KYPIPE

PIPE2010

QuickStart Guide for

Pipe2010: KYPipe
Pipe2010: Surge
Pipe2010: GoFlow
Pipe2010: Gas
Pipe2010: Steam
Pipe2010: SWMM

- Installation Instructions
- System Requirements
- Demonstration Examples

LICENSE AGREEMENT

This is a legal agreement between the user and KYPipe LLC. By accepting, using or installing any portion of this software the user agrees to be bound by the terms of this agreement.

SOFTWARE LICENSE

GRANT OF LICENSE: For each license purchased from KYPipe, LLC, or one of its authorized distributors, KYPipe LLC grants to the user the right to use one copy of the software program(s) on a single terminal connected to a single computer (i.e., with a single CPU). The user may not network non-network versions of the software or otherwise use single user versions on more than one computer terminal at the same time. Network versions are only to be used with one physical site (buildings at the same mailing address) and are not to be used in a WAN environment. WAN licenses may be used only for the number of sites licensed. The number of network licenses purchased for a network version is the maximum number of users permitted to run the software concurrently. If granted for an evaluation period by KYPipe LLC, user agrees not to use the software beyond the evaluation period specified by KYPipe LLC. The user agrees not to utilize features, options, or number of pipes beyond the license the user has purchased.

COPYRIGHT: The software and the documentation are owned by KYPipe LLC and are protected by United States copyright law and international treaty provisions. The user must treat the software like any other copyrighted material except that the user may make one copy of the software solely for backup or archival purposes or may transfer the software to a single hard disk and keep the original disk(s) sole for backup or archival purposes. The user may not copy the written materials accompanying the software without explicit written permission from KYPipe, LLC.

TRANSFER BY USER: The user may not rent, lease, assign or permit others to use the software but may transfer the software and accompanying materials on a permanent basis provided the user retains no copies and the recipient agrees to the terms of this agreement. As a condition to permit the recipient use the software under this License Agreement, when such a transfer is made, KYPipe, LLC must be notified, in writing, of the transfer, including the identity and address of the recipient, and the agreement of the recipient to the terms of this License Agreement.

OTHER RESTRICTIONS: The user may not modify the software. The user may not reverse engineer, decompile, disassemble, or otherwise attempt to determine the source code of the software. The user shall protect the software from unauthorized use, and shall protect the software and the intellectual property from infringement by others. The user shall notify KYPipe, LLC, in writing, immediately upon receiving any information that would indicate that the software is being used in an unauthorized manner or the intellectual property is being infringed.

DISCLAIMER

Although every reasonable effort has been made to ensure that the results obtained are correct, neither the author(s) nor KYPipe, LLC assumes any responsibility for any results or any use made of the results obtained with these programs. THE SOFTWARE IS SOLD AS IS WITH NO IMPLIED WARRANTIES, INCLUDING WARRANTIES OF MERCHANTABILITY AND FITNESS FOR ANY PARTICULAR PURPOSE. NO EXPRESS WARRANTY EXISTS EXCEPT AS SPECIFICALLY SET FORTH IN WRITING BY KYPIPE, LLC. IN NO EVENT, REGARDLESS OF THE NATURE OF ANY CLAIM, WILL KYPIPE, LLC, ITS MEMBERS OR AFFILIATES, BE LIABLE FOR ANY LOSS FOR PERSONAL INJURY, BUSINESS INTERRUPTION, LOST PROFITS, OR INCIDENTAL OR CONSEQUENTIAL DAMAGES, AND ITS LIABILITY, IF ANY, SHALL BE LIMITED TO THE PURCHASE PRICE OF THE SOFTWARE.

USE OF THE DOCUMENTATION AND PROGRAM

The documentation is provided for the use of individuals or companies which purchase it from KYPipe, LLC. Except for back-up copies, the program disks or documentation may not be copied, marketed, or distributed without explicit written permission from KYPipe, LLC. For users who wish to use the programs on networks or multiple computers or different locations, network copies and multiple copy discounts may be obtained. Please contact KYPipe, LLC for details.

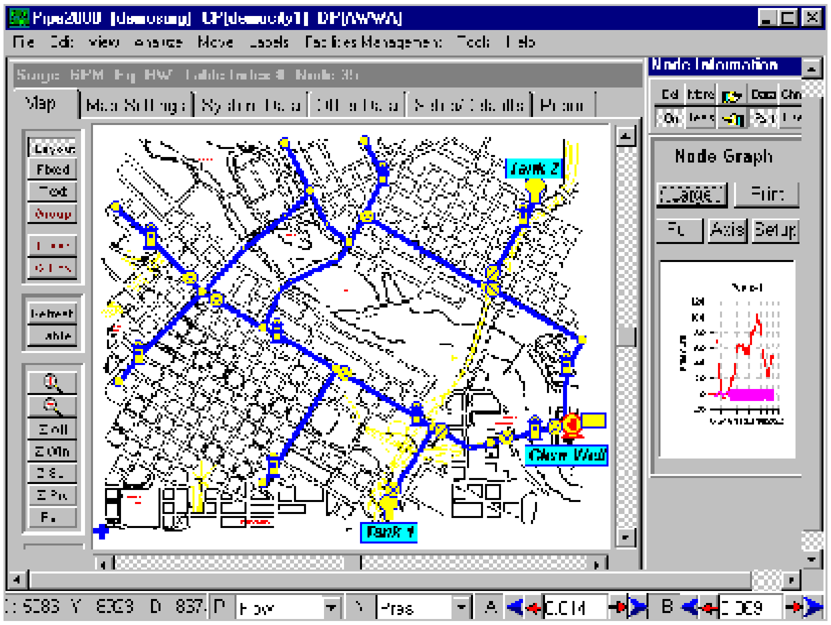
GOVERNING LAW AND VENUE FOR ENFORCEMENT AND DISPUTES

This Agreement will be governed by and construed in accordance with the substantive laws of the Commonwealth of Kentucky, and, to the extent federal law applies, to the laws of the United States. The state and federal courts of Fayette County, Kentucky, shall have exclusive jurisdiction over any claim brought against KYPipe, LLC, and the user agrees to submit to the jurisdiction of the state and federal courts of Fayette County, Kentucky, in the event any claim is brought against the user, and user waives all defenses to jurisdiction and inconvenience of forum.

Pipe2010 QuickStart Guide – Contents

What is A QuickStart Guide?

The purpose of this guide is to get you started using Pipe2010 as quickly as possible. Over 40 audio/video tutorials are provided to illustrate all aspects of Pipe2010.The Pipe2010 Online Help system provides extensive information regarding the program features and operations. Several Demonstration Examples and a QuickStart example are discussed in this guide, the Online Help and AVI tutorials. Pipe2010 is designed to provide very comprehensive capabilities while being intuitive and easy to use. When possible Pipe2010 employs standard Windows features to provide a familiar operating environment. Instructions for installing Pipe2010 and getting started follow.



WELCOME TO PIPE2010

Your PIPE2010 package consists of 3 main items:

1- The Program CD
2- CD #2
3- The USB Hardware Lock

When the Program CD is inserted into the drive a menu screen will pop up that allows you to install Pipe2010 or run the audio/video tutorial. The program CD contains the current image of the PIPE2010 software. The most current image is always posted on our website at www.kypipe.com in the Download area.

The Pipe2010 tutorial is accessed from the main Program CD but you may be prompted to insert CD #2 occasionally.

The third item, the hardware lock, is by far the MOST important. This key IS your software license. If you lose the key, you may need to purchase another copy of the software.

You may install the software on as many computers as you like (work, laptop, and home). Only the computer that has the key inserted will run the software. If you need to do a presentation for a client, for example, you can simply unplug the key from your main computer and insert it into your laptop.

We have worked very hard over the years to enhance and improve upon the software. Many of the features and changes are the result of feedback of our users. Please send us EMAIL or give us a call if something does not seem to be working correctly or if you have feature suggestions!

Do not plug in your USB key until software installation is complete.

INSTALLATION INSTRUCTIONS (Demo Version)

1. Insert the PIPE2010 Program CD into your CD drive.

2. If the PIPE2010 install program does not start automatically, use Windows Explorer to browse to your CD drive and run the SETUP.exe application. This will install the PIPE2008 software.

That's it! You should now be all set to run PIPE2010

If something does NOT work correctly, please call Jana Faith at 812-843-4145 or Bill Gilbert at 859-257-4941 and we will help you get up and running.

INSTALLATION INSTRUCTIONS (Stand Alone Version)

First, DO NOT insert the hardware key into your computer. It is important to run the software install (which will setup drivers) and THEN install the hardware key.

1. Insert the PIPE2010 Program CD into your CD drive.

2. If the PIPE2010 install program does not start automatically, use Windows Explorer to browse to your CD drive and run the SETUP.exe application. This will install the PIPE2010 software.

3. After the installation is complete insert the hardware key into your computer. If you are using a USB key and have multiple USB ports, it does not matter which port you use. If you are using a parallel port key and have other parallel port keys attached to your machine, please make sure to put ours first in the stack (closest to the computer).

4. Plug in the USB hardware key. Your operating system should recognize it. You may see some drivers being installed or configured or a New Hardware Wizard may appear. Follow through this wizard.

That's it! You should now be all set to run PIPE2010

If something does NOT work correctly, then please call Jana Faith at 812-843-4145 or Bill Gilbert at 859-257-4941 and we will help you get up and running.

NETWORK VERSION INSTALLATION INSTRUCTIONS

The network license for PIPE2010 does NOT apply to WAN use. This means that you are not permitted to use your network key to run PIPE2010 at multiple buildings, companies, or mailing addresses.

*** CLIENT MACHINE

First, DO NOT insert the hardware key into your computer. It is important to run the software install (which will setup drivers) and THEN plug in the hardware key.

1. Insert the PIPE2010 Program CD into your CD drive.

2. If the PIPE2010 install program does not start automatically, use Windows Explorer to browse to your CD drive and run the SETUP application. This will install the PIPE2010 software.

3. Open the control panel and run the Wibu-Key applet

4. Click the tab marked 'Network'.

5. In the subsystems box make sure that only Wk-Local and Wk-Lan are selected.

6. In the box below the server search list (lower right corner of this screen) Type in the IP address of the machine that is actually holding the physical WibuKey. Click the ADD button to put the server in the list. (The IP address of the server machine should be made static. To get an IP Address use a Command prompt (under Accessories) and type ipconfig).

SERVER INSTALLATION – Client machine also acting as server

If a machine where PIPE2010 is installed as a client is also going to act as the key server for the network then please do the following:

1. After installing the program files as described above, insert the hardware key into your computer. If you are using a USB key and have multiple USB ports, it does not matter which port you use. If you are using a parallel port key and have other parallel port keys attached to your machine, please make sure to put ours first in the stack (closest to the computer).

2. When the key is plugged in, your operating system should recognize it and you may see some drivers being installed or configured.

3. Click START | PROGRAMS | Wibu-KEY and select Network Server

4. A small icon should appear on your screen, click the right mouse button on it once (which will bring up a menu) and select INSTALL AS SERVICE.

5. If you do not want this icon to always be on your screen then click the right mouse button on it once (which will bring up a menu) and select SET INTO TASKBAR.

*** SERVER Installation – Stand Alone Server

Important: use "Client Machine as Server" installation instructions if these don't work. Any machine on the network may be used as a server.

To configure a new machine to act as the PIPE2010 license server, please do the following;

1. Insert your PIPE2010 Program Disk and when the menu pops up click the 'Network Install' button. This will start the installation of drivers for the USB 'wibu' key. (Note: if using Vista 64 bit, browse to the c:\Program Files\KYPIPE\Pipe2010\WIBU folder, then run WKRuntime64.exe instead.)

2. After the installation is complete insert the hardware key into your computer. If you have multiple USB ports, it does not matter which port you use. If you are using a parallel port key and have other parallel port keys attached to your machine, please make sure to put ours first in the stack (closest to the computer).

3. When the key is plugged in, your operating system should recognize it and you may see some drivers being installed or configured.

4. Click START | PROGRAMS | Wibu-KEY and select Network Server

5. A small icon should appear on your screen, click the right mouse button on it once (which will bring up a menu) and select INSTALL AS SERVICE.

6. If you do not want this icon to always be on your screen then click the right mouse button on it once (which will bring up a menu) and select SET INTO TASKBAR.

*** IMPORTANT - The machine that is acting as the server MUST have the hardware key inserted any time the machine is restarted for the network license to work properly.

If something does NOT work correctly, then please call Jana Faith at 812-843-4145 or Bill Gilbert at 859-257-4941 and we will help you get up and running.

Computer Requirements

The minimum system requirements for PIPE2010 are:
Pentium Processor
VGA monitor
512 Meg RAM
130 Meg free hard disk space
Windows NT4 (parallel port key only), Windows Vista, XP, 2000 or later

This software will NOT run on earlier Window versions.

For outstanding performance the minimum RECOMMENDED system for PIPE2010 is:
(1) Pentium IV
(2) VGA monitor (19" or larger)
(3) 1024 Meg RAM
(4) 130 Meg free hard disk space
(5) Windows Vista, XP, 2000 or later

Extra RAM will increase performance particularly on large systems with multiple background images. All of our in-house systems that we use for PIPE2008 consulting work have at least 512 Meg RAM.

Updating Pipe2010

You can update Pipe2010 to the current version by visiting the download area of our website (http://www.kypipe.com). Look for the Pipe2010 current version. Because we update the image frequently (to fix “bugs” which are reported or to add features) you should visit our www site regularly to update your software.

Windows Vista

Pipe2010 will work correctly under Windows Vista 32 bit and 64 bit editions. Because of extensive security measures of Vista there are several steps that may need to be taken to allow proper operation of PIPE2010. The result of these steps is to allow the user to write to the PIPE2010 directory.

- Make sure you are logged in as an administrator
- For Vista 64 bit, browse to the c:\Program Files\KYPIPE\Pipe2010\WIBU folder, then run WKRuntime64.exe instead.)
- Click START | COMPUTER to bring up the file explorer application
- Click on the C drive to browse the local files
- Right click on the Program_Files folder and select Properties
- Click the Read_Only checkbox until it appears empty, then click APPLY
- When prompted, choose ”Apply changes to this folder, subfolders and files“ and click OK
- If boxes appear about administrator permission, click Continue
- If errors occur, select Ignore All

Pipe2010 should now run correctly for you.

Display Settings

We suggest that you run Pipe2010 in as high a resolution as your monitor can display such that it can be comfortably read. We recommend the following settings:

Monitor Size	Setting
14" or 15"	1024 x 768
17"	1280 x 1024
21"	1600 x 1200

We recommend that you use a setting for your display of more than 256 colors. If you use a 256 color mode and load background pictures you may experience color distortion of the display. You can verify / set the resolution and number of colors within Windows by doing the following:

From any open space on your windows desktop (the background, not on top of a window) click the right mouse button and select PROPERTIES on the menu that appears. This will bring up the display properties dialog box. Click on the tab marked SETTINGS. There is a screen area slider, which you can move to the desired resolution. There is a drag down list marked COLORS. Verify that this setting is something higher than 256 colors (8 bit). If this is not the case please select a mode with more colors (greater than 8 bits per pixel) then click on OK.

Audio/Video Tutorials (AVI's) and Online Help

There are over 40 audio/video tutorials which address all aspects of using Pipe2010. The tutorials are contained on both of the CDs that came with Pipe2010. In addition the Help File is accessed from the Pipe2010 main menu under **HELP**. This provides extensive information on modeling and the Pipe2010 environment. You can access topics from the **Contents** or specific items from the **Index.**

Getting Started

Insert your **Pipe2010 Program CD** to begin your tutorial. When the menu pops up select **Start Tutorial**. If it does not pop up then use Windows Explorer to browse to your CD drive and run the **Pipe2010Tutorial** application. When the Tutorial Subject menu pops up select KYPipe, Surge, or the subject appropriate to you.

The first video of the tutorial entitled **Pipe2010 Overview** gives you a very brief look at the process of laying out piping systems and providing data. After viewing this introduction some users may wish to use the **Select Video** button to jump to the following videos **Information Windows, Elements, Building a Model 1, Building a Model 2, Laying Out a System, and Graphical Data Entry**. Most users should watch the first ten videos in order.

After completing the first ten videos find the **QuickStart Tutorial Example** on page 25 of this manual. Refer to this while you watch the next four videos **QuickStart Example 1** through **QuickStart Example 4**. Once you have run these 14 sessions you should review some of the Help File information as noted in the **Contents** section.

KYPipe and Surge users should study the Demonstration Examples provided on page 31 of this manual and included in the **Demo** subdirectory. KYPipe2010 users should change the tutorial subject to **KYPipe2010 Advanced** and then view the accompanying videos **Hydraulic Model Example**, and **Extended Period Simulation Example**. Surge users should watch the tutorial videos **Surge Analysis Example** and **Adding Surge Protection to a Model**.

Contacting Us - Software Development and Support Team

The following individuals are involved in the Software development including the 2010 Series models, KYPipe, Surge, Gas, Steam, GoFlow, and SWMM and directly support the software.

Don J. Wood	Ph.D, Civil Engineering	(859) 492-6097	Don@kypipe.com
Srinivasa Lingireddy	Ph.D, Civil Engineering	(859) 519-2494	Srini@kypipe.com
Bill Gilbert	BS, Civil Engineering	(859) 257-4941	Bill@kypipe.com
Doug Wood	MS, Computer Engineering	(859) 263-0401	Doug@kypipe.com
Jana Faith	BS, Civil Engineering	(812) 843-4145	Jana@kypipe.com

KYPIPE LLC
3229 Brighton Place Lexington, KY 40509-2314
Phone: (859) 263-2234 FAX: (859) 263-0401

Visit our WWW site at www.kypipe.com

Visit our www site regularly to check for updates which can be downloaded.
Demo versions of Pipe2010 also can be downloaded as can the AVI tutorials.

Continuous research and development over the past 20 years has resulted in the most advanced hydraulic modeling capability available. Some noteworthy results of this very high level of development include:

1. Development of the full equation set approach for network hydraulics utilizing the Newton-Raphson linearized approach for solving the network equations. This is the most robust algorithm available for solving the complex and sometimes ill conditioned hydraulic relationships.
2. Development of enhanced network equations which allow direct calculation of design, operation and calibration parameters.
3. Development of a powerful general approach for transient flow in simple or complex pipe networks.
4. Application of genetic algorithms to optimize network hydraulic and water quality calibrations and operations.
5. Development of an effective time averaging water quality model.
6. The hydraulic model incorporates devices such as automatically adjusting regulating valves (pressure and flow), variable speed pumps, flow meters, switching capabilities to control valves and pumps, etc.
7. Extension of our steady state network models to compressible flow (gas and steam).

Our focus for many years has been hydraulic modeling. The University of Kentucky team of academics and of engineers is, perhaps, the world's leading group of experts in this area. They have been most successful in quickly developing their advanced hydraulic modeling technology for use by practicing engineers

Over the last several years, high level computer engineers and engineering software developers have joined the team. They have developed advanced graphical interfaces to enhance the KYPipe and SURGE modeling environment. These engineers have worked very closely with our hydraulic modeling team of experts to develop a wide range of extremely advanced capabilities to simplify and speed up the essential modeling tasks and to provide additional useful capabilities. By incorporating suggestions and concepts provided to us by our large and knowledgeable user base, we have developed a truly outstanding environment for all aspects of hydraulic and water quality modeling. The ergonomics and capabilities of KYPIPE4 and SURGE are, by far, the best available anywhere at any cost.

The new Windows advanced graphical environment, PIPE2010, has been adapted to other models, analyzing gas (Pipe2010: Gas), steam (Pipe2010: Steam), fire sprinkler systems (Pipe2010: GoFlow) and transient flow (Pipe2010: Surge).

User support of our software is provided directly by our team of experts. This situation assures that the level of support is very high. Providing this level of support fosters a very close relationship between the development team and the users. Engineers who have used previous versions of our software will recognize that many of the new features and capabilities are ones they had wished for or suggested to us.

Pipe2010 Help File Contents

There are a series of operations necessary to develop a pipe system model, enter data, and analyze the piping system. Sections marked with an * are included in this manual.

ABOUT Pipe2010 ONLINE HELP
How to access and review important information

OVERVIEW
What is Pipe2010?

A FIRST LOOK AT Pipe2010- AUDIO/VIDEO TUTORIALS *
A number of short audio/video clips demonstrate how to use Pipe2010

NETWORK ELEMENTS *
What are the parts of a piping system model?

LAYING OUT A PIPING SYSTEM *
How do I make a piping system model with Pipe2010?

BACKGROUND IMAGES
Several types of backgrounds can be used to speed up and enhance your model development and use

QUICKSTART EXAMPLE *
Walk me through developing a pipe network model with Pipe2010

PIPE2010 OPERATIONS
Information on Menus, Tabs, and Buttons

INFORMATION WINDOWS
Boxes for entering pipe and node data and displaying information

DATA REQUIREMENTS
Access information on data requirements and units

VALVES AND HYDRANTS
Access information on valves and hydrants

DEMAND ALLOCATIONS / METERS
Pipe2010 has some very advanced features for handling demands

SOME SPECIAL FEATURES
Save lots of time and do some neat stuff

NETWORK ANALYSIS
How do I perform the analysis on my system?

PIPE2010 PRESENTATIONS
How can I review my data and see the results of my analysis?

DATA FILES / SCENARIO MANAGEMENT
Pipe2010 data files include Demand and Change Pattern selections to facilitate Scenario Management.

DATA TABLES

Pipe2010 data can be accessed and manipulated in Excel compatible spreadsheets

SETS AND GROUPS
How can I use this powerful capability?

ADVANCED CAPABILITIES
Show me some of the other advanced capabilities

EXTENDED PERIOD SIMULATIONS (EPS)
See examples of several extended period simulations

CALIBRATION
Optimized Calibration with Pipe2010 and other calibration approaches.

WATER QUALITY
Use EPANET with Pipe2010 to answer water quality questions.

FACILITIES MANAGEMENT
Pipe2010 has many useful Facilities Management features

REFERENCE MANUAL
Detailed information about modeling and the KYPipe analysis engine

UTILITIES
What extra programs come with Pipe2010?

RURAL WATER SYSTEMS
A specially designed network analysis approach to reflect the demand patterns of a Rural Water System.

SURGE
The Pipe2010 transient flow model

GAS
The Pipe2010 compressible flow model

STEAM
The Pipe2010 saturated steam flow model

GOFLOW
The Pipe2010 fire sprinkler system model

DEMO FILES *
Demonstration Files for KYPipe2010 and Surge

Pipe2010 Tutorial (Audio/Video)

Pipe2010 is designed to provide very rapid and intuitive model development. An extensive Help File is provided and topics can be access through the Contents page or a comprehensive Index. Multimedia presentations (audio/video clips) of operations can be found on the **Pipe2010 Tutorial**. Insert your **Pipe2010 Program CD** to begin the tutorial. When the menu pops up select **Start Tutorial**. If it does not pop up then use Windows Explorer to browse to your CD drive and run the **Pipe2010Tutorial** application.

When the **Tutorial Subject** menu pops up select KYPipe, Surge, GoFlow, Gas, Steam or SWMM. You may switch to a different subject by clicking the **Tutorial Subject** button at any time. For KYPipe users there is also an advanced tutorial entitled **KYPipe2010 Advanced**.

The tutorial menu has Play and Pause buttons and a Trackbar that allows you to back up or advance the presentations at any time. The list below groups the available audio/video clips according to their purpose. It is recommended that you review the **Pipe2010 GUI** and **Model Development** clips prior to using Pipe2010. The additional clips may be reviewed as you utilize the capabilities which they address.

Overview

• **Pipe2010 Overview**	Overview of Pipe2010 and the video tutorials

Pipe2010 Graphical User Interface

• **Buttons**	Using the buttons to the left of the map
• **Top Tabs**	Using the tabs at the top of the map
• **Main Menu**	Use of the main menu (top)
• **Information Windows**	Use of the Information windows (right side)

Model Development

• **Elements**	Model elements - pipes and nodes
• **Building a Model 1**	Operations for building a model
• **Building a Model 2**	Building a model (continued)
• **Laying Out a System**	Laying out a pipe model system
• **Graphical Data Entry**	Graphical data entry
• **QuickStart Example 1-4**	QuickStart example (4 clips)
• **Changes**	Additional data provides multiple simulations

Background Maps and Images

• **Grids and Vector Backgrounds**	Using grids or vector file backgrounds
• **Scaling and Raster Backgrounds**	Scaling and using raster file backgrounds
• **Bitmap Images**	Importing and displaying bitmap images

Extended Period Simulation (Pipe2010 Only)

• **Extended Period Simulation**	Overview of Extended Period Simulations
• **EPS Tanks**	Example setup - system data and tanks
• **EPS Control Switches**	EPS control switches

Customizing Data Entry and Precision (not for Gas or Steam)

• **Pipe Types**	Customizing pipe type data
• **Fittings**	Customizing fittings data
• **Precision and Sliders**	Customizing unit precision and sliders

Presenting Data and Results

• **Contours**	Generating and labeling contours

- **Map Labeling** Using labels for data and results

Other Applications (may not be applicable to some system types)

- **Group Editing** Group selection and editing
- **Meters** Using meters for demand allocation
- **Material and Power Costs** Material and power cost calculations
- **Pipeline and Head Profiles** Generating pipeline and head profiles

KYPipe2010 Advanced (change **Tutorial Subject** to access)

- **Constraint Parameter Calculations** Parameter calculations
- **Pump and System Curves** Producing pump curves and system curves
- **Rural Analysis Using PDD Curves** Hydraulic Analysis using peak demand requirements (PDD curves)
- **Hydraulic Model Example** Example hydraulic model
- **Extended Period Simulation Example** Extended period simulation of a model
- **Calibration - Parts 1-6** Calibrating a System

Surge

- **Intro to Surge Analysis 1** Introduction to Surge Analysis - Part 1
- **Intro to Surge Analysis 2** Introduction to Surge Analysis - Part 2
- **Surge Geometric Requirements** Surge model differences - geometric requirements
- **Surge Components** Surge model differences - components
- **Converting KYPipe to Surge** Converting steady state (KYPipe2010) to Surge model - example
- **Surge Control Devices** Adding Surge Control Devices - example
- **Features for Surge components** Features for Surge components
- **Surge Control Components** Surge control components
- **Variable Input Data (Changes)** Variable input data (changes)
- **Surge Analysis Example** Surge analysis of a hydraulic model
- **Adding Surge Protection To A Model** Adding surge protection to a model

GoFlow

- **GoFlow Elements**
- **Sprinkler System Layout**
- **QuickStart Example - System Layout**
- **QuickStart Example - Data Entry**
- **QuickStart Example - Analysis And Results**

Gas

- **Gas Overview**

Steam

- **Steam Overview**

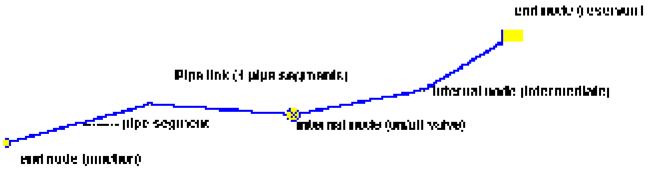
Network Elements

Insert the Pipe2010 program CD to see the **Elements** tutorial.

Pipe distribution systems are constructed using the following two elements:

1. Pipe Links
2. Nodes

All development is carried out using only these two elements. Important definitions are illustrated in the picture below and the following descriptions:



Pipe Links
Pipe links are uniform sections of pipes (same basic properties) following any route. A pipe link may be comprised of one or more pipe segments. A pipe segment is a straight run of pipe with no internal nodes.

Nodes
Nodes are located at the ends of pipe segments and include all distribution system devices that are modeled.

- **Internal nodes** - are located between two pipe segments.
- **End nodes** - are located at the ends of all pipe links and can connect other pipe links, represent a dead end or a connection to a supply.
- **Text nodes** - can be located anywhere on your map and are used for adding information to your map.

*End nodes count as nodes used for your model while internal and text nodes do not.

Internal Nodes

Internal nodes are located between two pipe segments of identical properties. The intermediate node is usually a point where a directional change occurs while the other internal nodes (valve, hydrant, in-line meter, metered connections, and check valves) are devices or model elements located in a pipe link. From the modeling viewpoint, internal nodes are essentially passive devices (they do not directly affect the calculation), although they do provide added modeling capabilities. Internal node types can be interchanged. They also can be changed to an end node at anytime. However, end nodes can be changed to internal nodes only if there are exactly two connecting pipe links with identical pipe properties.

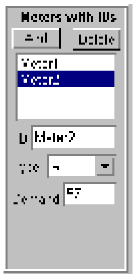
1. Intermediate Node -
No device at this location - usually represents a change of alignment. To delete all intermediate nodes see Deleting Intermediate Nodes

.2. Valve-
Indicates location of cut-off valves. The minor loss for this inactive valve is not automatically included in the network analysis or the report. To account for a minor loss due to a valve, the user may enter the loss as a pipe fitting or using the active valve element.

3. Hydrant -
Indicates location of fire hydrants.

4. In-line Meter -
Indicates presence of an in-line meter for pipe link. It is used for EPS reports of total flows.

5. Metered Connections -
Indicates location of metered connections. Meter ID may be specified to interface with meter records.

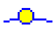


6. Check Valve (Directional) -
Indicates device in pipe link that prevents flow reversal. The correct direction (flow allowed in direction indicated) must be selected in the pipe link.

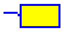
7. Customized Device -
Two additional internal nodes can be used to represent any desired devices (such as air release valves).

End Nodes

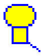
End nodes are located at each end of all pipe links. End nodes represent both passive connections, such as junctions and connections to supplies, and active elements, such as pumps. One or more pipe links can connect to a common end node. For non-directional end nodes (junctions, reservoirs, tanks, variable pressure supplies, and sprinklers), pipe links can be connected in any manner. For directional end nodes (pumps, loss elements, and regulators), an inlet and outlet connection point are shown and pipe links must be connected to the appropriate side of the element so that the direction indicated is correct. Pumps and loss elements (but not regulators) can connect (on one side) directly to a reservoir. This condition is modeled when no pipe link connections are made to one side of the element. This side is then modeled as a constant head reservoir and the reservoir head must be specified with the input data. All end node types can be interchanged. If a change is made from a non-directional to a directional node, the pipe links will connect arbitrarily. It is necessary to make sure that the direction is correct and the pipe links are properly connected. However, an end node can be changed to an internal node only if there are exactly two pipe links and the basic pipe link properties are the same (except length and minor coefficients). If the properties are not the same, the change to an internal node will be possible only if an option to utilize common properties is accepted.



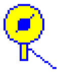
1. Junction -
A connection of one (dead end junction) or more pipe links.



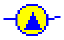
2. Reservoir -
A connection of one or more pipe links to a constant level reservoir. During a simulation, the reservoir level remains constant unless data is provided to change its value.



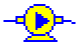
3. Tank -
A connection of one or more pipe links to a variable level storage node. For EPS (extended period simulations) level changes are calculated.



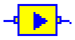
4. Variable Pressure Supply -
A connection of one or more pipe links to a supply where the supply pressure depends on the supply flow and is determined by using pressure flow data provided.



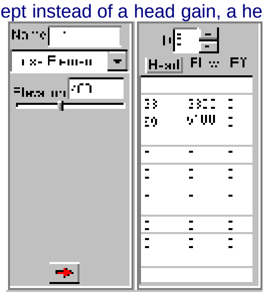
5. Sprinkler (Pressure Dependent Outflow) -
A connection of one or more pipe links to a point where flow is discharged based on the pressure in the distribution system. The characteristics of a connecting pipe may be defined (length, diameter, elevation change). This device can model a leak or a pressure sensitive demand.

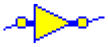


6. Pumps (Directional) -
A connection of one or more pipe links to a pump. The pump direction must be set and pipe links connected to the appropriate sides.

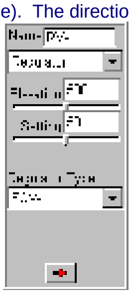


7. Loss Element (Directional) -
An element identical to a pump except instead of a head gain, a head loss occurs.



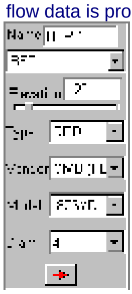


8. Regulator (Directional) -
A connection of one or more pipes is required to each side of the device that maintains downstream pressure (pressure regulating valve), upstream pressure (pressure sustaining valve) or flow (flow control valve). The direction must be set and the pipe links connected to the appropriate side.



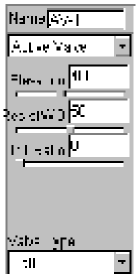
9. LE Library (Back Flow Preventer) -

A special loss element for which head flow data is provided based on manufacturer, model, and size. See Loss Elements Library.



10. Active Valve -

A valve which may be opened and closed and for which the minor loss is included in the network analysis. See **Active Valve**.



Laying Out a Pipe System

Pipe2010 is designed to provide a very simple, intuitive interface for your pipe system development. All development is done in ‘Layout’ mode. When you are not developing or modifying your system, you should select a different mode (usually ‘Fixed’) so you will not inadvertently modify the layout. The layout and subsequent modifications are done with the following operations. Insert the Pipe2010 program CD to see the **Building a Model 1 and 2** tutorials.

1. Select a Node or Pipe Link - Point to node or pipe and LC (left click).

2. Add Pipe Segment and Node - Select starting node (existing) and point mouse to ending node location (new) and RC (right click).

3. Add Pipe Segment - Select starting node (existing) and point mouse to ending node location (existing) and RC (right click).

4. Move Node - Point mouse to node and hold down left mouse button - drag to the new location.

5. Add Internal Node - Point mouse to desired location in pipe link and LC (left click). Click ‘Insrt’ (Pipe Information Window) and select internal node type from pop-up list.

6. Change Node Type - Select node and click on current node type selector (below name) and select from node type pop-up list (Node Information Window).

7. Delete Internal Node - Select internal node and click on ‘Del’ (delete) in the Node Information Window. **** Note that this will combine the two connecting pipe segments into one segment eliminating the internal node. To delete all intermediate nodes in a system, see Deleting Intermediate Nodes.

8. Delete End Node - Select end node and click on ‘Del’ (delete) in Node Information Window. **** Note that this will also delete ALL the pipe links connecting the node. If you do not wish to do this, change the node type to a junction.

9. Delete Pipe Link - Select pipe and click on ‘Del’ (delete) in Pipe Information Window.

10. Change Node Direction - For directional end nodes (pumps, loss elements and regulators), select node and click on ➔ in the Node Information Window. The symbol in the node icon will change direction. You can do this to correct your model or to improve the appearance of the directional node.

11. Change Pipe Direction - The positive pipe direction (for referencing flows, etc.) is from Node 1 to Node 2. To reverse this, click on Reverse (Pipe Information Window). It is necessary to insure pipes with check valves are in the correct direction.

12. Change Pipe Link Connection - For pipe link connections to directional nodes, click the ⟳ symbol adjacent to the directional node (Pipe Information Window). You will see the link connection change to the other side of the directional node.

As you layout your system (using operation 2), intermediate nodes are automatically inserted at all changes in alignment. These are automatially changed to junction nodes if only one or more than two pipes are connected or if the properties of the two connecting segments differ.

Quick Start Example

Step 1 - Initial Preparation
Step 2 - System Layout
Step 3 - Analyze System and Review Results
Step 4 - Some Additional Simulations

Insert the Pipe2010 CD to see the **QuickStart Example 1 - 4** tutorials.

This will guide you through the complete layout development, data entry and hydraulic analysis of a simple pipe network. We recommend that you run Pipe2010 in as high a resolution as your monitor can display such that it can be comfortably read. We recommend the following Windows 95/NT settings:

Monitor Size	Display Setting
14" or 15"	1024 x 768
17"	1280 x 1024
21"	1600 x 1280

Step 1 - Initial Preparation

Initial steps include file selection, background preparation and system data selections.

a. file selection

You can access an existing data file or, as for this demonstration, create a new one. Click on File (top menu box) and select New.

b. system data selection

The New File setup screen appears. As a minimum you need to specify the flow units and head loss equation to use.

Click on the Units drop down list and select GPM. The default head loss equation showing (Hazen-Williams) and the defaults showing for data features are all acceptable.

Click on MAP to return to the Pipe2010 map.

c. background preparation

You can import a drawing map, utilize grid lines or choose not to use a background. For this demonstration we will turn on a grid and use it to guide our layout letting Pipe2010 calculate pipe lengths.

Click on Map Settings / Grids - The default grid settings of 1000 (major) and 100 (minor) are good for our demonstration so we will use them.

Click on Major Grid and Minor Grid check boxes. This will display background grid lines.

Click on Map to return to the Pipe2010 map.

Step 2 - System Layout

The map area which appears on the screen will show a region approximately 1000 x 1000 feet with the 100 foot grid lines displayed. This area will be appropriate for the demonstration. A larger or smaller region can be displayed by clicking on the zoom in (+) or a zoom out (-) button on the left side.

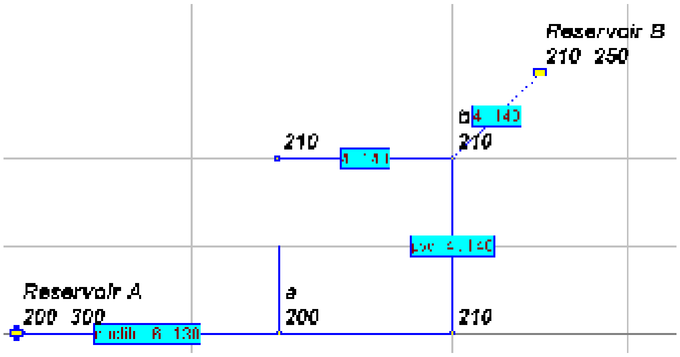


Figure 1 Example pipe system

The system we wish to lay out is shown above drawn on a 100 foot grid system. It is a loop fed by Reservoir A (HGL = 300) and discharges into Reservoir B (HGL = 250). The node elevations are noted. This is followed by the reservoir HGL's at the two reservoirs. The pipe material, diameter and roughness is noted for each pipe in a box. Points (a) and (b) are shown for reference in the discussion below. The development of the pipe system model is accomplished in three steps.

a. layout pipes and nodes

The entire piping system can be laid out using the mouse and a right click (RC) to add pipes and nodes and a left click (LC) to select a node. The following operations will produce the system layout:

1) RC on gridline intersection to make first node
2) move mouse 300 feet (3 blocks) to right and RC (a)
3) move mouse 200 feet up and RC
4) move mouse 200 feet right and RC
5) move mouse 200 feet down and RC (a)
6) move mouse 200 feet left (back to existing node) and RC
7) select node at (b) and move 100 feet up and 100 feet to left and RC

Now all the pipes and nodes are laid out. Note all nodes are either junction or intermediate nodes and Pipe2010 has assigned pipe and node names.

b. change node types

Select any nodes which are different than shown and change to the correct node type. To do this select the node and click on drop down node list (Node Information Window - below Name) and select desired type from list.

1) Select node at Reservoir A (LC) and change node type to Reservoir
2) Select node at Reservoir B and change node type to Reservoir

The system should now look as shown below.

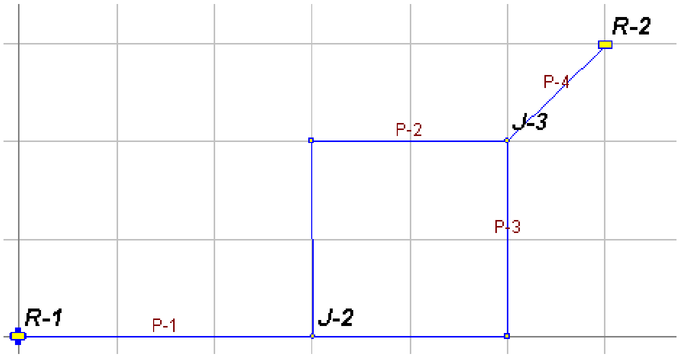


Figure 2 Completed pipe system layout

c. provide data

Select each pipe and end node and provide data

1. Select each pipe and click Pipe Type (Pipe Information Window) and select choice from drop down list. Select ductile: 250:6 for pipe from Reservoir A and pvc: 150:4 for the rest. Note that default roughness values are provided. Provide appropriate Fittings Data (elbow for pipes with 90o bend, for example
2. Select each Reservoir and provide values shown for Grade (HGL) and Elevation
3. Select each junction and intermediate node, enter Elevation

d. save data file

Provide a name and save your data file

Click on File (Main Menu) and Save As and provide a file name in the popup menu. Such as QSI (for Quick Start example 1).

Step 3 - Analyze System and Review Results

a. check data and run analysis

1. Click Analyze (Main menu) and select Error Check. If errors are flagged correct these. If the message "No Errors" appears proceed
2. Click Analyze (Main Menu) and select Analyze System and click Analyze on the popup menu to accept the defaults (Analyze with KYPipe, Use Current Year)

b. review results

The results can be reviewed on the schematic using Results Labels or by looking at the tabulated output.

1. Click on Output (Main tabs) and scroll through the tabulated summary of data and results. If the Page Up and Page Down keys don't work click anywhere on the screen to activate them. Click on Maps (Main tabs) to go back to your system graphical display.
2. Click on Labels (Main menu) and select Pipe Results | Pipe Result A and Node Results | Node Result A to show the results depicted in the Results Selection bar on the bottom right of the screen. You can click on the P selector to change the pipe results (to Flow, for example) and the N selector (Pressure for example) to change to the node results. A helpful selection is Loss (head loss) for pipes and HGL for nodes because it provides a very useful view of the system operation. Printouts based on these selections are shown (Figure 3 and 4).

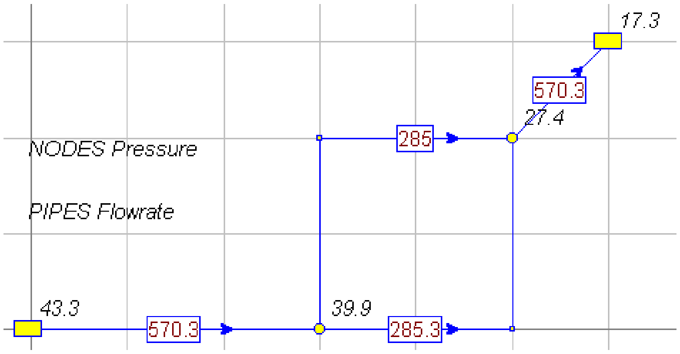


Figure 3 Case 1 - Pressure and Flow

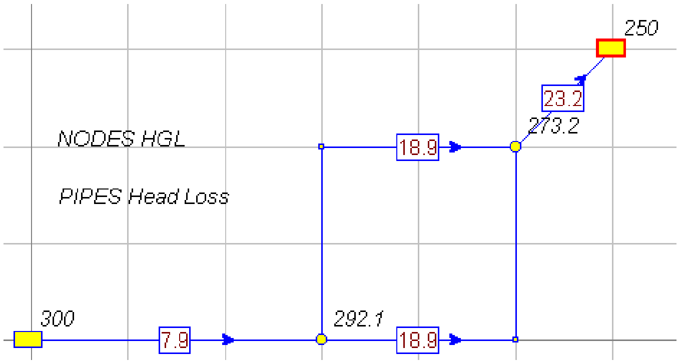


Figure 4 Case 1 - Loss and HGL

Step 4 - Some Additional Simulations

It is very easy to modify data and run a new simulation. Several are described:

a. **age based roughness**

Rerun the analysis but this time click on Use Current Year to remove that requirement and enter the year 2023 (25 years from now). The analysis now shows a significant change in pipe roughness due to aging and a substantial decrease in the capacity to transport water from Reservoir A to Reservoir B. A printout showing flows and pressures illustrate this (Figure 5).

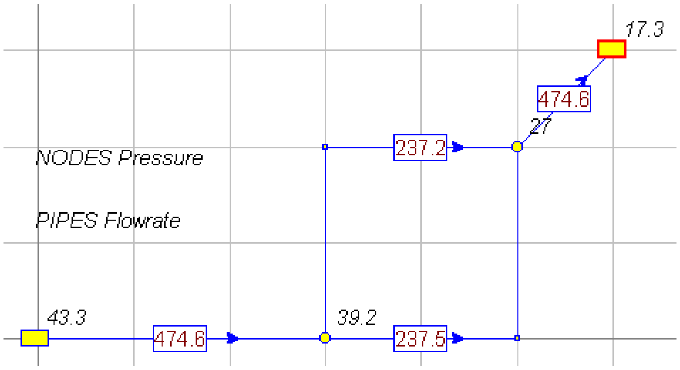


Figure 5 Case 2 - 25 years (2023)

b. **add a pump**

We want to add a 40 HP (useful horsepower) pump in the line leading form Reservoir A about 100 feet from the reservoir. To do this Click on (LC) the pipe at the desired location and click on Insrt (Pipe Information Window - button) select Intermediate Node . Now select the intermediate node (LC on node) and change node type to Pump. Select the pump and select Constant Pwr (power) for Pump Type and input 40 (HP) for the Power and 210 (ft.) for the elevation (Node Information Window). Now analyze the system and note the effect of this pump which provides around 136 feet of head and nearly doubles the flow. A printout showing flows and pressures is shown (Figure 6).

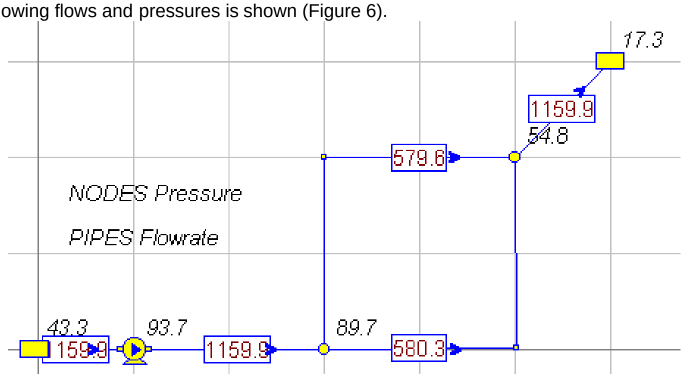


Figure 6 Case 3 - Added Pump

KYPipe and Surge Demonstration Examples

KYPipe - Regular Simulations
KYPipe - Extended Period Simulations
KYPipe - Other Capabilities
Surge
Surge Protection
Optimized Calibration
Water Quality (EPANET)

A simple pipe system representing the main pipes of a small municipal distribution system is shown in Figure 1. This system is used to demonstrate the use of KYPipe2010 for regular and extended period simulations and Surge for surge analysis. A number of modeling features may be demonstrated using the data files provided in the *DEMO* subdirectory. We suggest that you run the demonstration files with a screen resolution of 1024 by 768 or higher if possible.

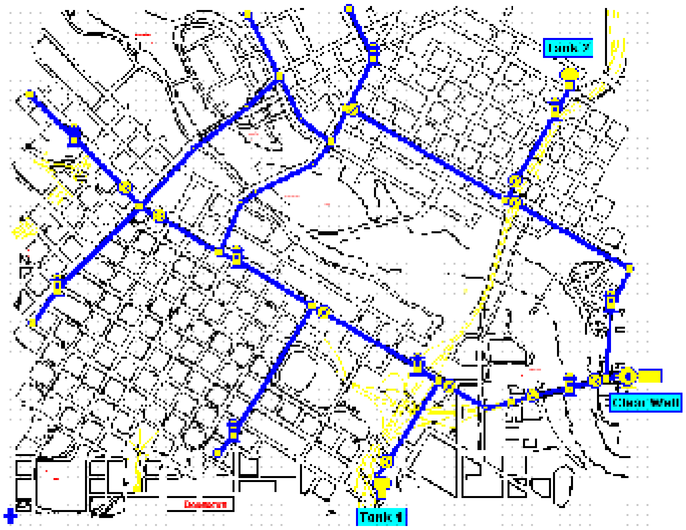


Figure 1 Pipe System Layout for Demonstration Examples

KYPipe - Regular Simulations

Click on ***File*** (Main Menu) and ***Open*** and select the file ***Demoreg*** (in the demo subdirectory) using the browser. You should get the pipe system and map shown in Figure 1. The ***Demoreg*** file sets up the baseline analysis (Case 0) and two additional scenarios (Cases 1 and 2).
Case 0 - The pump is running with normal demands
Case 1 - The pump is off and the tanks supply the system
Case 2 - The pump is off and a fire demand of 650 g.p.m. is placed Junction J-13

You can see normal demand patterns specified by clicking on ***Labels*** (Main Menu) and selecting ***Junction Demands*** and ***Type***.

To run the analysis, click on ***Analysis*** (Main Menu), select ***Analyze System*** and make sure that ***KYPipe*** is selected before you click ***Analyze***. Once the analysis is complete, you can click on ***Report*** to see the tabulated report. There are many advantages to viewing the results graphically using several KYPipe2010 features.

1) Results Labels*:* Click on ***Labels***, ***Pipe Results***, and ***Pipe Result A*** and repeat for ***Node Results*** and ***Node Results A***. This will display flow rates (in g.p.m.) for each pipe and the pressure (in p.s.i.) for each node for the baseline data (Case 0). Figure 2 shows this display You can use the ***Results Selector*** bar at the bottom of the screen to select different parameters for nodes (drop down list for ***N*** (node) box) and pipes (drop down list for ***P*** (pipe) box) and look at Cases 1 and 2 using the arrows in the ***A*** case/time selection box.

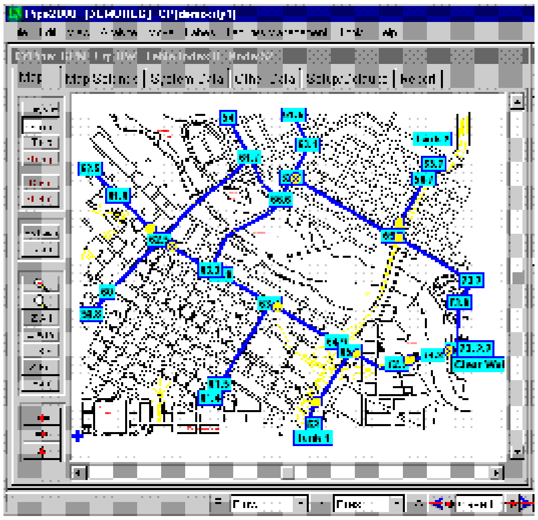


Figure 2 Results Labels, Case 0

2) Contours: Contours are very effective means for showing results. Show pressure contours for Case 2 to illustrate this feature. Make sure ***Pres*** (pressure) is selected in the ***N*** box and Case 2 in the ***A*** box (***Results Selector*** bar). Click on ***Map Settings*** and ***Emphasis/Contours*** and select ***Pressure*** (parameter). The contour values should be set at 20, 30, 40, 50, and 60. Click on ***Show Contour***, then ***Map*** (top tabs) to return to the map. The pressure contours should be displayed (if not, click the ***Refresh*** button). Figure 3 shows this display.

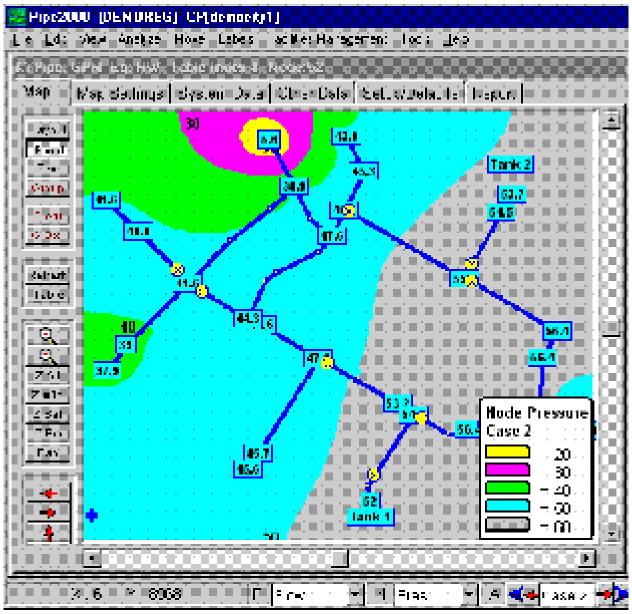
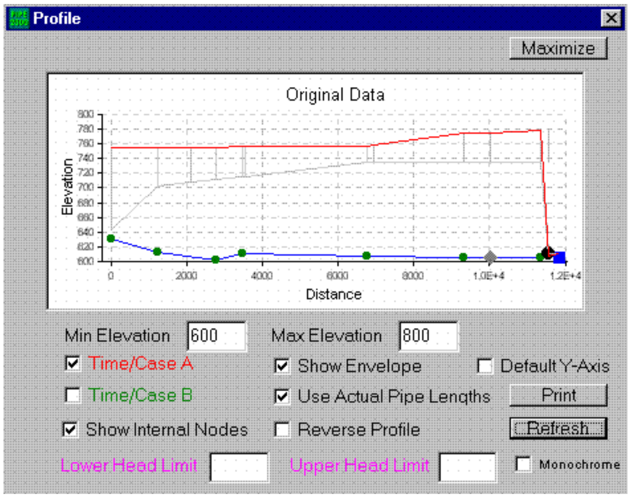


Figure 3 Contours, Case 2

3) Profiles: A profile plot showing the pipeline profile and head profiles provides a very useful tool. To display this, click the ***Group*** button and select a starting node (J-13), upper center- dead end, and an ending node (the clear well reservoir). Next, click ***Analyze*** (Main Menu) and ***Profile*** and ***Create Profile from Leftmost Selected Node***. The profile shown in Figure 4 will be displayed. The envelope of heads for the three cases will be displayed if ***Show Envelope*** is selected. Select ***Time/Case A*** and ***Time/Case B*** and the profiles for the cases selected in A and B (***Results Selector*** bar) will be displayed. You can provide an Upper (or Lower) Head Limit to see if your heads exceed the limits.

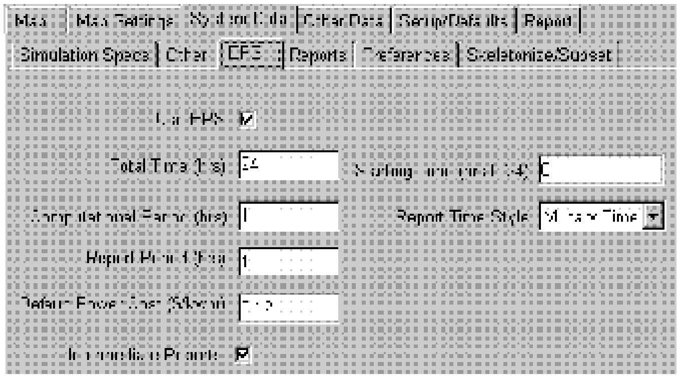


Min Elevation 600
Max Elevation 800
☑ Time/Case A
☑ Show Envelope
☐ Default Y-Axis
☐ Time/Case B
☑ Use Actual Pipe Lengths
☑ Show Internal Nodes
☐ Reverse Profile
Lower Head Limit
Upper Head Limit
☐ Monochrome

Figure 4 Profile, Case 0

KYPipe - Extended Period Simulations

Click on *File* and *Open* and select the file *Demoeps*. This file sets up a 24 hour EPS at hour increments. Select *System Data* and *EPS* to see this setup. A 24-hour demand pattern based on data provided by AWWA is used. This pattern can be viewed by clicking on *Setup/Default* and *Demand Pattern.* Click *Map* to return to the map. For this simulation, the pump is controlled by the level of water in Tank 1 (T-1). When the water level drops below 737 feet, the pump comes on and goes off when the water level reaches 753 feet. Click *Other Data* and *Control Switches* to see this setup.



Use EPS ☑

Total Time (hrs) 24

Starting [illegible] 0

Computational Period (hrs) 1

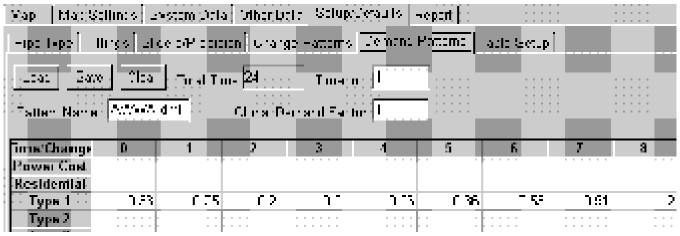
Report Time Style Military Time

Report Period (hrs) 1

Default Power Cost (%/kwh) [illegible]

Intermediate Reports ☑

Figure 5, System Data / EPS



Total Time 24 Time Incr 1

Pattern Name AWWA.dmd Global Demand Factor 1

Time/Change	0	1	2	3	4	5	6	7	8
Power Cost									
Residential									
Type 1	[illegible]	0.75	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]
Type 2									

Figure 6 Setup/Defaults / Demand Patterns - AWWA demand pattern

Switch Units HGL (ft or m)

controlled element	is	on/off	when	sensing node	is	below/between	low level	and	on/off	when	above	high level
[illegible]	is	on	when	T-1	is	below	737	and	off	when	above	753

Figure 7 Other Data / Control Switches

Analyze the system (click ***Analyze*** and select ***Analyze System***). KYPipe should be selected from the submenu. After the analysis is complete, the results can be viewed using the tabulated report, labels, contours, and profiles as described previously. An additional method of viewing results, which is particularly useful for EPS, is the use of Node Graphs and Results Tables, which are accessed as follows. Select a node and turn on the ***Rslt*** button (Node Information Window on right side of display). Turn off the other three buttons. You will see a Node Graph and a Results Table of a node result (the result type (pressure, head, etc.) will be determined by the parameter selected in the ***N*** box (Results Selector box). Click on ***Full*** to see a full screen display of the Node Graph or Results Table. Click ***Small*** to return to the map. If you carry out these operations with the ***Group*** button selected, you can produce graphs and tables with results for multiple nodes. Pipe graphs and tables are produced in a similar manner by selecting one or more pipes.

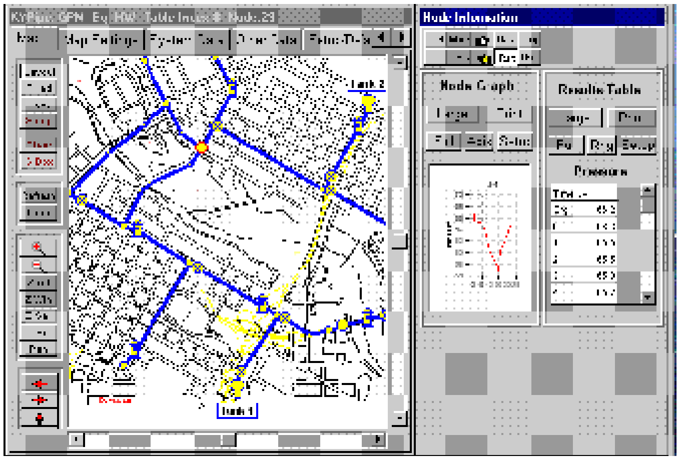


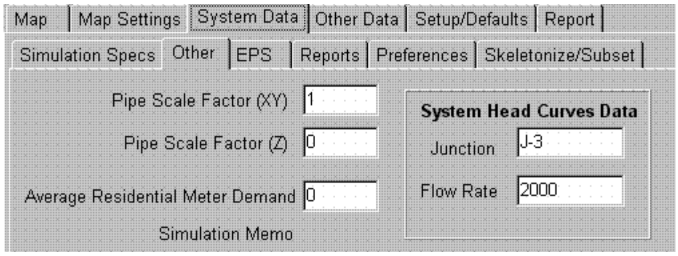
Figure 8 Results Table and Graph for Selected Node

KYPipe - Other Capabilities

The demonstration files may be used to demonstrate other Pipe2010 capabilities.

System Curves*:*

A system curve is a plot of required head vs. flow at a location (node) where a pump is to be positioned. KYPipe2010 will produce a system curve which can be plotted with pump curves to aid with pump selection. To illustrate this, load the ***Demoreg*** file. The junction downstream from the pump (J-13) is used for the system curve node and the setup may be seen by clicking ***System Data*** and ***Other***. The required data appears under System Head Curve Data and includes the junction (J-3) and the maximum flow rate (2,000 g.p.m.) used to develop the system curve. Click on ***Map*** to display the map. The pump should be shut off to develop the system curve so select the pumps (in Layout Mode) and click the ***On/Off*** switch in the upper left corner of the Node Information Window. A red ***X*** should appear through the pump indicating that it is off. To produce the System Curve, click ***Analyze***, ***Analyze System***, and select ***System Head Curve*** before clicking ***Analyze***. The analysis will do 11 simulations with flows 0 to 2,000 in increments of 200 g.p.m. The results for the system curve are summarized at the end of the ***Report*** (select ***Report*** and scroll to the end). The best way to view the results is with a plot of the system curve and any available pump curves. Click on ***Facility Management*** (Main Menu) and ***Pump Curves***. When the graph appears, select the items to display for the choices shown on the left side. Click on ***SC*** for the system curve and any numerical tags displayed for Pump ID's (in this case, there is just one). The graph shown below will appear. The intersection of the System Curve and a pump curve indicates the operating point for that pump. To return to the map, close this window.



Pipe Scale Factor (XY) 1

Pipe Scale Factor (Z) 0

Average Residential Meter Demand 0

System Head Curves Data

Junction J-3

Flow Rate 2000

Figure 9 System Data / Other - Setup for System Head Curves

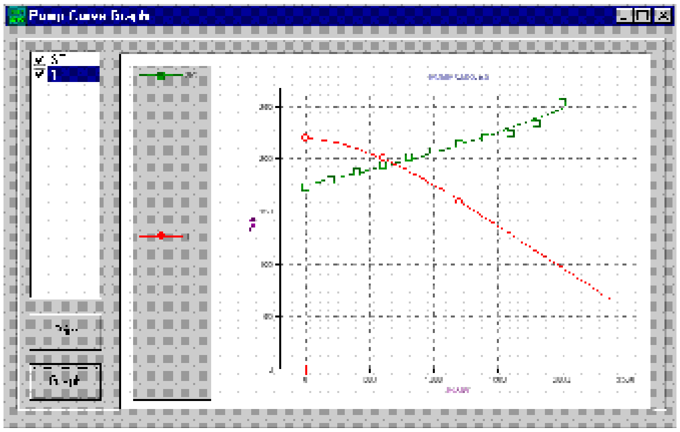


Figure 10 - System Curve

On/Off Valves - Pipe Break*:*

Pipe2010 models can include on/off valves (Ä) which can be used to control the open/closed status of any pipe link. In *Layout Mode*, select a valve and click on the *On/Off* switch in the upper left corner of the Node Information Box. A red *X* through the valve will indicate the valve and corresponding pipe is closed and a closed pipe will appear as a thin dashed line. In Group Mode, you can select multiple valves and check *On* or *Off* in the Edit Node Set box to set the status of the selected valves.

The Pipe Break feature will identify the valves which need to be closed to isolate the location in the pipe system which you indicate. This is done by clicking *Facility Management* and selecting *Pipe Break*. Then move the Ø symbol to the location to isolate and click. The display will show the area to be isolated and the valves to be closed. To obtain a report of the valves to be closed, click on *Facility Management* and *Pipe Break Report*.

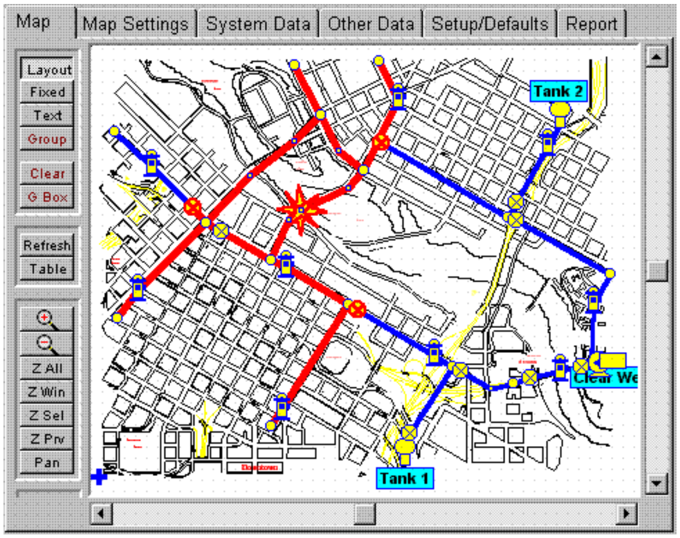


Figure 11 Pipe Break Simulation

Images:

A bitmap image (.BMP file) can be associated with each node. This feature will allow the user to provide additional information about each node. Three such images are loaded for the file *Demoreg*. In Layout mode, click on the valve in the upper center of the system. Click *Full* in the *Node Title* box on the left and you will see a hand drawn sketch showing the valve location in the field. Click *Small* (upper left) to return to the map. Click on the valve just to the left of the pump and repeat this process to see a schematic of the valve details. Click on the pump and repeat the process to see a photograph of a pumping facility.

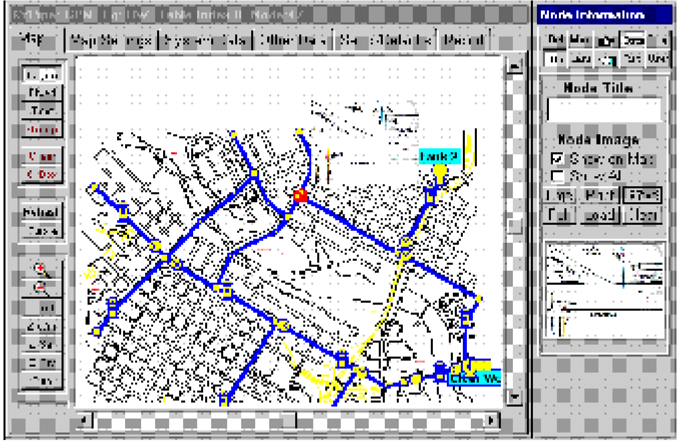


Figure 12 Node Image - Valve Map

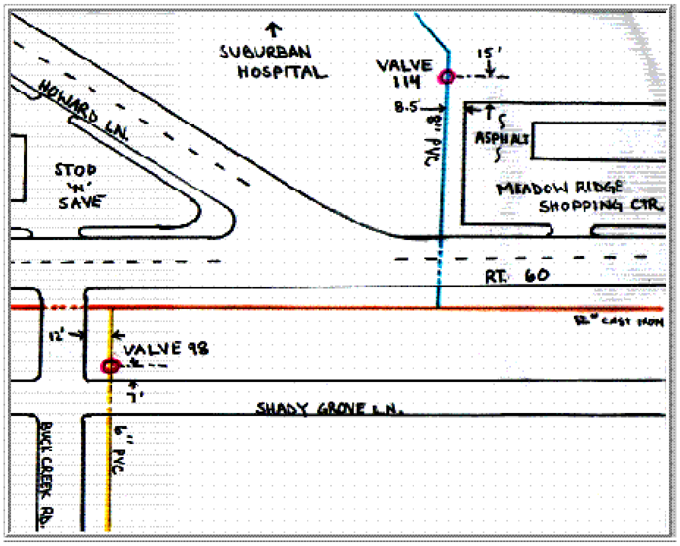


Fig 12a Valve Map - Large Size

Hydrants:

Pipe2010 models may include fire hydrants and has some special modeling capabilities for hydrants. This includes plotting test data and using the model to calculate fire flows. Eight fire hydrants are included in the *Demoreg* file. If you don't see the hydrants, click *View* and *Show Hydrants* to activate their display. In *Layout* mode, select the hydrant in the upper center of the system. In the Node Information window, you will see the pertinent hydraulic data (elevation, static and residual pressure, and residual flow). Make sure the *Data* button is on (the rest should be off). The pressures and flow inputs are for field measurements. Click on *Graph* and a plot will appear based on either the *Test Data* or *Calculated Data*. Select *Test Data* and you will see that AWWA recommended fire flow data plot projects a fire flow of around 840 g.p.m. at 20 p.s.i. If you change the selection to *Analysis Data*, you will get a similar plot based on model calculations. These calculations are obtained by going into Group mode and selecting the hydrants of interest and then performing an analysis selecting the *Fireflow Analysis* option.

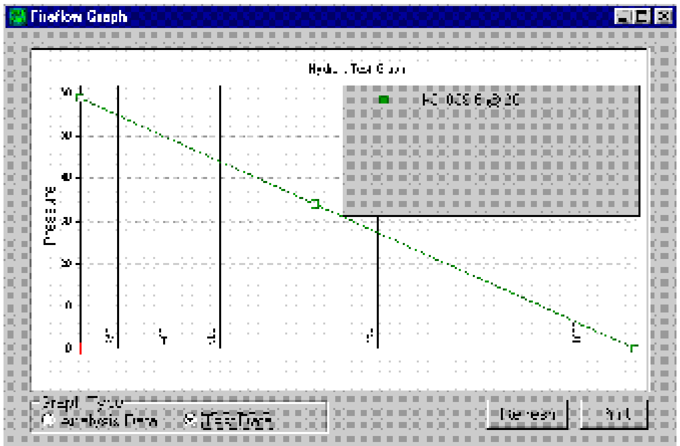
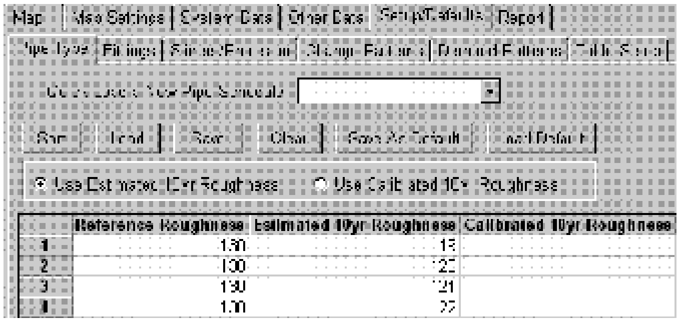


Figure 13 Fireflow Plot

Calculation Year (age based roughness):

Pipe2010 allows users to carry out simulations for a future date and projects the pipe roughness based on Pipe Type data provided by the user. This data includes a Reference Roughness (usually the new pipe roughness) and an Estimated 10-Year Roughness. To utilize this feature, a reference year is input for each pipe (the year the pipe roughness is the reference roughness - usually the year the pipe was installed). For the *Demoreg* file, the reference year is 2001 for all pipes. A reference roughness of 130 was input for the new ductile iron pipe and an estimated 10-year roughness of 119-122 was used based on the pipe size. You can see this data by clicking on *Setup/Defaults* and *Pipe Type*. Now you can do an analysis for a projected date of 2026 (25 years) by clicking on *Analysis*, *Analyze System* and turn off the *Use Current Year* switch so it will use the year 2026 shown in the box below. You can enter any year you want into this box.



● Use Estimated 10yr Roughness ○ Use Calibrated 10yr Roughness

	Reference Roughness	Estimated 10yr Roughness	Calibrated 10yr Roughness
1	130	[illegible]	
2	130	122	
3	130	121	
4	130	[illegible]	

Figure 14 Pipe Type Table Showing Roughness Data

After the analysis is complete, you can view the Report and see the calculated roughness values (106-113) and can note that due to the increase in roughness, the pressure at junction J-13 has dropped from around 20 p.s.i. in 2001 to 5.6 p.s.i. in 2026.

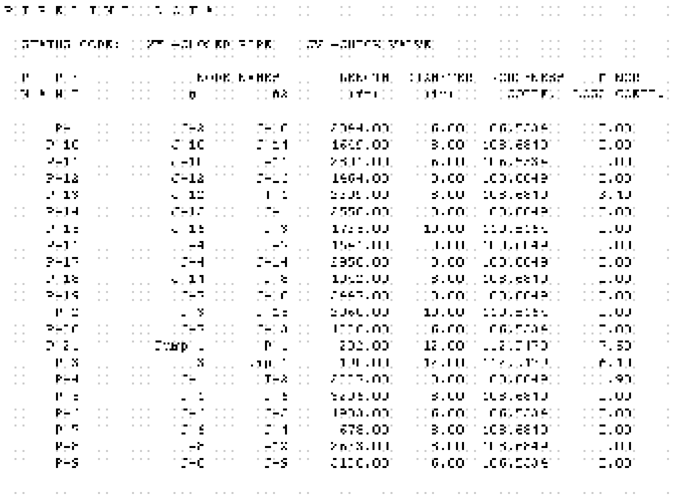


Figure 15 Calculated Roughness Values

Pipe2010 : Surge

Click on ***File*** and ***Open*** and select the file ***Demosurg***. This file is identical to the file Demoreg except for the addition of the data required for surge analysis. For this demonstration, the pump is shut down which will produce a transient that starts with the steady state conditions with the pump operating and terminates with the pump off and the tanks supplying. These are cases 0 and 1 for the Demoreg file for the steady state KYPipe2010 demonstration.

One additional pipe data item is required - wave speed. In Layout Mode, click on a pipe and the ***Data*** button (Pipe Information) and you will see the wave speed (Wv Spd) displayed. The value can be entered here or included in the Pipe Type table where it will be entered automatically when the Pipe Type is selected. A tool for calculating wave speed is provided. Click on ***Tools*** (Main Menu), then ***Wave Speed***, select ***Ductile Iron***, and use ***8 inch*** diameter with ***0.25 inch*** wall thickness and a wave speed of around 4,100 ft/s will be calculated.

Review the System Data to note differences for surge analysis. Click ***System Data*** and ***Simulation Specs***. The required entries are Units and Equations. The rest will default but you may wish to override these - especially the Total Simulation Time which defaults to 10 seconds but is entered as 20 seconds for this demonstration. Click ***Other*** to access a second system data screen. You should provide a node for the Screen Plot Node which appears while the transient is being calculated. The best way to do this is to select the desired node before you access this screen and then click ***Use Selected Node***. Click ***Map*** to return to the map.

The Change Data is very important data that defines the cause of the transient. For this demonstration, a 2 second pump shutdown is simulated starting 1 second into the simulation. Click on the pump and the Change Data (***Chng***) button in the Node Information window (turn off the other buttons so the Node Changes box can appear). You will see the setup for the pump speed ratio change which stays at 1 (speed/rated speed) for 1 second then ramps down to 0 at three seconds. Also note that a check valve is specified for the pump. Click on ***Data*** (Node Information) to see the Surge Device Data box where the check valve is selected and the closure time and resistance are defined.

The surge analysis has been already run for this demonstration file and there are extensive results to be reviewed. The most effective means are viewing pressure (or head) verses time plots and pipeline profiles with the head envelope displayed. In addition an extensive tabulated report is generated for both the transient and steady state results.

Time Plots:

Select a node and turn on the results (***Rslt***) button (Node Information). One of the Results boxes which appear in the Node Information Window is Node Graph. You may need to turn off other buttons to see the plot which is shown in this box. For the demonstration, select node J-13 (upper left center). Click on ***Full*** to see a full screen plot of the pressure transient. Note that there is cavitation (- 30 feet of head) at around 8 seconds during this transient. Click ***Small*** to return to the map.

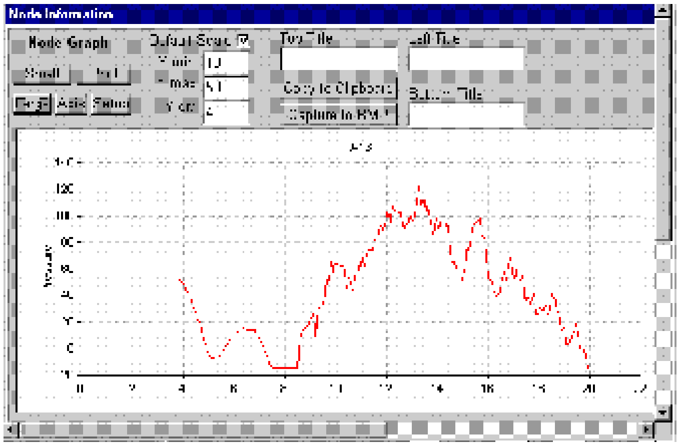


Figure 16 Node Results Graph

Profiles:

Click the *Group* button (left side) and select node J-13 and the reservoir to produce a pipeline profile between those nodes. Click *Analyze*, *Profile* and *Create Profile from Leftmost Selected Node*. The profile will appear. Click *Maximize*. Make sure *Show Envelope* and *Time/Case A* is selected. If you provide the y axis range of Minimum Elevation = 500 and Maximum Elevation = 1,000 (turn off *Default Y Axis* selection), the profile will be well scaled. You can watch the change in the head line by clicking the rightmost arrow in the A box (Results Selector - bottom). This steps forward in increments of 5% of the total simulation time. Close the profile window.

Tabulated Reports*:*

Click on *Report* to access the two tabulated reports. You can switch between the report for the initial steady state conditions and the transient analysis by clicking *Load/Swap*. Of particular interest is the table of maximum/minimum heads which appears at the end of the transient analysis report.

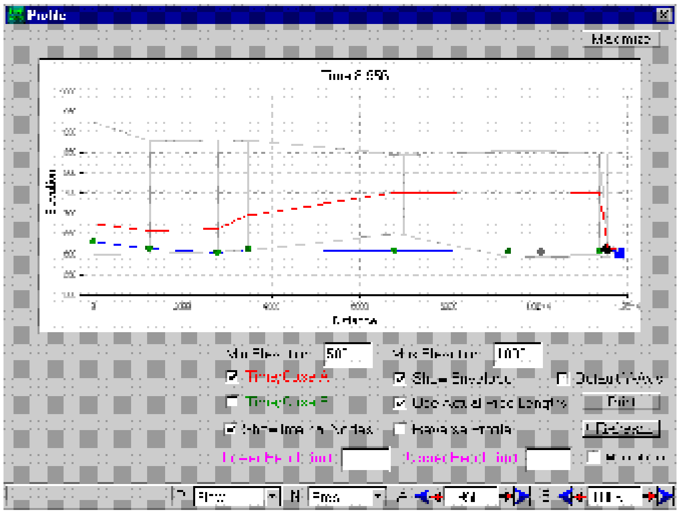


Figure 17 Profile

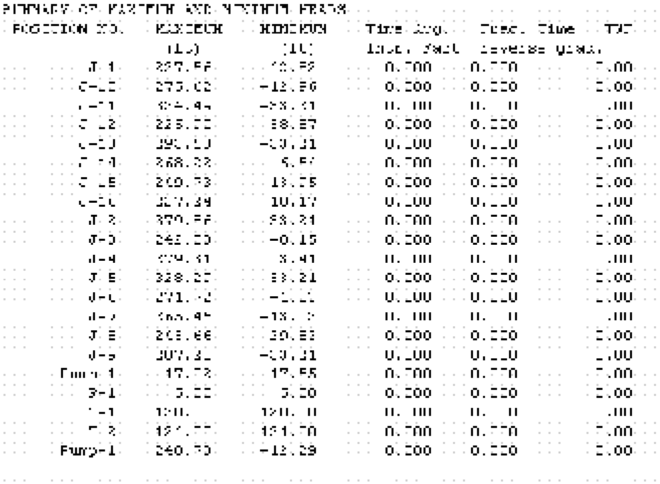


Figure 18 Maximum and Minimum Heads

Surge Protection

Because of low transient pressures which reached cavitation pressure at a number of nodes, a second surge analysis was carried out with a closed surge tank positioned just downstream from the pump. When the pump loses power with an associated rapid flow reduction, the surge tank supplies flow to compensate for the loss of flow and thereby reduce the pressure surge. A second demonstration data file is provided to illustrate this application. Click on ***File*** and ***Open*** and select ***Surgtank***. You should see a zoomed in view of the area of the pump showing the surge tank. This file is identical to the Demosurg file with the exception of the surge tank. The surge tank was added by inserting an intermediate node at the location and changing the Node Type to Closed Surge Tank. Click on the surge tank and the ***Data*** button (Node Information) and the surge tank data appears in two boxes including the Device Data box. The tank is a 4 foot vertical cylindrical vessel which is initially half full of air (62.8 ft³). The inflow and outflow resistance of 0.1 will give a 0.1 foot head loss at a flow of 1CFS.

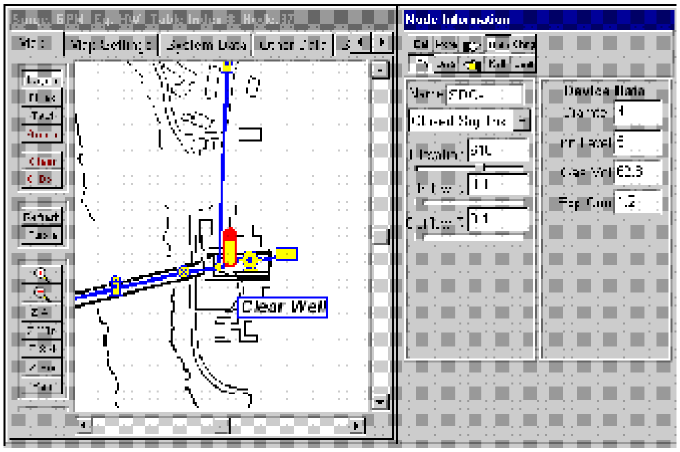


Figure 19 Surge Tank and Data

Click on the Zoom All (***Z All***) button to show the entire system. The analysis has been conducted so select junction J-13 to see the effect of the surge tank on the pressure surge. Click on the ***Rslt*** button (other buttons should be off) and click on ***Full*** to see the full screen plot. You can create the profile and view the tabulated results as described for the previous demonstration.

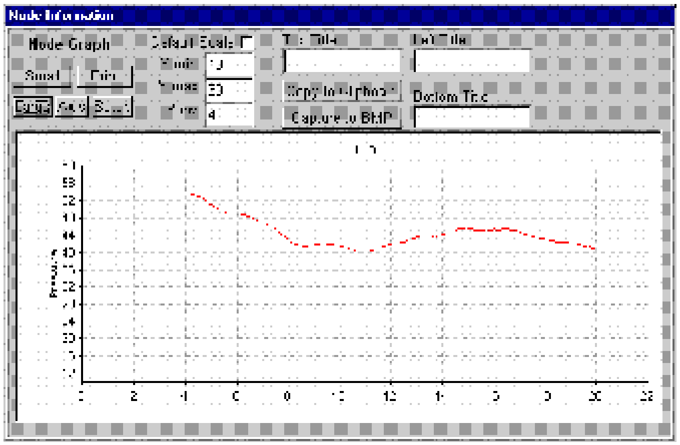


Figure 20 Node Results Graph

KYPipe - Optimized Calibration

Click on File and Open and select the DemoCal file to see a demonstration of the Pipe2010 Optimized Calibration module. You may wish to review the "What is a Pipe2010 Optimized Calibration?" topic before you go through the demonstration.
Figure 21 shows a network schematic with the test results of four fire flow tests displayed.

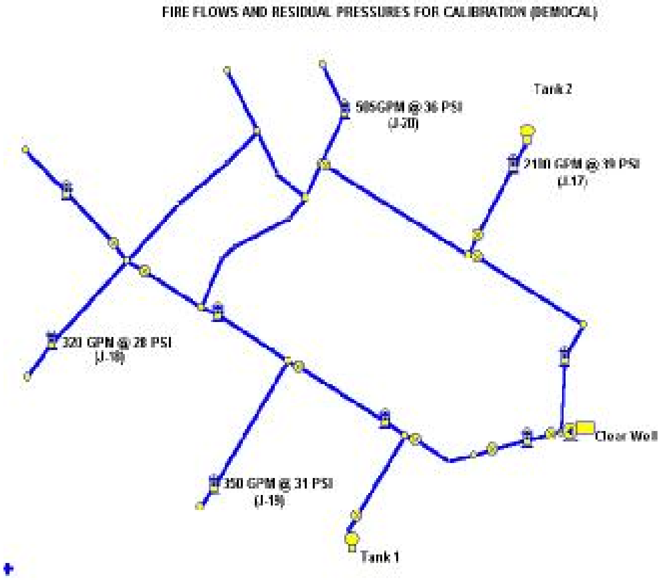


Figure 21 Fire Flow Test Results

These include the residual flow and pressure for each of the tests. For the calibration run, these four hydrants were converted to junctions as required to set up the calibration data. For the demonstration, it is assumed that the boundary conditions for each fire flow test were the same and that the baseline demands and the tank levels are those used for the DemoReg file and shown in Figure 22.

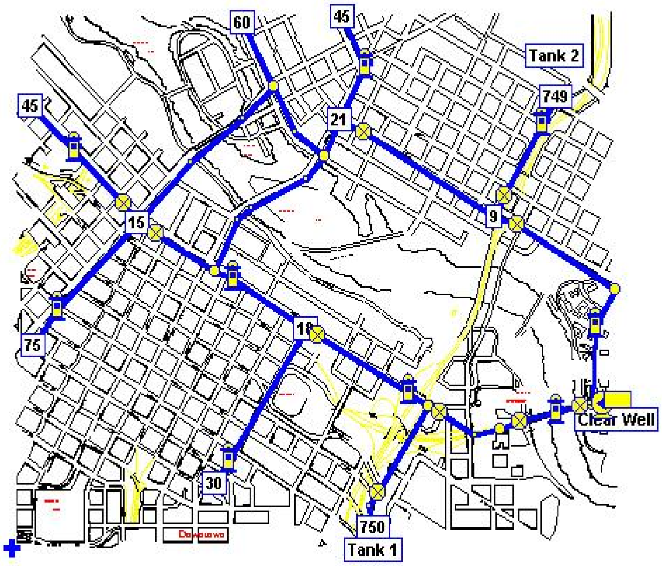
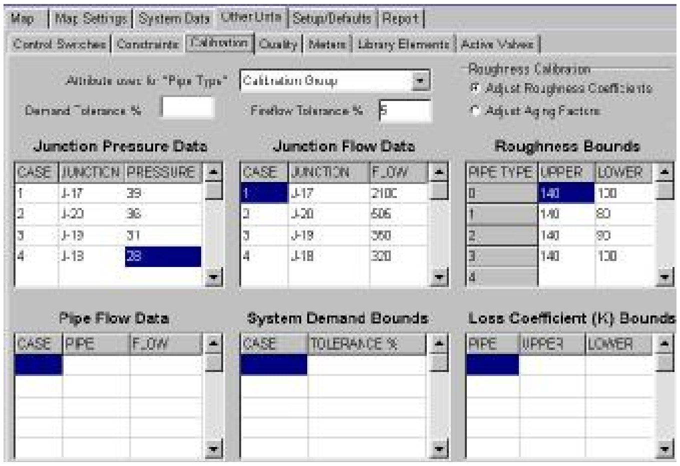


Figure 22 Tank Levels and Baseline Demands

Thus, it is not necessary to enter change data for the four separate fire flow tests. The only additional data required is the Calibration Data shown in Figure 23.



Attribute used for "Pipe Type": Calibration Group

Demand Tolerance %:

Fireflow Tolerance %: 5

Roughness Calibration

● Adjust Roughness Coefficients

○ Adjust Aging Factors

Junction Pressure Data

CASE	JUNCTION	PRESSURE
1	J-17	39
2	J-20	36
3	J-19	31
4	J-18	28

Junction Flow Data

CASE	JUNCTION	FLOW
1	J-17	2100
2	J-20	506
3	J-19	350
4	J-18	320

Roughness Bounds

PIPE TYPE	UPPER	LOWER
0	140	100
1	140	80
2	140	90
3	140	100
4		

Pipe Flow Data

CASE	PIPE	FLOW

System Demand Bounds

CASE	TOLERANCE %

Loss Coefficient (K) Bounds

PIPE	UPPER	LOWER

Figure 23 Fire Flow and Calibration Data

The roughness bounds were defined for four Calibration Groups selected using diameter as follows:

Group	Diameter
0	12
1	6
2	8
3	10

Two cases were run. The results for the first case are shown in Figure 24.

```
Percent Deviation between MEASURED and TARGET Values =   1.422

Percent Deviation between MEASURED and CALCULATED (uncalibrated) Values = 14.02


 OPTIMAL values for the Decision variables:

 Hazen William coefficients: for group number    0 =  115.  [140.0< >100.0]
 Hazen William coefficients: for group number    1 =   95.  [140.0< > 80.0]
 Hazen William coefficients: for group number    2 =   96.  [140.0< > 90.0]
 Hazen William coefficients: for group number    3 =  132.  [140.0< >100.0]


 Junction (Fire) Flow(s) for Change  1 are INCREASED by  5.00%
 Junction (Fire) Flow(s) for Change  2 are DECREASED by  5.00%
 Junction (Fire) Flow(s) for Change  3 are DECREASED by  4.03%
 Junction (Fire) Flow(s) for Change  4 are INCREASED by  5.00%


 Measured and Target pressures (psi or kPa):

    TEST        NODE      MEASURED        OPTIMAL
    CASE       NUMBER     PRESSURE       PRESSURE
  ------------------------------------------------
      1         J-17        39.0 (43.6)    39.0
      2         J-20        36.0 (37.5)    33.9
      3         J-19        31.0 (34.5)    31.0
      4         J-18        28.0 (35.7)    28.1


 Date & Time:  Mon Nov 26 08:22:52 2001


  ------- NETWORK CALIBRATION COMPLETED --------
```

Figure 24 First Case Results

For this calibration a 5% tolerance was introduced for the fire flows. This means that the fireflows can be +/- 5% of the measured residual flow and accounts for a small error in this measurement. The calibration run produced a calibration where the optimal pressure differed from the measured pressure by only 1.4% where the difference is greater than 1.4% for the uncalibrated model.

For the second case, a zero percent fireflow tolerance was used and, as expected, a larger difference of 4.5% was obtained. These results are shown in Figure 25.

```
Percent Deviation between MEASURED and TARGET Values =   4.534


 OPTIMAL values for the Decision variables:

 Hazen William coefficients: for group number    0 =  103.  [140.0< >100.0]
 Hazen William coefficients: for group number    1 =  101.  [140.0< > 80.0]
 Hazen William coefficients: for group number    2 =   92.  [140.0< > 90.0]
 Hazen William coefficients: for group number    3 =  140.  [140.0< >100.0]



 Measured and Target pressures (psi or kPa):

    TEST        NODE      MEASURED        OPTIMAL
    CASE       NUMBER     PRESSURE       PRESSURE
  ------------------------------------------------
      1         J-17        39.0           38.9
      2         J-20        36.0           32.6
      3         J-19        31.0           31.0
      4         J-18        28.0           31.6

 Date & Time:  Mon Nov 26 08:19:49 2001


  ------- NETWORK CALIBRATION COMPLETED --------
```

Figure 25 Second Case Results

KYPipe Water Quality Analysis

A water quality analysis is generally run using an EPS file. This is to determine the variance in the water quality parameters over a time period (generally 24 hours). Only one screen of additional data is required to set up the water quality analysis. To see this data, click on **File** and **Open** and select the file **Demoqual**. Click on **Other Data** and **Quality** to see the data screen shown below.

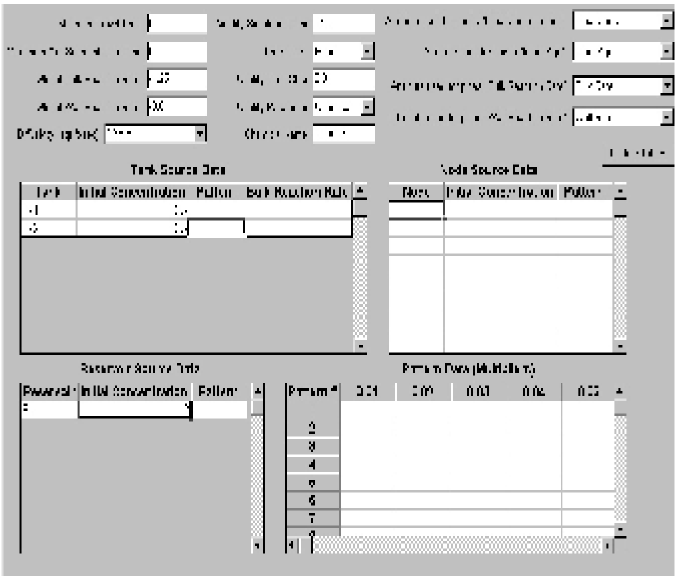
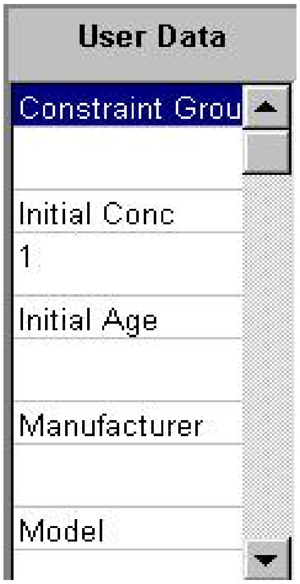


Figure 26 Water Quality Data

The Bulk and Wall Reaction Rates are set for all pipes using the global value shown rather than inputting values for each pipe. A Simulation Time of 144 hours is chosen to provide ample time for the solution to reach a repeatable condition. For this example a **Chemical** analysis is chosen and the chemical name input as **Chlorine** to determine the chlorine residuals. We could choose to calculate the age of the water (select Age) or trace the origin of the water (select Trace). One additional useful data input is the **Initial Concentration** of chlorine at each node. You can take no action and this parameter will be assigned an initial value of zero. However, a reasonable estimate of this value will provide the solution more quickly and accurately. Since the chlorine is supplied at 2 ppm, a value of 1 ppm is used for the initial concentration and this data is assigned by using the **Gbox** (Group Mode) to select the entire system and the **Edit Node Set** to assign a value of 1.0 to the Initial Concentration. When this is done the **User Data** for each node should display this data as shown below:


User Data

Constraint Grou

Initial Conc
1

Initial Age

Manufacturer

Model

Figure 27 User Data with Initial Concentration

The Water Quality Analysis is then run by selecting **Analyze / Analyze System** and **Water Quality**.

Once the analysis is completed the results are reviewed. Figure 28 shows the results for the minimum and maximum chlorine levels. This is obtained by selecting **Chlorine** in the **N Box** (Results Selector) and **Node Results/Node Results Min and Max** under Labels.

A plot of the variations in the chlorine residuals at various nodes can be shown as illustrated in Figure 29.

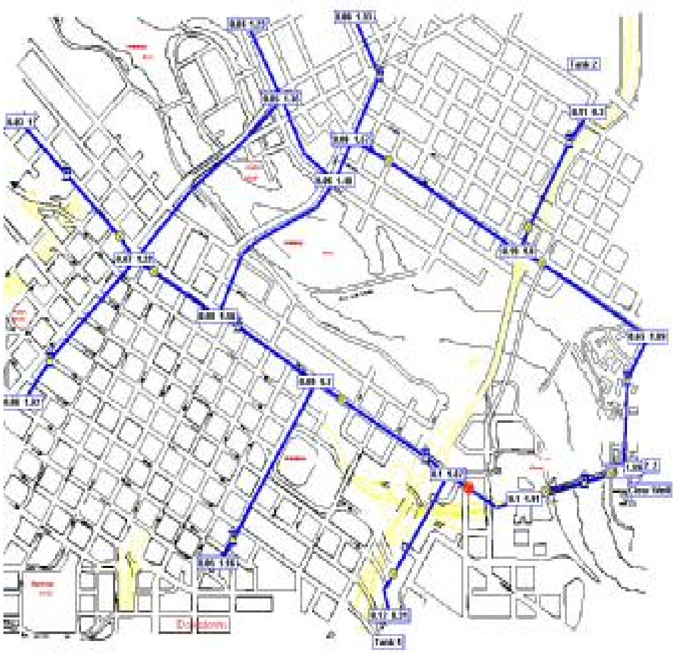


Figure 28 Min/Max Chlorine Residuals

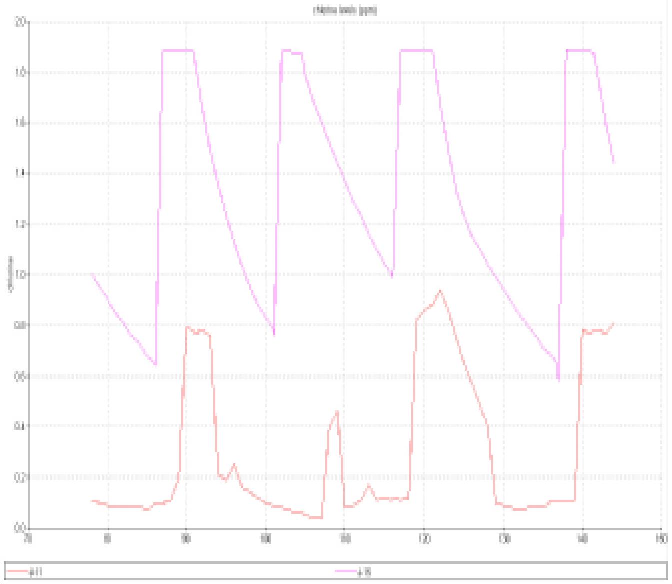


Figure 29 Chlorine Residuals at Selected Nodes