider the following hydraulic profile:

This profile represents a complex network of splits and flow combinations, with the treatment trains not running in parallel with matching elements. Redistributing return flows that are located midstream in a hydraulic profile such as this represent too many possibilities for the operation to be performed accurately.
Changing the section description

If a section has already been added to the hydraulic profile and the description needs to be changed, it’s a fairly simple process: located in the main screen’s “Profile” menu is the section description option:

This will display the section description form, which contains the current section description as well as a box for entering a new description:

Selecting “OK” will change the existing description to the one provided in the box above.

Undo/redo operations

Visual Hydraulics offers undo and redo commands which allow the most recent actions performed on the hydraulic profile to be undone, and redone if necessary. The undo command is available when a new section is added to the profile, an existing section is edited, a section is inserted or deleted, a flow condition is changed, and so on. Clicking on the undo button simply restores the profile to the way it was before the most recent change was made. Visual Hydraulics allows for five undo operations before the undo option becomes inactive. The most recent action that is undone can be redone once.
PART VI – SUMMARY DATA

A big part of the hydraulic design process is being able to analyze the results and determine if those results are acceptable for the given design conditions. Simply having a water elevation for a calculated section is not useful unless the designer can see where that value came from. In addition, data is not useful unless it can be summarized in some fashion for presentation. Visual Hydraulics has a variety of summary options that allow the designer to break down each section of the profile, analyze each section over any range of flows, catch unusual conditions, and summarize the data obtained by the program through the use of summary reports. Each of these options will be broken down in detail.

The summary forms

Once a section has been added to the hydraulic profile, the results can be checked by requesting a calculation summary from the program. This will report what values were calculated in determining the head loss or head as well as summarize the data that was provided by the user for that section. To generate the summary form for a particular section, the section must first be highlighted in the hydraulic profile table. The summary form can then be accessed by using the toolbar or the drop down menus:

For example, if a calculation summary is generated for a pipe section, it would appear similar to the following form:
All of the summary forms for each hydraulic feature available have two main tabs, the diagram tab and the general information tab:

The diagram tab provides a pictorial summary of the results obtained from the data that has been provided. Also on the bottom of the diagram tab are the option buttons:

The “Close” button simply closes the summary form and returns the designer to the main screen. The “View equations used” button and “Generate summary” button are discussed in more detail later in this section.

The general information tab for this pipe section appears as follows:
This tab provides a summary of the data provided by the designer and also a summary of the hydraulic features of the pipe that were calculated by the program.

The diagram tab may differ for some of the sections depending on the hydraulic features of that section. For example, for a rectangular weir operating under normal conditions, the diagram would appear similar to the following:
If the weir is submerged however, the diagram tab would show a submerged weir diagram:

**Generating the summary equations**

In addition to being able to view the summary data for a particular section, Visual Hydraulics also provides an option for the designer to view the equations and methodology used by the program to come up with that particular head or head loss value. This is done using the “View equations used” option, which is on every summary form:
When the “View equations used” button is chosen, the program displays the equation(s) used to determine that loss:

This allows the designer to see how the program calculated the head or loss for that section and make sure the values are reasonable.

Similar to the summary diagrams, the equations depend on the condition of the section that has been analyzed. Again, using the rectangular weir as an example, normal operation would generate the standard equations for calculating the head over a weir:

A submerged weir requires a different type of analysis and a different methodology:
A detailed discussion of the equations and methods used by the program for each hydraulic feature is provided in the appendix of this manual.

**The head loss graph**

Another summary feature available to the designer is the head loss graph option. Once a section has been added to the profile, the program can be instructed to generate a head loss graph for that section. The program will graph a flow vs. head loss (or head, if it’s a weir or flume) curve for a range of flows above and below the specified flow for this section. This is useful for determining how that section behaves hydraulically over a range of flows.

To generate a head loss graph for a section, highlight that section in the hydraulic profile table. The toolbar or main menu may then be used to display the graph:

A sample head loss graph is shown for a typical pipe section as follows:
Flow is graphed on the x-axis and head or head loss is graphed on the y-axis. The graph is constructed over a range that goes from 0 to twice the design flow for that section. The flow increment and range are displayed at the bottom of the form.

The default range of 0 to twice the design flow is initially used for the graph, but this range may be changed to any value the designer wishes. This is done by altering the flow increment. In this example, the range is from 0 to 14 mgd and the flow increment is 1.4 mgd. To alter the increment, simply change the value provided in the box, then hit the “Update graph” button:

Here the increment is being changed from 1.4 mgd to 3 mgd, which will make the graph range go from 0 mgd to 30 mgd:
The head loss graph is the most convenient summary method available for quickly determining how a hydraulic feature will behave over any range of flows.

**Operating conditions**

The operating conditions were mentioned very briefly in the discussion of the features available from the program. This feature was added to alert the designer to any unusual conditions that may be encountered while analyzing the profile. These “unusual conditions” may include extremely high head losses, submerged weirs, high velocities, etc. When an operating condition is encountered by the software that falls outside of the range of normal operation, the software provides to the user an alert. This alert is represented by a small red box with a white exclamation point, which is placed at the bottom right corner of the element:
If the program reports that a section is operating in a state other than “Normal”, a description of the operating condition can be generated for the section. This can be done by first highlighting the flagged element on the screen and then using either the toolbar or main menu:

For the “Fully submerged” weir condition shown in the profile above, the following message would be provided:

The operating conditions are provided for information only. Often unusual conditions such as submerged weirs may be perfectly acceptable and even desirable in some cases. They are merely provided as a tool to the designer.

**The summary reports**

Visual Hydraulics offers a range of reporting features, including full reports for an entire profile to reports of individual sections of the profile. If a section is analyzed in individual section mode, a report may be generated for that section as well.
Full profile report

The full profile report is a summary report for the entire hydraulic profile that has been analyzed. It includes the flow conditions specified and a detailed summary of each individual section of the profile. The summary reports may be accessed in two ways, either the toolbar or main menu:

This will display the report menu form:

This menu contains two option buttons, the full profile report option and individual report option. The individual report option is discussed later in this section. To generate a full profile report, make sure that option is chosen and then select the “Create Report” button at the bottom of the form:
The full profile report is then displayed:

![Full Profile Report](image)

The report format includes a listing of the design flows at the top of the report and then a summary of each section of the hydraulic profile. The top of the report provides an option for resizing the report and also provides options for printing and exporting the report to a variety of formats so that it may be saved. The exporting option is discussed later in this section. The scroll bar at the right of the report allows the designer to view the entire page, and the arrows provided at the top of the report are used for moving between each page in the report.

### Individual Reports

In addition to the full profile report, a detailed report of any section may also be generated:

![Individual Sections for this Profile](image)

To create an individual report, the “Individual Report” option button is first selected. This will activate the list box to the right, which contains all of the sections available for reports. As a note, individual reports cannot be generated for elevation steps/drops, since they are simply a
value provided to the program. To display an individual report, highlight the appropriate section in the list box provided and then select the “Create Report” button:

The individual reports include a diagram and detailed summary of the values provided and calculated by the program for that section. The report is pre-formatted and may be printed by simply using the icon provided at the top of the report.

Summary reports may also be generated directly from the summary forms. All of the summary forms have a “Generate Summary” button. This option is also available when working in individual section mode:

Graph / Summary Table Report

The final reporting option available is the graph and summary table report. This report is also performed on an individual section, and will display a summary of the hydraulic behavior of the selected section over a range of flows that is set by the designer. Going back to the report form, the final option should be selected to generate the graph / tabular summary. In addition, two input boxes are provided that allow the designer to set the range the summary will be performed over. A range must be entered before the “Create Report” button is selected and the report can be generated. The following steps are used to generate the summary: first highlight the section
to be summarized in the list provided on the form. Next select the "Graph / Table" option, and in the boxes provided enter the range of flows that should be analyzed and summarized by the program. In the example below, the section titled "24-inch filter effluent flume pipe 1" is to be summarized over the range of 2 to 20 mgd.

Choosing the "Create Report" option then generates the summary report for the selected section:
Exporting and saving a report

Once a particular report has been generated, it may be saved in a variety of formats. These include the rich text file format (.rtf), Microsoft® Word format (.doc), and Adobe® Reader format (.pdf). In order to save a report, simply click the export report option on the report toolbar:

This will bring up the exporting options dialog box, which supplies the option of choosing a particular file format for saving the report:

In this case, the .pdf format is chosen for saving the file. The program will then provide a standard save dialog box:

The report will then be saved in the chosen format at the location specified.
Generating the Water Surface Elevation Graph

In addition to the graphs and reporting features that were previously discussed, the user also has the option of generating an Excel® based graph of the entire hydraulic profile. The user’s machine must have Microsoft Excel® for this feature to work. Visual Hydraulics will pass the data from the current hydraulic profile to a spreadsheet, where it is stored and summarized in chart format. The water surface elevation graph is accessed from the “Summaries” menu item on the main screen:

This will bring up the water surface elevations graph options form. From this form, sections can be added or removed from the graph to be generated. When satisfied, the graph is created by selecting the “Create Graph” option:

The graph of the entire hydraulic profile is then displayed:
Another useful feature of the software’s summary options is the ability to save a copy of the main screen diagram that depicts the hydraulic profile. This diagram may be useful in drawings and/or reports, so the option was provided that allows the user to save this diagram as a JPEG (.jpg) file. The main screen diagram may be saved at any time by accessing its button on the toolbar:

When this is done, the software provides the user with the option of saving the main screen diagram at any location:
One note on saving the main screen diagram: the files that are saved may get quite large, depending on the size of the diagram to be saved. Picture manipulation may be performed (such as changing the file type to a .jpg or .gif) or the resolution may be modified to shrink the file size, if desired.
PART VII – DESIGN OPTIONS

In addition to the many design features already covered, Visual Hydraulics also offers some specialized design options for the user. This section is devoted to summarizing these features and providing examples regarding how they can be used.

Hydraulic profile flow management

One of the new features in Version 4.2 is the flow management tool. While flows for any individual hydraulic element can be changed at any time by editing that element, users have expressed an interest in being able to manage the flows for all of the hydraulic elements from one location. To access this feature, use the “Flows” icon from the toolbar or the “Options – System Flows” menu item:

This will bring up the system flows form, and the Element Flows tab should be selected:
As can be seen on the form, all of the hydraulic elements and their respective flows and flow percentages are displayed for the user. From this form the flow for any element can be manipulated. Simply highlight the element to change, and the flows for that element will be displayed in the individual boxes at the bottom of the form. Using the flow box or percentage box, the user can change the flow for that section:

![Hydraulic Profile System Flows](image)

It should be noted that flow values cannot be changed for flow combinations or flow splits, if one of those elements is selected, the following is displayed on the form:

![Flow Scenario: Forward Flow](image)

Flow values also cannot be changed if an element is off-line. If elements are off-line, they are also flagged on the form:

![Flow Scenario: Forward Flow](image)
Starting a flow path separate from the hydraulic profile

Another new feature in Version 4.2 is the ability to add a new flow path that is completely independent of the current hydraulic profile. Users of the previous version expressed a need for this capability, so it was added to the software. One of the most common examples of a separate path in a hydraulic profile is a flow bypass path that may have a separate outfall. Consider the following sketch:

In this case, under certain flow conditions, flow would be diverted from the main stream and sent to the “bypass” stream. This scenario can be modeled in Version 4.2 by following these steps. Consider the very basic hydraulic profile below:

If the user wants to start a new flow path separate from the existing hydraulic profile, the “Profile – Start Separate Flow Path” option is chosen from the main menu:
This brings up the following form:

![Starting elevation for separate flow path](image)

The user provides the new flow path starting water elevation (which can also be determined from a weir or drop, as can be seen from the form) and a description of the downstream starting point, in this case “Bypass discharge point”. When complete, the new path can be started on the main screen:

![Diagram](image)

Once the new path has been started, additional hydraulic elements can be added:
In this case, the separate path can then be connected to the existing hydraulic profile at the “Bypass flow split” element simply by editing the flow split element:

The hydraulic profile is then updated accordingly:
PART VIII – USING THE SPECIALIZED DESIGN TOOLS

Critical Elevations

Is it not uncommon for there to be elevations that are crucial to the proper analysis of a hydraulic profile, elevations that must be maintained or minimized to insure that the overall hydraulic behavior of the system is not compromised. Examples are submerged weirs, overflowing channels, and manufacturer required elevation set points for equipment. Visual Hydraulics has a built-in feature that allows the user to set a critical elevation for any section of the hydraulic profile, and that feature will flag the user if that critical elevation is ever reached during an analysis. The critical elevations option can be accessed from the “Options” menu on the main screen:

Selecting this option will in turn bring up the critical elevations form:
A critical elevation can easily be specified for any section of the hydraulic profile by selecting the desired section and then providing an elevation in the box provided. In this example, the user selects the “UV effluent channel 1 weir” section and provides a critical elevation of 908.5. This critical elevation is then saved in the software, and if the elevation is encountered during an analysis, it is flagged as an operating condition:

As can be seen from the diagram above, the UV effluent channel 1 weir is flagged with a red boxed exclamation point. If a description of the flag is requested from the software, the following message is supplied:

The special loss regression feature

It’s highly unlikely that every type of loss encountered during design will be available in this program. Treatment plants often have specialized applications or equipment that do not have standardized equations or methods for determining their head loss while they are in operation. Often this information will be provided by the equipment manufacturer.

Visual Hydraulics has a unique built in subprogram that allows the creation of head loss equations based on data provided to the program. The designer specifies various flow vs. head
loss points for the special loss being analyzed, and the program uses regression analysis to correlate the data into a head or head loss best fit equation. Visual Hydraulics offers four different types of regression analyses: linear, 2nd degree polynomial, 3rd degree polynomial, and 4th degree polynomial regressions. Once the data points have been entered, the regression type to be used by the program is chosen, and the program will determine the best fit equation for that data. Once an analysis has been performed, a data correlation factor (R squared value) will also be provided that reports how well the data fits the regression equation. Different regression equations can be sampled to determine which equation best represents the data that has been provided.

In addition to this section of the manual, the help files provided with the program discuss the special loss in detail, and a tutorial is provided as well.

Regression analysis background

Regression analyses are performed using the “least squares” method, which calculates the best-fit equation that represents the minimum amount of deviations from a set of data points. The data points provided are the flow vs. head loss points. Using these points, the program performs the selected regression analysis to determine the equation that minimizes the deviations (errors) from this set of data.

Linear regression equation:

\[ y = mx + b, \]

where

- \( y \) = head loss point
- \( x \) = flow point
- \( m \) = slope of line
- \( b \) = y-intercept

Polynomial regression equation:

\[ y = C_0 + C_1x_1 + C_2x_2 + \ldots + C_4x_4, \]

where

- \( y \) = head loss point
- \( x \) = flow point
- \( C_0, C_1, C_2, C_3, C_4 \) = calculated constants

The methodology behind determining the constants for the various regression equations can be a lengthy and complex process. For more information on regression analysis, any linear algebra text may be consulted.

Getting started

A special loss is added to a hydraulic profile in the same fashion as any other hydraulic feature, by using the new section toolbar on the main screen:
This will display the special loss type form, which includes a regression analysis and pump analysis. In this case, regression analysis is being examined, so the first option should be chosen:

The special loss design form has a variety of sections. The top left frame is for specifying the type of regression analysis to be performed. The top right frame is where the flow vs. head loss data are provided by the designer. The bottom right frame is where the calculated regression equation and R squared value is displayed by the program once an analysis has been performed.

The first step in performing an analysis is to enter the data points to be analyzed. Consider the following data for a vortex grit unit supplied by the equipment manufacturer:

<table>
<thead>
<tr>
<th>Flow (mgd)</th>
<th>Head loss (feet)</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>0.02</td>
</tr>
<tr>
<td>20</td>
<td>0.05</td>
</tr>
<tr>
<td>30</td>
<td>0.09</td>
</tr>
<tr>
<td>40</td>
<td>0.16</td>
</tr>
<tr>
<td>50</td>
<td>0.26</td>
</tr>
<tr>
<td>60</td>
<td>0.41</td>
</tr>
</tbody>
</table>
To enter this data, simply place each flow and corresponding head loss value into the boxes provided, and click the “Add Point” button to add the data point to the list:

When “Add Point” is selected, the data point is added to the storage list:
This is done for every data point in the data set. Once all 6 points have been added, the design form appears as follows:

Data points that have been added to the storage list may be manipulated at any time. To clear the entire list of points, simply select the “Clear All” button. To delete a point, highlight that point in the storage list and select the “Delete Point” button. To change one of the values for a point in the list, highlight that point and select the “Edit Point” button. This will bring up the edit point form:

Simply change either value as needed and hit the “Finished” button to update the point in the storage list.

Once all of the data points have been entered onto the form, the best-fit equation for the data can be generated. The first step in having the computer calculate the best-fit equation is to select the type of equation for the data. Again, four options are available, linear, 2nd degree polynomial, 3rd degree polynomial, and 4th degree polynomial. Choosing the right equation is a trial and
error process, and it’s not a bad idea to check all of the different equations to see what kind of results they will give. In this case, assume a linear equation is tried first. Make sure the “Linear” option is selected, and then hit the “Calculate equation” button:

The program will then perform a linear regression analysis on the data, and display the resulting equation:

In addition to the equation being displayed, the program also displays the R squared value calculated for the data set based on the type of equation chosen. The R squared value is a measure of how well the data correlates to the analyzed regression equation. An R squared value of 1 indicates perfect correlation, or no deviation from the equation for each point in the data set. Values less than 1 indicate deviations from the equation: the lower the R squared value, the greater the deviations. In this case the R squared value is 0.886, which is not bad, but not very good, either. For this type of application R squared values should be very close to one, in the range of 0.98-0.99 for reasonable accuracy.

Another measure of how well the equation performs is to simply test it. This option is also provided by the program. On the left side of the form is a “Test Calculated Equation” button,
which will allow the designer to use the calculated equation to see how well corresponding head loss values are calculated from flow values. When this button is selected, the program provides the following input box:

In this case, a flow rate of 50 mgd is to be checked. Clicking “OK” will display the result:

The linear equation predicts a value of 0.30 feet. Checking the data points, the actual head loss value at this flow is 0.26 feet. While this correlation isn’t that far off, it’s obvious that as the flow rates get higher the head loss values begin to increase significantly. A linear equation will not accurately predict the hydraulic behavior of this unit at higher flows, therefore another equation should be used.

Next the polynomial equations should be tested. Select the “Polynomial” option on the form, and the polynomial degree drop down box is activated:

For a 2nd degree polynomial, the result is much better:
This equation produces an R squared value of 0.9908, which is very good and indicates a very high degree correlation. Again, the data should still be tested:

This result should be satisfactory. However, if even more correlation is desired, the 3rd degree and 4th degree polynomials will provide an even higher degree of correlation.

Checking the validity of the equation

There is still one more thing to consider when using the special loss feature. Once a special loss has been added to the profile, a calculation summary should be performed on that section. Again, this can be done by highlighting the special loss in the hydraulic profile table and either using the toolbar or main menu:
When this is done for a special loss, the program will display a graph of the equation over a wide range of flows. For the previous example, this graph would appear as follows:

![Graph of equation](image)

This is an important step because many equations, such as 2\textsuperscript{nd} degree polynomials, can follow a parabolic path. This means that the head loss values will continue to increase to a maximum value, then decrease from that point on. Head loss values will obviously not decrease at higher and higher flows, so this should be checked once an equation is chosen. One way to ensure this will not be a problem is to provide data points beyond the expected maximum flow the unit will be handling. For this example, the equation does not follow a parabolic path and is perfectly acceptable for this set of data. Now when the designer changes flow conditions and reanalyzes the hydraulic profile, the program will accurately account for this with this special loss as well.

**The pump analysis tool**

Almost every treatment plant has at least one or more sets of pumps for raising the hydraulic head of the flow in the plant and allowing the flow to proceed through the plant by gravity. Pumps actually represent drops in the hydraulic profile, since the analysis of hydraulic profiles typically proceeds from downstream working upstream. Visual Hydraulics allows the designer to enter flow vs. head data points for a pump and then uses a second degree polynomial regression on those data points to formulate a pump curve that will represent the drop in head from upstream to downstream over any range of flows. Consider the following illustration:
The elevations shown above represent the water levels upstream and downstream of the pump at different flow conditions. The following table summarizes the difference in head at the varying flow conditions:

<table>
<thead>
<tr>
<th>Flow (mgd)</th>
<th>Head (ft)</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>54.5</td>
</tr>
<tr>
<td>10</td>
<td>49.5</td>
</tr>
<tr>
<td>15</td>
<td>40.5</td>
</tr>
</tbody>
</table>

In order to add a pump to the hydraulic profile, the special loss option should be selected from the new section toolbar:

This will display the special loss type form, which includes a regression analysis and pump analysis. In this case, a pump analysis is being examined, so the second option should be chosen:
The pump design form provides the designer with the ability to enter flow vs. head points for the pump being analyzed. Up to 5 points may be entered, but only 3 points are required. No less than 3 points may be entered. The more points that are entered, the more accurate the pump equation will be over the entire range of flows. The head values are the difference in water level between the downstream suction level and the upstream water level the pump is pumping to. If the values in the table shown previously are entered into the design form, it would appear as follows:
In almost any pump analysis the difference between the upstream and downstream water levels should decrease as the flow increases. This is because a pump’s head capacity decreases as the flow level increases. Again, a minimum of three points are required to generate the pump curve being analyzed. Once the points have been added to the form and the appropriate flow has been specified on the “Flows” tab, hitting “Finished / update profile” will add the pump to the hydraulic profile:

As can be seen from the hydraulic profile above, the pump actually drops the level of the hydraulic profile by the head calculated for the pump.

Specifying a constant head condition

Because it may be difficult to quantify exactly how the head changes with flow for a pump or pumps, it may be more desirable to set up the hydraulic profile to have a constant head drop condition for the pump over any range of flows. In other words, no matter what the flow is, the head drop from downstream to upstream will remain constant.

This is easily accomplished with the pump option in Visual Hydraulics. Simply specify the same head drop for the varying flow conditions on the form. For example, if the designer wants to assume a constant head drop for a set of pumps of 50 feet, then 50 feet should be entered for all of the flow conditions on the pump design form:
As can be seen above, a head value of 50 feet is entered for the entire range of flows. This allows the pump drop to be 50 feet, no matter what the flow is through the pump(s).
The flow split distribution analysis tool

The biggest advancement in Visual Hydraulics 4.2 is the development of the flow split distribution analysis tool. It also represents one of the most challenging tasks in system hydraulic analysis. Although proper design procedures stress the need for measures to be implemented to insure appropriate flow split between multiple units, this often does not occur and may not even be possible.

The hydraulic process of analyzing how flow is distributed among various paths can be extraordinarily complex, depending on the nature of the flow paths and the number of flow paths that must calibrated throughout the hydraulic profile. Visual Hydraulics uses the concept of “equalized head”, which is the principle that flow will be distributed among various paths until the water elevation upstream of all of those paths will seek an equal water surface elevation. Therefore if more head is required to pass flow through one path, less flow will be conveyed to that path and more flow will be conveyed to the flow path that requires less head. A balanced system has equal head losses across all flow paths and the sum of the flows through each flow path will add up to the total flow:

\[ \Sigma Q_1 + Q_2 + Q_3 + \ldots + Q_n = Q_{\text{total}} \]  

(1)

\[ \text{WSEL}_1 = \text{WSEL}_2 = \text{WSEL}_3 = \text{WSEL}_n \]  

(2)

Consider the following very basic flow path split:

In the scenario above, this system is considered to be balanced. The WSELs downstream of the flow split are equal, therefore the amount of flow being conveyed to each path is correct. This is a very simple case, with the two pipes comprising the separate flow paths being exactly equal. If a simple change is made to the profile so that the pipes are not equal, the WSELs for each pipe will vary:
In this case, the WSELs upstream of the two pipes are not equal, and this system is therefore unbalanced in terms of flows. More flow should be conveyed through flow distribution pipe 2 than flow distribution 1, and vice versa. The challenge lies in determining how much flow should be distributed through each path, and that’s when the flow distribution tool provided within Visual Hydraulics should be used. In order to implement this option, the user would access the “Options” menu item from the main screen:

Selecting this option from the menu will instruct the software to perform a flow distribution analysis on the current hydraulic profile. The following confirmation message will be provided before proceeding:

This will instruct the software to perform a flow distribution analysis of the entire hydraulic profile. This may take some time depending on the complexity of the hydraulic profile and the number of flow splits. Please be patient.
Something to note from this confirmation message, depending on the size of the hydraulic profile, number of flow splits, and complexity of the flow distribution, the analysis may take some time. During the analysis, the software will provide an update progress of the analysis:

![Flow Distribution Progress](image1)

The progress screen provides the number of iterations performed as well as a progress bar that summarizes how many of the flow paths have been calibrated successfully. When all of the flow splits have been calibrated, the analysis is complete and the results will be displayed in tabular format. This summary form is shown as follows for the simple flow split just analyzed:

![Flow Distribution Analysis Results](image2)

Highlighted on the summary form above are the two varying pipe sections, and it can be seen just how the flows through each path vary in order to provide an equal upstream water surface elevation.
Also provided on the summary form are three other summary values, the “original downstream flow”, “downstream flow after iterations”, and the “difference”. These values are provided as a check on the overall analysis. Extremely complex analyses may result in very slight total flow differences, although it is rare. If a difference does occur, these values let the user know how significant the difference may be, and if the analysis is acceptable. The hydraulic profile screen is then updated to show the revised elevations of the pipe sections, which are now equal:

The example presented previously is the most simplistic case of a flow distribution analysis. Most flow distribution analyses are much more complex and involve items such as weirs, flow paths off-line, and many flow splits and flow paths to consider. Some of these scenarios are further discussed below.

**Weir considerations and flow splits**

Looking back at the flow split hydraulic element, there are two options for the type of flow split specified, a weir flow split and a user defined flow split, where the user defines the head loss for the system, assuming there is no weir present:
If a weir split is specified for a flow split element, additional info is requested from the user, including the length of the weir and weir invert:

In the case of a flow split with a weir, the software must also look at the elevation of the weir when determining the flow split downstream of the weir. Consider the previous simplistic hydraulic profile, but with a flow split containing a weir with the characteristics above. In this example, the weir invert is set at 702.00. If a flow split analysis is now run on the system with the weir flow split, the following results are presented:
As can be seen from the results form, the flow values for the two flow paths did not change. This is because the two downstream water elevations are lower than the invert elevation of the weir, therefore the weir invert governs. The flows to both flow paths will remain the same. If the weir elevation had been lower, say 701.00, then the weir would be submerged and the flows through each path would be affected.

Flow paths off-line

Visual Hydraulics offers users the ability to take entire flow paths off-line downstream of a flow split, with the flow being distributed through the paths that are on-line. If a flow distribution analysis is performed on a profile that has one or more flow paths on-line, the flow path or flow paths are simply not included in the flow distribution analysis (as would be expected). So if a hydraulic profile such as the one below is examined:

In this case, the flow path that is off-line is not included in the flow distribution analysis if an analysis is performed:
Zero flow scenario

One of the more useful applications of the flow split analysis is the determination of flow to bypass paths, which is quite common in treatment plant applications. Often flow splits are set up to allow a certain flow value to be conveyed through a portion of the treatment process, and if that flow value is exceeded, the additional flow will be conveyed around that process (bypassed). Visual Hydraulics can be used to analyze this type of scenario and determine when bypasses occur. Consider the following sample hydraulic profile:

In the example above, the normal mode of operation would be for all flow to follow the “normal path”, or lower path. The flow path with the “overflow weir” element would only come on-line if high flows were encountered. In this case, the user guesses that portion of the flow will go through the bypass path, with the remaining flow being conveyed through the normal path. To check that assumption, a flow distribution analysis can then be performed:

As can be seen from the adjusted hydraulic profile, it turns out that the bypass weir (“overflow weir”) actually does not receive any flow because the other flow path can safely pass all of the flow before the weir elevation is reached. If this situation is encountered by the software, the path will display a “No flow” label above the hydraulic element so the user is made aware that flow is not conveyed to that section of the profile. If higher flows are encountered at the flow split, eventually the overflow weir would become active:
In this case, the weir was lowered significantly, and does become active if another flow distribution analysis is run.
The manifold/diffuser tool

Manifolds and diffusers are often found in treatment plants where the flow is to be distributed over a wide area. This is usually accomplished by discharging the flow through a variety of ports or orifices that are spaced some length apart. Common examples are ocean outfalls (diffusers) and trickling filter distributors. Manifolds and diffusers actually represent one of the more challenging hydraulic problems to solve, and the head loss through the entire system is based on many factors, including the number of ports/orifices, port/orifice size and spacing, diffuser/manifold diameter, riser diameter (if any), and of course system flow. The entire system must be balanced by analyzing the flow through each orifice and determining the head loss over the entire system. For a more detailed discussion on the hydraulics of diffusers and manifolds, please see the appendix section. The diffuser/manifold tool is located under “Options” on the main menu:

![Diffuser/Manifold Tool](image)

This will bring up the diffuser/manifold analysis form:

![Diffuser/Manifold Form](image)
This form has three tabs. The General tab contains information pertaining to the entire diffuser/manifold, including the flow through the system, the roughness of the system (Darcy-Weisbach roughness value), specific gravity of the receiving water, whether or not the system has risers, and how many ports are at each discharge location, either one or two. This is clarified in the diagram below:

![Diagram showing two examples of port configurations: one with two ports per discharge location and one with one port per discharge location.]

In the diagram above, the first example shows two ports per discharge location, whereas the second example shows only one port per discharge location.

To demonstrate the use of this tool, consider the following example. A lake outfall has risers that discharge to one port per discharge location (one side only) and the diffuser is comprised of the following sections. Note that the sections listed below start from the furthest end of the diffuser and work back in the direction opposite of the flow.

<table>
<thead>
<tr>
<th>Diff diameter</th>
<th>Riser diameter</th>
<th>Riser length</th>
<th>Port spacing</th>
<th>Port diameter</th>
<th>No. of ports</th>
</tr>
</thead>
<tbody>
<tr>
<td>36 in</td>
<td>12 in</td>
<td>8 ft</td>
<td>6 ft</td>
<td>3.25 in</td>
<td>30</td>
</tr>
<tr>
<td>42 in</td>
<td>12 in</td>
<td>8 ft</td>
<td>6 ft</td>
<td>3.0 in</td>
<td>45</td>
</tr>
<tr>
<td>48 in</td>
<td>12 in</td>
<td>8 ft</td>
<td>6 ft</td>
<td>2.75 in</td>
<td>75</td>
</tr>
<tr>
<td>48 in</td>
<td>12 in</td>
<td>8 ft</td>
<td>6 ft</td>
<td>2.5 in</td>
<td>90</td>
</tr>
</tbody>
</table>

The plant flow is 40 mgd. The analysis is initiated by providing the required information on the “General” tab.
In this case a value of 0.0015 feet is chosen for the roughness. Since the lake is fresh water, the specific gravity will be 1. Once the general characteristics have been provided, the individual sections can be added for this diffuser. This is done on the second tab, “Diffuser Sections”:

To add the first section, the characteristics of that section are entered onto the form, and then the “Add to List” button is chosen:
When this button is selected, the new diffuser section is added to the list on the form:

This is how the individual diffuser/manifold sections are added to the form. The other diffuser sections are added in the same fashion, and the form should appear as follows when all the sections have been specified:
A diffuser section can be removed from the list at any time simply by highlighting the section to be removed on the form and selecting the “Remove from List” button. In addition, all of the diffuser sections can be removed by selecting the “Clear All” button. Once all the sections have been properly added, the analysis can begin. This is done by selecting the “Calculate” button on the form. The analysis is an advanced iterative process, and may take a few seconds depending on the complexity of the system. Once the program has reached a balanced condition for flow through each port, it will display the results on the “Results” tab:

<table>
<thead>
<tr>
<th>Diffuser System Results</th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Port number</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>3.25</td>
<td>0.30282</td>
<td>5.253</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>3.25</td>
<td>0.30282</td>
<td>5.253</td>
<td>0</td>
</tr>
<tr>
<td>3</td>
<td>3.25</td>
<td>0.30282</td>
<td>5.253</td>
<td>0.0001</td>
</tr>
<tr>
<td>4</td>
<td>3.25</td>
<td>0.30282</td>
<td>5.253</td>
<td>0.0002</td>
</tr>
<tr>
<td>5</td>
<td>3.25</td>
<td>0.30283</td>
<td>5.253</td>
<td>0.0002</td>
</tr>
<tr>
<td>6</td>
<td>3.25</td>
<td>0.30283</td>
<td>5.253</td>
<td>0.0003</td>
</tr>
<tr>
<td>7</td>
<td>3.25</td>
<td>0.30285</td>
<td>5.253</td>
<td>0.0005</td>
</tr>
<tr>
<td>8</td>
<td>3.25</td>
<td>0.30285</td>
<td>5.254</td>
<td>0.0006</td>
</tr>
<tr>
<td>9</td>
<td>3.25</td>
<td>0.30288</td>
<td>5.254</td>
<td>0.0009</td>
</tr>
<tr>
<td>10</td>
<td>3.25</td>
<td>0.30291</td>
<td>5.255</td>
<td>0.0009</td>
</tr>
<tr>
<td>11</td>
<td>3.25</td>
<td>0.30274</td>
<td>5.255</td>
<td>0.0011</td>
</tr>
<tr>
<td>12</td>
<td>3.25</td>
<td>0.30276</td>
<td>5.256</td>
<td>0.0014</td>
</tr>
<tr>
<td>13</td>
<td>3.25</td>
<td>0.30283</td>
<td>5.257</td>
<td>0.0016</td>
</tr>
<tr>
<td>14</td>
<td>3.25</td>
<td>0.30289</td>
<td>5.259</td>
<td>0.0019</td>
</tr>
<tr>
<td>15</td>
<td>3.25</td>
<td>0.30296</td>
<td>5.259</td>
<td>0.0021</td>
</tr>
<tr>
<td>16</td>
<td>3.25</td>
<td>0.30303</td>
<td>5.26</td>
<td>0.0024</td>
</tr>
<tr>
<td>17</td>
<td>3.25</td>
<td>0.30312</td>
<td>5.262</td>
<td>0.0027</td>
</tr>
</tbody>
</table>

The characteristics of each port are summarized, including port flow, discharge velocity, head loss between ports, and head required at the port to produce the port’s discharge flow. Also included at the bottom of the tab is the calculated overall flow (as a check) and total head loss for the entire system. In this case the head loss was only 1.18 feet, which is quite low for a diffuser of this type.

As a note on the affects of the specific gravity of the receiving water, outfalls are very common in oceans and bodies of salt water where the specific gravity of the receiving water is slightly greater than that of the water being discharged. Sea water, for example, has an average specific gravity of around 1.02. If the diffuser has a slope and is not laid horizontally, the specific gravity of the receiving water will have an affect on the hydraulics of the diffuser. The specific gravity is specified on the “General Features” tab of the diffuser analysis form.
Part IX – Dealing with Unexpected Errors

As much as the software is tested and re-tested in an attempt to account for every scenario encountered, errors or anomalies may pop up from time to time. Error messages are provided throughout the software if errors are encountered, and this section addresses issues that could occur that are unexpected. Options have been built into Visual Hydraulics 4.2 that allow the user to rectify some of the issues that have been encountered in the past without having to contact Innovative Hydraulics.

Element becomes “disconnected” from the profile

One of the anomalies that has popped up on rare occasion is a hydraulic element actually becoming disconnected from the rest of the hydraulic profile. Consider the following example screenshot:

As can be seen above, the element titled “42-inch trickling filter effluent pipe to final clarifiers” has become disconnected from the rest of the profile. This element should be just upstream and connected to the “final clarifier flow splitter weir” element. Visual Hydraulics has built in features to deal with anomalies such as this. Under “Project Options” in the main menu, there is an “Element Properties” option:
This option will display the following properties form for the selected hydraulic element:

As can be seen from the properties above, the “final clarifier flow splitter weir” isn’t registering a downstream or upstream dependency, which is incorrect. It should have the element titled “final clarifier flow split” as its downstream dependency and “42-inch trickling filter effluent pipe to final clarifiers” as its upstream dependency. To fix this, the “Change” button is selected. Looking first at the downstream dependency:

The correct downstream dependency is chosen from the list of hydraulic elements provided, in this case the “final clarifier flow split” element. This then corrects the downstream dependency:
The correct downstream dependency is now shown. The same process can be followed for the upstream dependency, and when the “Finished/Update” option is chosen, the hydraulic profile is corrected:
Pressure pipe or conduit

Pipes flowing under pressure represent the most common losses typically encountered at a treatment plant. Many methods and equations exist for determining the head loss through a full flowing pipe. In addition, many other factors exist that may contribute to this head loss. These may include solids in the flow, flow temperature, and corrosion due to pipe age.

Head loss theory

The study of pipe head loss has been one of the most widely documented hydraulic problems since the analysis of flow began. The method for determining the losses in pipes is fairly universal. A pipe will experience two main types of head loss, losses from friction and losses from pipe fittings (minor losses). For calculating the friction loss in pipes, there are three methods that are typically acceptable. They are the Darcy-Weisbach equation, the Hazen-Williams equation, and the Manning’s equation. The Darcy-Weisbach and Hazen-Williams equation are typically more often used for full pipe flow, while the Manning’s equation is the method of choice for conduits flowing partially full, or open channels. Manning’s equation is the method used by this program for open channel analysis and is discussed in more detail in the open channel section of this manual.

Friction loss

The Darcy-Weisbach equation is:

\[ H_L = f \frac{L V^2}{D 2g} \]

The variable “f” is the Darcy friction factor, which may be determined from the Moody Diagram or the following equations:

For a Reynolds number less than 2000, the flow is laminar and the friction factor may be determined by:

\[ f = \frac{64}{R_e} \]

For a Reynolds number between 2000 and 4000 the flow is in the transition phase between laminar and turbulent flow, and the friction factor may be determined by:

\[ f = \left( \frac{0.25}{\text{LOG}(\frac{e}{3.7D} + \frac{5.74}{R_e^{0.9}})} \right)^2 \]

\( e = \) relative pipe roughness
\( D = \) pipe diameter or height
\( R_e = \) Reynolds number
For a Reynolds number greater than 4000, the flow is considered turbulent. Almost all pipe flows encountered will exhibit turbulent flow behavior. For turbulent flow conditions, the friction factor may be determined by:

\[ f = \left( \frac{1}{1.14 - 2 \times \log\left(\frac{e}{D}\right)} \right)^2 \]

The program will determine the Reynolds number and then select the appropriate equation depending on the range that the Reynolds number falls into. This value is then used to calculate the friction head loss in the pipe.

The other popular method for determining the friction loss in a full flowing pipe is the Hazen-Williams equation. This method is more simplistic and does not account for the flow condition in the pipe. The equation is:

\[ H_L = \left[ \frac{Q}{(0.432 \times C \times D^{2.63})} \right]^{1.85} \times L \]

\[ Q = \text{flow} \]
\[ C = \text{loss coefficient} \]
\[ D = \text{pipe diameter}^* \]
\[ L = \text{pipe length} \]

The Hazen-Williams method provides a quick, simple calculation for coming up with a reasonable estimate of the friction loss in a pipe. The loss coefficient values (C) range from 140 for brand new smooth pipe to 80 for older pipe. Values between 100-110 are typically used for average pipe.

* It should be noted that only circular pipes may be analyzed using the Hazen-Williams method. Rectangular or irregularly shaped conduits must be analyzed using either the Darcy-Weisbach method or Manning’s equation.

The Manning’s equation is the final friction loss method available with the program. The Manning’s head loss equation is:

\[ H_L = \left[ \frac{(V \times n)}{(1.49 \times R^{0.67})} \right]^2 \times L \]

\[ V = \text{velocity} \]
\[ n = \text{Manning’s loss coefficient} \]
\[ R = \text{pipe hydraulic radius} \]
\[ L = \text{pipe length} \]

Manning’s “n” value ranges from 0.011 for new, smooth pipes to 0.016 for older pipe.
Fitting (minor losses)

The other type of loss that a pipe will experience will be the fitting or minor losses. The universally accepted method for determining the head loss from a pipe fitting is to multiply the velocity head through the fitting by an empirically determined $K$ value. The head loss equation is:

$$H_L = K \times \frac{V^2}{2g}$$

$K$ = fitting $K$ value  
$V$ = velocity  
$g$ = gravitational acceleration

The $K$ values are unique for each fitting, and have been established over time through numerous tests and observations. Some $K$ values are fairly universal, such as the $K$ values for bends and elbows, while other less common fittings may have $K$ values that vary depending on the source of information that is consulted. The following table provides a list of the fittings and $K$ values that are initially included in the program.

**Table of fittings available from Visual Hydraulics**

<table>
<thead>
<tr>
<th>Fitting</th>
<th>Kvalue</th>
<th>Fitting</th>
<th>Kvalue</th>
</tr>
</thead>
<tbody>
<tr>
<td>90 degree elbow, standard</td>
<td>0.25</td>
<td>Entrance, flush</td>
<td>0.50</td>
</tr>
<tr>
<td>90 degree elbow, long radius</td>
<td>0.20</td>
<td>Entrance, projecting inward</td>
<td>1.0</td>
</tr>
<tr>
<td>45 degree elbow or bend</td>
<td>0.20</td>
<td>Entrance, slightly rounded</td>
<td>0.23</td>
</tr>
<tr>
<td>22.5 degree elbow or bend</td>
<td>0.15</td>
<td>Sluice gate as submerged port in wall</td>
<td>0.80</td>
</tr>
<tr>
<td>11.25 degree elbow or bend</td>
<td>0.13</td>
<td>Sluice gate as contraction in conduit</td>
<td>0.50</td>
</tr>
<tr>
<td>Non-standard elbow or bend</td>
<td>-----</td>
<td>Sudden contraction, $d = 1/4D$</td>
<td>0.42</td>
</tr>
<tr>
<td>Gate valve, fully open</td>
<td>0.19</td>
<td>Sudden contraction, $d = 1/2D$</td>
<td>0.33</td>
</tr>
<tr>
<td>Globe valve, fully open</td>
<td>4.0</td>
<td>Sudden contraction, $d = 3/4D$</td>
<td>0.19</td>
</tr>
<tr>
<td>Swing check valve</td>
<td>2.5</td>
<td>Tee, straight run to branch</td>
<td>1.8</td>
</tr>
<tr>
<td>Tilting disk check valve</td>
<td>0.34</td>
<td>Tee, branch to straight run</td>
<td>1.5</td>
</tr>
<tr>
<td>Butterfly valve</td>
<td>0.30</td>
<td>Flow past tee branch (run of tee)</td>
<td>0.60</td>
</tr>
<tr>
<td>Angle valve</td>
<td>2.5</td>
<td>Cross, line flow</td>
<td>0.50</td>
</tr>
<tr>
<td>Plug valve, fully open</td>
<td>0.75</td>
<td>Cross, branch flow</td>
<td>0.75</td>
</tr>
<tr>
<td>Diaphragm valve, fully open</td>
<td>2.3</td>
<td>Wye branch</td>
<td>1.0</td>
</tr>
<tr>
<td>Return bend, regular</td>
<td>0.38</td>
<td>Strainer</td>
<td>4.6</td>
</tr>
<tr>
<td>Return bend, long radius</td>
<td>0.25</td>
<td>90 degree box turn</td>
<td>1.0</td>
</tr>
<tr>
<td>Exit</td>
<td>1.0</td>
<td>Float controlled valve</td>
<td>2.0</td>
</tr>
</tbody>
</table>

This is a list of the most common pipe fittings, and obviously many others exist. The program offers you the ability to tailor the fitting database to your needs by allowing you to change $K$ values.
values, add fittings, and delete fittings. The fitting database is covered in more detail in the program options portion of Part II – Program features.

Other factors

In addition to the friction and fitting losses that occur from flow in a pipe, other factors can also contribute to this head loss. Two of these factors are the additional friction loss experienced by pipes carrying flows with solids and the affect of pipe age. Both of these factors are available when using the program. The program accounts for these other factors by determining a friction multiplier based on the input you provide. If a sludge is being conveyed by the pipe, you would provide the program with the type of sludge and an estimate of the percentage of solids in the flow, and the program will determine a friction multiplier based on these characteristics. The graph used by the program to determine these multipliers is shown below.

**Sludge friction multiplying factors**

![Sludge friction multipliers graph](image)

For the affects of pipe age, the inputs you would provide are the pipe diameter and age of the pipe. Using the table on the next page, the program then determines the appropriate friction multiplier based on these characteristics. It should be noted that the multiplication factors provided in this table and the program are highly theoretical and should be used with care. The process used by the program to account for solids in the pipe flow or pipe age is covered in more detail in the discussion of the pipe/conduit analysis portion of this manual.
### Pipe age multiplication factors

<table>
<thead>
<tr>
<th>Pipe age (years)</th>
<th>Small pipes (4”-10”) (102mm – 254 mm)</th>
<th>Large pipes (12” – 60”) (305 mm – 1524 mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>New</td>
<td>1.00</td>
<td>1.00</td>
</tr>
<tr>
<td>5</td>
<td>1.40</td>
<td>1.30</td>
</tr>
<tr>
<td>10</td>
<td>2.20</td>
<td>1.60</td>
</tr>
<tr>
<td>15</td>
<td>3.60</td>
<td>1.80</td>
</tr>
<tr>
<td>20</td>
<td>5.00</td>
<td>2.00</td>
</tr>
<tr>
<td>25</td>
<td>6.30</td>
<td>2.10</td>
</tr>
<tr>
<td>30</td>
<td>7.25</td>
<td>2.20</td>
</tr>
<tr>
<td>35</td>
<td>8.10</td>
<td>2.30</td>
</tr>
<tr>
<td>40</td>
<td>8.75</td>
<td>2.40</td>
</tr>
<tr>
<td>45</td>
<td>9.25</td>
<td>2.60</td>
</tr>
<tr>
<td>50</td>
<td>9.60</td>
<td>2.86</td>
</tr>
<tr>
<td>55</td>
<td>9.80</td>
<td>3.26</td>
</tr>
<tr>
<td>60</td>
<td>10.00</td>
<td>3.70</td>
</tr>
<tr>
<td>65</td>
<td>10.05</td>
<td>4.25</td>
</tr>
<tr>
<td>70</td>
<td>10.10</td>
<td>4.70</td>
</tr>
</tbody>
</table>

### Open Channel

Open channels are fairly common in treatment plants and are usually used to convey flows between unit processes. This program assumes that the channels that are being analyzed exhibit gradually varied flow behavior. Hydraulic abnormalities such as hydraulic jumps are not considered and are usually not recommended for open channel flow in treatment plants. Most open channels that are designed in treatment plants have minimal or zero slope, and usually flow in the subcritical state, with the exception of ramps and chutes. This program analyzes open channels in the following way:

1) The critical depth in the channel is calculated for the specified flow and channel characteristics.

2) The depth of flow in the channel is calculated by subtracting the invert of the channel from the previous water surface elevation.

3) If the depth of flow determined in 2) is greater than the critical depth, the flow in the channel is subcritical and this depth is used as the depth downstream in the channel. Otherwise, critical depth is used as the depth downstream in the channel.

4) Once the appropriate downstream depth is determined, the standard step method for calculating water surface profiles is employed by the program to determine the water surface elevation upstream in the channel.
The Standard Step Method

The Standard Step Method is a trial and error process that involves taking a known depth at one point in a channel (in this case, downstream) and guessing a second point upstream. The equation is:

\[ \Delta L = \frac{[(V_1^2 - V_2^2)/2g + y_{upstream} - y_{downstream}]}{(S - S_o)} \]

- \( \Delta L \) – length of reach, or length of channel
- \( V_1 \) – Velocity of flow upstream
- \( V_2 \) – Velocity of flow downstream
- \( g \) – gravity
- \( y \) – depth of flow, downstream and upstream
- \( S \) – average friction slope over the length of the channel = \((nQ/1.49AR^{2/3})^2\)
- \( S_o \) – bottom slope of the channel

The characteristics of the channel are known downstream. The value of \( y \) upstream is then guessed, and \( \Delta L \) is computed. This value is then compared to the actual length of the channel, and if they differ, a new guess is performed and the equation re-evaluated. This process continues until \( \Delta L \) and the channel length differ by an acceptable amount (less than 0.1%).

Flow profiles

Many types of flow conditions exist in open channels. Channel slopes may be mild, steep, horizontal, and even adverse. The program will analyze the flow conditions in the channel and determine the flow condition that will prevail based on the starting depth (\( y \)), critical depth (\( y_c \)), and normal depth (\( y_n \)) of flow in the channel.

Mild slope

A channel is termed mildly sloped when \( y_n > y_c \).

<table>
<thead>
<tr>
<th>Case I (M1 curve)</th>
<th>Case II (M2 Curve)</th>
<th>Case III (M3 Curve)</th>
</tr>
</thead>
<tbody>
<tr>
<td>– ( y &gt; y_n &gt; y_c )</td>
<td>– ( y_n &gt; y &gt; y_c )</td>
<td>– ( y_n &gt; y_c &gt; y )</td>
</tr>
<tr>
<td>– Depth decreases upstream</td>
<td>– Depth increases upstream</td>
<td>– Depth decreases upstream</td>
</tr>
</tbody>
</table>

Steep slope

A channel is termed steeply sloped when \( y_c > y_n \).
Case I (S1 curve)          Case II (S2 Curve)          Case III (S3 Curve)
- \( y > y_c > y_n \)          - \( y_c > y > y_n \)          - \( y_c > y_n > y \)
- Depth decreases upstream          - Depth increases upstream          - Depth decreases upstream

**Horizontal slope**

A channel is termed horizontal when \( S_o = 0 \). Normal depth in this case is infinite.

Case I (H2 curve)          Case II (H3 Curve)
- \( y_n > y > y_c \)          - \( y_n > y_c > y \)
- Depth increases upstream          - Depth decreases upstream

**Adverse slope**

A channel is termed adversely sloped when \( S_o < 0 \). Normal depth is also infinite in this case.

Case I (A2 curve)          Case II (A3 Curve)
- \( y_n > y > y_c \)          - \( y_n > y_c > y \)
- Depth increases upstream          - Depth decreases upstream

**How the program determines the starting or downstream depth**

Determining the downstream channel depth is the first critical step in analyzing an open channel. For gradually varied flow profiles, downstream controls typically govern and the channel depth upstream is determined by accounting for the friction losses the flow experiences along the channel. Two possible scenarios exist for the downstream depth in an open channel – the downstream depth may be controlled by the previous water surface elevation, or the downstream depth may be based on critical flow in the channel. The following diagrams illustrate these scenarios:

Scenerio I – Previous water surface elevation governing depth in channel. In this case, the flow over the weir sets the water depth for the flow coming from the open channel. Under these conditions, the water depth is greater than the depth of flow that would occur naturally in the channel, therefore the backwater created by the weir sets the depth of flow.
Innovative Hydraulics

Once the correct downstream channel depth is determined, the program uses the standard step method to account for friction losses along the length of the channel and determine the upstream channel depth.

**Channel bends**

In addition to the friction losses that will occur in open channels, losses may also occur when the flow goes around a channel bend. This loss is a function of the angle of the channel bend and the velocity of flow in the channel. The bend loss is determined in the same way that a fitting loss is determined for a pipe, by multiplying the velocity head by an empirical $K$ value. $K$ values are provided in the program for varying degrees of channel bends.

And the bend loss is then determined by:

$$H_L = K \frac{V_{down}^2}{2g}$$

It should be noted that the velocity used in the bend calculation is considered to be the downstream velocity in the channel. Therefore, whenever a channel bend is specified in the program, a bend should only be included with a channel section when that bend is on the downstream end of the channel.

This loss is included in the final upstream channel depth. The process of specifying channel bends is covered in more detail in the discussion of the open channel analysis portion of this manual.

**Scenario II**

- depth of flow governed by naturally occurring flow. In this case, water flows from an open channel and the falls into a drop box before exiting a pipe placed at the invert of the drop box. The depth of flow and subsequent channel velocity are based on the critical depth. The water surface elevation upstream the channel is then equal to the downstream elevation of flow in the channel (in this case, critical depth) plus channel losses.
**Channel transition loss**

Channel transitions can be either expansions or contractions. The applicable equations for the energy balance associated with a channel transition loss are as follows:

**Channel contraction**

\[ H_L = K_c \frac{(V_{down}^2 - V_{up}^2)}{2g} \quad \text{Head loss equation} \]

\[ V_{down}^2 / 2g + y_{down} + H_L = V_{up}^2 / 2g + y_{up} \quad \text{Energy balance equation} \]

These two equations are solved simultaneously to determine the head loss (\(H_L\)) that will balance the energy equation. The default \(K_c\) value for a channel contraction is 0.3.

**Channel expansion**

\[ H_L = K_e \frac{(V_{up}^2 - V_{down}^2)}{2g} \quad \text{Head loss equation} \]

\[ V_{down}^2 / 2g + y_{down} + H_L = V_{up}^2 / 2g + y_{up} \quad \text{Energy balance equation} \]

These two equations are solved simultaneously to determine the head loss (\(H_L\)) that will balance the energy equation. The default \(K_e\) value for a channel expansion is 0.5.

**Orifice/baffle/gate losses**

Head loss occurs when flow passes through an orifice or opening due to the constriction of flow. These losses are dependent on the shape of the opening, velocity of flow through the opening, and whether or not the opening is submerged, because unsubmerged openings with no downstream control will actually behave like weirs. There are three possible scenarios for flow behavior through a constricted opening:

**Scenario I – Unsubmerged, no downstream control**

In the case of the unsubmerged opening not affected by the downstream water level, the flow through the opening will behave like a contracted weir. Specialized equations have been
developed for analyzing the head loss through an unsubmerged opening, for both circular and rectangular openings:

**Circular opening equation:**

\[ H = D \times \left\{ \frac{Q}{\left[1.885 \times (2 \times G) \times (D/2)^{0.5} \times (2.5)^{0.546}\right]} \right\} \]

**Rectangular opening equation:**

\[ H = \left[\frac{Q}{(2/3 \times L \times (2 \times G)^{0.667})}\right]^{0.5} \]

- \( Q \) = flow through orifice, cfs
- \( H \) = resulting head, ft
- \( D \) = diameter or height of opening, ft
- \( G \) = gravity

### Scenario II – Unsubmerged with downstream control

A rare case can occur when an opening is unsubmerged upstream of the opening but the water level downstream of the opening is actually higher than the water level that would result from natural flow through the opening. This is demonstrated in the diagram below:

In this case, shape is not a factor because the opening is still unsubmerged and the loss is treated as a sharp entrance loss. This loss is a function of the velocity through the opening only, and is similar to a fitting analysis for a pipe. The applicable equation is:

\[ H_L = 0.5 \times \frac{V^2}{2g} \]

Where \( V \) is the velocity of flow through the opening.

### Scenario III – Completely submerged

\[ Q = \text{flow through opening, cfs} \]
\[ H = \text{resulting head, ft} \]
\[ L = \text{width of opening, ft} \]
Complete submergence is the most common hydraulic condition that occurs with the various opening types encountered in treatment plant applications. The equation for head loss for flow through a submerged orifice is:

\[ H_L = \frac{1}{C^2} \cdot \frac{V^2}{2g} \]

\( C \) = orifice loss coefficient
\( V \) = velocity through orifice
\( g \) = gravitational acceleration

The orifice loss coefficient \( C \) is typically taken as 0.6 for round orifices and 0.62 for rectangular or square orifices.

**Weirs**

Weirs are very common in treatment plant applications. Not only do they offer flow measuring capability but also serve as control points throughout the plant. Weirs are typically used at the outlet of treatment processes but also serve to provide equal flow distribution between multiple treatment units. Visual Hydraulics offers you the capability of analyzing six different types of weirs – V-notch, Rectangular, Sharp crested, Cipolletti, Broad crested, and Rectangular contracted weirs. The water surface elevation for flow upstream of a weir is determined by calculating the head over the weir. The theory of each is as follows:

**V-notch weirs**

The equation for the head over a V-notch weir is:

\[ H = \left[ \frac{Q}{C \cdot \tan\left(\frac{\theta}{2}\right)} \right]^{0.4} \]

\( H \) = head over weir
\( Q \) = flow per notch
\( \theta \) = angle of v-notch
\( C \) = flow coefficient – 2.5 for English, 1.38 for SI

**Rectangular, Sharp crested, and Cipolletti weirs**

The equation for these three weirs is basically the same, with the exception of the “C” coefficient, which is unique for each weir. The equation for the head over these weirs is:

\[ H = \left[ \frac{Q}{C \cdot L} \right]^{2/3} \]

\( H \) = head
\( Q \) = flow
\( L \) = weir length
\( C \) = weir coefficient

For a rectangular weir, \( C = 3 \) (1.656 for SI), for a sharp crested weir, \( C = 3.33 \) (1.838 for SI), and for a Cipolletti weir, \( C = 3.367 \) (1.859 for SI).
Broad crested weirs

Determining the head over a broad crested weir is a little more complicated than the equations provided for v notch and rectangular weirs, because it’s a trial and error process. The broad crested weir equation is:

\[
Q = C \times \left[ \frac{d_1}{(d_1 + h)} \right]^{1/2} \times L \times H^{1.5}
\]

H = head over weir
Q = flow
h = weir height
L = weir length
d_1 = h + H
C = flow coefficient, 3.475 for English, 1.918 for SI

The only unknown is H, but since d_1 is also dependent on H, a trial and error solution must be used to satisfy the equation. The program iterates through values of the head until the equation is satisfied.

Rectangular contracted weirs

Similar to the broad crested weir, the rectangular contracted weir requires a trial and error process to determine the head over the weir. The equation for the rectangular contracted weir is:

\[
Q = C \times (L - n \times \frac{H}{10}) \times H^{1.5}
\]

H = head over weir
Q = flow
L = length
n = number of contracted sides (1 or 2)
C = flow coefficient, 3.33 for English, 1.838 for SI

Again, the only unknown is H, and the program iterates through values of H until the equation is satisfied.

Rectangular notched weirs

Rectangular notched weirs are very similar to the v-notch weir, with the exception of the notch shape. Rectangular weirs are much less common than v-notch weirs. They do offer the advantage of being able to convey more flow through each notch than a v-notch weir because the area of a rectangular notch is greater than that of a v-notch. The equation for a rectangular notched weir is:

\[
Q = \frac{2}{3} \times C_d \times B \times (2 \times G)^{0.5} \times H^{1.5}
\]

H = head over weir
Q = flow per notch
C_d = discharge coefficient, default of 0.62
B = width of notch
G = gravity
Submerged weirs

In addition to weirs operating under normal conditions, weirs may operate under “submerged” conditions, which occur when the downstream water level reaches the invert or top of the weir. This may be a result of intentional operation, but usually weir submergence is the unwanted result of high flows or inadequate design.

Weirs behave differently hydraulically once a submerged condition is reached. The downstream water elevation now becomes the controlling factor in determining the head over the weir. The normal head equations no longer apply, and the upstream depth must be determined using either submerged weir head loss equations or graphs.

For rectangular and sharp crested weirs, the Villamonte and Mavis equations are utilized to determine the upstream flow depth over a submerged rectangular type or sharp crested weir. The Villamonte equation for rectangular type weirs is:

\[
\frac{Q}{Q_1} = \left[1 - \left(\frac{H_2}{H_1}\right)^n\right]^{0.385}
\]

- \(Q\) = actual flow over weir
- \(Q_1\) = free discharge flow at \(H_1\)
- \(H_1\) = weir upstream depth
- \(H_2\) = depth downstream of weir
- \(n\) = exponent in the free discharge equation (dependent on type of weir)
For triangular or V notch weirs, a graph plotting $Q/Q_1$ vs. $H_2/H_1$ is used to determine the head over the V notch weir.

**Bar racks / Screens**

Bar racks and screens are commonly found in the headworks of treatment plants and are used to remove large objects from the influent flow. Flow passes through a set of bars, where the flow is constricted and may result in head loss. This head loss is dependent on the size (width of the rack or screen), the spacing of the bars, the width of the bars, and the depth of flow through the rack or screen. Visual Hydraulics 4.2 was modified to include two different theories for analyzing the hydraulics of bar racks because both are widely used. Version 4.2 now offers the Kirschmer theory, which is in addition to the Bernoulli theory that has been used in previous versions. Both theories are now discussed.

**Bernoulli theory**

The Bernoulli head loss equation used by the program for a bar rack / screen is:

$$H_L = 0.0222 \times (V^2 - v^2)$$

Here, $V$ = velocity through the bars and $v$ = velocity in the approach channel.

To find the velocity through the bars, you have to know the free open area of the rack or screen. The program determines the free area of the rack or screen by requesting the overall width of the unit, the width of the bars, and the spacing of the bars. The program also allows for a blockage percentage, which restricts the open area of the rack or screen. Using this information, the free open area of the unit is calculated. Then the velocity through the bars can be determined by:

$$V = \frac{Q}{\text{open area}}$$

The velocity in the approach channel must also be determined. This is more difficult, because it’s a trial and error process. The depth upstream in the channel will be equal to the depth downstream plus the head loss of the unit. But since the head loss is dependent on the upstream depth, the head loss and upstream depth must be determined simultaneously so that both equations correctly work out. The program iterates through upstream channel depths until the upstream depth satisfies the conditions of the bar rack or screen and head loss. The velocity in the upstream channel can then be determined by:

$$v = \frac{Q}{\text{depth}_{\text{up}} \times \text{channel width}}$$
Once the two velocities are determined, the head loss can then be calculated. It should be noted that the bar spacing (s) must always be greater than the actual width of the bars (w). If you try and enter a bar spacing less than the width of the bars, the program will not allow you to continue. Also, if a rack invert is specified that is higher than the previous water surface elevation (i.e. if the flow exits the rack and drops to a lower level), the natural flow downstream of the rack will be used as the downstream depth, which will be the critical depth in the channel.

**Kirschmer theory**

The Kirschmer theory considers a few more factors than the Bernoulli theory. The Kirschmer head loss equation used by the program for a bar rack / screen is:

\[
H_L = k_s \left( \frac{W_b}{\text{Open area}} \right)^{1.33} \frac{V_{\text{approach}}^2}{2g \sin(\Theta)}
\]

- \(H_L\) = head loss
- \(k_s\) = shape coefficient
- \(W_b\) = bar width
- \(V_{\text{approach}}\) = approach velocity
- \(g\) = gravity
- \(\Theta\) = angle of bar rack

To find the velocity through the bars, you have to know the free open area of the rack or screen. The program determines the open area of the rack or screen by requesting the overall width of the unit, the width of the bars, and the spacing of the bars. The program also allows for a blockage percentage, which restricts the open area of the rack or screen. Using this information, the free open area of the unit is calculated.

The velocity in the approach channel must also be determined. This is more difficult, because it’s a trial and error process. The depth upstream in the channel will be equal to the depth downstream plus the head loss of the unit. But since the head loss is dependent on the upstream depth, the head loss and upstream depth must be determined simultaneously so that both equations correctly work out. The program iterates through upstream channel depths until the upstream depth satisfies the conditions of the bar rack or screen and head loss. The velocity in the upstream channel can then be determined by:

\[
v = \frac{Q}{\text{depth}_{\text{up}} \times \text{channel width}}
\]
Filters

Filters are operated by spreading the flow over a large surface area for treatment. The flow passes at a fairly slow velocity through the filter media. The head loss associated with a filter is dependent on a variety of factors, including filter surface area, type of filter particle, velocity of flow, and porosity of the filter media. Three main steps are used in analyzing the head loss for a filter. They are as follows:

First, the Reynolds number of the flow is determined:

\[ N_R = \frac{SF \times d \times V_s}{\nu} \]

\( SF = \) particle shape factor  
\( d = \) depth of filter media  
\( V_s = \) velocity of flow through media  
\( \nu = \) kinematic viscosity of water

Using the Reynolds number, the coefficient of drag is then calculated:

\[ C_D = \frac{24}{N_R} + \frac{3}{N_R^{1/2}} + 0.34 \]

\( C_D = \) coefficient of drag  
\( N_R = \) Reynolds number

Once the Reynolds number and coefficient of drag have been determined, the filter head loss can be calculated using the following equation (Kozeny equation):

\[ H_L = \frac{1.067}{SF} \times C_D \times \frac{1}{\Phi^4} \times \frac{L}{d} \times \frac{V_s^2}{g} \]

\( SF = \) shape factor  
\( C_D = \) coefficient of drag  
\( \Phi = \) porosity of media  
\( L = \) media depth  
\( d = \) particle diameter  
\( V_s = \) velocity through media  
\( g = \) gravity

Flumes

Flumes are other common flow measuring/splitting devices commonly found in treatment plants. Most flumes channel flow into a diverging “throat” section, which creates a type of hydraulic jump and establishes a set upstream depth of flow through the flume. This depth is dependent on the flow rate through the flume and characteristics of the flume. These characteristics depend on the type of flume. Visual Hydraulics contains four different types of flumes for analysis – Parshall flumes, rectangular flumes, trapezoidal flumes, and cutthroat flumes.

Parshall flumes

Parshall flumes are the most common types of flumes used in flow measuring applications. Their flow to head relationships are well documented and fairly simple to calculate. The level of flow through a flume is dependent only on the flow rate and the size of the throat. The Parshall flume equation is:
The empirical constants “e” and “m” are dependent on the throat width. For a throat width of less than 10 feet (3.05 m), \( m = 4 \times W \), where \( W \) is the throat width. For Parshall flumes with a throat width greater than 10 ft, \( m = 3.688 \times W + 2.5 \).

Values for “e” may be found in almost any hydraulic source. A table of e values for Parshall flumes is listed below:

<table>
<thead>
<tr>
<th>Flume throat width (W), ft</th>
<th>Flume “e” value</th>
</tr>
</thead>
<tbody>
<tr>
<td>0 – 0.25 ft (0 – 0.08 m)</td>
<td>1.55</td>
</tr>
<tr>
<td>0.25 – 0.5 ft (0.08 – 0.15 m)</td>
<td>1.547</td>
</tr>
<tr>
<td>0.5 – 0.75 ft (0.15 – 0.23 m)</td>
<td>1.58</td>
</tr>
<tr>
<td>0.75 – 1 ft (0.23 – 0.3 m)</td>
<td>1.53</td>
</tr>
<tr>
<td>1 – 1.5 ft (0.3 – 0.46 m)</td>
<td>1.522</td>
</tr>
<tr>
<td>1.5 – 2 ft (0.46 – 0.61 m)</td>
<td>1.538</td>
</tr>
<tr>
<td>2 – 3 ft (0.61 – 0.91 m)</td>
<td>1.55</td>
</tr>
<tr>
<td>3 – 4 ft (0.91 – 1.22 m)</td>
<td>1.566</td>
</tr>
<tr>
<td>4 – 5 ft (1.22 – 1.52 m)</td>
<td>1.578</td>
</tr>
<tr>
<td>5 – 6 ft (1.52 – 1.83 m)</td>
<td>1.587</td>
</tr>
<tr>
<td>6 – 8 ft (1.83 – 2.44 m)</td>
<td>1.595</td>
</tr>
<tr>
<td>8 – 10 ft (2.44 – 3.05 m)</td>
<td>1.607</td>
</tr>
<tr>
<td>&gt; 10 ft (3.05 m)</td>
<td>1.6</td>
</tr>
</tbody>
</table>

Rectangular flumes (adapted from ISO 4359 – 1983, 1999)

The process for determining the head through a rectangular flume is a fairly lengthy one, and requires a trial and error process. Three equations are involved, and they all require the value of the head flowing through the flume, so they must be solved simultaneously. The three equations are:

\[
C_d = \left[ 1 - \frac{0.006 \times L}{b} \right] \times \left[ 1 - \frac{0.003 \times L}{h} \right]^{1.5} \tag{I}
\]

\[
\left[ C_v^{2/3} - 1 \right]^{1/2} = 0.385 \times \left[ b \times h \times C_v \times \frac{C_d}{A} \right] \tag{II}
\]
The methodology involved with solving for the head flowing through a trapezoidal flume is exactly the same as that used for a rectangular flume. The equations are very similar, with the exception of accounting for the side slope of the flume. In addition, one other factor is included in the calculation, that being a shape coefficient, $C_s$. $C_s$ is a function of not only the shape of the flume but also the head flowing through the flume, so the calculation becomes quite a complex trial and error process of simultaneously solving multiple equations. For the trapezoidal flume, the three equations are as follows:

$$Q = b \times C_d \times C_v \times \left[ \frac{2 \times h}{3} \right]^{3/2} \times g^{1/2} \quad \text{(III)}$$

$C_d = \text{coefficient of discharge}$  
$L = \text{flume throat length}$  
$b = \text{flume throat width}$  
$h = \text{measured head}$  
$C_v = \text{approach velocity coefficient}$  
$A = \text{approach channel cross-sectional area}$  
$Q = \text{flow through flume}$  
$g = \text{gravity}$

Trapezoidal flumes (adapted from ISO 4359 – 1983, 1999)

$$C_d = \left[ 1 - \frac{0.006 \times k \times L}{b} \right] \times \left[ 1 - \frac{0.003 \times L}{h} \right]^{1.5} \quad \text{(I)}$$

$$\left[ C_v^{2/3} - 1 \right] = 0.385 \times \left[ b \times h \times C_v \times \frac{C_s}{A} \right] \quad \text{(II)}$$

$$Q = b \times C_d \times C_s \times C_v \times \left[ \frac{2 \times h}{3} \right]^{3/2} \times g^{1/2} \quad \text{(III)}$$

$C_d = \text{coefficient of discharge}$  
$C_s = \text{shape coefficient}$  
$k = (1 + m^2)^{1/2} - m$  
$m = \text{side slope of flume throat}$  
$L = \text{flume throat length}$  
$b = \text{flume throat width}$  
$h = \text{measured head}$  
$C_v = \text{approach velocity coefficient}$
Cutthroat flumes

The final type of flume offered with this program is the cutthroat flume. The cutthroat flume is very simple in terms of construction and also very simple to analyze hydraulically, at least when compared to the rectangular and trapezoidal flumes. The method is very straightforward and involves only the following equation:

\[
\text{Head} = \left[ \frac{Q}{K \cdot W^{1.025}} \right]^{1/n} \\
Q = \text{flow} \\
W = \text{throat width} \\
n, K = \text{empirically determined constants that are dependent on the flume length, } L.
\]

Tank launders and troughs are special open channel applications. Almost all launders are rectangular in shape and have either zero or a very minimal slope from their high point to the launder outlet. The hydraulic “worst case scenario” point for a launder is established as the point in the launder that is furthest away from the launder outlet (half the launder perimeter in the case of tanks with perimeter weirs). The following open channel flow equation is used for determining the upstream depth in a rectangular launder:

\[
\text{Depth upstream} = \left[ \frac{2h_c^3}{h_o} + (h_o - 1/3iL)^2 - 2/3iL \right]^{1/2} \\
h_c = \text{Critical depth} \\
h_o = \text{Downstream depth} \\
L = \text{Launder length} \\
i = \text{Launder slope}
\]

The depth downstream in the launder at the launder outlet is very important in determining the final or upstream depth in the launder. If the outlet is unsubmerged and flow free falls from the launder into the outlet box or pipe, then the critical depth is used as the downstream starting depth in the launder. If the launder outlet is submerged and the submerged depth is greater than the critical depth in the launder, the submerged depth is used as the starting depth.

Junctions

Contraction/Enlargement

Contractions and enlargements are specialized pipe losses that occur when the pipe transitions from one size to another. The change in velocity associated with this transition results in a head loss, and that head loss is dependent on the change in pipe size as well as the angle of transition, or how fast the pipe changes from one size to the other. The following equations are used to analyze contractions and enlargements:

\[
\text{Head} = \frac{Q}{K \cdot W^{1.025}}^{1/n} \\
Q = \text{flow} \\
W = \text{throat width} \\
n, K = \text{empirically determined constants that are dependent on the flume length, } L.
\]
Contraction / enlargement equations:

\[ K = 0.8 \times \sin(0.01745 \times \theta/2) \times \left[ 1 - \frac{D_{\text{small}}^2}{D_{\text{large}}^2} \right] \quad \text{For transition angles } \leq 45 \text{ degrees} \]

\[ K = 0.5 \times \sin(0.01745 \times \theta/2)^{1/2} \times \left[ 1 - \frac{D_{\text{small}}^2}{D_{\text{large}}^2} \right] \quad \text{For transition angles } > 45 \text{ degrees} \]

Once the appropriate K value is determined, this K value is multiplied by the velocity head through the appropriate pipe section to determine the head loss:

\[ \text{Loss} = K \times \left( \frac{V_{\text{small}}^2}{2g} \right) \quad \text{Contraction} \]

\[ \text{Loss} = K \times \left( \frac{V_{\text{large}}^2}{2g} \right) \quad \text{Expansion} \]

**Basic Tee Connection**

Tee connections exist where two pipes either combine or separate into one single pipe. A tee connection is considered a “basic” tee connection by the software if the flow through the tee is one directional, in other words, the flow does not combine or divide at the tee. If the tee has combining or dividing flows, a combining/dividing tee element should be used.

Although a tee can be added as a fitting when analyzing a pipe section, sometimes a detailed analysis of a tee is required. The head loss that is associated with a tee connection is mostly dependent on how the flow passes through the tee itself. The head loss through a tee is calculated by multiplying a loss coefficient “K” by the velocity head through the tee. The “K” coefficient is based on the flow path through the tee and size of the pipes which comprise the tee. The applicable equation is:

\[ \text{Loss} = K \times \left( \frac{V_{\text{tee}}^2}{2g} \right) \]

“K” loss coefficient values are summarized in the following table:

<table>
<thead>
<tr>
<th>Tee Type</th>
<th>Tee “K” Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Branch to Line</td>
<td>1.5</td>
</tr>
<tr>
<td>Line to Branch</td>
<td>1.8</td>
</tr>
<tr>
<td>Run of Tee</td>
<td>0.6</td>
</tr>
</tbody>
</table>
In addition to the K values shown in the table above, additional loss coefficients are applied if the tee results in a contraction of expansion of the pipe size where the tee junction exists.

**Combing/Dividing Tees**

Combining and dividing tees are unique tee junctions where the flow is split into multiple directions or the flow is combined from more than one direction into one pipe. The hydraulics with determining the head loss associated with these types of tees is much more complex than assigning a K value to the tee based on its configuration (see basic tee above). Although the general head loss equation is the same as that for a basic tee, the determination of the K value is based on multiple factors, including the type of connection, angle of the tee, ratio of flow in to flow out of the tee, and area ratio of the tee diameters along the flow path. Extensive research has been performed by Miller, and that research has been resulted in numerous graphs that represent the varying scenarios that could occur at the tee connections. These graphs are used by Visual Hydraulics to determine the K value for the tee that is being analyzed. A sample of one of these graphs presented by Miller is as follows:
Manhole

Manholes are junction points at which one or more pipes enters and leaves an open structure. The head loss due to a manhole is based on the configuration of the pipes relative to each other (if only one pipe enters and one pipe leaves the manhole) and how many pipes enter the manhole. The overall head loss equation for a manhole is:

\[
\text{Loss} = K \times \left( \frac{V_{\text{out}}^2}{2g} \right)
\]

\(V_{\text{out}}\) is the velocity of flow in the pipe exiting the manhole, and “K” is the loss coefficient based on the pipe configuration and number of pipes entering the manhole. If only one pipe enters the manhole and one pipe leaves the manhole, the angle between the pipes determines the “K” value. The following table summarizes how the “K” value is determined for various manhole configurations.

<table>
<thead>
<tr>
<th>Manhole configuration</th>
<th>Manhole “K” Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>One pipe entering manhole, pipe angle &gt; 150°</td>
<td>0.30</td>
</tr>
<tr>
<td>One pipe entering manhole, 135° &lt; pipe angle &lt;= 150°</td>
<td>0.45</td>
</tr>
<tr>
<td>One pipe entering manhole, 120° &lt; pipe angle &lt;= 135°</td>
<td>0.75</td>
</tr>
<tr>
<td>One pipe entering manhole, 90° &lt; pipe angle &lt;= 120°</td>
<td>0.85</td>
</tr>
<tr>
<td>One pipe entering manhole, pipe angle &lt;= 90°</td>
<td>1.0</td>
</tr>
<tr>
<td>Two pipes entering manhole, one pipe leaving</td>
<td>0.80</td>
</tr>
<tr>
<td>Three pipes entering manhole, one pipe leaving</td>
<td>1.0</td>
</tr>
</tbody>
</table>

Sample single pipe entering, single pipe leaving angle configurations are shown below:
Venturi Meter

Venturi meters are placed directly in pipe lines and are used to measure flow. A pressure drop is created by constricting the flow area through the throat section. This pressure drop can be correlated to a flow value. Because a constriction in pipe area is required to create the pressure drop, head losses are usually common with Venturi meters. A standard equation is used to evaluate this head loss, and is provided below:

\[
\text{Loss} = \frac{\left(\frac{Q}{0.95 \times A}\right)^2 \times (1 - B^4)}{2g}
\]

Q = Flow through meter  
A = Throat area  
B = Throat to inlet diameter ratio  
g = gravity

Flow Control Valve

Flow control valves are in-line pipe valves that often provide a significant reduction in area to reduce the flow through a particular section or to provide a desired water elevation upstream of the valve. The reduction in area through a flow control valve creates a drop in pressure, which is then easily correlated to a head loss across the valve. The head loss calculation for a flow control valve is very similar to that of a standard pipe fitting. Instead of a K value for a pipe, flow control valves have flow coefficients, or “Cv” values. These values are manufacturer specific and are based on the pressure drop that will occur across a particular valve at a given flow value. The head loss equations for a flow control valve are as follows:

English system

\[
\text{Loss} = 2.31 \times \left(\frac{Q \times 448.43}{C_v}\right)^2
\]

\[
Q = \text{Flow through meter}  
C_v = \text{Valve flow coefficient}  
448.43 = \text{conversion to gpm}  
2.31 = \text{conversion from psi to feet}
\]

SI system

\[
\text{Loss} = 0.102 \times \left(\frac{Q \times 1000}{C_v}\right)^2
\]

\[
Q = \text{Flow through meter}  
C_v = \text{Valve flow coefficient}  
1000 = \text{conversion to l/s}  
0.102 = \text{conversion from kPa to meters}
\]
Special Loss / Pump

The special loss portion of the program provides the user with the ability to analyze any type of hydraulic feature if flow and head loss data are available. This is especially useful for equipment or specialized applications that do not have standard equations or methods for a hydraulic analysis. The user provides the program with flow and head loss data (such as data obtained from a manufacturer on a piece of equipment), and the program will then perform a regression analysis on that data to determine a best fit loss equation. Four regression analyses are available: linear, 2\textsuperscript{nd} degree, 3\textsuperscript{rd} degree, and 4\textsuperscript{th} degree regressions. Once an analysis is performed, a correlation factor (R squared value) will be provided to the designer to see how well the equation fits the data. This allows the designer to choose the best equation for the data. The regression analysis procedure is discussed below:

**Linear regression equation:**

\[ y = mx + b, \text{ where} \]
\[ y = \text{head loss point} \]
\[ x = \text{flow point} \]
\[ m = \text{slope of line} \]
\[ b = \text{y-intercept} \]

**Polynomial regression equation:**

\[ y = C_0 + C_1x_1 + C_2x_2 + \ldots + C_4x_4, \text{ where} \]
\[ y = \text{head loss point} \]
\[ x = \text{flow point} \]
\[ C_0, C_1, C_2, C_3, C_4 = \text{calculated constants} \]

If a polynomial regression is the chosen method (which it will most likely be), the C factors must be determined by the program. This is done by performing the least squares method for solving multiple equations simultaneously. The process can be quite lengthy and complex, and any linear algebra or matrices textbook may be consulted for a detailed discussion on the theory behind regression analyses.

**Manifold/Diffuser**

The analysis of a multiple discharge application such as a manifold or diffuser is one of the more complex problems encountered when performing hydraulic calculations. The analysis is a trial and error process that involves balancing the overall head of the system with the available head and flow at each discharge location. With some manifolds/diffusers having up to 1000 discharge points (ports), a computer generated model provides the only practical solution to such complex systems.

Consider the following diffuser diagram:
This diagram shows a diffuser that discharges flow through ports located on riser pipes. Many manifolds or diffusers simply discharge flow through ports located right in the main manifold wall. Manifolds with risers are slightly more complex because of the additional head loss the flow will experience passing through the riser pipes.

The flow through each port is a direct function of the pressure head at that port. The equation for port discharge is:

\[
Q = 0.04377 \times D^2 \times (H / Kr)^{0.5}
\]

Where:

- \(Q\) = port flow, cfs
- \(D\) = port diameter, in
- \(H\) = head at port, ft
- \(Kr\) = riser loss coefficient

The riser loss coefficient \(Kr\) is 1 if there is no riser (only an exit loss). If there is a riser, this loss coefficient is based on the friction and fitting losses associated with the riser. The equation is:

\[
Kr = K_{exit} + \left[(K_{entrance} + f \times (L_{riser} / D_{riser})) \times (D_{port} / D_{riser})\right]^4
\]

Where:

- \(K_{entrance}\) = Riser entrance coefficient (typically taken as 0.5)
- \(f\) = Darcy friction factor
- \(L_{riser}\) = Riser length, ft
- \(D_{riser}\) = Riser diameter, ft or in
- \(D_{port}\) = Port diameter, ft or in
- \(K_{exit}\) = Port exit coefficient, typically taken as 1

In addition to the head loss experienced by the port (and possibly the riser), there will also be a head loss in the main manifold pipe itself. And this head loss will actually increase the closer it gets to its point of origin (point 1 in the diagram), because most of the flow is being conveyed by the manifold at the origin end, while only a small portion of the flow is being conveyed by the manifold near its termination (point 2 in the diagram). This is why manifolds/diffusers typically have tapering sections, because the flow in the manifold near the end of termination is much less than the flow at its point of origin.
Visual Hydraulics performs a manifold/diffuser analysis using the following trial and error procedure:

1) Starting at the terminal end of the manifold (point 2), the head required at the last port is guessed by the program.
2) Using the port flow equation, the flow through the end port is determined for this head value.
3) The head loss through the port and riser for this flow is then determined.
4) The head loss through the manifold section between the last port and next port upstream is then determined by calculating the friction loss in this pipe at the calculated flow value.
5) The head loss through the port, riser, and manifold section is then added to the head value for the previous port downstream. This is the head for the next port upstream.
6) Based on this new head, the flow through the next port is calculated.
7) This process continues all the way upstream, with the program constantly tabulating the required head all the way upstream as well as the total flow through each port.
8) Once the first port upstream is reached, the program compares the sum of all port flows to the actual design flow. If they differ, the program starts over and adjusts the guess for the head required at the last port downstream (step 1 above). The program then runs through steps 2 – 7 again. This continues until the sum of all the port flows equals the actual design flow specified for the system.
9) The program then reports the overall head loss experienced by the manifold and also reports in tabular format the flow, head required, port discharge velocity, and head loss for every single port along the entire manifold.

For additional information on multi-port diffusers or manifolds, a number of hydraulic sources may be contacted.