## Table of Contents


1. Table of Contents 19-22
   1.1. NETWORK ANALYSIS 19-22
   1.2. Company Information 22
      1.1.1. Contact Us 22-23
      1.1.2. History and Development 23-24
   1.3. Software Information 24
      1.2.1. License Agreement 24-25
      1.2.2. Installation Instructions 25-28
      1.2.3. Version Features 28-36
      1.2.4. International Decimal Settings 36-37
   1.4. Pipe2018 - All Models 37
      1.3.1. Help and Tutorials 37
         1.3.1.1. About the Online Help 37-38
         1.3.1.2. Overview and Getting Started 38-40
         1.3.1.3. Quick Start 40-47
         1.3.1.4. KYPipe and Surge Demo files 47-89
         1.3.1.5. PIPE2000 Tutorial (Audio/Video) 89-92
         1.3.1.6. Reference Manual 92
            1.3.1.6.1. Introduction to Modeling 92-94
            1.3.1.6.2. Method of Analysis 94-95

1.3.1.6.3. Model Simplification 95-96
1.3.1.6.4. Pipe System Geometry 96-101
1.3.1.6.5. Pipe System Components 101-102
1.3.1.6.6. Pressure and Flow Specifications 102-103
1.3.1.6.7. Multiple Scenarios - Changes 103-104
1.3.1.7. Some Special Features 104-105
1.3.1.8. Warnings and Error 105-110
1.3.1.9. Hotkeys and Shortcuts 110-113
1.3.2. Files, Printing, Menus 113
1.3.2.1. Menus and Tabs 113
1.3.2.1.1. KYnetic Configuration Menu 113-114
1.3.2.1.2. Classic Menu in KYnetic Interface 114-116
1.3.2.1.3. Analyze (Main Menu) 116-120
1.3.2.1.4. Edit (Main Menu) 120-127
1.3.2.1.5. Facilities Management (Main Menu) 127-129
1.3.2.1.6. File (Main Menu) 129-135
1.3.2.1.7. Help (Main Menu) 135-137
1.3.2.1.8. Labels (Main Menu) 137-141
1.3.2.1.9. Main Menu 141
1.3.2.1.10. MAP SETTINGS 141
1.3.2.1.11. Other 141-145

1.3.2.1.12. OTHER DATA 145-146
1.3.2.1.13. Preferences 146-152
1.3.2.1.14. REPORT 152-156
1.3.2.1.15. Reports (System Data) 156-159
1.3.2.1.16. SETUP / DEFAULTS 159
1.3.2.1.17. SYSTEM DATA 159-160
1.3.2.1.18. Tools (Main Menu) 160-162
1.3.2.1.19. View (Main Menu) 162-165
1.3.2.1.20. Hotkeys and Shortcuts 165-168
1.3.2.2. Information Windows and Data Boxes 168
1.3.2.2.1. Information Windows 168-169
1.3.2.2.2. Node Change Box 169-170
1.3.2.2.3. Node Data Boxes 170-174
1.3.2.2.4. Node Information Window 174-180
1.3.2.2.5. Node Results Boxes 180-183
1.3.2.2.6. Node User Box 183-184
1.3.2.2.7. Pipe Change Box 184-185
1.3.2.2.8. Pipe Data Boxes 186-190
1.3.2.2.9. Pipe Information Window 190-196
1.3.2.2.10. Pipe Results Boxes 196-199
1.3.2.2.11. Pipe User Box 199-202

1.3.2.3. Printing 202-206
1.3.2.4. Files 206
1.3.2.4.1. Backup Files 206-207
1.3.2.4.2. Data Files 207-208
1.3.2.4.3. Data Records 208-209
1.3.2.4.4. Table Setup 209-210
1.3.3. Creating and Editing a Model 210
1.3.3.1. Pipe System Layout and Modification 210-213
1.3.3.2. Scenarios, Changes, Demand Patterns 213
1.3.3.2.1. Change Data 213-214
1.3.3.2.2. Change Patterns 214-217
1.3.3.2.3. Generate Change Pattern 217-220
1.3.3.2.4. Demand Allocations/Meters 220-221
1.3.3.2.5. Demand Patterns 221-223
1.3.3.2.6. Demand Specification 223-224
1.3.3.2.7. Scenario Management 224
1.3.3.3. Copy and Paste 224-228
1.3.3.4. Automatic Demand Distribution 228-232
1.3.3.5. Connect Crisscross Pipes 232-234
1.3.3.6. Data Requirements 234-235
1.3.3.7. Data Tables 235-242

1.3.3.8. Data Table - Quickstart Example 242-247
1.3.3.9. Deleting Intermediate Nodes 247-248
1.3.3.10. Fittings 248-254
1.3.3.11. Group Operations 254
1.3.3.12. Groups, Subsets, Minimizing 254-255
1.3.3.13. Input and Editing Shortcuts 255-257
1.3.3.14. Interpolate Elevations 257-259
1.3.3.15. Logical Text Inputs 259-260
1.3.3.16. Materials and Rating 260
1.3.3.17. Meter Based Demands 260-261
1.3.3.18. Operating Modes 261-264
1.3.3.19. Pipe Diameter 264
1.3.3.20. Pipe Type Data 264-267
1.3.3.21. Sets and Group Mode 267-276
1.3.3.22. Simulation Specs 276-281
1.3.3.23. Snap Hydrants Valves and Meters 281-284
1.3.3.24. User Data 284-299
1.3.4. Units and Reference Tables 299
1.3.4.1. Analyzing for liquids other than water 299
1.3.4.2. Darcy-Weisbach Table 299-301
1.3.4.3. Hazen-Williams Table 301-304

1.3.4.4. Modified Hazen Williams Formula 304-305
1.3.4.5. Minor Loss Coefficients Table 305-309
1.3.4.6. Tools (Main Menu) 309-311
1.3.4.7. Units 311-313
1.3.4.8. Units Help - Hover Buttons 313
1.3.4.9. Units Pipe2000 : KYPipe / Surge 313-315
1.3.5. Elements and Devices 315
1.3.5.1. Active Valve 315-318
1.3.5.2. All About Pipes 318-319
1.3.5.3. All about nodes 319-323
1.3.5.4. Annulus Pipe Flow 323-324
1.3.5.5. Blowoff Hydrant 324-325
1.3.5.6. Check Valve 325
1.3.5.7. Closed Loop Systems 325-326
1.3.5.8. Compressors 326
1.3.5.9. End Nodes 326-328
1.3.5.10. Flow Control Valve (FCV) 328-329
1.3.5.11. Holding Tank 329-331
1.3.5.12. Hydrant Data 331-332
1.3.5.13. Hydrant Monitor 332-334
1.3.5.14. Hydrotank 334-336

1.3.5.15. Hydrotank Using Tank Element 336-338
1.3.5.16. Inline Meter 338
1.3.5.17. Internal Nodes 338-340
1.3.5.18. Internal Node Basic Data 340-341
1.3.5.19. Internal Node Data 341-344
1.3.5.20. Junction Data 344-346
1.3.5.21. Libraries 347-349
1.3.5.22. Loss Element Data 349-351
1.3.5.23. Loss Element Library 351-352
1.3.5.24. LPS Tank 352-355
1.3.5.25. Metered Connection Data 355-358
1.3.5.26. Meters 358-359
1.3.5.27. Minor Loss Components 359-360
1.3.5.28. Network Elements 360-361
1.3.5.29. Node Types and Properties 361-364
1.3.5.30. On/Off Valves 364-365
1.3.5.31. Positive Displacement Pump in Surge 365-368
1.3.5.32. Pressure Supply Data 369-373
1.3.5.33. Pressure Regulating Valve (PRV) 374
1.3.5.34. Pressure Sustaining Valve (PSV) 374-375
1.3.5.35. Pressure Relief Valve in KYPipe 375-376

1.3.5.36. Pump Data 376-383
1.3.5.37. Pump Minimum Flow Valve 383
1.3.5.38. Pumps 383-388
1.3.5.39. Regulating Valves 388
1.3.5.40. Regulator Data 388-390
1.3.5.41. Reservoir Data 390-393
1.3.5.42. Residential Meters 393-394
1.3.5.43. Sprinkler Data 394-396
1.3.5.44. Tank Data 396-401
1.3.5.45. Vacuum Breaker element 401-404
1.3.5.46. Valves 404-405
1.3.6. Backgrounds and Map Tools 405
1.3.6.1. Node and Pipe Glyphs 405-413
1.3.6.2. Plan View-Profile View 413-414
1.3.6.3. Isolation Zones 414-418
1.3.6.4. Internet Background Maps and Automatic Elevation Retrieval 418-466
1.3.6.4.1. Driving Directions - Internet Maps 466-471
1.3.6.5. Animate Map 471-473
1.3.6.6. Background Images 473-475
1.3.6.7. Backgrounds 475-480
1.3.6.8. Complex CAD Formats Background Method 480-481

1.3.6.9. Colors 481-482
1.3.6.10. Contours - Main Menu 482-484
1.3.6.11. Cursor Notation 484-486
1.3.6.12. Emphasis / Contours - Nodes 486-495
1.3.6.13. Emphasize Pipes as Nodes 495-498
1.3.6.14. Grids 498-501
1.3.6.15. Labels 501-504
1.3.6.16. Legend 504-507
1.3.6.17. Map Link 507-511
1.3.6.18. Map Screen 511-513
1.3.6.19. Minimum View Scale 513
1.3.6.20. Node Images and Text Nodes 513-518
1.3.6.21. Panning Controls 518-520
1.3.6.22. Pipe Emphasis 520-523
1.3.6.23. Pushpin 523-525
1.3.6.24. Scaling Background Maps 525-532
1.3.6.25. Scroll Bar 532
1.3.6.26. Text Nodes 532-534
1.3.6.27. Text Node Data 534-537
1.3.6.28. Text and Notes menu 537-538
1.3.6.29. URL and SCADA for nodes 538-542

1.3.7. Analysis and Design Tools 542
1.3.7.1. Calibration 542-543
1.3.7.1.1. Calibration of Hydraulic Networks 543-575
1.3.7.1.2. Calibration 575-576
1.3.7.1.3. Calibration Wizard 576-595
1.3.7.1.4. Calibration Examples 595-618
1.3.7.1.5. Optimized Calibration 618-619
1.3.7.1.6. Optimized Calibration Data 619-634
1.3.7.2. Pressure Sensitive Analysis 635-640
1.3.7.3. Constraints Data 640-643
1.3.7.4. Cost and Inventory Calculations 643-652
1.3.7.5. Direct Parameter Calculation 652-662
1.3.7.6. Network Analysis 662-664
1.3.7.7. Operational Control Settings 664-669
1.3.7.8. Optimal Pipe Design 669-671
1.3.7.9. Optimal Pump Scheduling 671-676
1.3.7.10. Optimal Pump Scheduling Algorithms 676-697
1.3.7.11. Pipe Break 697-701
1.3.7.12. Profile 701-706
1.3.7.13. Profile Animated 706-717
1.3.7.14. Creating Profiles 717-721

1.3.7.15. Pump and System Curves 721-737
1.3.7.16. Skeletonize/Subset 738
1.3.7.17. Skeletonize 738-740
1.3.7.18. Travel Time Path and Distance 740-744
1.3.7.19. Former Analysis Method 744-745
1.3.8. Results 745
1.3.8.1. User Information in Report 745-747
1.3.8.2. Saved Results 747-748
1.3.8.3. Customized Reporting 748-753
1.3.8.4. Presentation Generator 754-776
1.3.8.5. Result Selector Bar 776-777
1.3.8.6. Results Presentations 777-790
1.3.8.7. Selected Output 791
1.3.8.8. Sliders/Precision 791-793
1.3.9. Import/Export 793
1.3.9.1. ArcView Import Utility 793-804
1.3.9.2. ArcView Export Utility 804-807
1.3.9.3. AutoCad Exchange 807-812
1.3.9.4. Convert WaterCAD 812
1.3.9.5. Copy and Paste 812-816
1.3.9.6. Cybernet Import 816

1.3.9.7. DAT Import 816-819
1.3.9.8. Data Exchange 819-820
1.3.9.9. DEM Elevations Import Tool 820-821
1.3.9.10. EPANET Conversion 821-824
1.3.9.11. EPANET Export 824-825
1.3.9.12. Excel Import and Export 825-827
1.3.9.13. Execute GenFile 827
1.3.9.14. Google Earth Import and Export 828-832
1.3.9.15. Import DT2 File 832-833
1.3.9.16. Import KY 833
1.3.9.17. Profile Data Import 833-835
1.3.9.18. Surge5 Conversion 835-836
1.3.9.19. To TIFF 836-837
1.3.9.20. Utility Programs 837-838
1.3.9.21. WaterCad Conversion 838-839
1.3.10. Tools - Calculators 839
1.3.10.1. 10 Year HW Coefficient 839
1.3.10.2. Calculator Tool 839-840
1.3.10.3. C Factor Calculator Tool 840
1.3.10.4. Headloss Calculator Tool 840-842
1.3.10.5. Hose and Nozzle Constants Tool 843-844

1.3.10.6. Power (HP or KW) Calculations Tool 844
1.3.10.7. Pump Characteristic Curves Tool 844-846
1.3.10.8. Pump Selection Tool 846
1.3.10.9. Residual Pressure Adjustment Tool 846-847
1.3.10.10. Resistance Calculations Tool 847-849
1.3.10.11. Sprinkler or Blowoff Constant Tool 849-850
1.3.10.12. Units Converter Tool 850
1.5. KYPipe - Water 850
1.4.1. Water Quality/EPANET 851
1.4.1.1. EPA Stage 2 DBP Rule 851-853
1.4.1.2. EPANET 853-878
1.4.1.3. Water Quality Calibration 878-880
1.4.1.4. Water Quality Sensor Placement Tool 880-901
1.4.2. Extended Period Simulation 901
1.4.2.1. Extended Period Simulations (EPS) 901-907
1.4.2.2. EPS (Extended Period Simulation) 907-909
1.4.2.3. Control Switches Data 909-912
1.4.2.4. Pressure Switch 912-914
1.4.3. Rural Analysis 914
1.4.3.1. Rural Water Systems 914-957
1.4.3.2. Quick Guide to Running Rural Analysis 957-958

1.4.3.3. Rural Water Systems (Peak Demand Requirements) 958-959
1.4.3.4. Calculate Branch Diameters 959-961
1.4.4. Hydrants, Fire Flows, Flushing 961
1.4.4.1. Fire Flows (Calculated) 961-974
1.4.4.2. Flushing Pipes 974-985
1.4.4.3. Hydrants, Fire Flows, and Flushing Pipes 985-986
1.4.5. Low Pressure Sewer Models 986-1028
1.4.6. Temperature Dependent Liquid 1028-1033
1.4.6.1. Density and Viscosity Table Tool 1033-1034
1.4.7. Age Based Roughness 1034-1042
1.4.8. Intrusion Calculator 1042-1044
1.6. Surge 1044
1.5.1. System Curve for Surge 1044-1052
1.5.2. Surge - System Data / Simulation Specs 1052-1055
1.5.3. Surge Modeling Requirements 1055-1056
1.5.4. Surge Calculations and Sign Conventions 1056-1058
1.5.5. Surge Elements and Data Requirements 1058-1120
1.5.5.1. Siphon Breaker 1120-1123
1.5.5.2. Check Valve Node 1123-1127
1.5.5.2.1. Dynamic Characteristic of Check Valves 1127-1137
1.5.5.2.2. Deceleration in Check Valve Dynamics Example 1137-1151

1.5.5.3. Creating Custom Pump Files 1151-1156
1.5.5.4. Dynamic Air Valves 1156-1160
1.5.5.5. Electrically Actuated Surge Anticipation Valve 1160-1164
1.5.5.6. Hydraullically Actuated Surge Anticipation Valve (HydSAV) 1164-1166
1.5.5.7. Hybrid Surge Tanks 1166-1175
1.5.5.8. Turbines 1175-1191
1.5.5.9. Zero Velocity Valves 1191-1193
1.5.6. Surge Pipe Data (Wave Speed) 1193-1203
1.5.7. Surge Components 1203-1209
1.5.8. Surge Control and Control Devices 1209-1220
1.5.9. Surge Transient Setup Examples 1220-1244
1.5.10. Surge - System Data / Other 1244-1245
1.5.11. Surge - System Data / Reports 1245-1247
1.5.12. Surge - Reviewing and Presenting Surge Results 1247-1261
1.5.13. Surge QuickStart Example 1261-1274
1.5.14. Surge Pipe Break Analysis 1274-1278
1.5.15. Units Pipe2000 : KYPipe / Surge 1278-1279
1.5.16. Positive Displacement Pump in Surge 1279-1283
1.5.17. Stabilization Settings for Surge 1284
1.5.18. Surge Preferences 1284-1285
1.5.19. Surge Tank Data 1285-1298

1.5.20. Modeling Air in Pipeline for Surge 1298-1299
1.5.21. Time Step in Surge 1299-1301
1.5.22. Surge Report Definitions 1301-1302
1.5.23. Role of Pump Inertia Values in Transient Analysis of Pipe Line System 1302-1319
1.5.24. EPA Surge 1319-1321
1.5.25. Surge Tools 1321
1.5.25.1. Air Slam Pressure Surge Tool 1321-1322
1.5.25.2. Air Valve Orifice Size Tool 1322-1323
1.5.25.3. Air Valve Placement Tool 1323-1332
1.5.25.4. Bladder Precharge Tool 1332-1353
1.5.25.5. EPS2Surge 1353
1.5.25.6. Equivalent Surge Tank Converter Tool 1353-1354
1.5.25.7. Force Calculations Tool 1354-1356
1.5.25.8. Generate Air Valvel Locations Tool 1356-1357
1.5.25.9. Generate Intermediate Pump File Tool 1357-1358
1.5.25.10. Modulating (Regulating) Valve Tool 1358-1359
1.5.25.11. Pump File Characteristics Tool 1359-1360
1.5.25.12. Select Pump File Calculate Inertia Tool 1360-1362
1.5.25.13. Spike Track Tool 1362-1366
1.5.25.14. Surge5 Conversion 1366-1367
1.5.25.15. Valve Stroking Tool 1367-1368

1.5.25.16. Wave Speed Tool 1368-1369
1.7. GoFlow/KYPipe Fire Sprinkler Analysis 1369
1.6.1. NFPA Standards in KYPipe and GoFlow 1369-1405
1.6.2. User Information in Report 1405-1407
1.6.3. GoFlow 1407
1.6.4. GoFlow - Quickstart Example 1407-1429
1.6.5. GoFlow - Nodes 1429-1432
1.6.6. GoFlow - System Data 1432-1435
1.6.7. GoFlow - Generate System 1435-1440
1.6.8. GoFlow - Auto Orthogonalize 1440-1442
1.6.9. GoFlow - Booster or Fire Pump Water Supply 1442-1444
1.6.10. GoFlow - Tree Generator Example 1444-1458
1.6.11. GoFlow - Grid Generator Example 1458-1474
1.6.12. GoFlow - Equivalent K-factor with a Sprinkler on a Riser Nipple 1474-1475
1.6.13. Locate Remote Sprinkler Area 1475-1478
1.6.14. Units GoFlow 1478-1479
1.6.15. GoFlow 1479
1.6.16. Required Capacity 1479-1481
1.8. SWMM Stormwater 1481
1.7.1. SWMM (Stormwater) 1481-1563
1.7.2. Storm Example 1563-1586

1.7.3. SWMM Error Messages 1586-1594
1.9. Gas and Steam 1594
1.8.1. Gas 1594-1599
1.8.2. Constant Density Gas 1599-1600
1.8.3. Ideal Gas 1600-1602
1.8.4. Non-Ideal Gas 1602-1603
1.8.5. Lookup Properties for Gas 1603-1604
1.8.6. Active Valve for GAS 1604-1605
1.8.7. Steam 1605-1607
1.8.8. Temperature Sensitive Analysis 1607-1614
1.8.9. Units Gas and Steam 1614-1615
1.8.10. Tools - Gas and Steam 1615
1.8.10.1. Gas K Calculations Tool 1615-1616
1.8.10.2. Gas Cv Calculations Tool 1616
1.10. GoPlot 1616
1.9.1. GoPlot 1616-1633
2. Index 1634-1671

# 1 Table of Contents

## CONTENTS

### Pipe2018 Help System

Pipe2018 is a powerful graphical user interface (GUI) for the development and simulation of models of piping systems. Pipe2018 supports calculations for incompressible flow (KYPIPE and EPANET), compressible flow (Gas and Steam), transient flow (Surge), fire sprinkler analysis (GoFlow), and stormwater modeling (SWMM). For all applications, there are a series of operations necessary to develop a pipe system model, enter data, and analyze the piping system. Online information regarding modeling and related information is available by clicking on the items listed below:

## Pipe2018 New Features and Capabilities

## What's new with Pipe2018?

ABOUT Pipe2018 ONLINE HELP

How to access and review important Help information

OVERVIEW AND GETTING STARTED

What is Pipe2018?

**Pipe2018 TUTORIAL (AUDIO/VIDEO)**

A number of short audio/video clips are available demonstrating how to use Pipe2018

**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 Pipe2018?

**BACKGROUND IMAGES**

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

**QUICKSTART EXAMPLE (KYPipe)**

Walk me through developing a pipe network model with Pipe2018

**INFORMATION WINDOWS**

Boxes for entering pipe and node data and displaying information

**DATA REQUIREMENTS (KYPipe)**

Access information on data requirements and units

**USER DATA**

User provides many capabilities for enhancing models

**VALVES**

Pipe2018 has many capabilities associated with valves

**HYDRANTS, FIREFLOWS, AND FLUSHING**

Pipe2018 has very advanced features for fire flow predictions

**DEMAND ALLOCATIONS / METERS**

Pipe2018 has some very advanced features for handling demands

**SOME SPECIAL FEATURES**

Save lots of time and do some neat stuff

**1.1 NETWORK ANALYSIS**

How do I perform the analysis on my system?

**Pipe2018 PRESENTATIONS**

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

**DATA FILES / SCENARIO MANAGEMENT**

Pipe2018 data files include Demand and Change Pattern selections to facilitate Scenario Management.

**DATA TABLES**

Pipe2018 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 Pipe2018 and other calibration approaches.

**WATER QUALITY**

Use EPANET with Pipe2018 to answer water quality questions.

**FACILITIES MANAGEMENT**

Pipe2018 has many useful Facilities Management features

**SKELETONIZATION**

How to skeletonize a model

**REFERENCE MANUAL**

Detailed information about modeling and the KYPIPE analysis engine

**UTILITIES**

What extra programs come with Pipe2018?

**RURAL WATER SYSTEMS**

A specially designed network analysis approach to reflect the peak demand requirements of a Rural Water System.

**Pipe2018: Surge**

The Pipe2018 transient flow model

Gas Model

The Pipe2018 compressible flow model

**Steam Model**

**The Pipe2018 saturated steam flow model**

**GoFlow Model**

**The Pipe2018 fire sprinkler system model**

**DEMO FILES**

Demonstration Files for KYPipe and Surge

## 1.2 Company Information

### 1.1.1 Contact Us

**Contacting Us - Software Development and Support Team**

| | | | |
|---|---|---|---|
| **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 |
| **Jana Faith** | BS, Civil Engineering | (812) 619-1901 | Jana@kypipe.com |
| **Doug Wood** | MS, Computer Engineering | (859) 263-0401 | Doug@kypipe.com |

| | | | |
|---|---|---|---|
| **Eric Liebenauer** | MS, Mechanical Engineering | (859) 263-2234 | Eric@kypipe.com |
| **Ferran Guillen** | MS, Mechanical Engineering | (+034)938 712 483 | Ferran@kypipe.com |

KYPIPE LLC
710 Tom's Creek Rd. Cary, NC 27519
Phone: (469) 250-1362


## 1.1.2 History and Development

Continuous research and development over the past 35 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 added their expertise to our product. Advanced graphical interfaces 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 advanced graphical environment, Pipe2018, has been adapted to other models, analyzing gas (Pipe2018 : Gas), steam (Pipe2018 : Steam), fire sprinkler systems (Pipe2018 : GoFlow), transient flow (Pipe2018 : Surge), and stormwater (Pipe2018 : SWMM).

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.

## 1.3 Software Information

### 1.2.1 License Agreement

**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.

## 1.2.2 Installation Instructions

**Pipe2018 General and Network Installation with a USB or Hard Drive Key**

**Note: These directions do NOT apply to the blue green WibuKey device.**

**Stand-Alone (local) Installation**

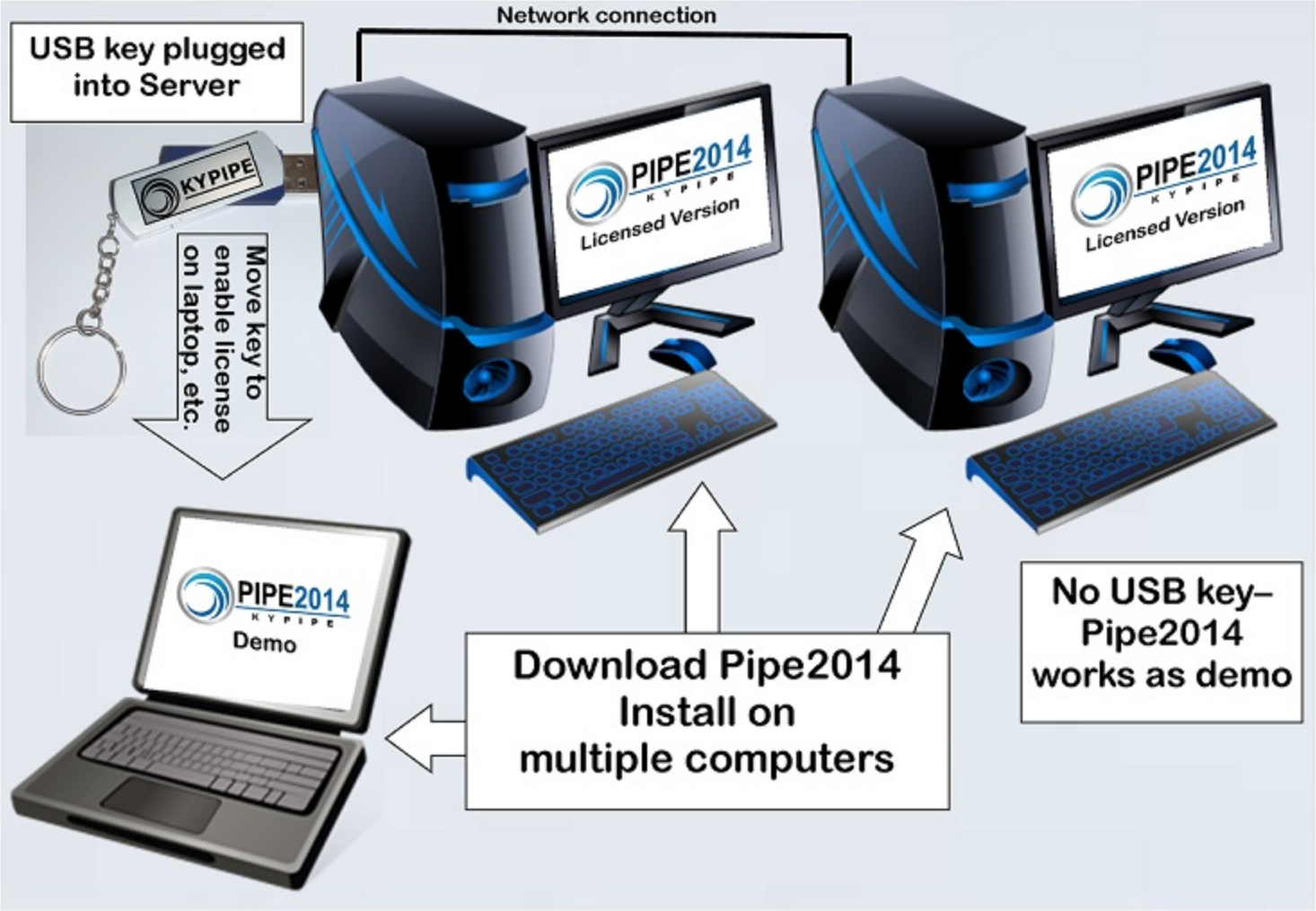
**Install Pipe2018 on each computer that will run it**

**Download Pipe2018 from www.kypipe.com under Support | Download Software, or install it from a CD or silver USB device.**

**Plug in USB key. For Hard Drive key, see "Remotely Create New Key" below**

**All users running Pipe2018 for the first time:**

**When you run Pipe2018 for the first time you will be asked if you want to search for USB keys. Select "Yes". As it searches various drives which may not have disks or are not accessible, you may get error messages. Click "Continue" any time this occurs until the USB key has been found. You will not need to repeat this process when Pipe2018 is launched the next time.**




**Network Installation**

1. First Install Pipe2018 on each computer that will run it

Download Pipe2018 from www.kypipe.com under Support | Download Software, or install it from a CD or silver USB device.

2. Next setup the computer that will act as the network server

Select a drive or folder on the network server where the key will be placed. If you have a USB device select the computer where the USB device will be plugged in. The selected drive or folder must be accessible with read/write privileges to all Clients on the network**. If you do not have a USB device with a key already on it then contact KYPipe about how to get the key onto the selected device/location.**

Browse to folder or USB device in Windows Explorer. Right click.

XP: Select "Sharing and Security"

Select the "Sharing" tab. If it asks you if you are sure you want to share, click the link. In the Network Sharing and Security area, check the box which says "Share this folder on the network"

Check the box which says "allow network users to change my files" Click ok.

Vista: Select "Share" (Vista)

Under Advanced Sharing click the Advanced Sharing button

Then check the box "Share this Folder"

Click on "Permissions". Check the "Allow" box next to "Change" and "Full Control". Click OK.

Win7: Select "Share With".

Under "Advanced Sharing" click the Advanced Sharing button.

Then check the box "Share this Folder".

Click on "Permissions". Check the "Allow" box next to "Change" and "Full Control". Click OK.

3. Next set up the client computers that will run Pipe2018

Pipe2018 should have been installed on each computer in the first step.

On each client machine, map the folder or USB drive over the network to a single drive letter.

XP / Vista / Win 7

Go to the client machine. Using Windows Explorer, navigate over the network to USB device. Right click on it and select map network drive. Assign it a single, unique drive letter.

4. All users running Pipe2018 for the first time

When you run Pipe2018 for the first time you will be asked if you want to search for USB keys. Select "Yes". As it searches various drives which may not have disks or are not accessible, you may get error messages. Click "Continue" any time this occurs until the USB key has been found. You will not need to repeat this process when Pipe2018 is launched the next time.

**Remotely Create New Key**

To create a new USB or Hard Drive key remotely

- Go to a computer where Pipe2018 is installed. If you have your own USB device you would like to use to hold a license key, plug it in first.
- Go to Pipe2018 in the Start menu, run Remote Update. Select 'USB Key (not Wibukey)'. Specify the drive letter where the USB device is located or where the hard drive key location is mapped for that computer. Click OK.
- Next under the 'Get USB License" click the button "Generate an Activation Request Code" and **send it to KYPipe**.
- Keep the application openand we will send back an Activation Code which will create your Key when you paste it in and click "Activate".

After the key has been placed on the Network then each computer must run 'Diagnostic Settings' and set the Key type to 'USB' and then specify the drive letter where the key location is mapped on that computer.

**Modify Existing Key**

To modify an existing USB or Hard Drive key remotely

- Go to a computer where Pipe2018 is installed. If you have a USB device with a key, make sure is it plugged in.
- Go to Pipe2018 in the Start menu, run Remote Update. Select 'USB Key (not Wibukey)'. Specify the drive letter where the USB device is located or where the hard drive key location is mapped for that computer. Click OK.
- Next click on the tab "Change USB License". Click the button "Generate a License Change Request Code" and **send it to KYPipe**.
- Keep the application openand we will send back an Activation Code which will modify your Key when you paste it in and click "Update License".

## 1.2.3 Version Features

### Pipe2018 Features

#### Node and Pipe Graphics

**PID Symbols (Piping and Instrumentation Diagram)**

**Node Symbol Library (Glyphs)**

**3D Pipes** – shading provides a 3D appearance

**Variable Pipe Thickness**

**Analysis and Reports**

**Save Results Files** (KYPipe and Surge): After analysis, save a results file. File can then be loaded and compared with other results files.

**Pressure Sensitive Demands (KYPipe)**

**User Information Report**: In the System Data menu, feature allows the user to add information to the analysis report and create custom tables. Especially useful for NFPA 15 reporting.

**Internet Maps Feature**

**Convert Coordinates:** Converts pipes to a different coordinate system while keeping the current latitude and longitude so the pipes will appear at the same geographical location.

**Pump, Pump Curve, System Curve Options**

**Plot pump curve for Rated type pumps and single-point pump curve IDs**

**Export pump curve data to Excel (CSV)**

**Change to Series or Parallel Pump Curves Graph**: Curves for 2-9 pumps are displayed instead of 6-9.

**System Head Curves for Surge**

**Automatically Calculate Pump Inertia and File Number (KYPipe and Surge):** for File Pumps and File/Table Pumps

**TranSurge**

**Improved Excel Profile Import Tool (see TranSurge Manual)**

**Labels, Contours, and Emphasis**

**Multi-line Pipe Labels (KYnetic):** One item per line instead of everything on one line.

**Isolation Zone Emphasis and Isolation Zone Report**

**Loss-Loss/1000 option for Pipe Emphasis and Labels**

**Compare Two Parameters for Node Emphasis/Contours**

**Map View Features and Controls**

**Plan View -> Profile View:** Allows the system to be seen in a side view for any defined profile. System data and results are still accessible but editing is not allowed. Fast switching between plan and profile view.

**Copy Node Changes to Legend:** Creates a map legend with the current list of changes for one node or all nodes

**Zoom Lock (KYnetic):** Right Click Zoom Windows or Ctrl-Shift-W

**Increased number of Saved Views from 7 to 20**

**Display all Results in Info Windows for Nodes (Pressure, HGL, Head, and Flow) and Pipes (Flow, Velocity, Loss, Loss/1000)**

**Interface**

**Recent Folder Option in the File Open/Close Menu**

**Sort Items Descending in Node Group Select (KYnetic)**

**Animated Profile Options**

**Animated Profile Export to EXCEL:** exports Node Name, Elevation, Distance, HGL, Pressure, Min HGL and Max HGL to EXCEL.

**Case C option**

**Cavitation Emphasis**

**Simple Mode: Symbols On/Off**

**Show Axis Lines**

**Box Around Graph**

**Display X-Coordinate**

## Pipe2016 Features

Air Valve Placement Tool (Surge Only)

New DXG/DWG display option - supports more recent file formats

Google Earth Export (KYnetic only)

Google Earth Import (KYnetic only)

Travel Distance - displays the travel distance from a selected node to all other nodes in the system (KYnetic only)

Travel Time/Path - shows animation of the travel time from a selected end node to all reachable nodes in the system (KYnetic only)

Emphasize Pipes as Nodes - emphasizes pipe with colors and values from Node Contours and Node Emphasis

Generate Change Pattern - generates a change pattern for a selected group of nodes in which each on/off combination is included (KYPipe, Gas and Steam; KYnetic only)

Check Valve Dynamics Tool (Surge Only)

Snap Hydrants, Valves, and Meters - connects floating nodes to the nearest pipe

Connect Crisscross Pipes - connects disconnected pipes (KYnetic only)

Use 'Classic' Main Menu in KYnetic (KYnetic only)

nterpolate Elevations - use existing elevation data to update nodes with missing elevation data.

New universal features:

Quick Profile on Longest Path - alternative to standard Quick Profile.

Google Map of Current Area - Opens Google Maps in web browser at the location of the model.

Modified Hazen Williams Equation - Recommended by the Indian government agency CPHEEO.

Head Loss Factor - will increase/decrease computed frictional headloss in a pipeline by Headloss factor

# Pipe2014 Features

1) Detailed PPT/WORD/HTML Presentation Generator (KYNetic)
2) Ability to use PDF as a background image
3) Google maps and Google elevation data for Internet Maps
4) Specific coordinate systems (UTM, State Plane) for Internet Maps
5) More than 75 Fittings allowed
6) Improved material Inventory/cost feature
7) Driving directions for nodes (KYNetic)
8) Surge Pipe Break analysis
9) SCADA menu (KYNetic)
10) URL association (KYNetic)
11) Improved contour resolution and accuracy
12) Polygon Based Automatic Demand Distribution13) For KYPipe, Overflowing, One-Way and Non-Draining Tank options.

## Pipe2012 Features

Internet Maps and Elevation Retrieval

EPA Surge

Siphon Breaker

Dynamic Air Valve

Electronically Activated Surge Anticipation Valve

Horizontal Closed Surge Tanks

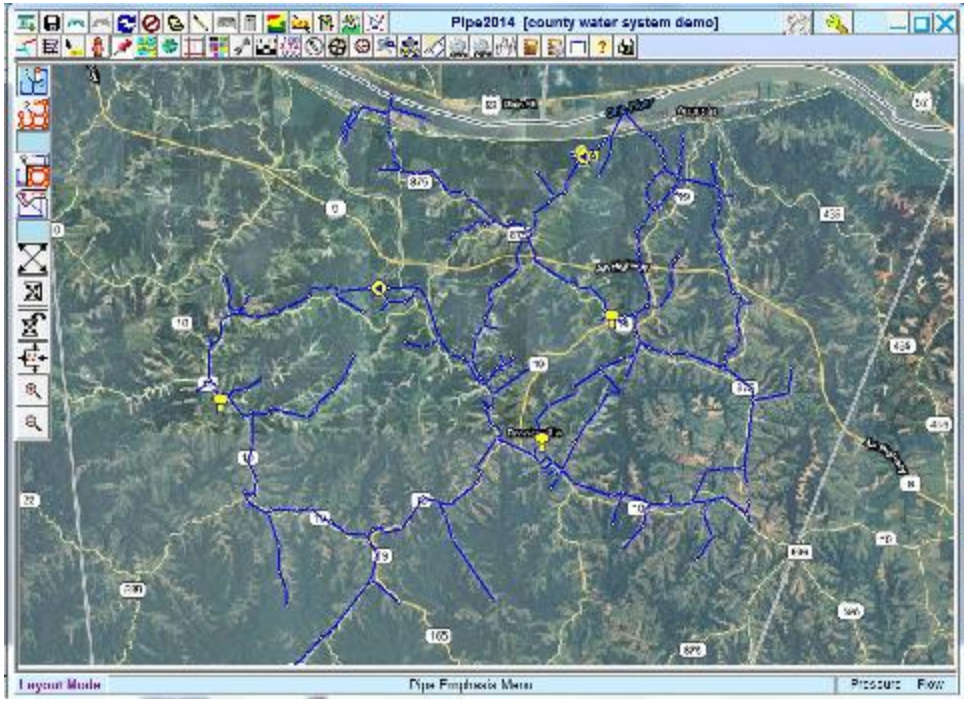
Holding Tank

File/Table Pumps

Check Valve Element

Pipe Crossover

Fittings Table




KYnetic Interface - All New, Redesigned KYnetic Interface Features includes

Water Quality Sensor Tool

Pipe Intrusion Calculator

Polygon Area Select

Multiple Languages

Advanced Display Features

Comprehensive Editing and Layout Tools (Group Edit)

Improved Contouring

## Pipe2010 Features

Optimal Pump Scheduling Tool - Optimize pumping operations and cost over time.

Optimal Design Module for KYPipe. Optimal selects the correct for any portion of a system.

Automatic Demand Distribution - Distributes total residential demand throughout the model automatically

PDF output for all Pipe2012 graphs, tables, maps, reports and profiles. Create paperless reports.

JPG output for most graphs, tables, maps, reports and profiles.

Animated Profiles for KYPipe, Surge, Gas, Steam and SWMM. Depicts flows and devices operations. Save AVI files.

New Elements:

- Constant Pressure Pumps, Constant Flow Pumps
- Pressure Reservoirs, Depth Reservoirs (for KYPipe and Surge)
- Hydrant Monitors - analyze hydrants for up to three valves open
- Active Valve for Gas
- Hydraulically Actuated Surge Anticipation Valve (Surge)

Expanded Rural Analysis for KYPipe

Text Labels may be rotated to any angle

Cursor Notation - Hover the mouse cursor over any pipe or node and a pop-up will display critical information and analysis results (example below).

# Pipe2008 Features

Pushpin - Ability to attach Notes to the map or to any node (All Engines).

Pipe2008: SWMM (Stormwater Engine) is available. Analyzes storm water and sanitary systems for pressure and partial flow scenarios using the latest EPA SWMM 5.0 engine.

Units help hover buttons. Just move the mouse over a word like 'Diam' and a hint will pop up that shows what units it is in. (All Engines)

LPS Tanks with pressure switches and inflow demand pattern (KYPipe and Surge)

Hydropnuematic tank - pump and tank combined uses air pressure to supply pressure to move water out of tank, when low add water to build up pressure. (KYPipe and Surge)

Demand patterns for all tank inflows (KYPipe and Surge)

CAD layers in backgrounds - Enhanced support dwg as background (All Engines), ability to turn layers on and off.

Add/remove check valves to anything with group edit. (KYPipe and Surge)

Copy/paste pump IDs and tank shape IDs (All Engines)

Group edit/select multiple and regular junction demands and multiple and regular demand types.

When laying out a system, right click adds node, adds pipe and changes in-line node to intermediate. Right or left click a second

time (same location) node will not be automatically made into intermediate node if it is in-line. (All Engines)

Improved print backgrounds (new method of 'Lighten' is slower, less pixilated /smoother and old method is faster, more blocky) (All Engines)

Calibration wizard (KYPipe)

250 sets of results available for Gas and Steam (Gas, Steam and SWMM)

Pipe2006/2008 New gradients on contours (All Engines)

Pipe2006/2008 - Contours may be accessed from main menu (All Engines)

Pipe2006/2008 - For Multiple Demand Types, displaying Demand labels on the map shows total of all demands, displaying both Demand and Demand Type labels shows a list of demands and types (All Engines)

Under Edit | Copy Map to Clipboard – identical to screen capture, but no bmp is created, just copied to the clipboard. Prompts user for size and orientation.

Save report as doc file.

Edit | Copy can now copy/paste part of the report.

Significantly reduced file size.

NFPA color coding – after running a hydrant analysis, go to node emphasis click button that says "NFPA Hydrant color coding" Or

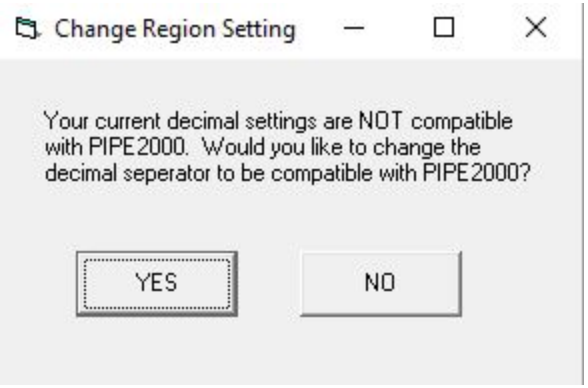
you may hit ctrl – Alt – H, or to go to Labels | Results A or B | Fireflow/Static/color code NFPA. Emphasizes the hydrants.

EPANET export without having to run Quality analysis.

## 1.2.4 International Decimal Settings

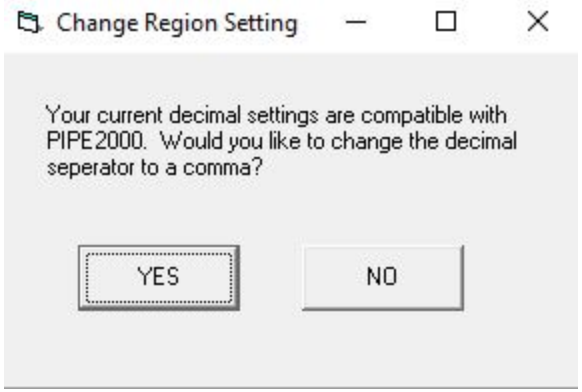
A comma "," decimal is not supported. If the decimal setting is incompatible, these messages will appear upon launching the Pipe program:

To correct the settings, run the International Decimal Settings utility.




Answer yes. You will get this reminder:

When you run International Decimal Settings utiliyt and if settings are correct, you will see this:




Answer No to use Pipe program. If decimal is to be changed back to a comma after use of Pipe program is complete, answer Yes.

## 1.4 Pipe2018 - All Models

### 1.3.1 Help and Tutorials

#### 1.3.1.1 About the Online Help

About the Online Help

Back to Contents

Utility Programs

---

Extensive information on Pipe2018 modeling and model development can be accessed online by selecting items from the Contents page or the Index. The following topics provide basic information on model development:

- Network Elements
- Laying Out a Piping System
- Quickstart Example

Details on the Pipe2018 Graphical User Interface (GUI) are included in:

- Information Windows

Access to data requirements is provided through:

Data Requirements

Details on the purpose and how to use various auxiliary modules is covered in:

Utility Programs

The pipe network modeling reference manual is included as part of the online information:

Reference Manual

A number of additional assorted topics are covered in the Online Help. These include:

Background Images

Demand Allocations/Meters

Some Special Features

Pipe2018 Presentations

Network Analysis

Data Files/Scenario Management

Data Tables

Sets and Group Mode

Advanced Capabilities

## 1.3.1.2 Overview and Getting Started

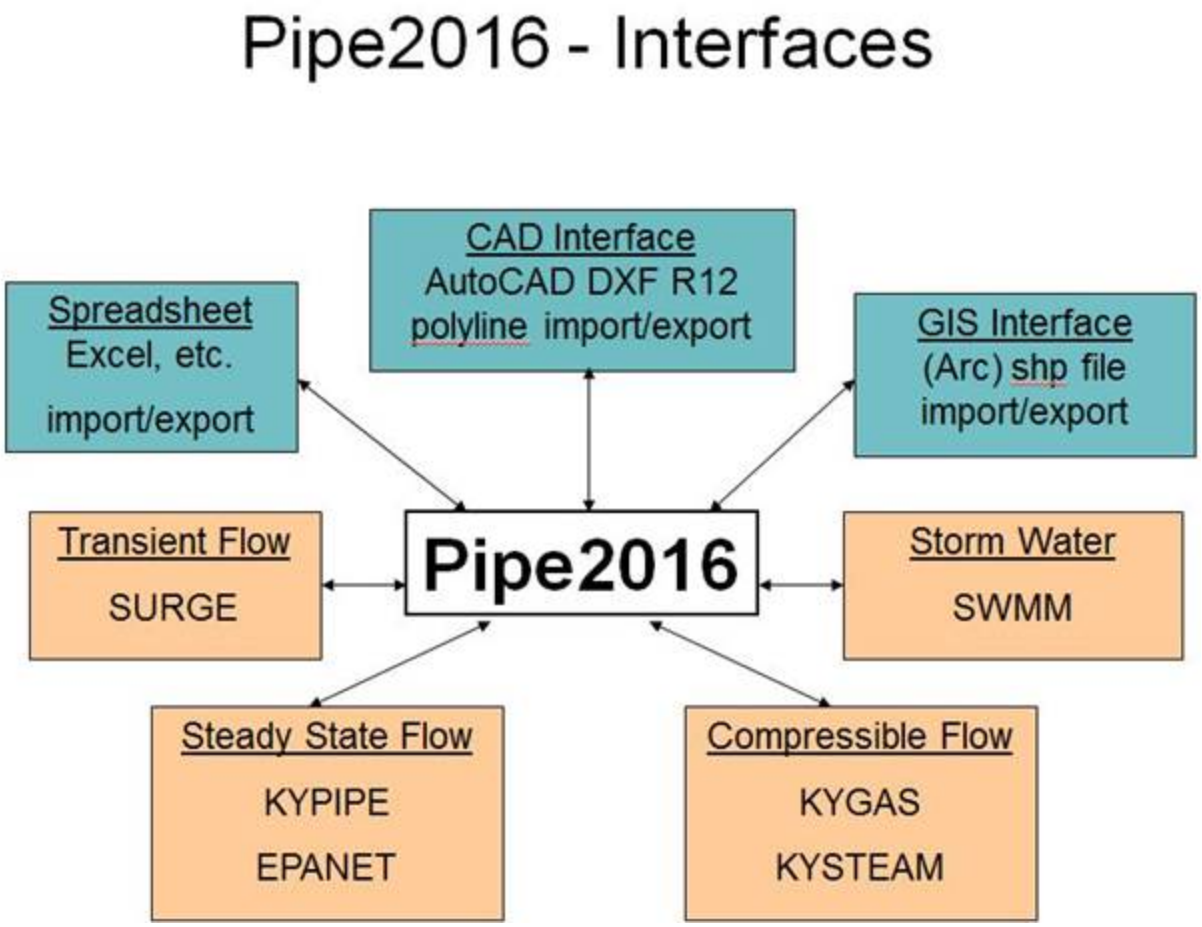
Overview and Getting Started

---

Pipe2018 is a powerful graphical user interface for creating comprehensive pipe system models, accessing and running associated engineering analysis engines and presenting results in a variety of ways. The models are entirely made up of pipe links, end nodes and internal nodes. Using this approach only a few simple steps are required to develop and modify pipe systems and define the associated data.

Pipe2018 can input a background map and drawings in a variety formats. In addition scaled grid lines may be used. Using a scaled background map or grid lines will allow pipe links to be precisely scaled (length calculated) as they are created.

A wide variety of pipe distribution system devices are supported and users can maintain an associated extensive table of data and records which can be customized to their specifications. In this way Pipe2018 can serve the multiple purposes of providing and maintaining extensive GIS records, generate up-to-date data files for hydraulic and water quality models and provide facilities management capabilities. The chart below illustrates some of the capabilities Pipe2018 incorporates.




**Audio/Video Tutorials (AVI's) and the Help File**

There are over 40 audio/video tutorials which address all aspects of using Pipe2018. The tutorials are contained on the KYPipe web site or USB device that comes with a Pipe2018 package**.** In addition, the Help File is accessed from the Pipe2018 main menu under **HELP**. This provides extensive information on modeling and the Pipe2018 environment. You can access topics from the **Contents** or specific items from the **Index.**

**Getting Started**

Insert **USB Device** license key 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 **Tutorial and Setup** application. When the Tutorial Subject menu pops up select Pipe2000: KYPipe, Surge, or the subject appropriate to you.

Some users may wish to use the **Select Video** button to jump to the topic of choice. Most users should watch the first ten videos in order.

After completing the first nine videos find the **Quickstart Example** in the Example Manual (this may be downloaded from the KYPipe web site). Refer to this while you watch the next four videos **Quickstart Example 1** through **Quickstart Example 4**. Once you have run these 13 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 in the Examples Manual. Copies of the model data files are included in the **Demo** subfolders. KYPipe users should change the tutorial subject to **Pipe2000 : KYPipe 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**.

## 1.3.1.3 Quick Start

Quickstart Example

Step 1 - Initial Preparation

Step 2 - System Layout

Step 3 - Analyze System and Review Results

Step 4 - Some Additional Simulations

See the Quick Start videos on the KYPipe web site.

This will guide you through the complete layout development, data entry and hydraulic analysis of a simple pipe network.

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. Specify the flow units.

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

Click on Return to Map icon or Map tab (Classic) to return to the Pipe2018 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 Pipe2018 calculate pipe lengths.

Click on Grids icon or Map Settings / Grids (Classic) - 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.

Return to the Pipe2018 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.

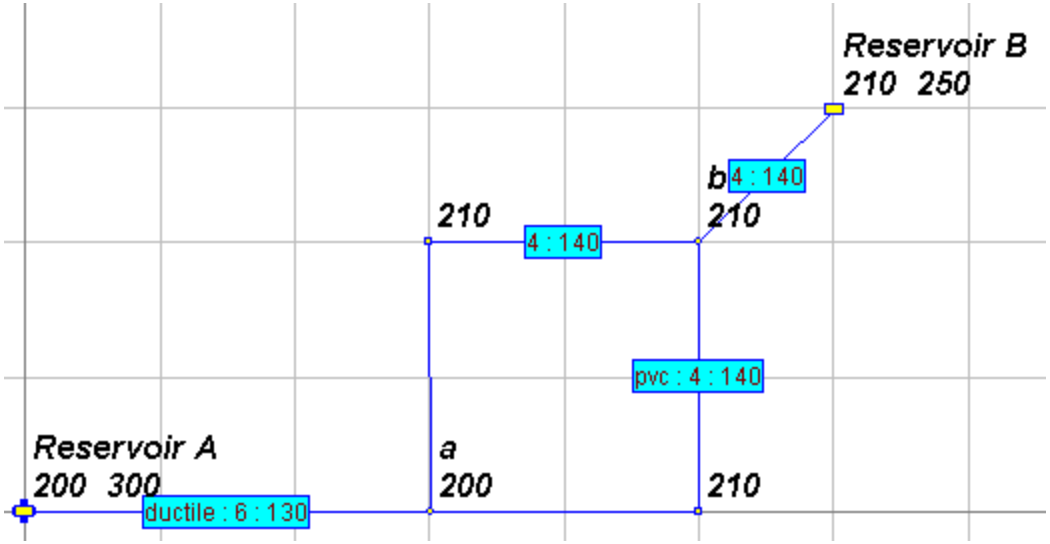



Figure 1 Example pipe system

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 Pipe2018 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, in the Node Info window, click on the Node Type cell or drop down node list (Classic) 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.

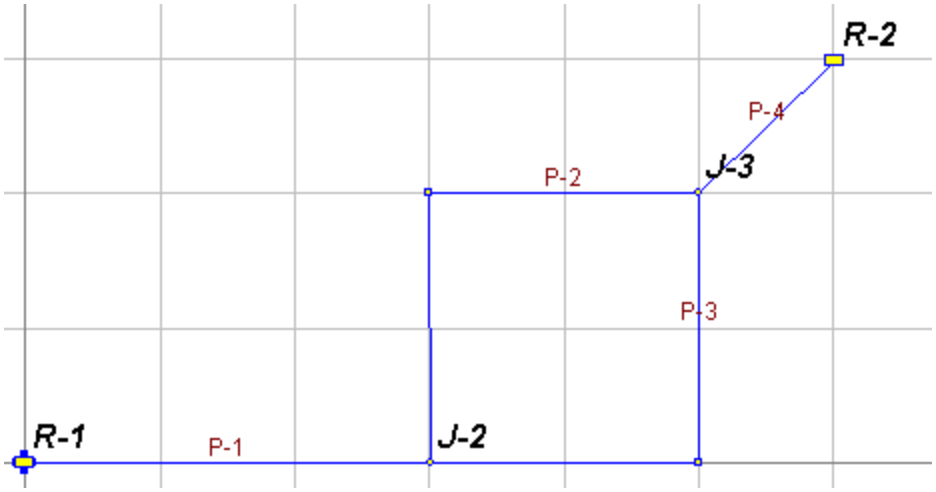



Figure 2 Completed pipe system layout

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 and provide 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 Report 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 Result A and Node Result A to show the results depicted in the Results Selection bar on the bottom right of the screen. 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).

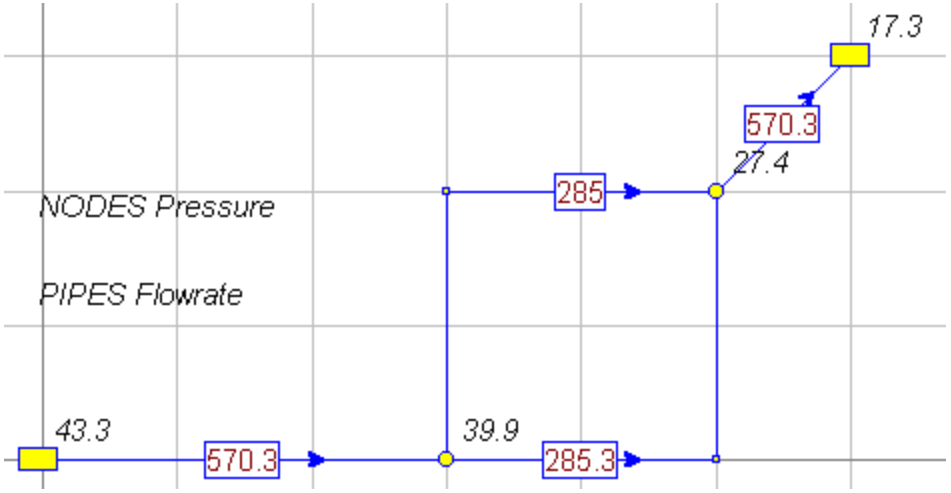



Figure 3 Case 1 - Pressure and Flow

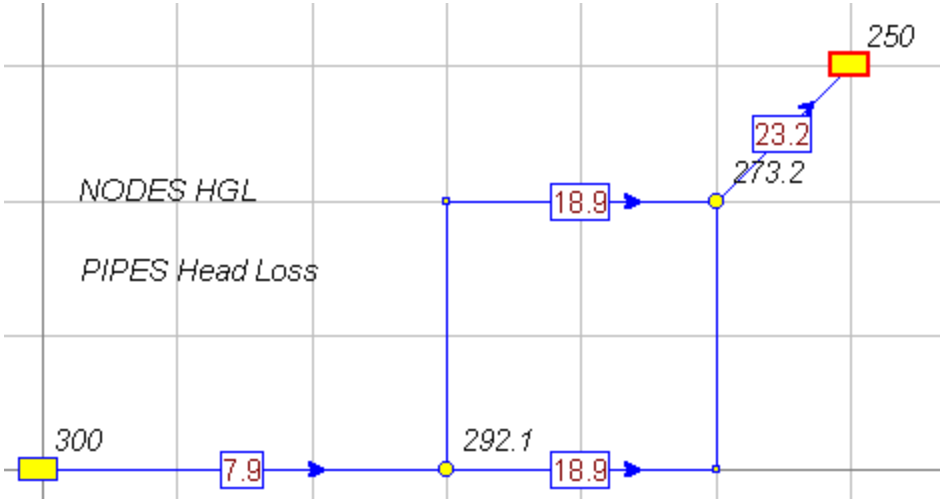



Figure 4 Case 1 - Loss and HGL

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). See Age-Based Roughness.

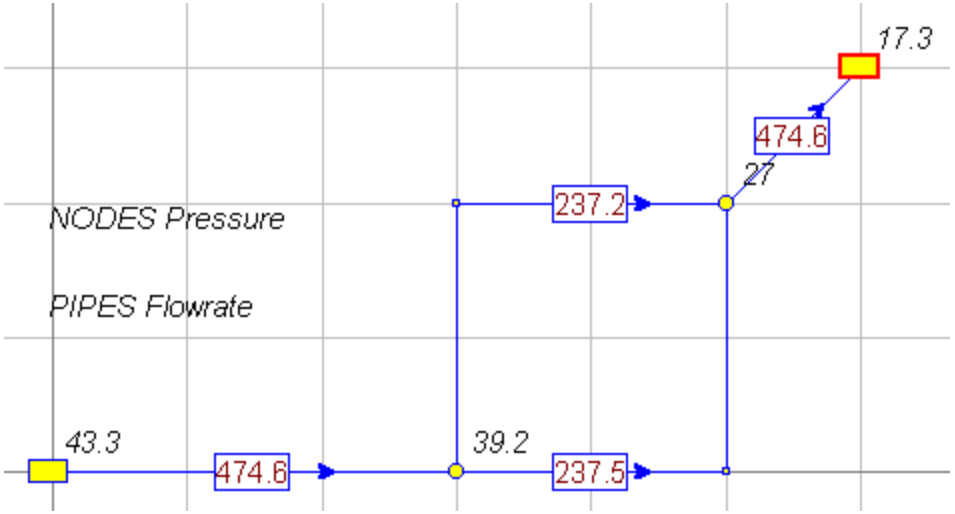



Figure 5 Case 2 - 25 years (2023)

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).

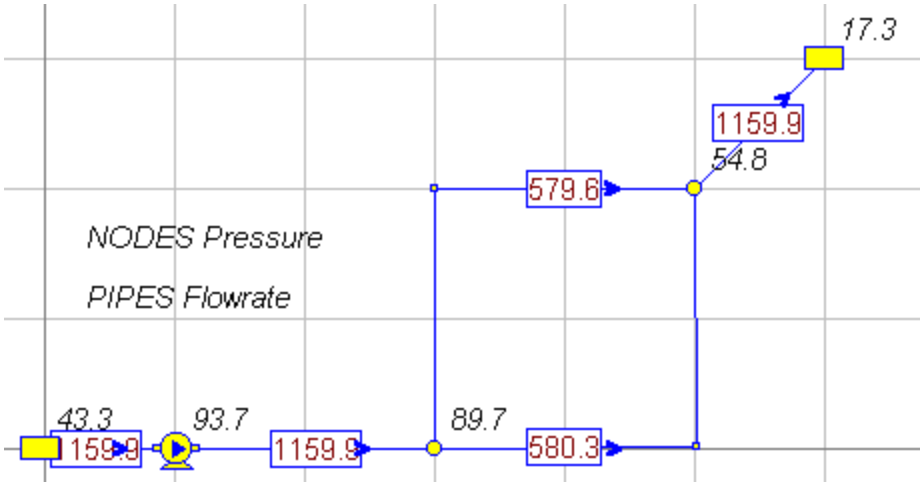



Figure 6 Case 3 - Added Pump

## 1.3.1.4 KYPipe and Surge Demo files

KYPipe and Surge Demo files

- **KYPipe - Regular Simulations**
- **KYPipe - Extended Period Simulations**
- **KYPipe - Other Capabilities**
- **Surge**
- **Surge Protection**
- **KYPipe - Optimized Calibration**
- **KYPipe - Water Quality**

---

KYPipe and Surge Demonstration Files

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 KYPipe for regular and extended period simulations and Surge for transient 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.

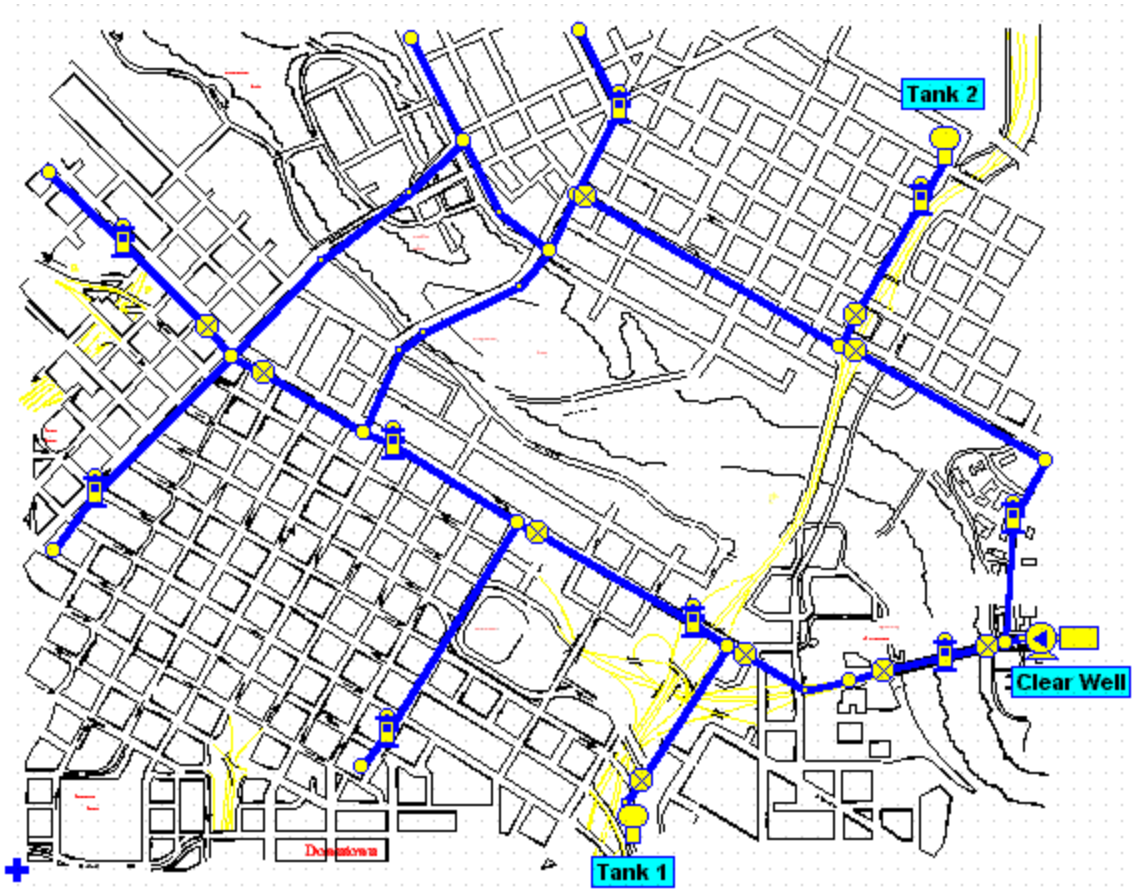



Figure 1 Demoreg

KYPipe - Regular Simulations

Click on ***File*** (Main Menu) and ***Open*** and select the file ***Demo Model*** (in the demo subdirectory) using the browser. You should get the pipe system and map shown in Figure 1. The ***Demo Model*** 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 specified at Junction J-13

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

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 KYPipe 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.

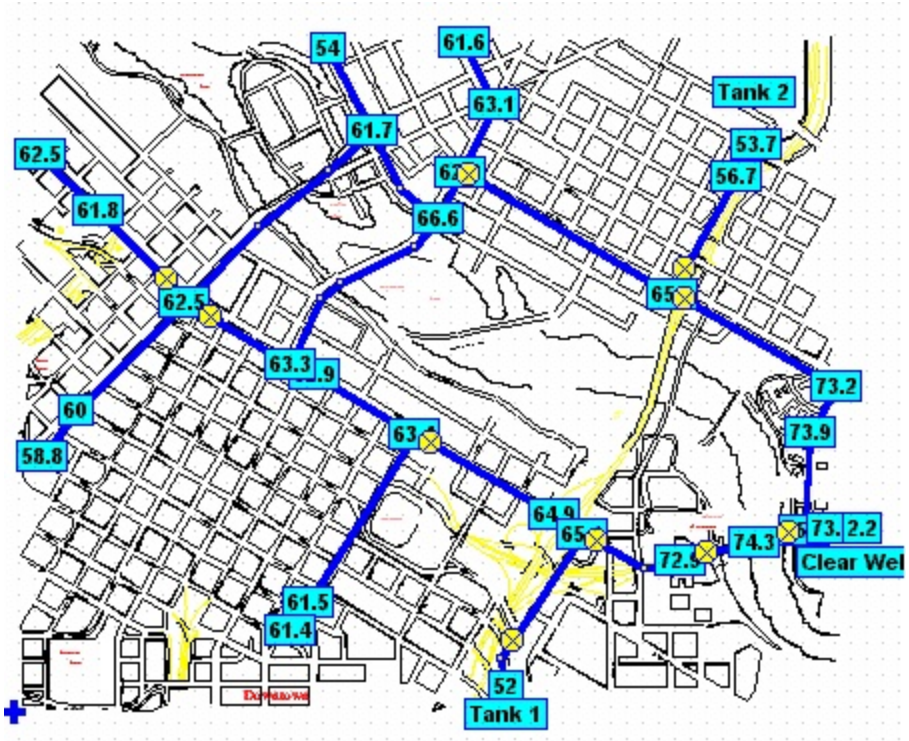



Figure 2 Results Labels, Case 0

**2) Contours*:*** Contours are very effective means for showing results. Show pressure contours for Case 2 to illustrate this feature. Make sure Pressure is selected for Node Results and Case 2 in the ***Results Selector*** bar at the bottom right corner of the Program window. Click on ***Emphasis/Contours*** icon or ***Map Settings*** and ***Emphasis/Contours*** (Classic) and select ***Pressure*** (parameter). The contour values should be set at 20, 30, 40, 50, 60. In Classic, check the ***Show Contour*** box and go back to the ***Map***. The pressure contours should be displayed (if not, click the ***Refresh*** button). Figure 3 shows this display.

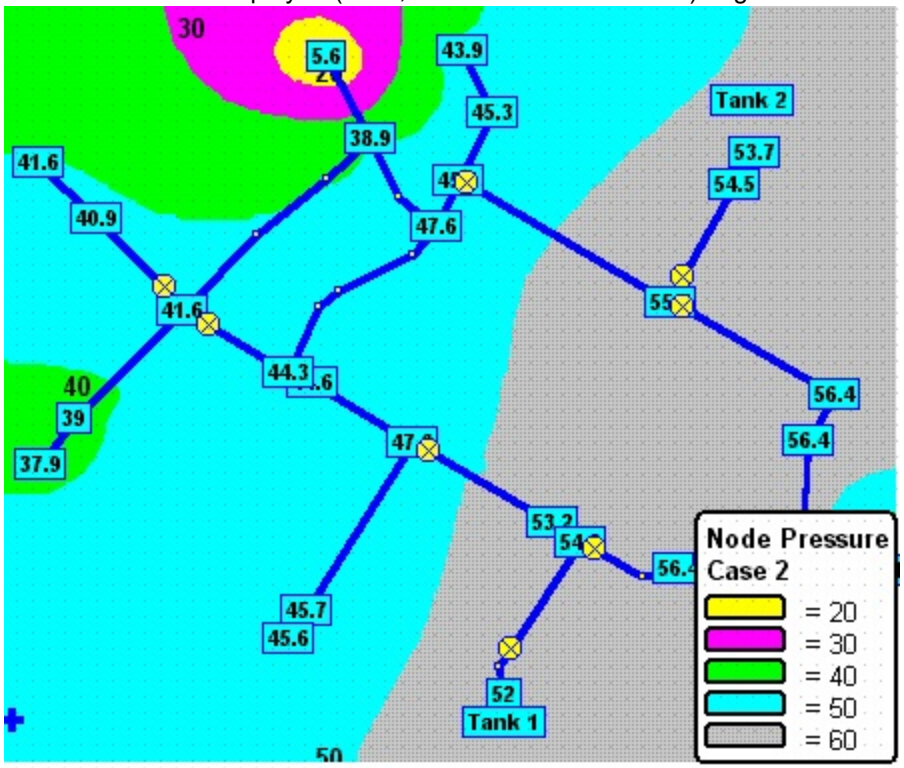
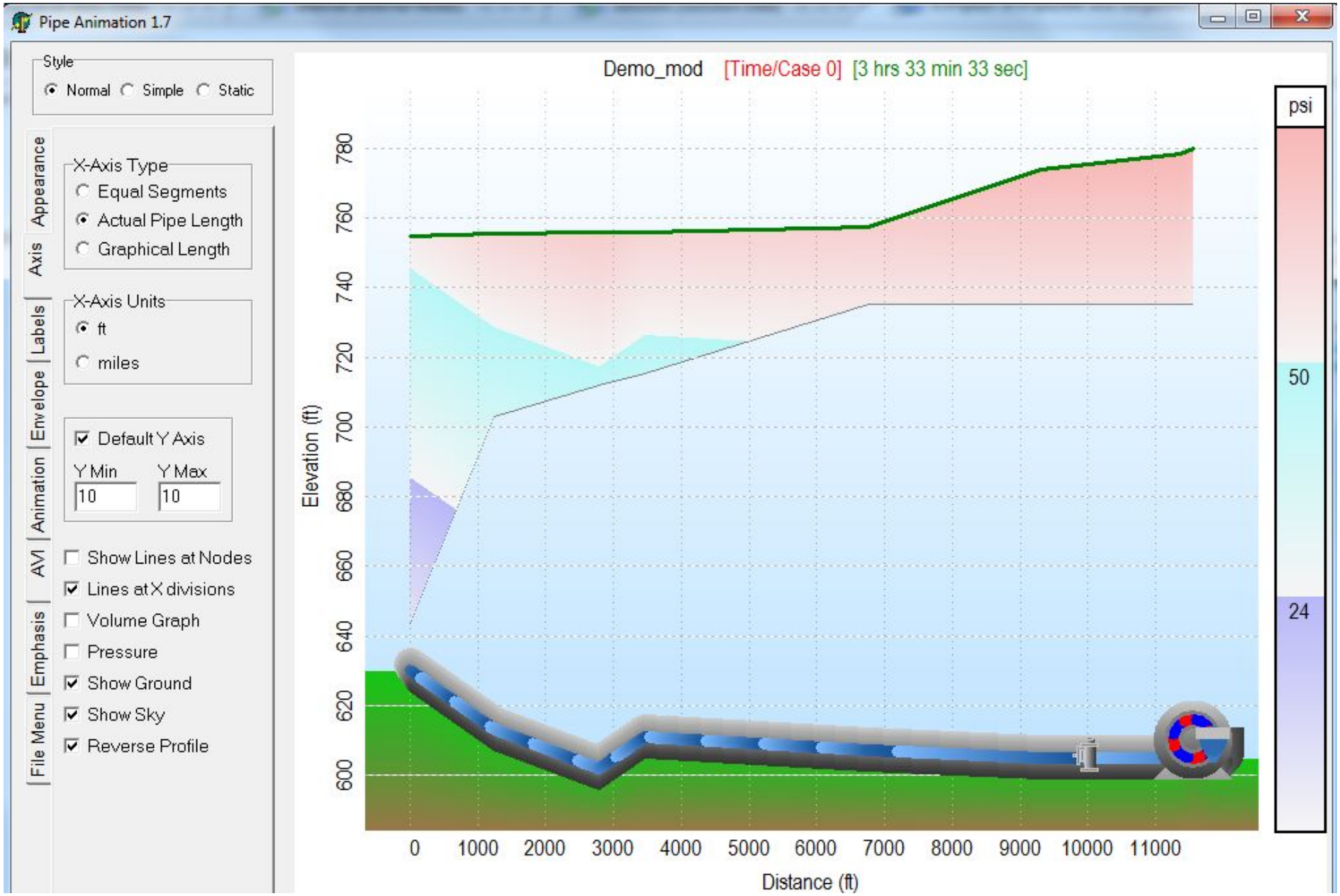
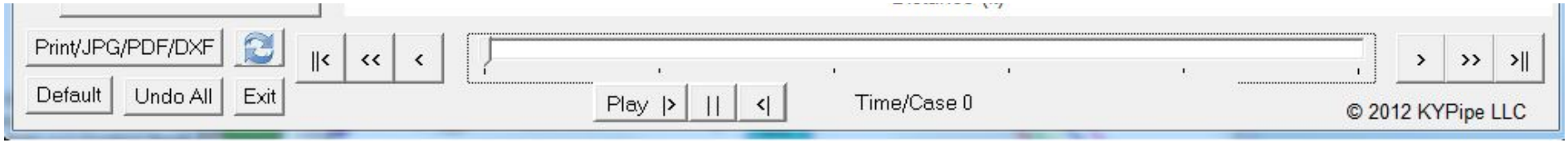



Figure 3 Contours, Case 2

**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.

Animated Profile


Pipe Animation 1.7
Style
Normal
Simple
Static
Appearance
Axis
Labels
Envelope
Animation
AVI
Emphasis
File Menu
X-Axis Type
Equal Segments
Actual Pipe Length
Graphical Length
X-Axis Units
ft
miles
Default Y Axis
Y Min
Y Max
10
10
Show Lines at Nodes
Lines at X divisions
Volume Graph
Pressure
Show Ground
Show Sky
Reverse Profile
Demo_mod [Time/Case 0] [3 hrs 33 min 33 sec]
psi
50
24
Elevation (ft)
600
620
640
660
680
700
720
740
760
780
0
1000
2000
3000
4000
5000
6000
7000
8000
9000
10000
11000
Distance (ft)




Static Profile:

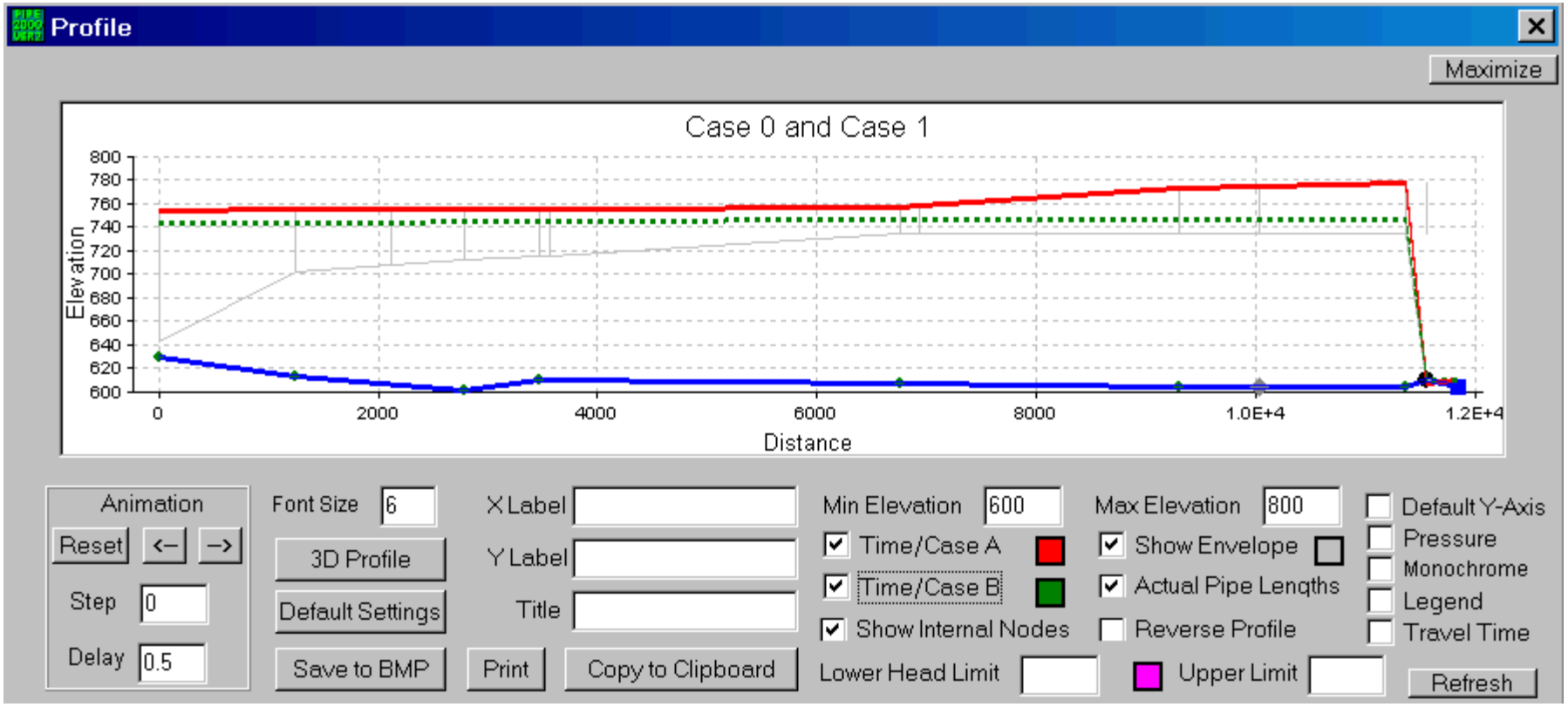



Figure 4 Profile, Case 0

**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 the Demand Pattern icon or ***Setup/Default*** and ***Demand Pattern (Classic)***. 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 EPS icon and Control Switches or ***Other Data*** and ***Control Switches*** (Classic) to see this setup.

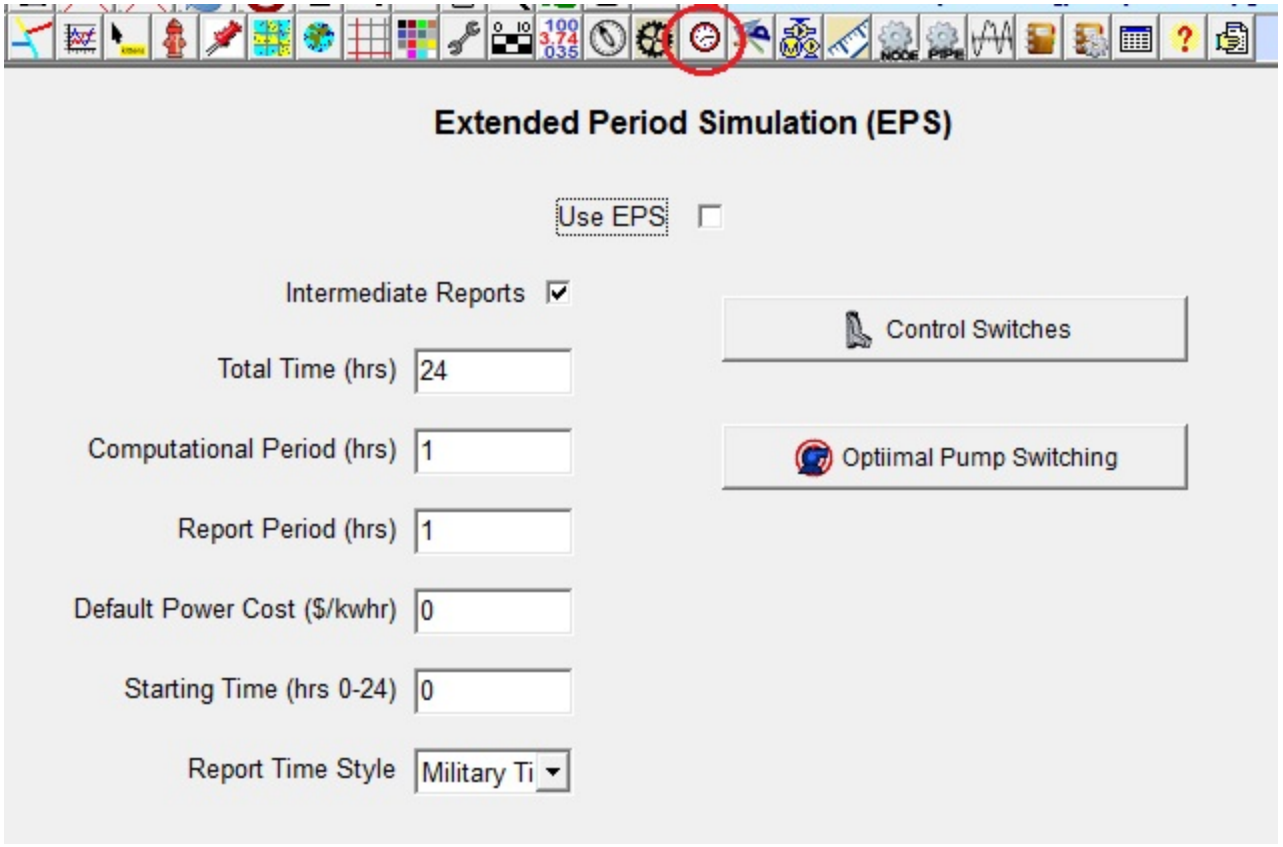



Figure 5, System Data / EPS

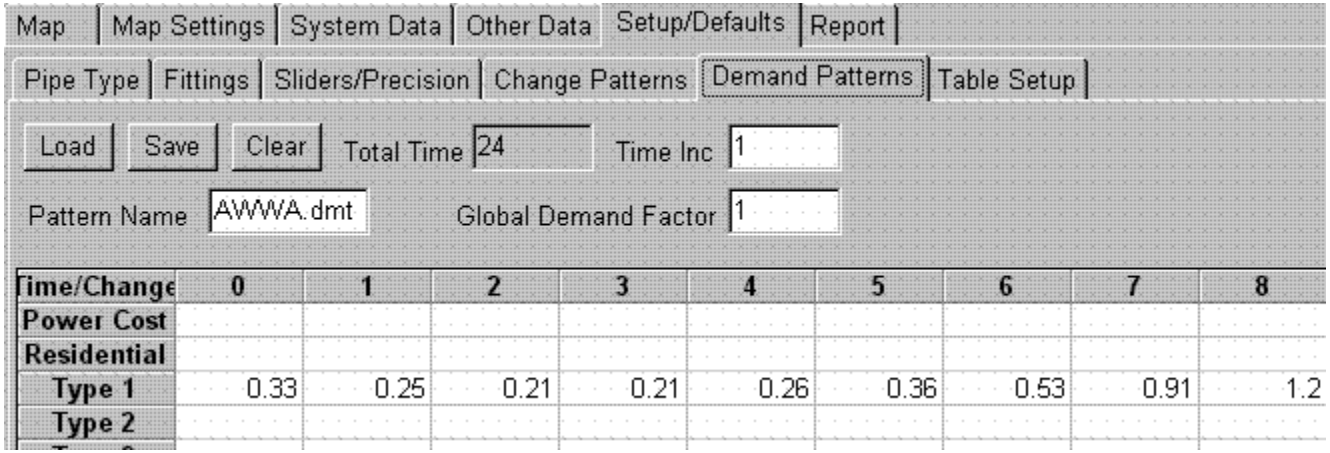



Figure 6 Setup/Defaults / Demand Patterns - AWWA demand pattern

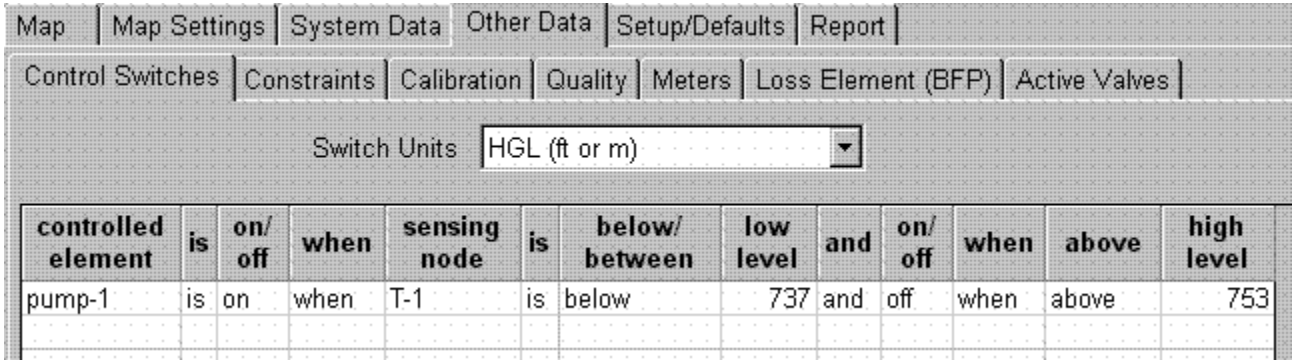



Figure 7 Other Data / Control Switches

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. In the Classic interface, these are 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 use the ***Group*** button to select multiple nodes, 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.

KYnetic Graph and Table Results icon:

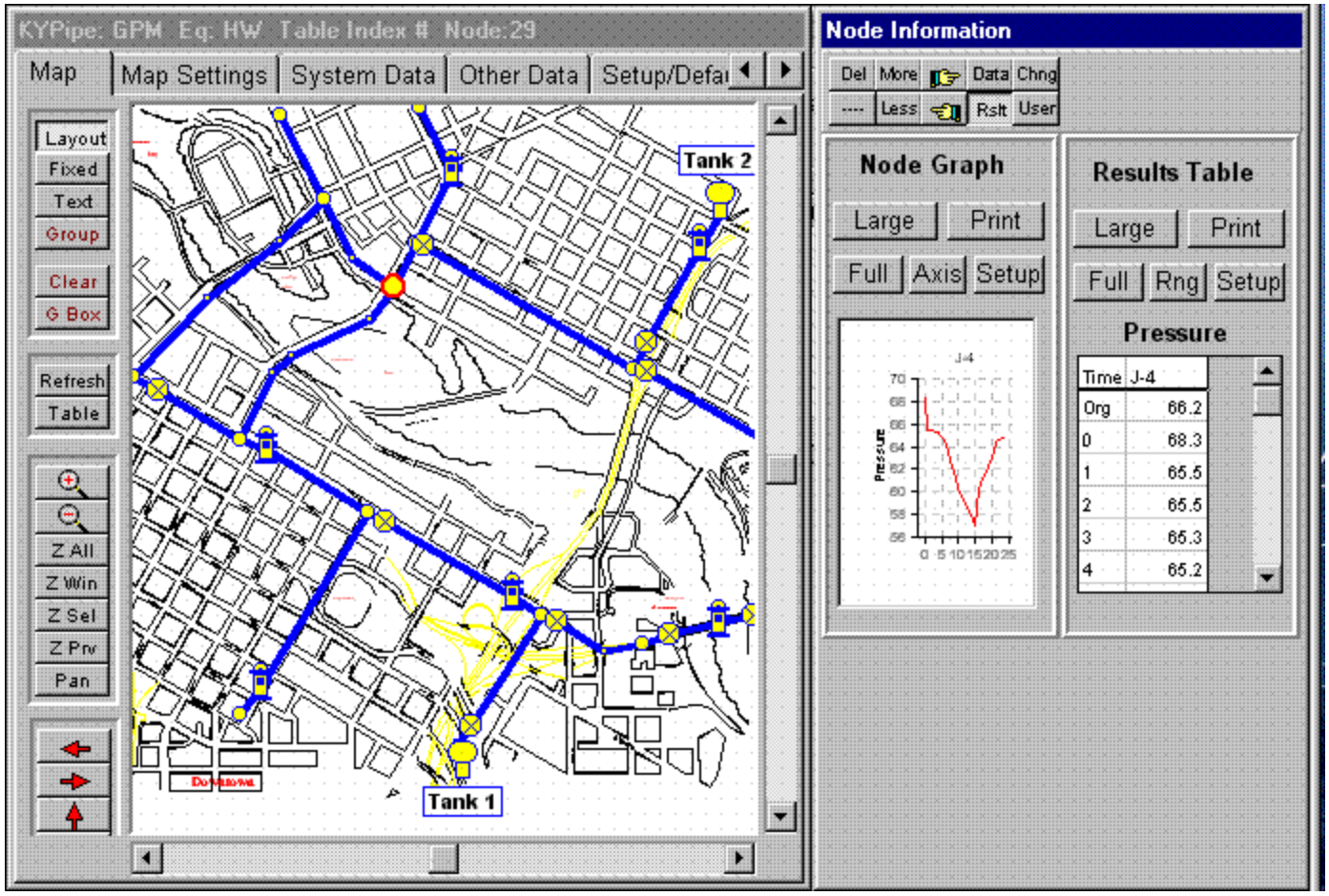



Figure 8 Results Table and Graph for Selected Node

**KYPipe - Other Capabilities**

The demonstration files may be used to demonstrate other Pipe2018 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. KYPipe 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 Other Data icon or ***System Data*** and ***Other*** (Classic). 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. Go back to 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 (upper left corner of the Node Information Window in Classic). 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, check the **System Curve** box to display the system curve. To display pump curves also, use the drop-down selectors at the bottom of the window. 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.

KYnetic Other Data menu icon:

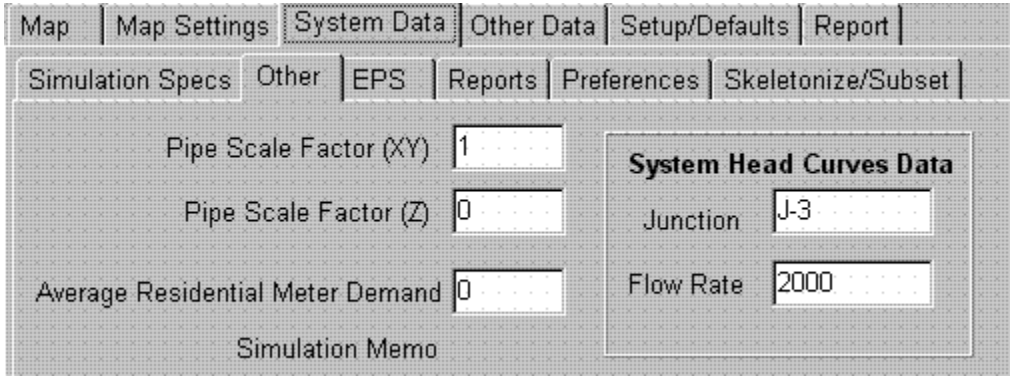



Figure 9 System Data / Other - Setup for System Head Curves

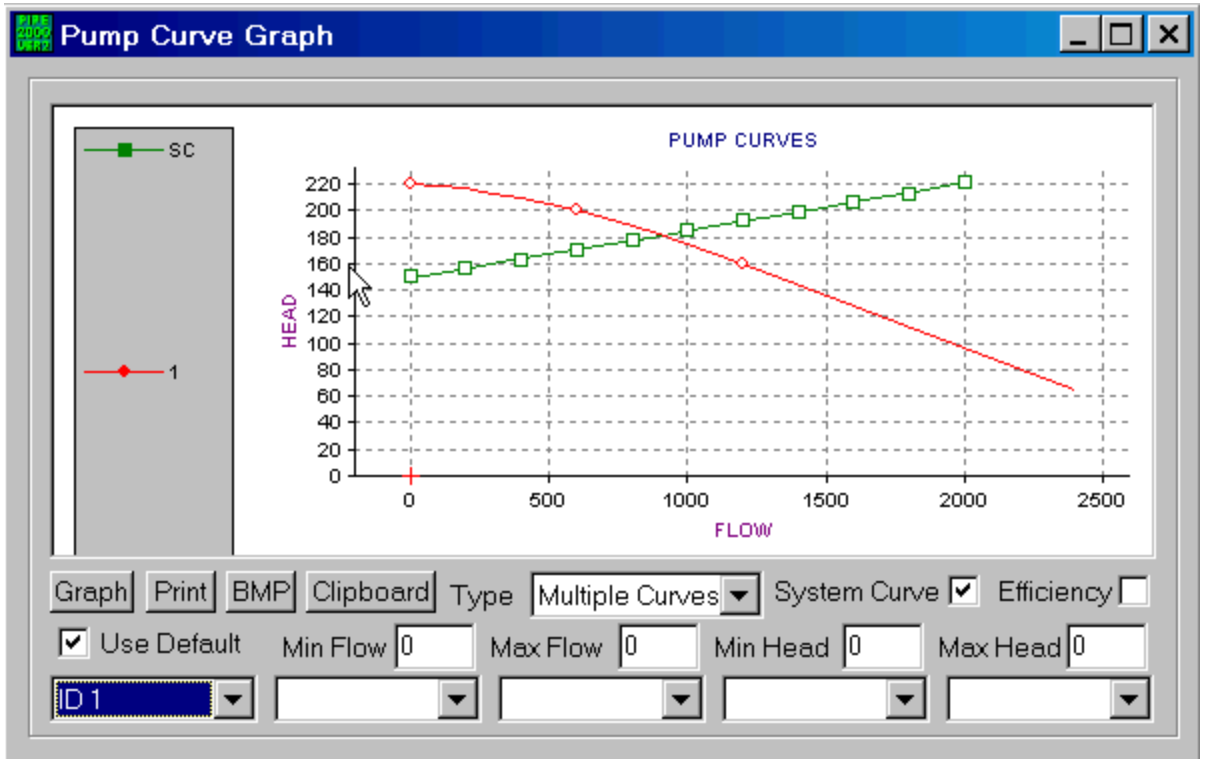



Figure 10 - System Curve

**On/Off Valves - Pipe Break*:***

Pipe2018 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 (upper left corner of the Node Information Box in Classic). 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 select ***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***.

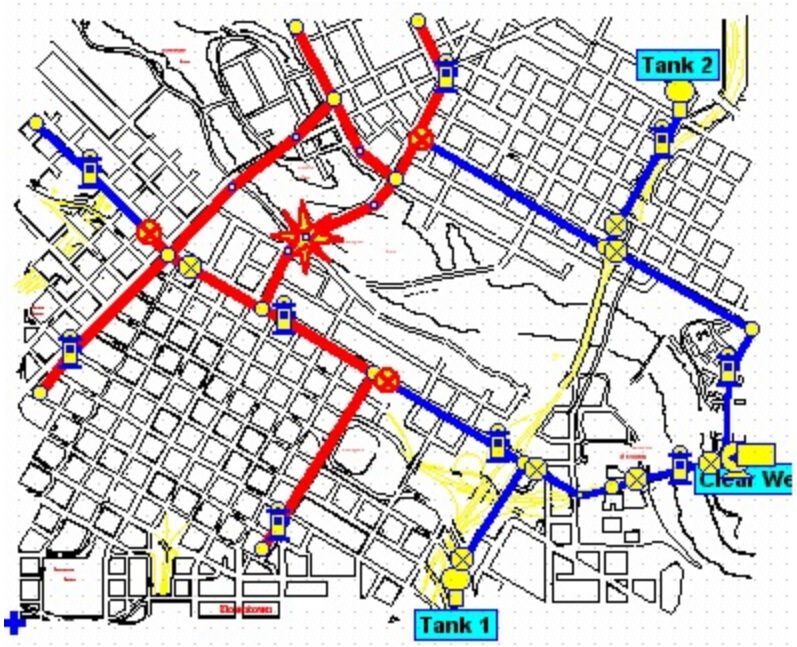
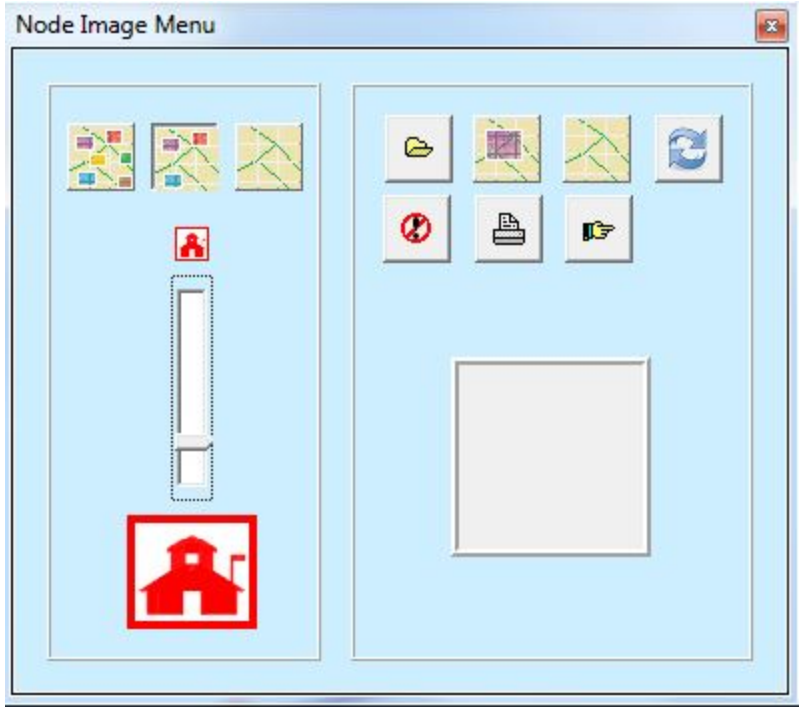



Figure 11 Pipe Break Simulation

**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 ***Demo Model***. In Layout mode, click on the valve in the upper center of the system. In Classic, click ***Full*** in the ***Node Title*** box on the left, in KYnetic click this Image menu icon and you will see a hand drawn sketch showing the valve location in the field. Click on the valve just to the left of the 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.

KYnetic Image Menu icon:




Classic Data box with Image information:

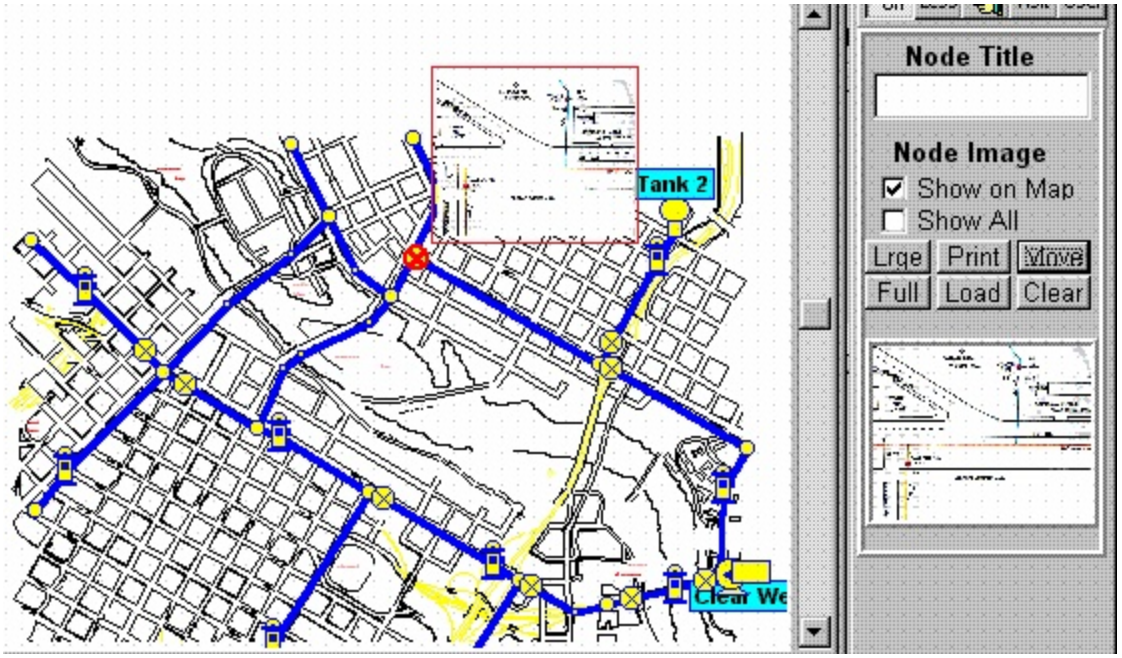



Figure 12 Node Image - Valve Map

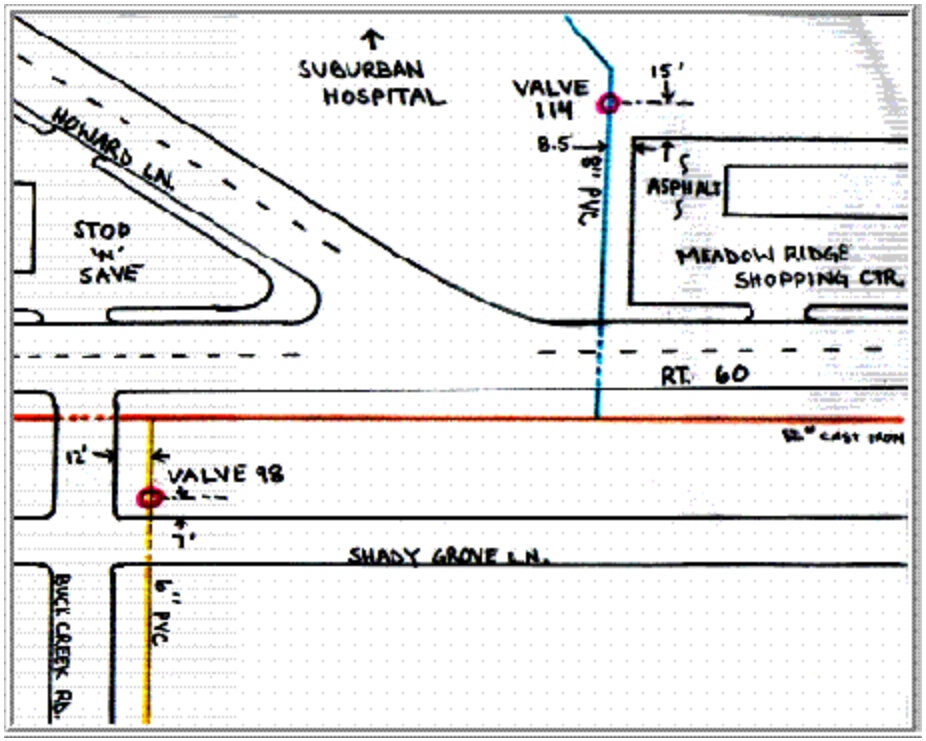



Fig 12a Valve Map - Large Size

**Hydrants:**

Pipe2018 models may include fire hydrants and have 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 ***Demo Model*** file. If you don't see the hydrants, click ***View*** menu 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.

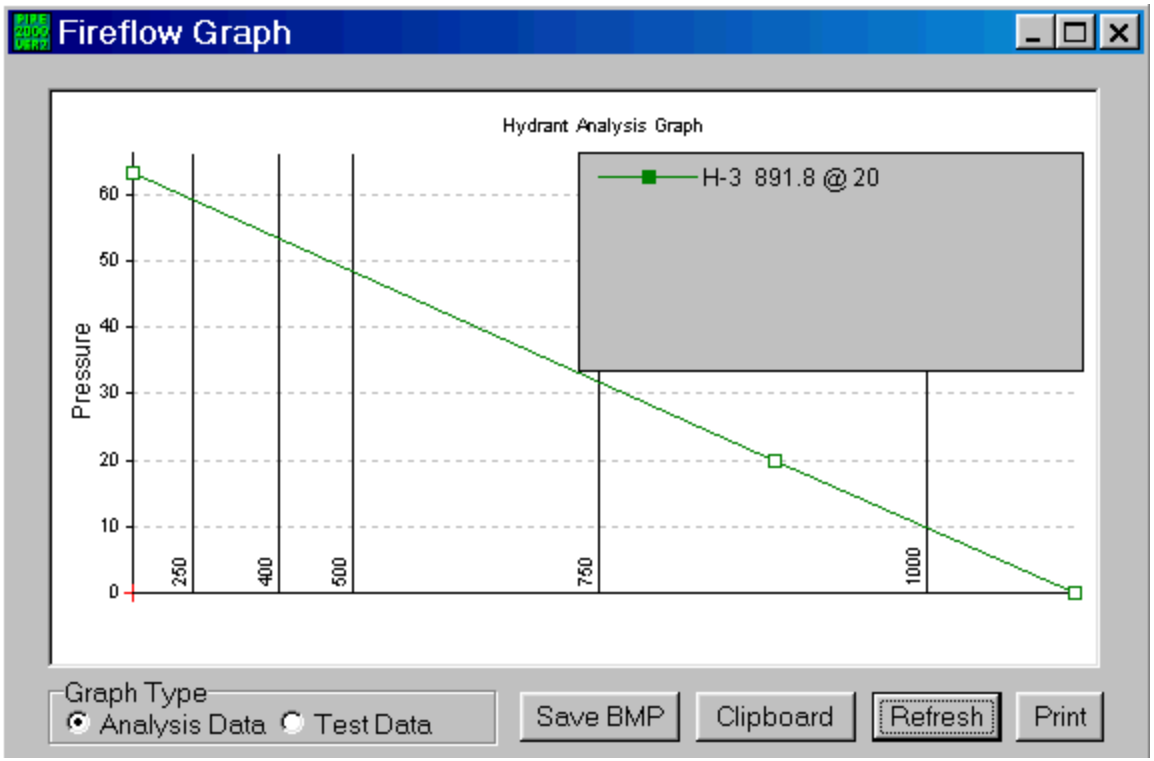



Figure 13 Hydrant Graph

**Calculation Year (age based roughness):**

Pipe2018 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 Components menu or ***Setup/Defaults*** and ***Pipe Type*** (Classic). 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.

KYnetic Components menu icon:

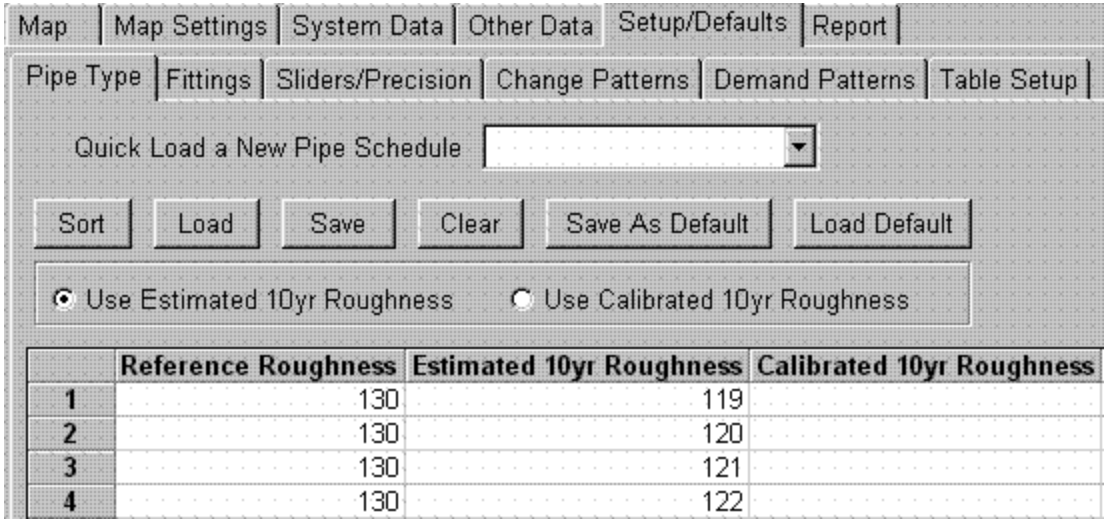



Figure 14 Pipe Type Table Showing Roughness Data

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.

P I P E L I N E  D A T A

STATUS CODE: XX -CLOSED PIPE CV -CHECK VALVE

| PIPE NAME | NODE NAMES #1 | #2 | LENGTH (ft) | DIAMETER (in) | ROUGHNESS COEFF. | MINOR LOSS COEFF. |
|---|---|---|---|---|---|---|
| P-1 | J-2 | J-10 | 2844.00 | 6.00 | 106.5534 | 0.00 |
| P-10 | J-10 | J-14 | 1649.00 | 8.00 | 108.6849 | 0.00 |
| P-11 | J-10 | J-11 | 2801.00 | 6.00 | 106.5534 | 0.00 |
| P-12 | J-12 | J-16 | 1464.00 | 8.00 | 108.6849 | 0.00 |
| P-13 | J-12 | T-1 | 2399.00 | 8.00 | 108.6849 | 3.40 |
| P-14 | J-15 | J-1 | 2550.00 | 8.00 | 108.6849 | 0.00 |
| P-15 | J-16 | J-3 | 1735.00 | 10.00 | 110.8164 | 0.00 |
| P-16 | J-4 | J-7 | 1546.00 | 8.00 | 108.6849 | 0.00 |
| P-17 | J-4 | J-14 | 2950.00 | 8.00 | 108.6849 | 0.00 |
| P-18 | J-14 | J-8 | 1942.00 | 8.00 | 108.6849 | 0.00 |
| P-19 | J-7 | J-10 | 3447.00 | 8.00 | 108.6849 | 0.00 |
| P-2 | J-3 | J-15 | 2060.00 | 10.00 | 110.8164 | 0.00 |
| P-20 | J-7 | J-13 | 1238.00 | 6.00 | 106.5534 | 0.00 |
| P-21 | Pump-1 | R-1 | 292.00 | 12.00 | 112.9479 | 7.50 |
| P-3 | J-3 | Pump-1 | 190.00 | 12.00 | 112.9479 | 6.40 |
| P-4 | J-1 | T-2 | 2337.00 | 8.00 | 108.6849 | 1.90 |
| P-5 | J-1 | J-6 | 3296.00 | 8.00 | 108.6849 | 0.00 |
| P-6 | J-6 | J-5 | 1983.00 | 6.00 | 106.5534 | 0.00 |
| P-7 | J-6 | J-4 | 678.00 | 8.00 | 108.6849 | 0.00 |
| P-8 | J-8 | J-12 | 2633.00 | 8.00 | 108.6849 | 0.00 |
| P-9 | J-8 | J-9 | 3138.00 | 6.00 | 106.5534 | 0.00 |

Figure 15 Calculated Roughness Values

**Surge**

Click on ***File*** and ***Open*** and select the file ***Pump Run Down***. This file is identical to the file ***Demo Model*** 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 KYPipe demonstration.

One additional pipe data item is required - wave speed. In Layout Mode, click on a pipe (in KYnetic click the Wave Speed icon ) and you will see the wave speed 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*** icon

(and ***Simulation Specs*** in Classic). 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***. 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*** in Classic) 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 appears 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.

KYnetic Graph and Table Results icon:

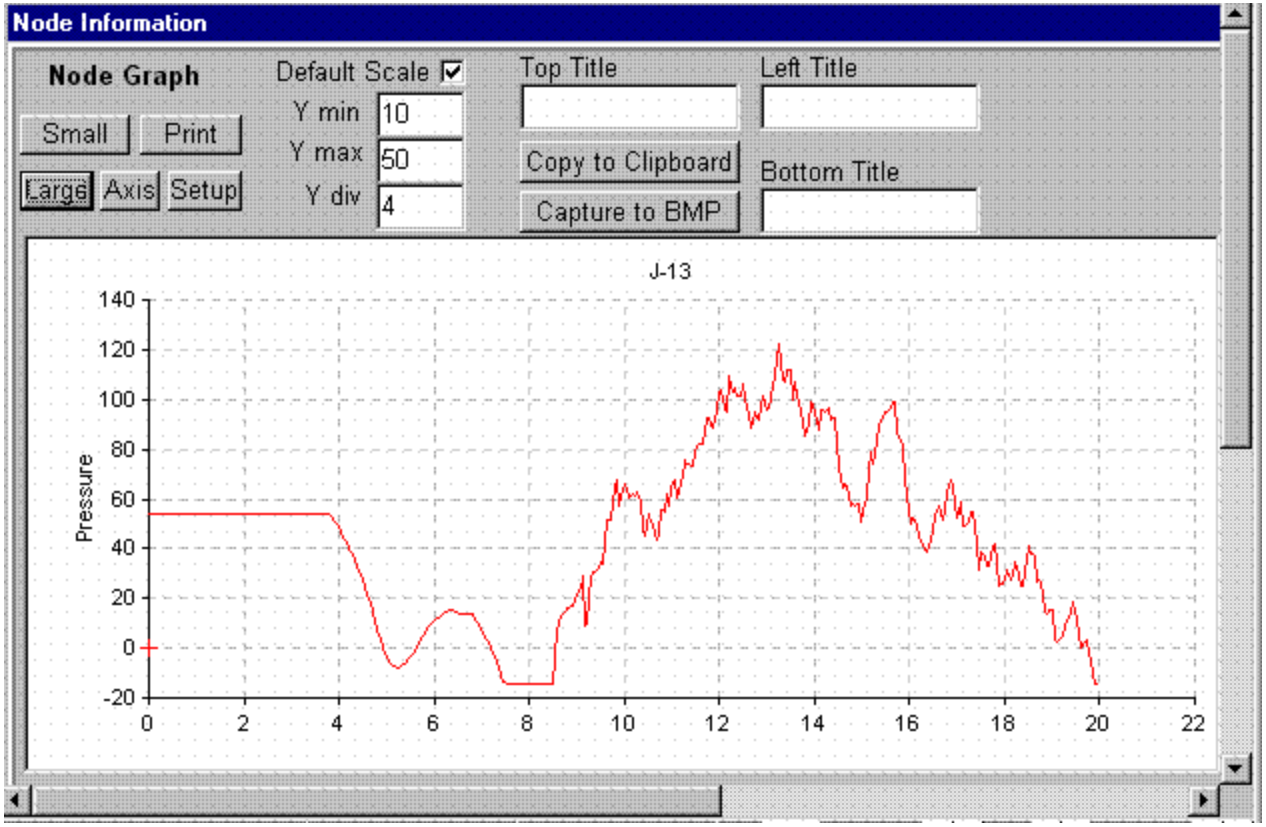
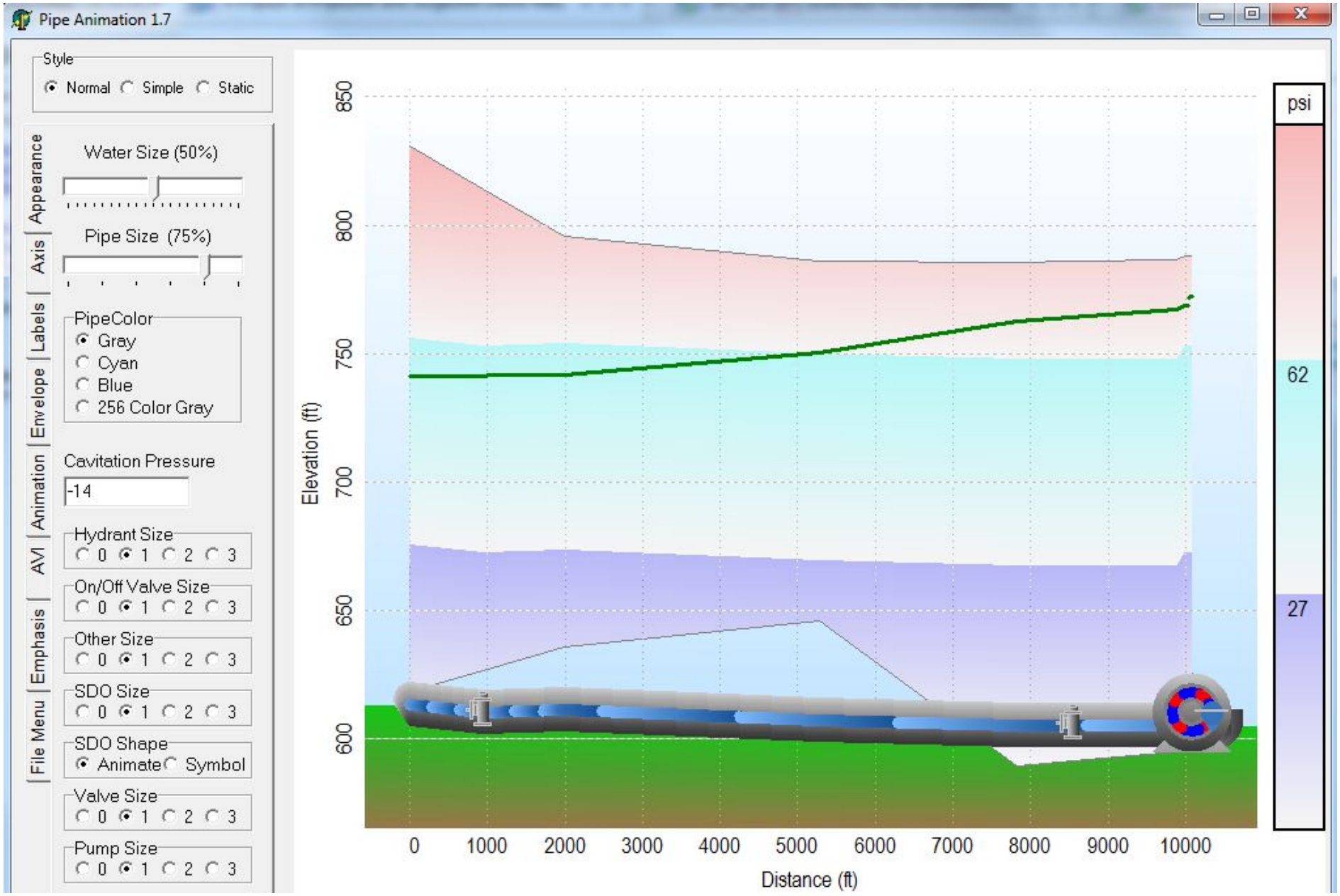


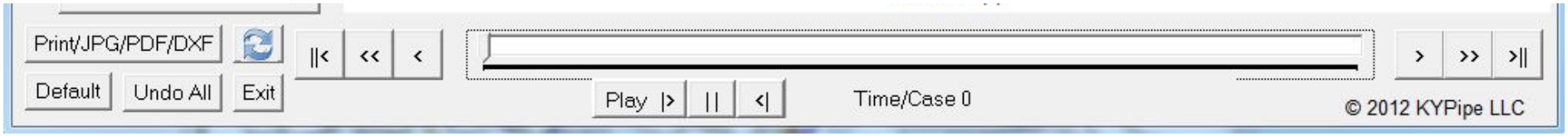

Figure 16 Node Results Graph

**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.

Animated Profile:


Pipe Animation 1.7
Style
Normal
Simple
Static
Appearance
Axis
Labels
Envelope
Animation
AVI
Emphasis
File Menu
Water Size (50%)
Pipe Size (75%)
PipeColor
Gray
Cyan
Blue
256 Color Gray
Cavitation Pressure
-14
Hydrant Size
On/Off Valve Size
Other Size
SDO Size
SDO Shape
Animate
Symbol
Valve Size
Pump Size
0 1 2 3
Elevation (ft)
850
800
750
700
650
600
0
1000
2000
3000
4000
5000
6000
7000
8000
9000
10000
Distance (ft)
psi
62
27




Static Profile:

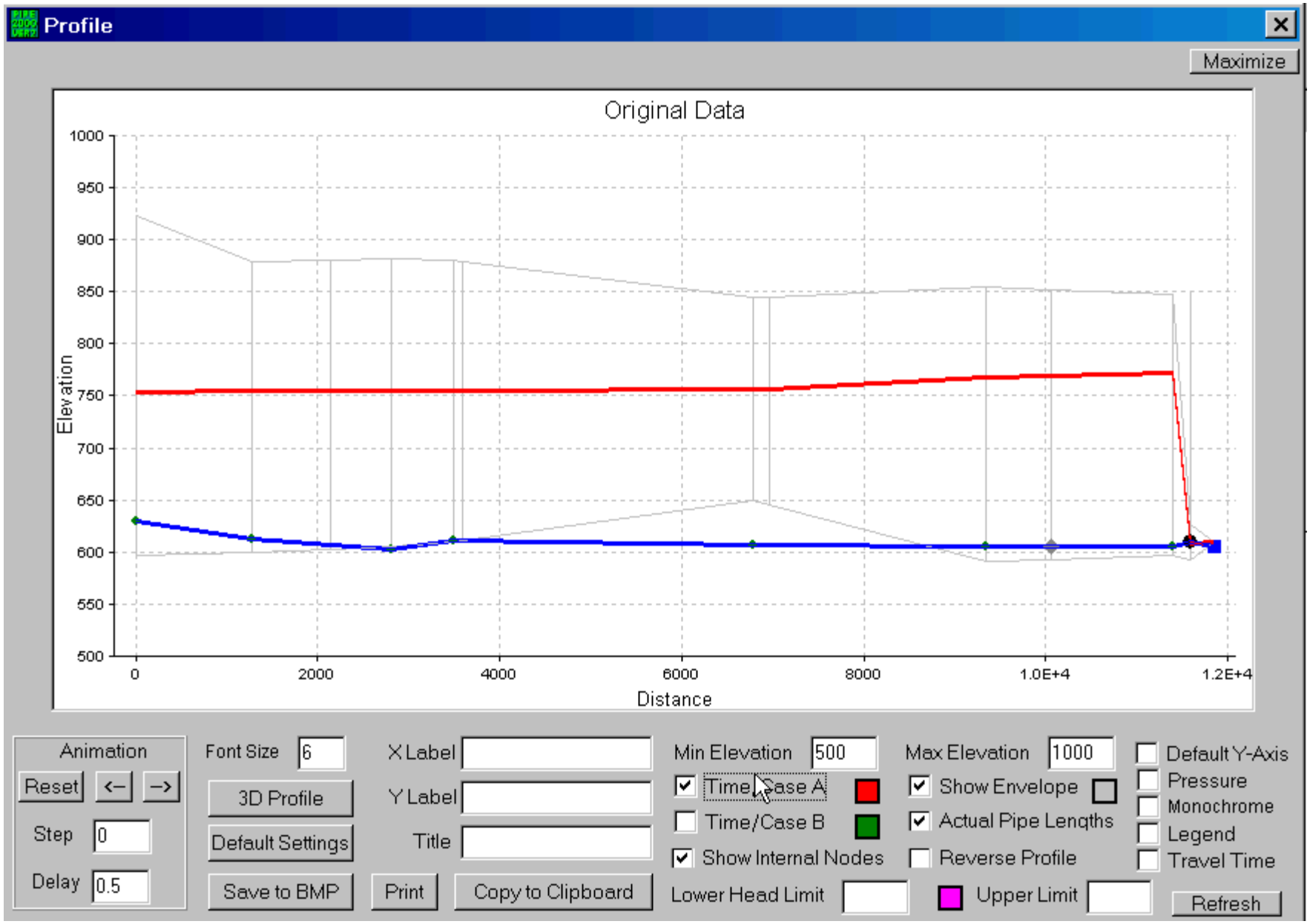

Profile
Maximize
Original Data
1000
950
900
850
800
750
700
650
600
550
500
Elevation
0
2000
4000
6000
8000
1.0E+4
1.2E+4
Distance
Animation
Reset
<-
->
Step
0
Delay
0.5
Font Size
6
3D Profile
Default Settings
Save to BMP
X Label
Y Label
Title
Print
Copy to Clipboard
Min Elevation
500
Max Elevation
1000
Time/Case A
Time/Case B
Show Internal Nodes
Show Envelope
Actual Pipe Lengths
Reverse Profile
Lower Head Limit
Upper Limit
Default Y-Axis
Pressure
Monochrome
Legend
Travel Time
Refresh

Figure 17 Profile

**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.

SUMMARY OF MAXIMUM AND MINIMUM HEADS

| POSITION NO. | MAXIMUM (ft) | MINIMUM (ft) | Time Avg. Intr. Fact | Frac. Time reverse grad. | TWF |
|---|---|---|---|---|---|
| J-1 | 237.56 | 42.82 | 0.000 | 0.000 | 0.00 |
| J-10 | 275.62 | -12.96 | 0.000 | 0.000 | 0.00 |
| J-11 | 324.49 | -33.21 | 0.000 | 0.000 | 0.00 |
| J-12 | 225.90 | 38.87 | 0.000 | 0.000 | 0.00 |
| J-13 | 293.53 | -33.21 | 0.000 | 0.000 | 0.00 |
| J-14 | 268.22 | 6.54 | 0.000 | 0.000 | 0.00 |
| J-15 | 249.73 | -13.95 | 0.000 | 0.000 | 0.00 |
| J-16 | 227.24 | 10.17 | 0.000 | 0.000 | 0.00 |
| J-2 | 379.86 | -33.21 | 0.000 | 0.000 | 0.00 |
| J-3 | 242.83 | -8.15 | 0.000 | 0.000 | 0.00 |
| J-4 | 279.31 | 3.41 | 0.000 | 0.000 | 0.00 |
| J-5 | 328.29 | -33.21 | 0.000 | 0.000 | 0.00 |
| J-6 | 271.72 | -1.03 | 0.000 | 0.000 | 0.00 |
| J-7 | 265.45 | -13.03 | 0.000 | 0.000 | 0.00 |
| J-8 | 243.66 | 29.83 | 0.000 | 0.000 | 0.00 |
| J-9 | 287.28 | -33.21 | 0.000 | 0.000 | 0.00 |
| Pump-1 | 17.02 | -17.55 | 0.000 | 0.000 | 0.00 |
| R-1 | 5.00 | 5.00 | 0.000 | 0.000 | 0.00 |
| T-1 | 120.00 | 120.00 | 0.000 | 0.000 | 0.00 |
| T-2 | 124.00 | 124.00 | 0.000 | 0.000 | 0.00 |
| Pump-1 | 240.73 | -12.29 | 0.000 | 0.000 | 0.00 |

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 ***Pump Shutdown with Surge Tank***. 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 1 CFS.

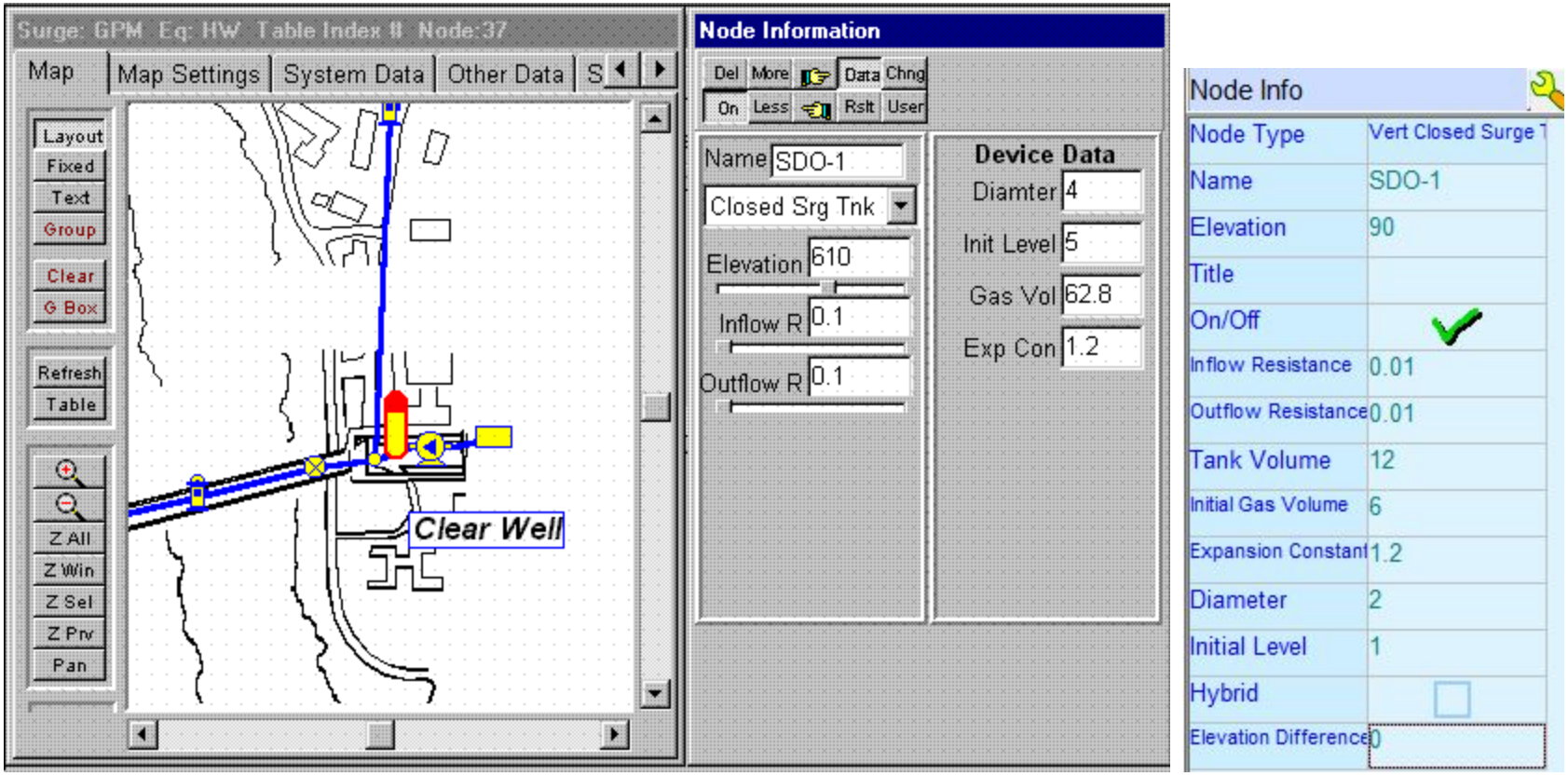



Figure 19 Surge Tank and Data

Click on the Zoom 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.

KYnetic Graph and Table Results icon:

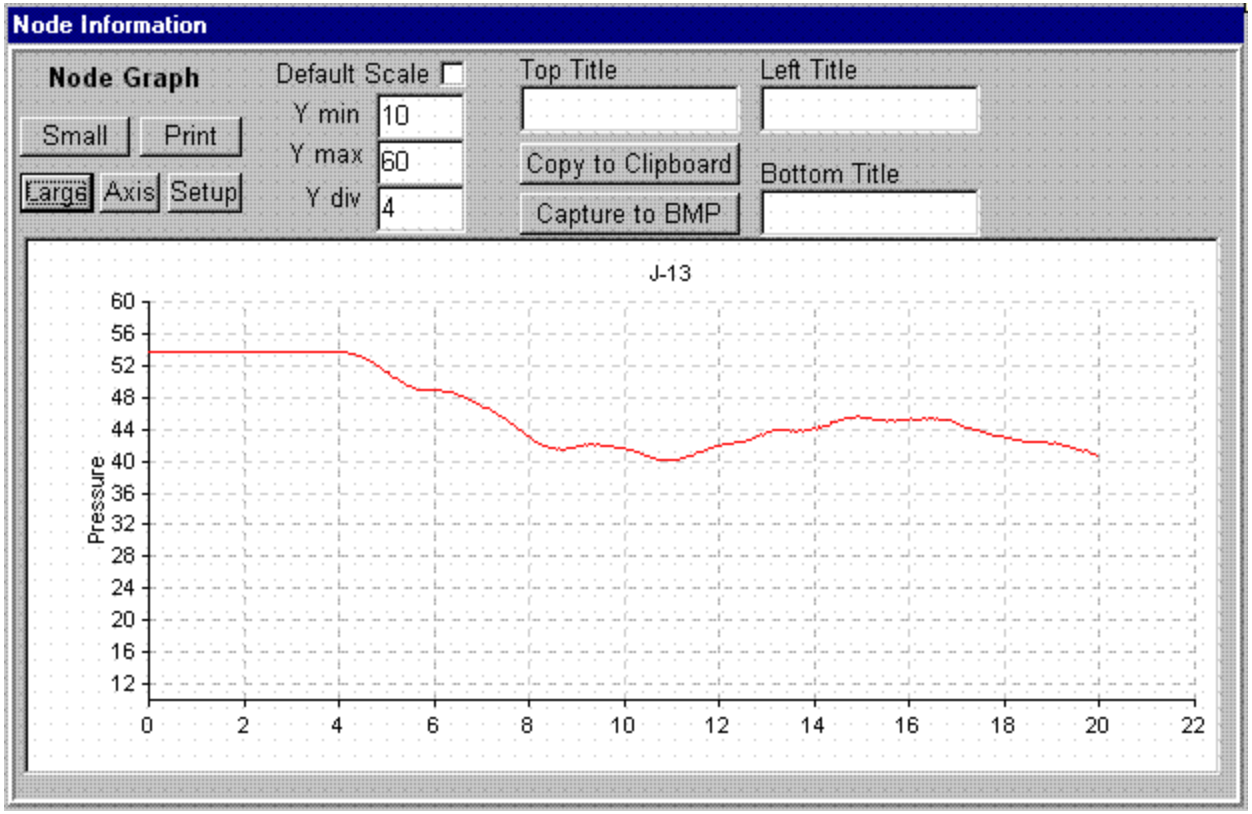



Figure 20 Node Results Graph

**KYPipe - Optimized Calibration**

Click on **File** and **Open** and select the **Demo Calibration** file to see a demonstration of the Pipe2018 Optimized Calibration module. You may wish to review the "Optimized Calibration Data" topic before you go through the demonstration.

Figure 21 shows a network schematic with the test results of four fire flow tests displayed.

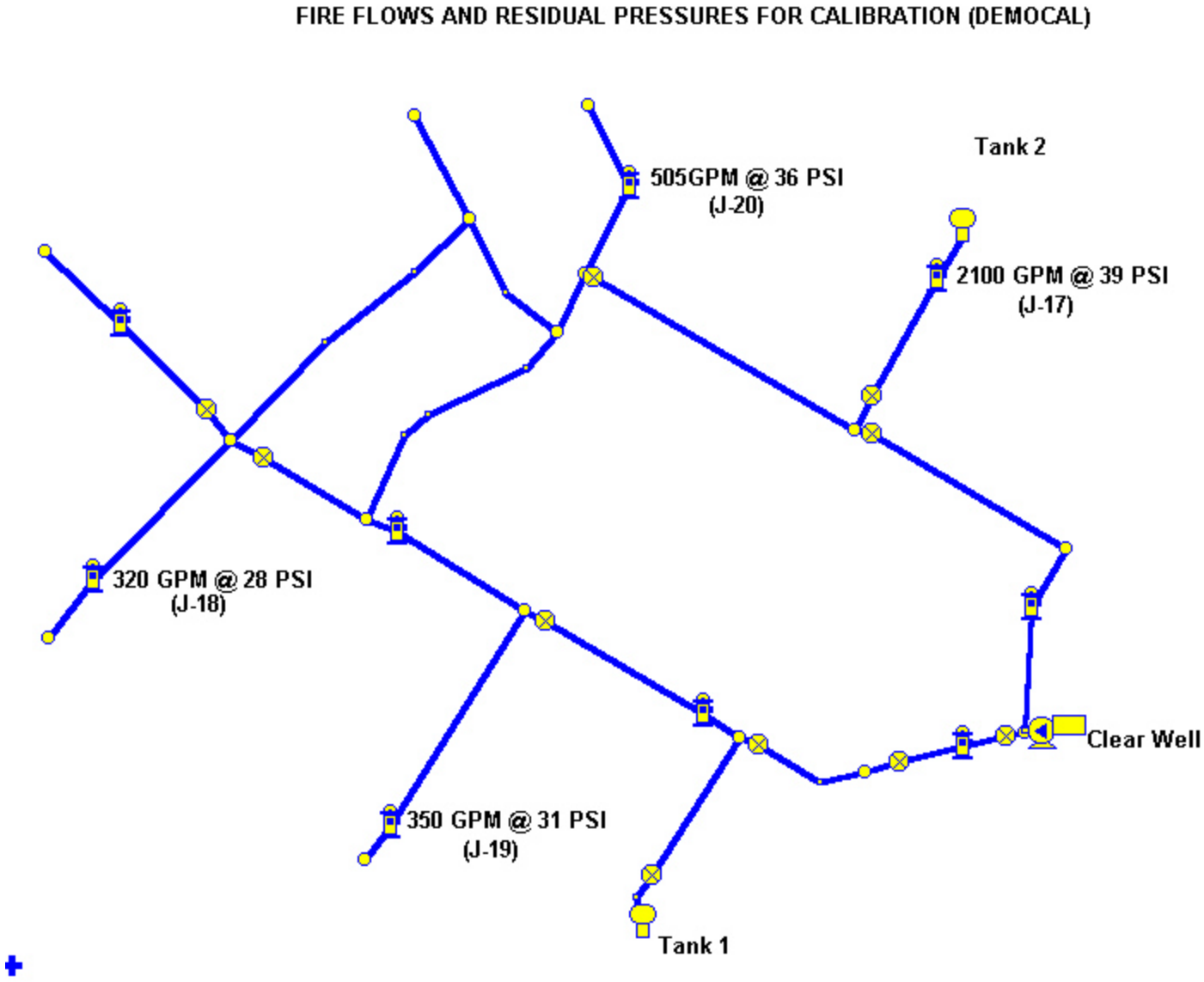


Figure 21 Fire Flow Test Results

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.

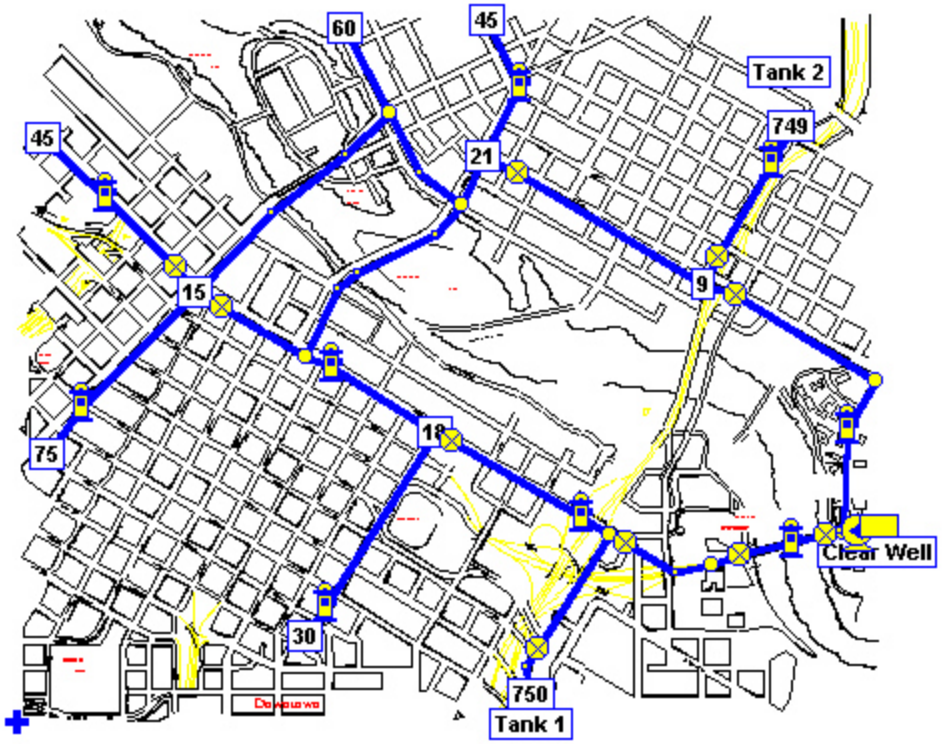
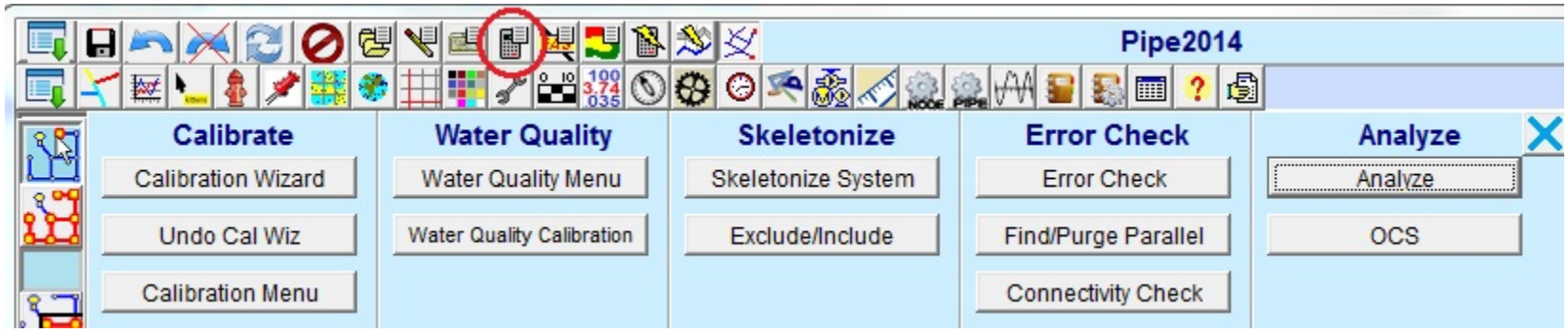



Figure 22 Tank Levels and Baseline Demands

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.


Pipe2014
Calibrate
Calibration Wizard
Undo Cal Wiz
Calibration Menu
Water Quality
Water Quality Menu
Water Quality Calibration
Skeletonize
Skeletonize System
Exclude/Include
Error Check
Error Check
Find/Purge Parallel
Connectivity Check
Analyze
Analyze
OCS

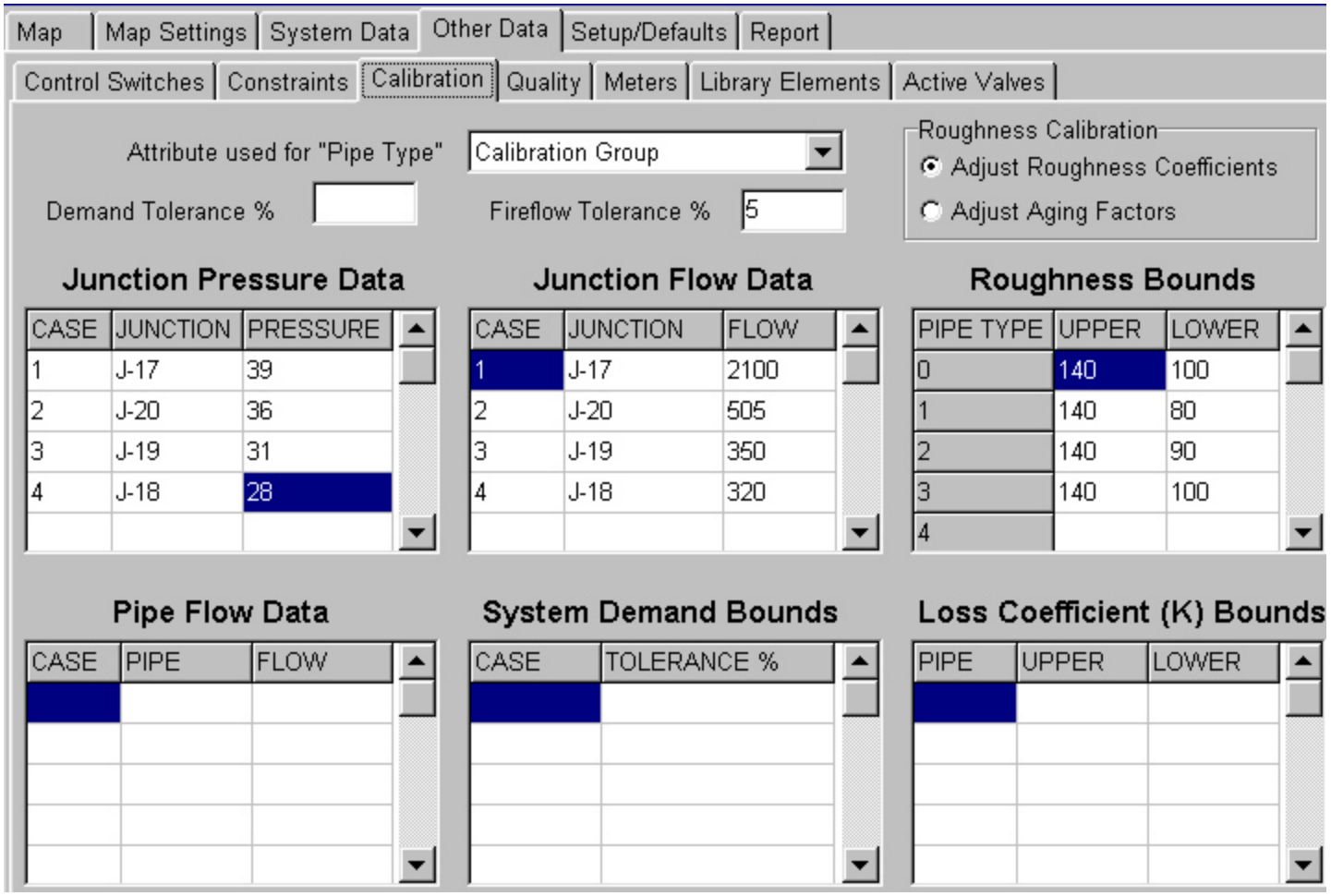



Figure 23 Fire Flow and Calibration Data

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**

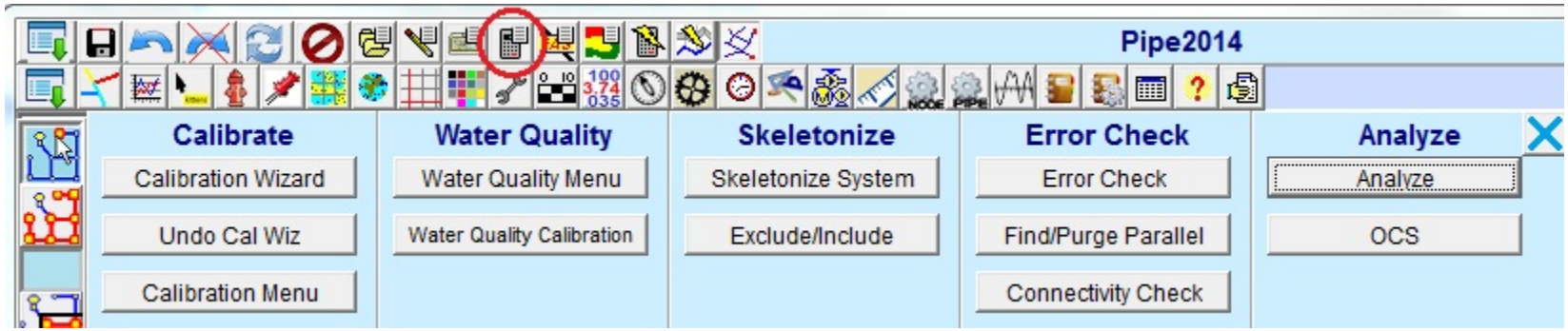
**Date & Time: Mon Nov 26 08:19:49 2001**

**------- 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 **Demo Water Quality**. Click on Analyze | Quality menu or **Other Data** and **Quality** (Classic) to see the data screen shown below.


Pipe2014
Calibrate
Calibration Wizard
Undo Cal Wiz
Calibration Menu
Water Quality
Water Quality Menu
Water Quality Calibration
Skeletonize
Skeletonize System
Exclude/Include
Error Check
Error Check
Find/Purge Parallel
Connectivity Check
Analyze
Analyze
OCS

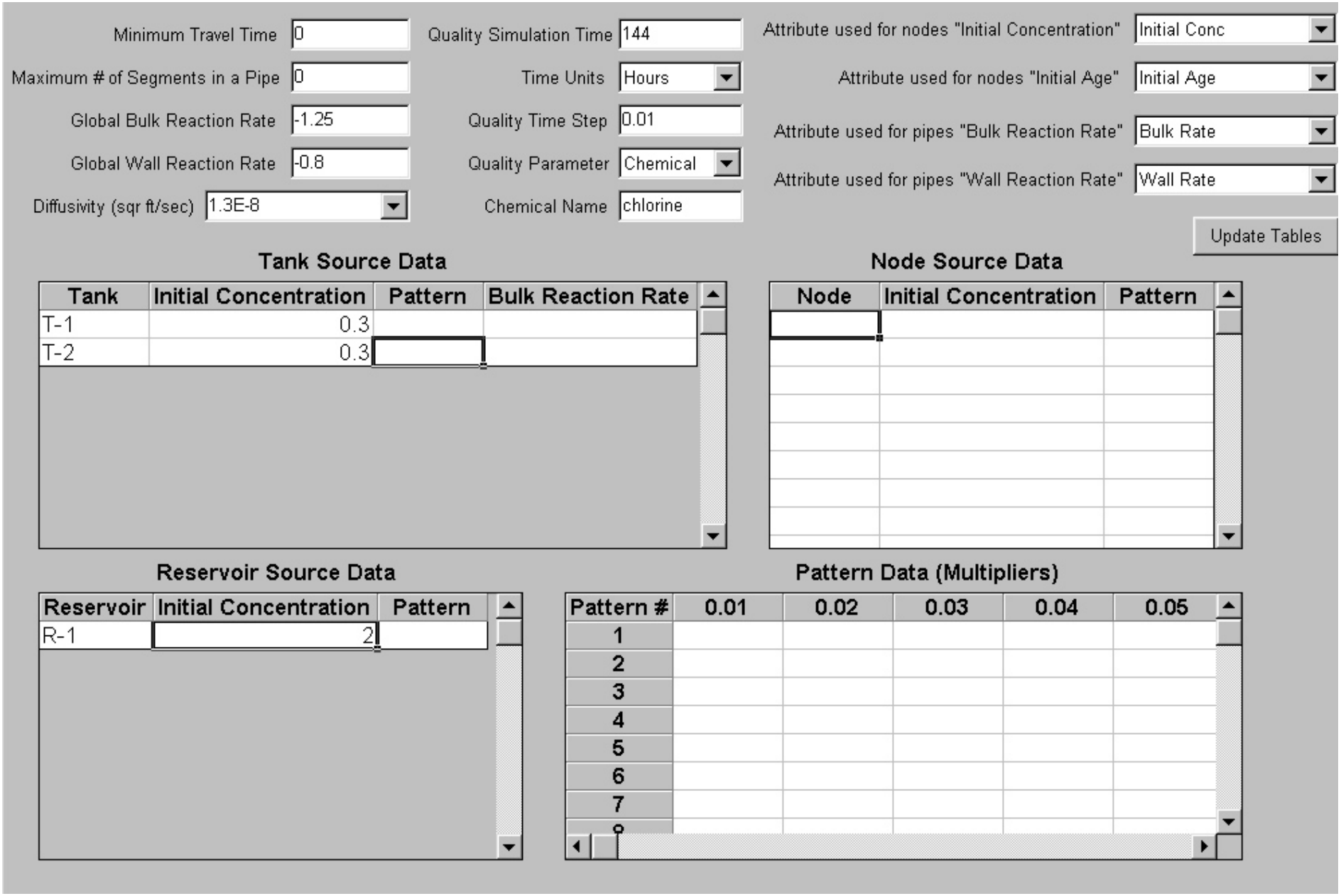

Minimum Travel Time 0
Maximum # of Segments in a Pipe 0
Global Bulk Reaction Rate -1.25
Global Wall Reaction Rate -0.8
Diffusivity (sqr ft/sec) 1.3E-8
Quality Simulation Time 144
Time Units Hours
Quality Time Step 0.01
Quality Parameter Chemical
Chemical Name chlorine
Attribute used for nodes "Initial Concentration" Initial Conc
Attribute used for nodes "Initial Age" Initial Age
Attribute used for pipes "Bulk Reaction Rate" Bulk Rate
Attribute used for pipes "Wall Reaction Rate" Wall Rate
Update Tables
Tank Source Data
Tank Initial Concentration Pattern Bulk Reaction Rate
T-1 0.3
T-2 0.3
Node Source Data
Node Initial Concentration Pattern
Reservoir Source Data
Reservoir Initial Concentration Pattern
R-1 2
Pattern Data (Multipliers)
Pattern # 0.01 0.02 0.03 0.04 0.05
1
2
3
4
5
6
7

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. EPANET requires the units for bulk and wall reaction rates to be per "day" basis. If the bulk/wall decay rates are zero the program assigns a default value. 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:

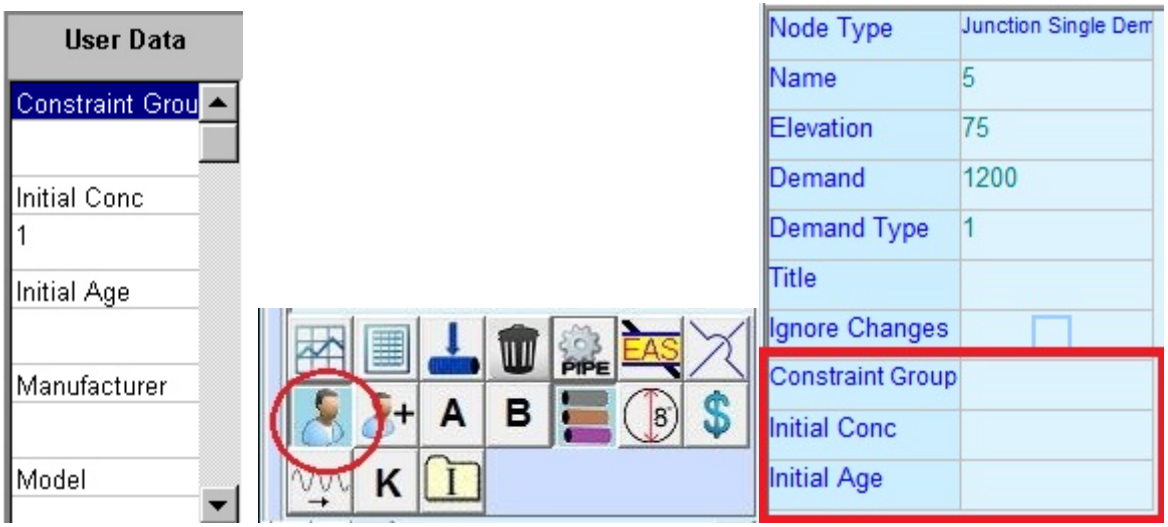
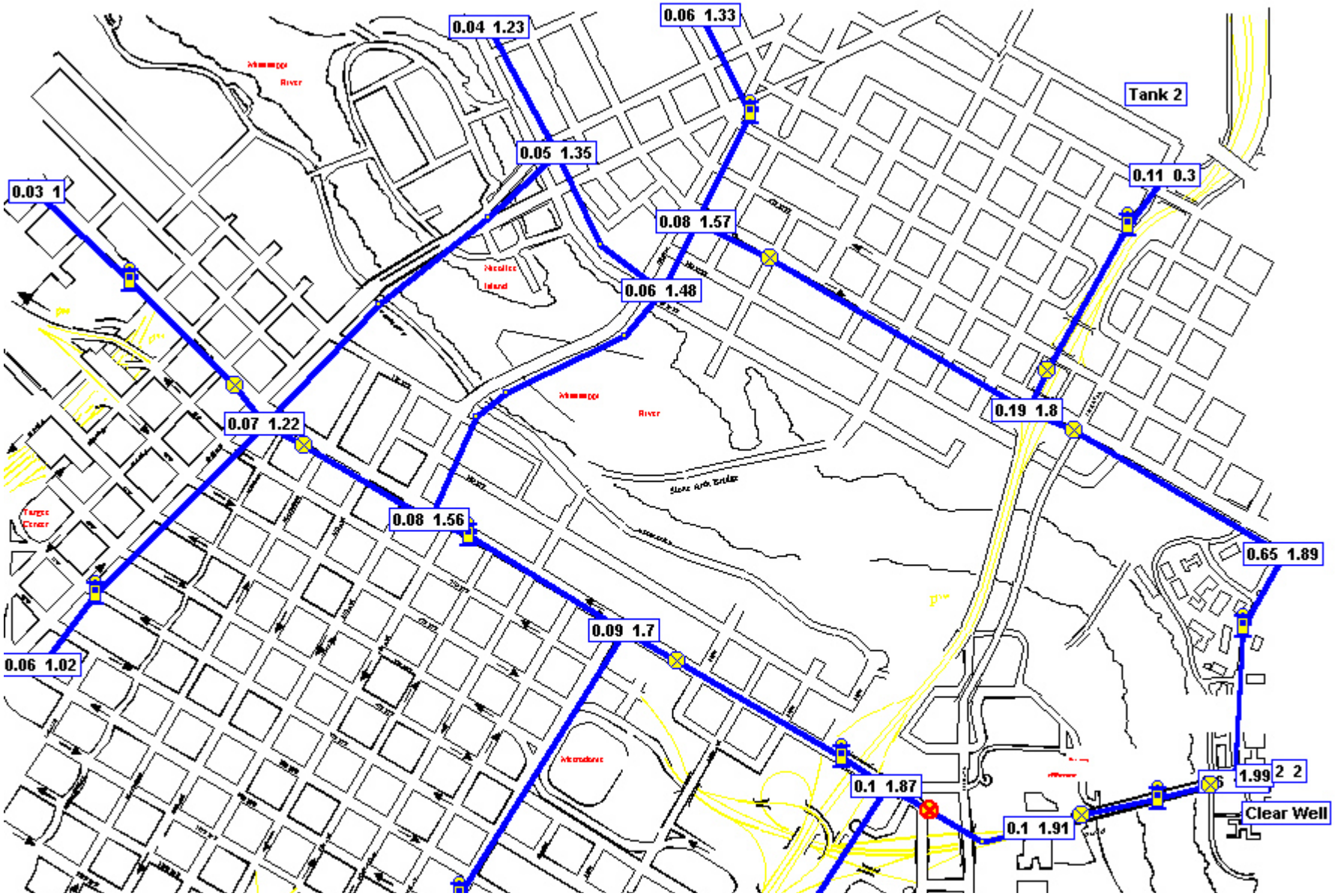



Figure 27 User Data with Initial Concentration

The Water Quality Analysis is then run by selecting **Analyze** 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 Results menu (bottom right of program window) 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.


0.04 1.23
0.06 1.33
Tank 2
0.05 1.35
0.11 0.3
0.03 1
0.08 1.57
0.06 1.48
0.19 1.8
0.07 1.22
0.08 1.56
0.65 1.89
0.09 1.7
0.06 1.02
0.1 1.87
1.99 2 2
Clear Well
0.1 1.91

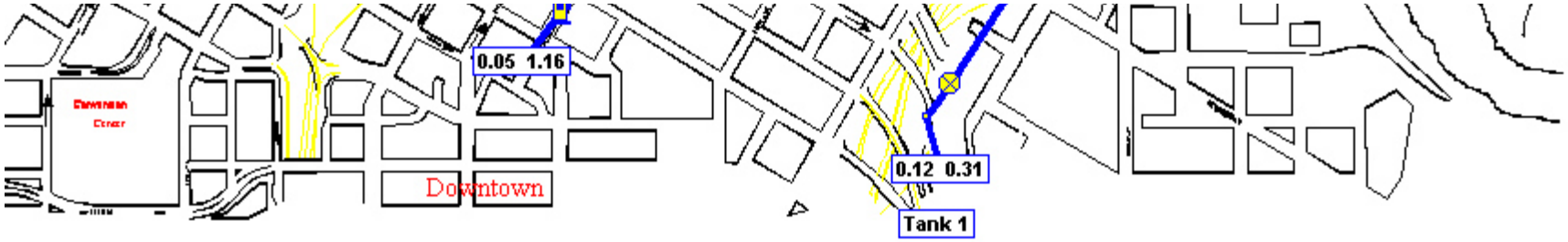



Figure 28 Min/Max Chlorine Residuals

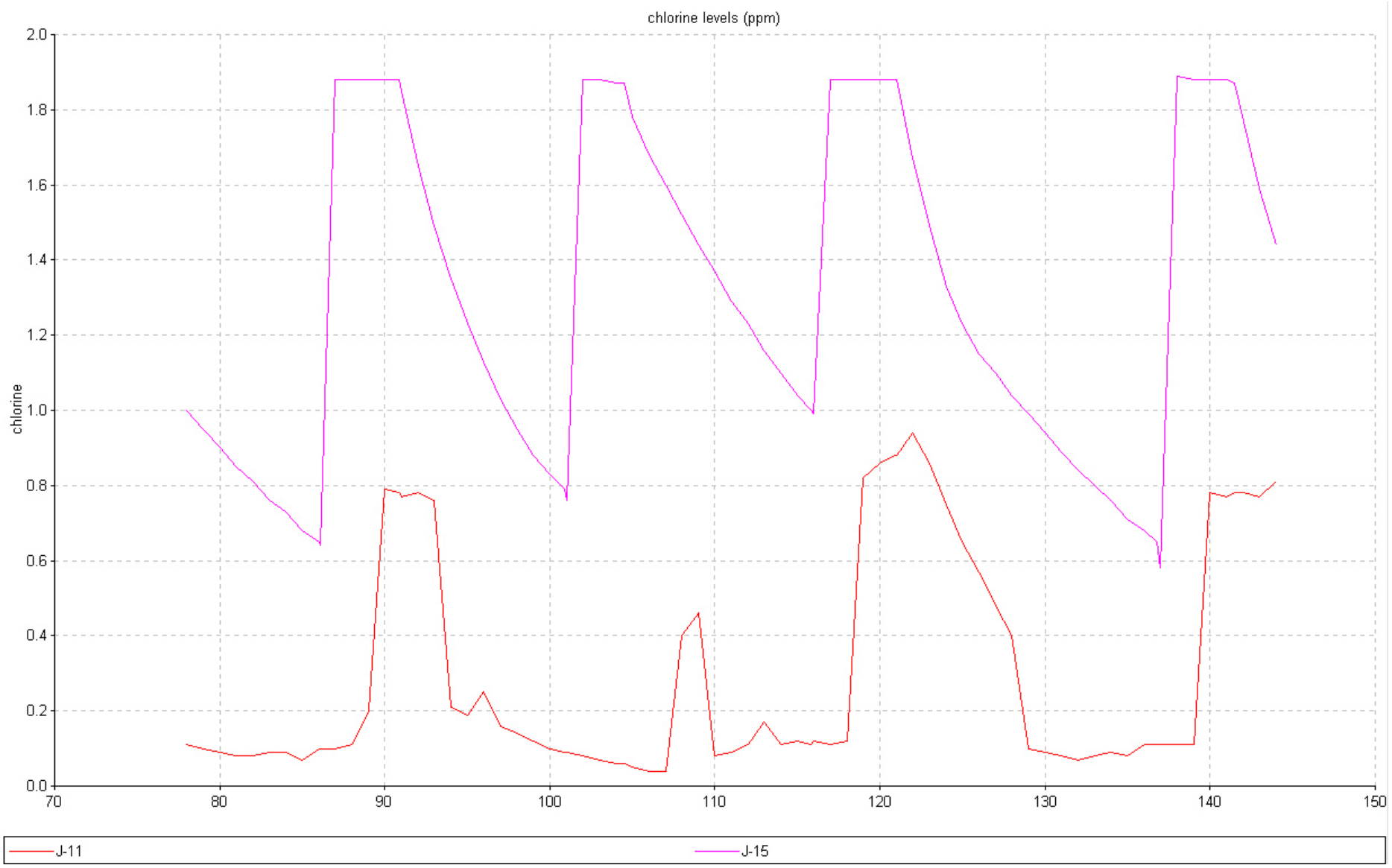



Figure 29 Chlorine Residuals at Selected Nodes

## 1.3.1.5 PIPE2000 Tutorial (Audio/Video)

Pipe2018 Tutorials (Audio/Video)

---

Pipe2018 is designed to provide very rapid and intuitive model development. An extensive Help File is provided and topics can be accessed through the Contents page or a comprehensive Index. Multimedia presentations (audio/video clips) of operations can be found on the **Pipe2018 Tutorial**. Insert your Pipe2018 USB Device or click the Tutorials icon in the upper right corner of the KYnetic interface window 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 USB device and find the **Tutorial and Setup** applications.

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 **KYPipe 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 **Pipe2018 GUI** and **Model Development** clips prior to using Pipe2018. The additional clips may be reviewed as you utilize the capabilities which they address.

**Pipe2018 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 Systm** 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** (KYPipe Only)

- **Extentded 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
- **Precison 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

**KYPipe 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
- **Exented Period Simulation Example** Extended period simulation of a model
- **Calibration - Parts 1-6** Cailbrating 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 (KYPipe) 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**

## 1.3.1.6 Reference Manual

Reference Manual

---

Introduction to Modeling

Method of Analysis

Model Simplification

Model Calibration

Pipe System Model Geometry

Pipe System Components

Pressure and Flow Specifications

Multiple Scenarios - Changes

Direct Parameter Calculations - Constraints

## 1.3.1.6.1 Introduction to Modeling

Introduction to Modeling

Modeling refers to the process involved in representing your piping system in the manner required for engineering calculations to be made. The engine refers to the module which sets up and solves the basic engineering equations. Engines for pipe system hydraulic calculations are designed to calculate the flows in all the pipes and the pressures of all nodes. In addition to the basic calculations, a number of additional calculations are important and useful The capabilities of the model depend on the scope of these calculations and the range of pipe system features handled by the engine.

The KYPIPE engine has been developed to calculate steady state flows and pressures for pipe distribution systems. The engine can be applied to any liquid, but does not generally apply to gas flow unless the assumption of constant density is acceptable. The engine is written to accommodate any piping configuration and a wide variety of hydraulic components such as pumps, valves (including check valves and regulating valves), any component or fitting which produces significant head loss (such as elbows, orifices, etc.), flow meters and storage tanks. Computations can be carried out using both English and SI units.

The KYPIPE engine is also capable of carrying out an extended period simulation (EPS) considering storage tank levels which vary over the simulation period. Storage tanks may have any shape and have upper and lower surface levels which define the range of operation of the tanks. Lines leading to storage tanks will close if the liquid surface levels reach these limits (altitude valve). As a feature of the extended period simulation the open-closed status of designated pipes may be controlled by the hydraulic grade line at a specified location in the network (pressure switch). This feature will allow, for example, bringing a booster pump on line if the pressure at a specified location drops below a specified switching value. This pump will operate until the pressure is increased above a second specified value. The same feature can be employed to use the water level in a storage tank to control a pump.

In this program the elevation plus the pressure head is referred to as the hydraulic grade line (HGL). The value of the hydraulic grade line is used for various data inputs rather than specifying both elevation and pressure.

The use of the features available for EPS will allow you to solve various transient pipe flow problems. This applies to a large class of slowly varying transients where acceleration forces are insignificant. Draining and filling of tanks are examples of this type of problem. Using an EPS, the analysis of flooded surcharged storm sewers can be made. The detention pools for the flooded regions at the inlets for the storm sewers are modeled as storage tanks which have a specified inflow which is determined from the run off hydrograph. The computer simulation will determine how high the water will rise at each detention basin and how the sewer system handles the flow, and the analysis can be carried out until all the detention pools have emptied.

Normal pipe network modeling involves the calculation of the flow in each pipe and the pressure at each node for a particular operating condition. In addition to carrying out these calculations, KYPIPE has been enhanced to allow you to directly calculate a variety of additional design, operation and calibration parameters which will exactly meet stated pressure requirements. This powerful, state of the art capability, greatly increases the usefulness of the current KYPIPE engine as a pipe network modeling tool by eliminating the trial and error procedure normally associated with such calculations. The following parameters can be selected for calculations:

1 pump speed
2 pump power
3 HGL settings for supplies or storage tanks
4 HGL settings for regulating valves
5 control valve settings (loss coefficients)
6 diameters
7 roughnesses
8 demands, flow requirements

One of these parameters can be selected for each pressure requirement specified. Additional details on the use of this feature is presented in under the topic Direct Parameter Calculations - Constraints.

SEE ALSO: Introduction to Modeling
Method of Analysis
Model Simplification
Model Calibration
Pipe System Model Geometry
Pipe System Components
Pressure and Flow Specifications
Multiple Scenarios - Changes
Direct Parameter Calculations - Constraints

## 1.3.1.6.2 Method of Analysis

Method of Analysis

---

KYPIPE is based on solving the full set of mass continuity and energy equations utilizing efficient linearization schemes to handle non

linear terms and a very powerful spare matrix routine developed by A.R. Curtis and J.K. Reid of the Theoretical Physics Division, UKAEA Research Group, Harwell, England. This approach accommodates elements such as closed lines, check valves, and regulating valves in a direct and very efficient manner. The approach also effectively handles data with widely varying parameter values. Extensive testing of various algorithms for pipe network analysis led to the conclusions that the approach used by KYPIPE is the most powerful and has the best convergence characteristic of the commonly used approaches

SEE ALSO: Wood, D. J. and Rayes, A.G. "**Reliability of algorithms for pipe network analysis.**" *J. Hydr. Div.* ASCE, 107(10), 1145-1161. (1981)

Wood, D. J. **KYPIPE Reference Manual**, Civil Engineering Software Center, University of Kentucky (1985)

SEE ALSO: Introduction to Modeling

Method of Analysis

Model Simplification

Pipe System Model Geometry

Pipe System Components

Pressure and Flow Specifications

Multiple Scenarios - Changes

Direct Parameter Calculations - Constraints

## 1.3.1.6.3 Model Simplification

Model Simplification

---

Before analyzing a pipe distribution system you should consider any possible simplifications which will not significantly affect the solution. This is particularly important for large distribution systems. It may be possible to save considerable computer time and reduce

office time needed to develop and manage the computer model and enter data. The most obvious simplification is to model a skeletonized distribution system comprised of fewer pipes than the actual system. The most common method of skeletonizing the distribution system is to only consider pipes above a certain minimum size. If this is done, flow demands for the regions not considered should be shown at junctions in the vicinity of these regions. Also, it is often possible to eliminate tree type pipe regions from a system. Demands to these regions can be specified at junctions leading to the region eliminated. Eliminating regions of this type will not affect pressures and flows in the main system. Series and parallel pipes can be replaced by single equivalent pipes. If the system has distinct low pressure regions these can be analyzed separately. If more detailed information is needed on portions of a simplified system these portions can be analyzed separately using the results of the analysis of the main system.

SEE ALSO: Introduction to Modeling

Method of Analysis

Model Simplification

Model Calibration

Pipe System Model Geometry

Pipe System Components

Pressure and Flow Specifications

Multiple Scenarios - Changes

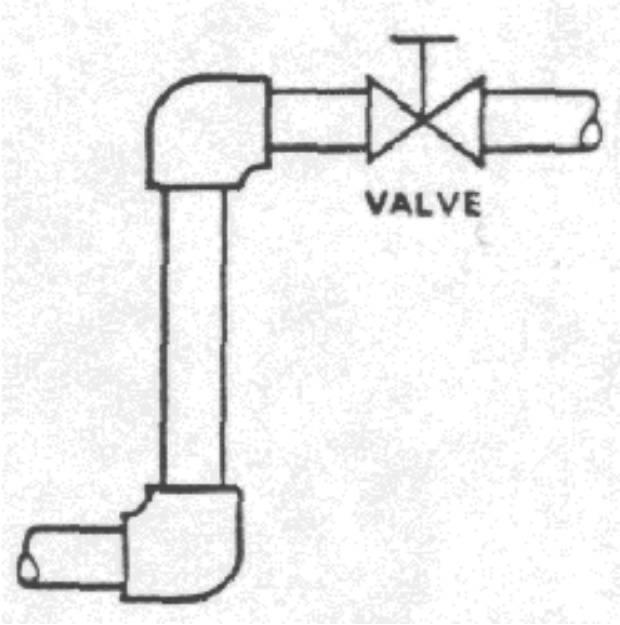
Direct Parameter Calculations - Constraints

## 1.3.1.6.4 Pipe System Geometry

Pipe System Geometry

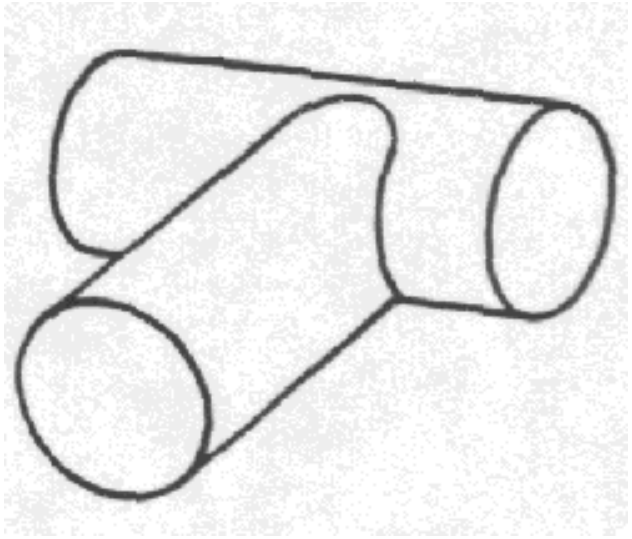
---

The principal elements in the pipe system are pipe sections. These are constant diameter sections which can contain pumps and fittings such as bends and valves as depicted below

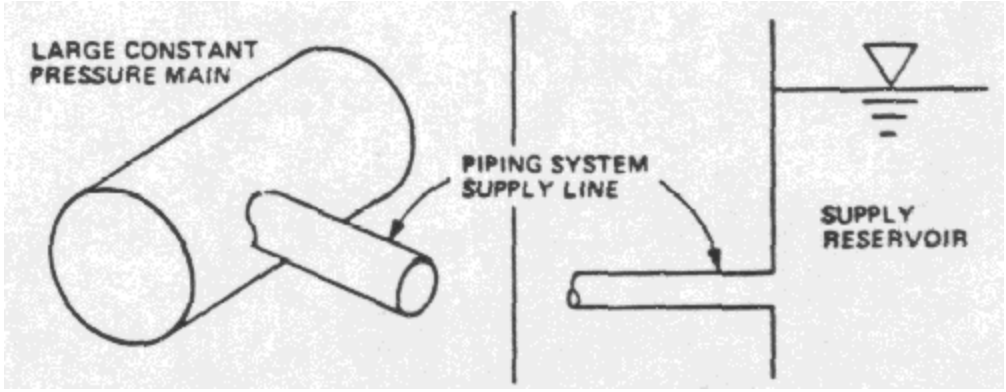



The end points of pipe sections are called nodes and are classified either as junction nodes or fixed grade nodes (FGNs).

junction node - A node where two or more pipes meet or where flow is put into or removed from the system. If a pipe diameter change occurs at a component such as a valve or a pump, this point is a junction node.



fixed grade nodes - A node in the system where both the pressure head and elevation (HGL) are known. This is usually a connection to a storage tank or reservoir or a source or discharge point operating at a specified pressure. Each system must have at least one fixed grade node (FGN).




In addition, pipe networks include primary loops which are defined as follows:

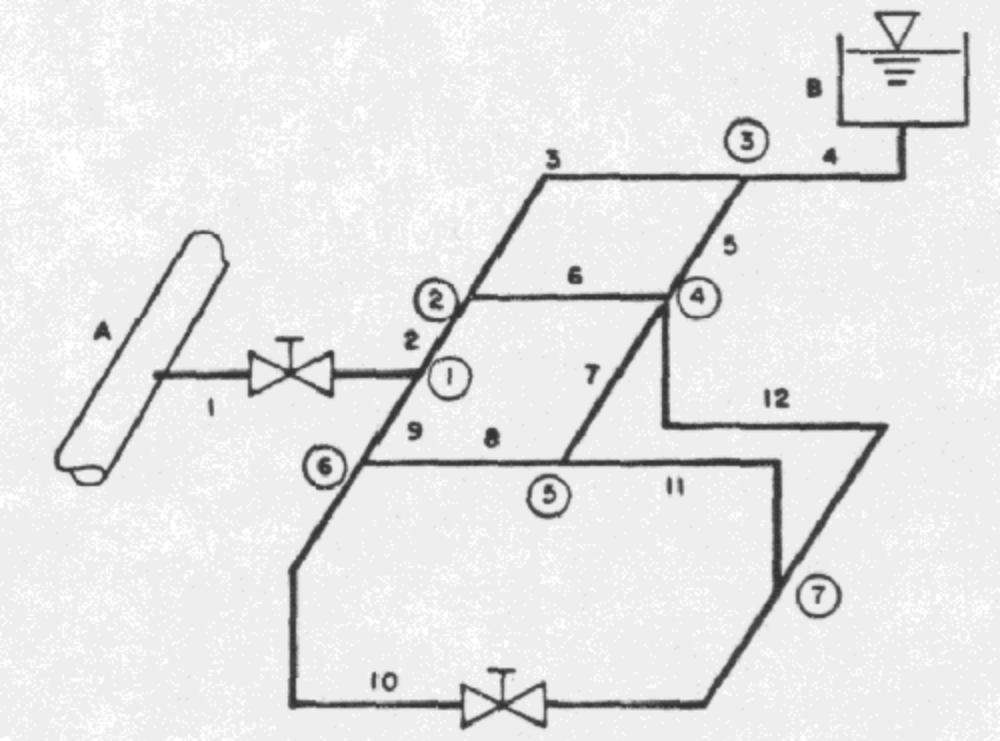
primary loop - A closed pipe circuit with no other closed pipe circuits contained within it.

If the junctions, primary loops, and fixed grade nodes are identified as described above, the following holds for all pipe systems:

$p = j + \mu + f - z$ (1)

where p = number of pipe section
j = number of junction nodes
μ = number of primary loops
f = number of fixed grade nodes
z = number of separate zones

Separate zones are ones which can not be accessed from another zone through a pipe section and, therefore, operate as independent systems. The picture below illustrates this concept.




p = number of pipe section=12
j = number of junction nodes=7
μ = number of primary loops=4
f = number of fixed grade nodes=2
12=7+4+2-1

SEE ALSO: Introduction to Modeling

Method of Analysis

Model Simplification

Model Calibration

Pipe System Model Geometry

Pipe System Components

Pressure and Flow Specifications
Multiple Scenarios - Changes
Direct Parameter Calculations - Constraints

## 1.3.1.6.5 Pipe System Components

Pipe System Components

---

Data regarding the physical characteristics of the components in the pipe system must be obtained prior to creating a model for computer analysis.

MODELING

Pipe Sections
Pumps
Check Valves
Regulating Valves
Variable Pressure Supply
Minor Loss Components
Storage Tanks
Pressure Switches
Flow Meters

SEE ALSO: Introduction to Modeling

Method of Analysis

Model Simplification

Model Calibration

Pipe System Model Geometry

Pipe System Components

Pressure and Flow Specifications

Multiple Scenarios - Changes

Direct Parameter Calculations - Constraints

## 1.3.1.6.6 Pressure and Flow Specifications

Pressure and Flow Specifications

---

Certain data are required to describe boundary pressure and flow specifications. The most important of these are the flows entering or leaving the distribution system at the junction nodes (demands). For some systems, analyses are carried out with no inflows or outflows (demands) specified. For most systems, however, demand requirements are specified at designated junction nodes and the pressure and flow distribution is determined for this situation. At any junction node, the external inflow (negative) or outflow (positive) demand may be specified. For each different case or time (EPS) any change in these demands from the initial specifications must be input.

Variations in demands represent very important data. Pipe2018 allows multiple global demand factors associated with up to ten junction demand types to enable you to easily create multiple demand patterns. In this manner the demands at junctions which may represent residential, commercial or industrial users can be changed using different demand factors to represent different types of demand variations which occur for regular simulation changes or throughout an EPS.

The elevations of junction nodes must be specified if the pressures (or pressure heads) are to be calculated. Values for the elevation of junction nodes are not required to compute the flow distribution and only affect the pressure calculation at the junction nodes. Thus, elevations need only be specified where calculated values of pressure are desired. Elevations are required if an accurate representation of pressure contours are to be displayed.

At each FGN, including variable level storage tanks for (EPS only), the initial HGL (pressure head + elevation) is an operating condition which must

be specified. This means that the elevation of surface levels in reservoirs and the initial levels for storage tanks must be specified for regular simulations. Also, if there are pressure requirements at fixed grade nodes, these are incorporated into the value specified for the HGL maintained by the FGN. If there are pressure regulating valves or pressure sustaining valves in the system HGL representing the setting must be specified. The regulated pressure is incorporated into the calculation of the HGL representing the valve setting (pressure head + elevation).

**Normal Flow Directions** - Flow directions for lines with pumps, check valves, and pressure regulating valves must be correctly specified in the data input and this is done by the order which the connecting nodes for the pipe section are input. The normal flow direction is assumed to be from the first node input to the second node input. If the calculated flow is in the opposite direction it will be tabulated with a negative sign.

SEE ALSO: Introduction to Modeling

Method of Analysis

Model Simplification

Model Calibration

Pipe System Model Geometry

Pipe System Components

Pressure and Flow Specifications

Multiple Scenarios - Changes

Direct Parameter Calculations - Constraints

## 1.3.1.6.7 Multiple Scenarios - Changes

Multiple Scenarios - Changes

---

The program is designed to perform a simulation using the original data and carry out additional simulations using specified changes. These changes include both changes which are made to alter the original data and specify new conditions for additional regular simulations, and changes specified to occur at designated times during an extended period simulation. The change data is coded using the same specifications for both applications.

Changes to any of the original data except connecting nodes are allowed. All pipeline characteristics such as length, diameter, roughness and pump characteristics can be changed. HGL changes for FGN's may be specified. Demands may be changed at designated junction nodes and global demand changes based on the original data may be made. For this application all the demands of a given type are changed by a specified global demand factor. The global demand factors are applied before any designated specific demand changes at specified nodes. Thus, specific changes supersede the global

changes.

When a series of changes are specified for a regular or EPS all changes other than junction node demands are incorporated into the system data, these changes remain in effect throughout the remaining simulations unless the same parameter is subsequently changed again. Junction node demands, however, are always referenced back to the original data for each simulation and changes based on the original demands must be specified.

SEE ALSO: Introduction to Modeling

- Method of Analysis
- Model Simplification
- Model Calibration
- Pipe System Model Geometry
- Pipe System Components
- Pressure and Flow Specifications
- Multiple Scenarios - Changes
- Direct Parameter Calculations - Constraints

## 1.3.1.7 Some Special Features

Some Special Features

---

This section draws your attention to some Pipe2018 features which can save you a lot of time. Some of these can be customized to provide even greater efficiency. Other features provide some very useful capabilities which you should know about before you set up your models.

- Pipe Type Data
- Fittings Data
- Using Meters
- Input and Editing Shortcuts
- Logical Text Inputs

Operating Modes

Node Images and Text Nodes

Group Operations

Age-Based Roughness

Pump and System Curves

Copy and Paste System

## 1.3.1.8 Warnings and Error

Here is a list of common errors and messages and issues.

- Printing problems
- Can't open clipboard
- Error when analyzing
- Error upon launching Pipe2018
- Invalid Floating Point Error
- A Portion of the system is disconnected from a FGN
- Unsupported new DWG version ID: "AC1024" error
- Access Violation Error
- Exception EOleSYS Error
- Error Can't Open File
- Fortran Error
- Object is not of the expected type
- Invalid Argument
- Floating Point Division by Zero
- Network Key is showing an active user but no users are logged on
- Print to PDF Problems
- I/O Error

**Printing problems**

Make sure you have the most current version of the Pipe2018 GUI. For Print Options select PDF or BMP instead of Printer. When the file is displayed there will be an option to print it. Do not use a PDF printer driver, use the PDF option built into Pipe2018.

**Can't Open Clipboard**

If you are running *Google Earth* or *MultiMon Taskbar* or *GoToMyPC* close it before starting Pipe2018.

**Error when analyzing**

Always make sure you have the most recent version of the Pipe2018 installation (not necessarily the lastest license key). If that does not resolve your problem, please contact Tech Support. Attaching the Pipe2018 data file with which you are having trouble gives Tech Support much more insight into the possible cause of the problem. The only file needed is the one with a .p2k extension.

**Error when launching Pipe2018**

This problem is usually caused by key drivers being incorrectly installed. Make sure the installation files are not in a compressed format (zipped), and that they are located in a full-access read/write folder such as on the computer **Desktop**, then re-run the installation with the latest downloaded installation files.

**Invalid Floating Point Error**

Send model file with .p2k extension to Tech support, include the following information:

- When the error occurs (e.g. upon analysis, launching the program, clicking a map element, etc)
- which version is being used
-

**A portion of the system is disconnected from a fgn...**

**Disconnected Regions**

The steady state hydraulic equations for a pipe system can not be solved if there are sections with pressures that can not be calculated. This occurs if a section of the system is closed off from the rest of the system with no supply to that section. When this occurs the pressure is unknown and can't be calculated from the basic hydraulic equations. This situation produces an invalid set of hydraulic equations and must be corrected before a solution can be obtained. Figure 1a shows the

type of modeling situation which leads to disconnected sections. In the left picture there are two disconnected sections. In the center picture there is one disconnected section. Even with the pump operating the hydraulics between the two closed valves is not solvable. The picture on the right does not have a disconnected section and is an acceptable arrangement.

When a disconnected section is encountered Pipe2018 will print the following warning message. The attempted fix will involve opening one or more of the closed elements which often will not change the basic operation of the piping system. For example if both valves are opened the flow will still be zero because the pump is shut off. However, this action could alter the operation of the piping system and users should carefully review the implication of the fix.

***** WARNING *****
**A PORTION OF THE SYSTEM IS DISCONNECTED FROM A FGN BY CLOSED LINES**

****** A FIX WILL BE ATTEMPTED**

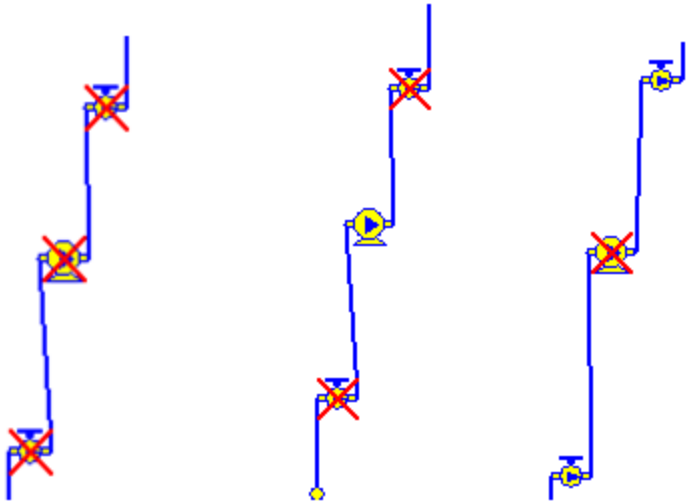


**Figure 1 Modeling Arrangements Producing Disconnected Sections**

Figure 2a and 2b depicts an alternate method for addressing disconnected sections. The picture on the right depicts an unacceptable situation resulting in a disconnected section. To correct this either the valve or pump must be opened. An alternative approach is to add a connection to a reservoir as shown in the picture on the right. Because the pump and/or valve are closed the reservoir will not affect the hydraulics. It will set the initial pressure in that section which is necessary to obtain a valid hydraulic analysis. If during the course of an EPS or surge analysis the valve and pump are opened, then the connection to a reservoir will affect the hydraulics. To minimize this effect the connecting pipe can be made very small with a large resistance. Then only a very small flow to or from the reservoir will result. This approach can, in fact, represent a leak which allows the hydraulic analysis to be completed with a minimal effect.

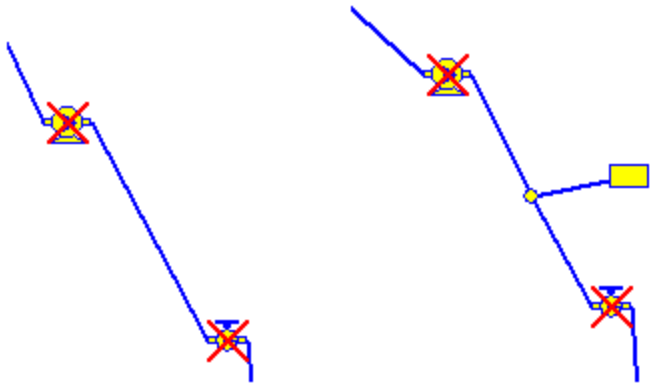


**Figure 2a Disconnected Sections Fixed Using a "Leak" Reservoir**

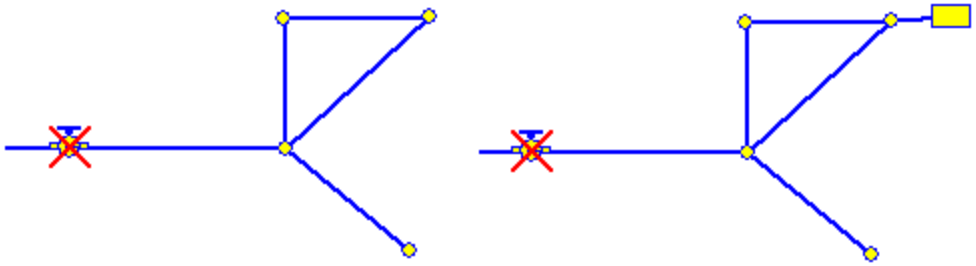


**Figure 2b Disconnected Sections Fixed Using a "Leak" Reservoir**

**Unsupported new DWG version ID: "AC1024" error**

Using a background from a newer version of AutoCAD, this error message can appear. If the AutoCAD drawing was xreferenced to another drawing, saving back to an earlier format alone will not work. However if the xreference is removed and a "Save As" to 2004 or other earlier format, it solves the problem. Using the 2012 format may experience the Unsupported new DWG version ID: "AC1024" error without solution. Turn off any unneeded layers before adding background.

**Access Violation Error**

If files are stored on a network, external storage device or CD, copy down to a local folder. Make sure the folder is a "User" folder (under the user account name folder) such as My Documents. If files are already stored locally and accessibility is not an issue, program version is up to date, then try text and excel file formats under File | Open - File Type.

**Exception EOleSYS Error**

This problem is usually caused by key drivers being incorrectly installed. Make sure the installation files are not in a compressed format (zipped), and that they are located in a full-access read/write folder such as on the computer **Desktop**, then re-run the installation with the latest downloaded installation files.

**Error Can't Open File**

If files are stored on a network, external storage device or CD, copy down to a local folder. Make sure the folder is a "User" folder (under the user account name folder) such as My Documents. If files are already stored locally and accessibility is not an issue, program version is up to date, then try text and excel file formats under File | Open - File Type.

**Fortran Error**

Send p2k file to tech support

**Object is not of the expected type**

Contact Tech Support

**Invalid Argument**

If files are stored on a network, external storage device or CD, copy down to a local folder. Make sure the folder is a "User" folder (under the user account name folder) such as My Documents. If files are already stored locally and accessibility is not an issue, program version is up to date, then try text and excel file formats under File | Open - File Type.

**Floating Point Division by Zero**

Send p2k model file to Tech Support

**Network Key is showing an active user but no users are logged on**

A network WIBU key incorrectly shows a logged-on user (due to an improper shut down while using Pipe2018).

Run the Wibu-Key license manager by clicking START | PROGRAMS | WIBU-KEY | Server-Monitor. When the Server Monitor starts, click the 2nd speed button from the left (with the magnifying glass) to Browse Network. In the left window pane, click on the name of the server (that has the key plugged in) and a list of active users will appear below it. Click on the user that has terminated abnormally, then click the button marked "Cancel User"

A network USB key incorrectly shows a logged-on user.  In this case, simply click on the user.

**Print to PDF Problem**

Repeat the attempt to print to PDF at least two more times.  If the problem does not self-correct, go the to C:\Program Files(x86)\KYPipe\Pipe2018\PDF and run the batch file called "RegisterPDF.bat".

**I/O Error**

Send the p2k model file to Tech Support

## 1.3.1.9 Hotkeys and Shortcuts

| | |
|---|---|
| List of Shortcuts | Ctrl-Alt-T |
| **Editing** | |
| Apply | Alt-A |
| Undo to last apply | Ctrl-U |
| Copy | Alt-C |
| Paste | Alt-V |
| Delete | Alt-Del |
| Insert node | Alt-Ins |
| Error Check | Ctrl-E |
| Connectivity Check | Ctrl-Shift-E |
| Orthogonalize | Ctrl-G |
| Redo last map change | Alt-R |
| Undo last map change | Alt-U |

| | | |
|---|---|---|
| Repeat pipe | Ctrl-R | |
| Move down | Ctrl-down | |
| Move left | Ctrl-left | |
| Move right | Ctrl-right | |
| Move up | Ctrl-up | |
| *Toggle Fixed Mode 2 | Ctrl-F | |
| *Select Everything - Nodes and Pipes | Ctrl-Alt-E | |

**Map View**

| | | |
|---|---|---|
| Show everything | Alt-E | |
| Pan | Alt-N | |
| Pan Arrows | Pans the map to the left (Ctrl+Left arrow), right (Ctrl+Right arrow), up (Ctrl+Up arrow), down (Ctrl+Down arrow). | |
| Pan - Layout Mode | LC and drag. | |
| Pan - Group Mode | Alt + LC and drag | |
| Cursor Notation Toggle | Alt-T | |
| Show background | Alt-S | |
| Hide labels | Ctrl-H | |
| Create contours | Ctrl-Alt-C | |
| Hide contours | Alt-H | |
| Animate | Ctrl-M | |
| Pump Status Emphasis Toggle | Ctrl-Alt-P | Not on Main Menu |
| NFPA Hydrant Color Coding | Ctrl-Alt-H | Not on Main Menu |
| Find Node | Ctrl-N | |
| Find pipe | Ctrl-P | |
| *Node Info Window | Ctrl-Shift-N | Only accesible by short-cut |
| *Pipe Info Window | Ctrl-Shift-P | Only accesible by short-cut |

| | | |
|---|---|---|
| Zoom All | Ctrl-A | |
| *Zoom In | Ctrl-I | |
| *Zoom Out | Ctrl-O | |
| Zoom Previous | Ctrl-Z | |
| Zoom Selected | Ctrl-S | |
| Zoom Window | Ctrl-W | |
| * Saved Views | Ctrl-Alt-V | |
| * Toggle All Node Images | Ctrl-Alt-Z | |
| Toggle Intermediate Nodes | Ctrl-Shift-I | |

**Features**

| | | |
|---|---|---|
| Add to presentation | Ctrl-Alt-A | |
| Load optimal diameters | Ctrl-D | |
| Show screen.txt file | Ctrl-F7 | |
| Create Profile | Hold down the Ctrl key and select two nodes, release and type Alt-L | |
| Pipe Break | Ctrl-B | |
| *Settings Menu | Ctrl-Alt-S | Only accesible by short-cut |
| Surge Output Table | Ctrl-T | Not on Main Menu |
| Google Earth Export | Alt-G | Not on Main Menu |
| Show SCADA Status Charts | Ctrl-Shift-S | Not on Main Menu |
| Add to Presentation | Ctrl-Alt-A | Not on Main Menu |
| *Go from Classic to Kynetic | Shift-F3 | Only accesible by short-cut |
| *View Test.txt | Ctrl-F8 | Only accesible by short-cut |
| *Complete Inventory | Alt-I | |
| *Selected Inventory | Ctrl-Alt-I | |
| *Display Node Graph | Ctrl-Alt-G | Hidden (Kynetic Only) |
| *Surge Analysis | Ctrl-Shift-F7 | |

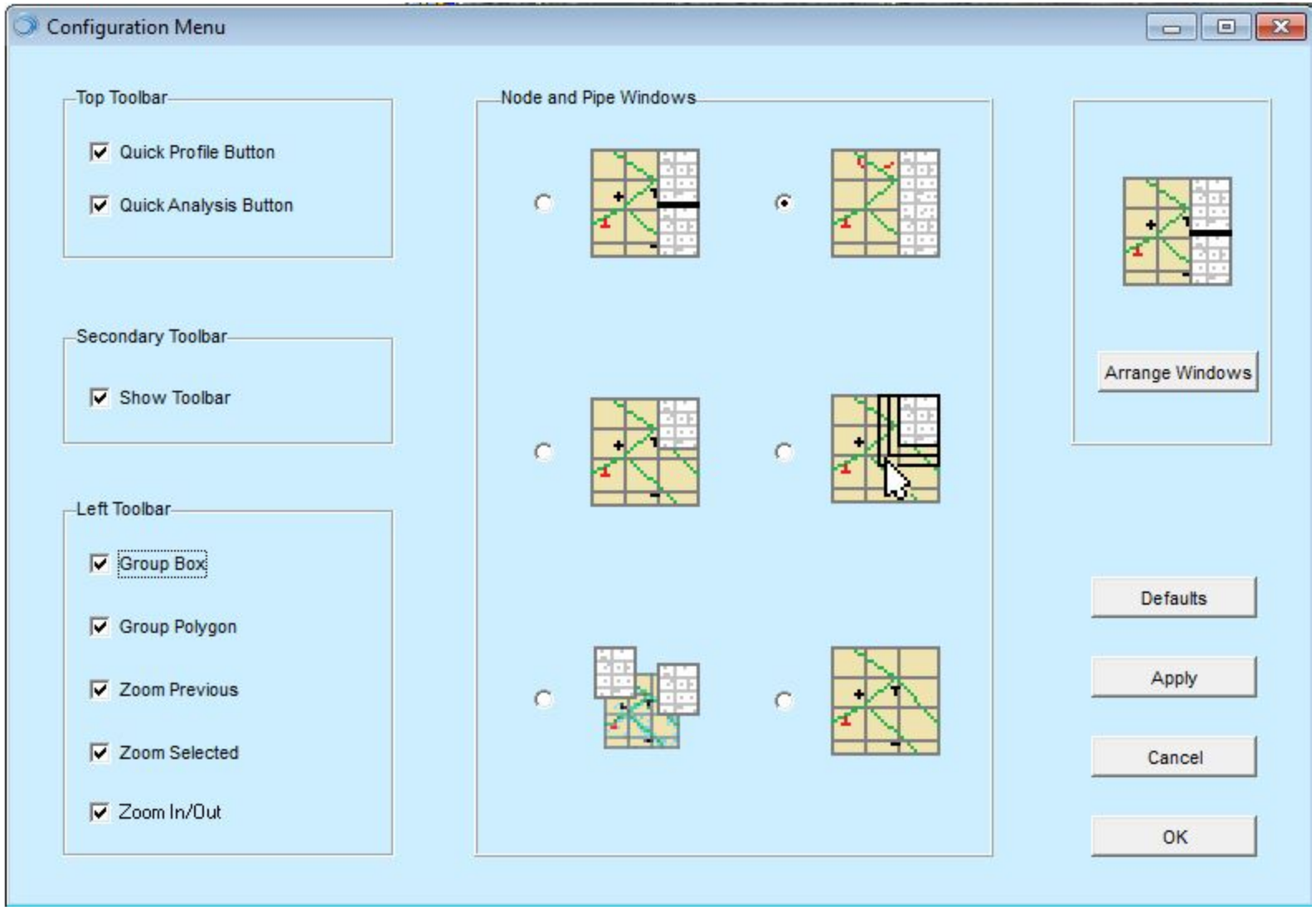
| | |
|---|---|
| *Surge Steady State | Ctrl-Shift-F8 |

* Hidden

## 1.3.2 Files, Printing, Menus

### 1.3.2.1 Menus and Tabs

#### 1.3.2.1.1 KYnetic Configuration Menu




Turn on/off Toolbar icons and Secondary Toolbar

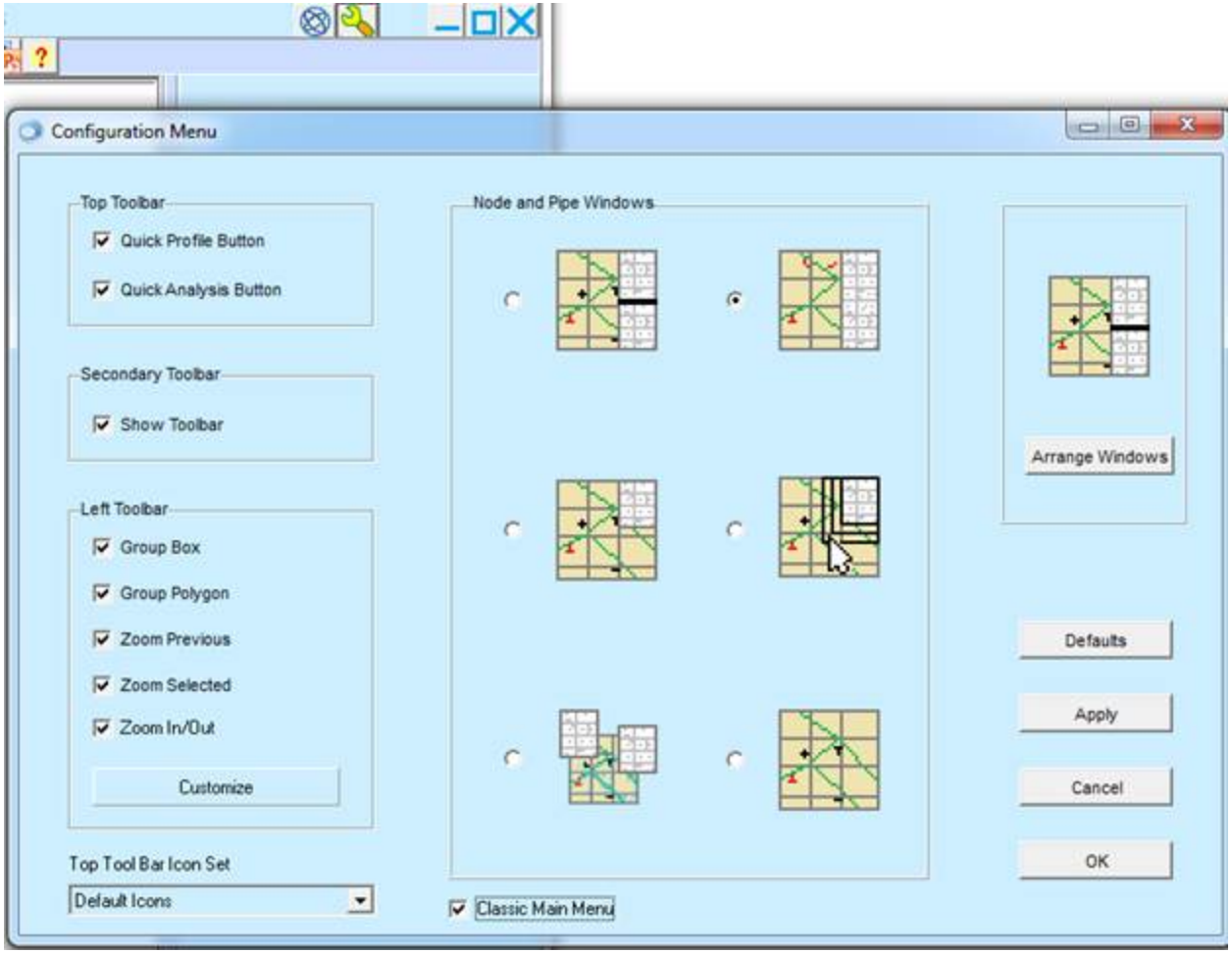
Set the position and appearance of the Node and Pipe Info windows.

### 1.3.2.1.2 Classic Menu in KYnetic Interface

# Classic Menu in KYnetic Interface

Click on Configuration menu .

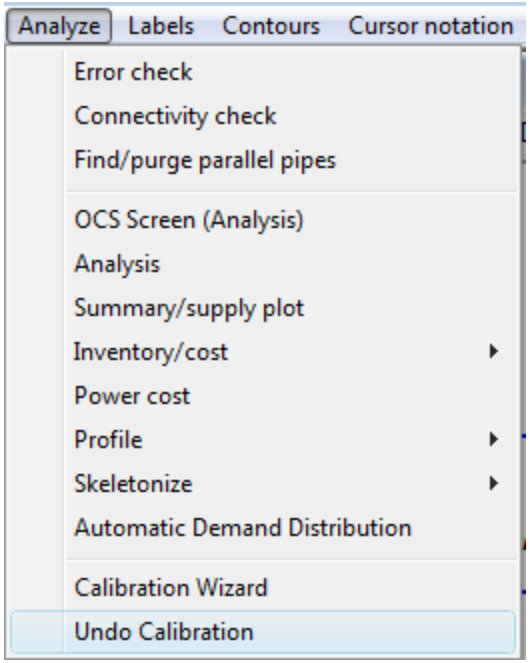
Check box at bottom “Classic Main Menu”

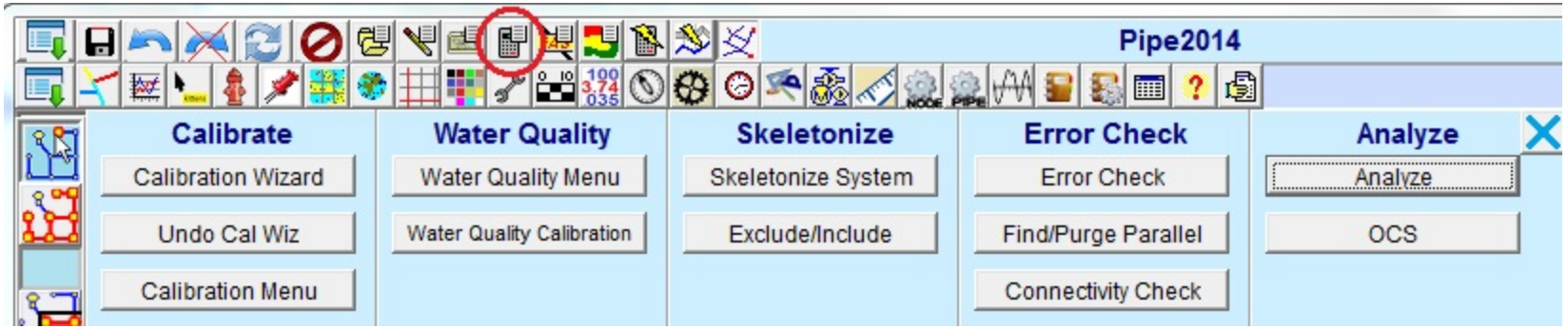


Classic main menu bar now is added to KYnetic interface:

## 1.3.2.1.3 Analyze (Main Menu)

Analyze (Main Menu)






Error Check

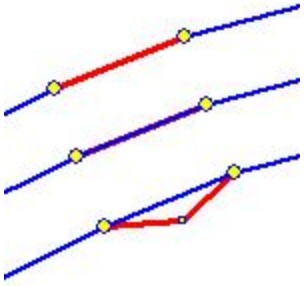
This selection will intelligently evaluate your data file and check for various errors in the system layout or data.

Connectivity Check

When a pipe is selected in a system, this feature checks to make sure every other pipe in the system is connected to the selected pipe. This is particularly useful in checking newly imported data from another source (e.g. Excel, AutoCAD). The disconnected pipes will also be noted in the Data Table (click on the Table button to the left of the Map Screen). In the Data Table, select Pipes. Check the Selected Items Only box to list disconnected pipes. Alternately, under Hidden data, look in the ~Selected column and disconnected pipes will be noted with a 1.

Find/Purge Parallel Pipes

Sometimes a parallel pipe is hard to see. It might accidently be added to the model:

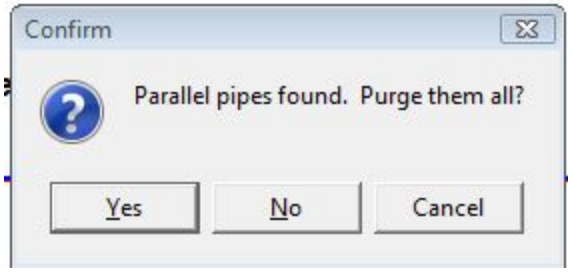


The top pipe is single.

Second pipe is two parallel, except for slight discoloration when the pipe is selected, you would not be able to tell there are two pipes.

Third pipe is also two parallel pipes. An intermediate node has been added and dragged to the side so it is easy to see.

Find/purge parallel pipes deletes parallel pipes. If you use this command Program will say "Parallel pipes found. Purge them all?"



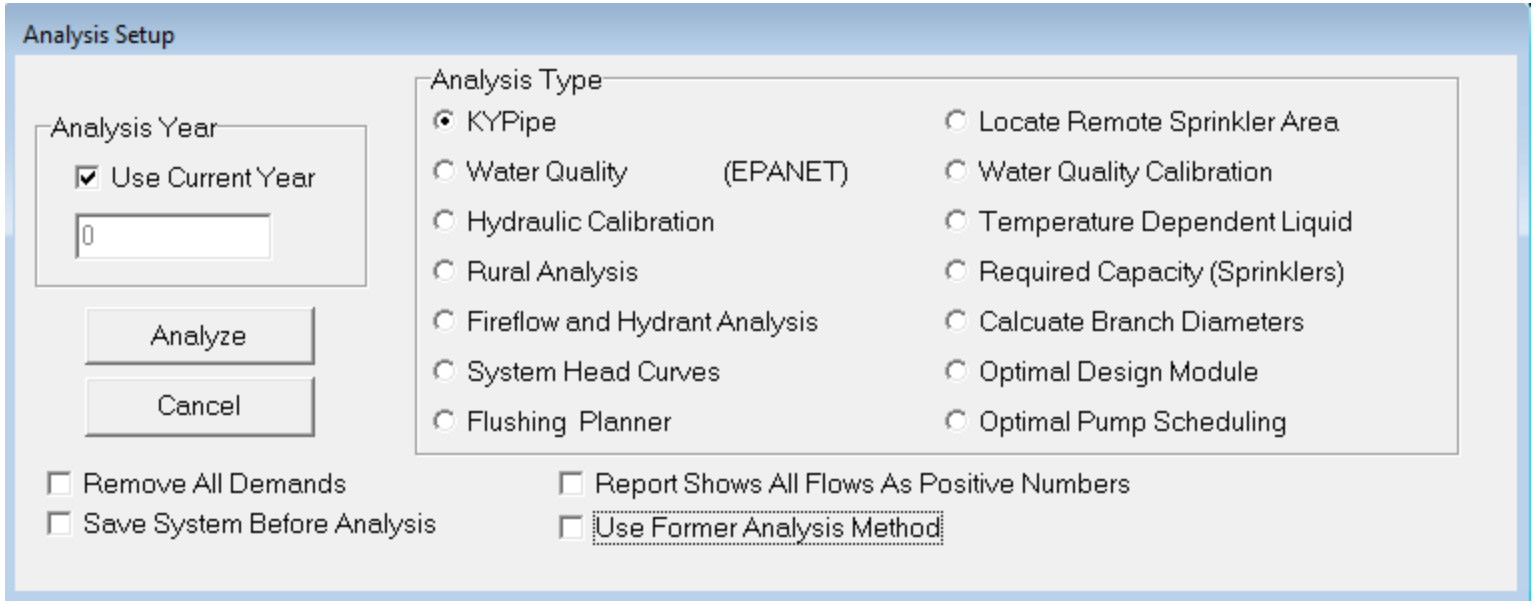

If you say "No" you will have the chance to delete them one at a time.

OCS Screen (Analysis)

**Several options for Hydraulic Analysis are available through the Operational Control Settings screen.**

Analysis

This selection brings up the window below that allows you to perform an analysis of your system. Click on any one of the analysis types for more information.




**Analysis Year** - Year to be used for pipe roughness projection calulations. See Age-Based Roughnesses.

**Remove All Demands** - Allows the user to create a simulation where demands are not applied, without permanently removing demand values from the model.

**User Former Analysis method**

The main difference is the ability to manipulate the loaded set with the following added menu items:

**Load sets of results** - Mainly for EPS or Surge analyses, this is useful to limit the number of cases in the output report when there are a large number of cases.

**Load all times** - Allows a all or a range of cases to be loaded into the report.

**Save System Before Analysis** - prompts the user to save before each analysis is launched.

**Report Shows All Flows as Positive Numbers** - Flow results may have a negative sign to indicate direction of flow. Each pipe has a Node 1 and a Node 2. If flow goes from Node 2 to Node 1 flow is reported as negative. Checking this box will remove any negative signs from the flow results.

Summary/supply plot

Used for GoFlow (Fire Sprinkler) Systems. Industry standard graphs and report. In KYnetic this is under Tables, Graphs and Profiles menu

Inventory/Cost - see Cost and Inventory Calculations

This selection causes a calculation to be performed that tabulates the following:

1) for each pipe type; the total length used and number of pipes
2) total cost for each pipe type
3) total cost for all pipes in system
4) an inventory of system elements

Power Cost - see Cost and Inventory Calculations

This selection performs a calculation that tabulates the costs (of electric power) of the operation of the pumps in your system. These calculations are based on the efficiency data entered for each pump as well as the electricity costs. This option is only available for EPS simulations as the cost is based on the time of operation.

Profile - see Profile In KYnetic this is under Table, Graphs and Profile menu

Skeletonize - see Skeletonize

Automatic Demand Distribution

Water Quality

Water Quality Calibration

Calibration Wizard / Undo Calibration

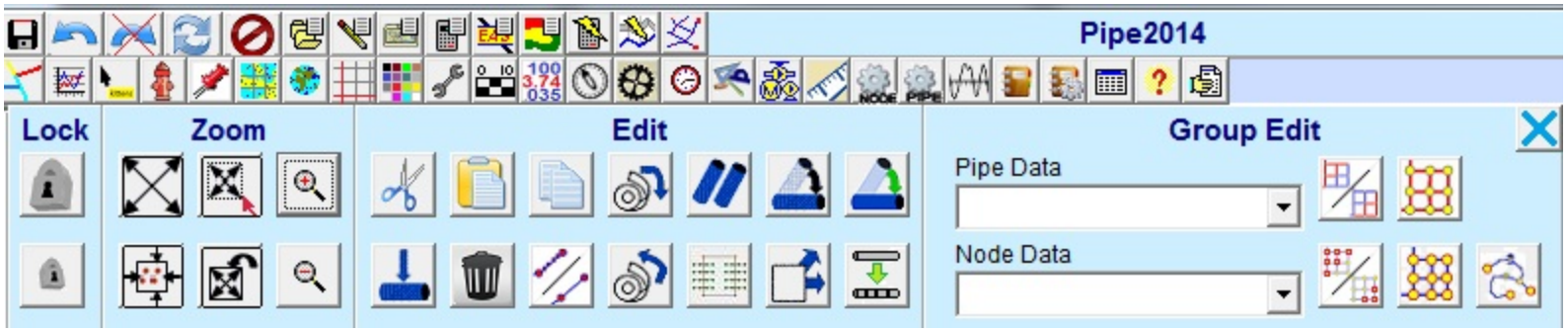
## 1.3.2.1.4 Edit (Main Menu)

Edit (Main Menu)

File Edit View Analyze Move Labels Facilities Management Tools Help

---

KYnetic Interface:




Lock

Fixed Mode - Cannot change or edit map view and some input data.
Fixed Mode 2 - Fixed but can add and delete nodes

Zoom - Standard Zoom functions

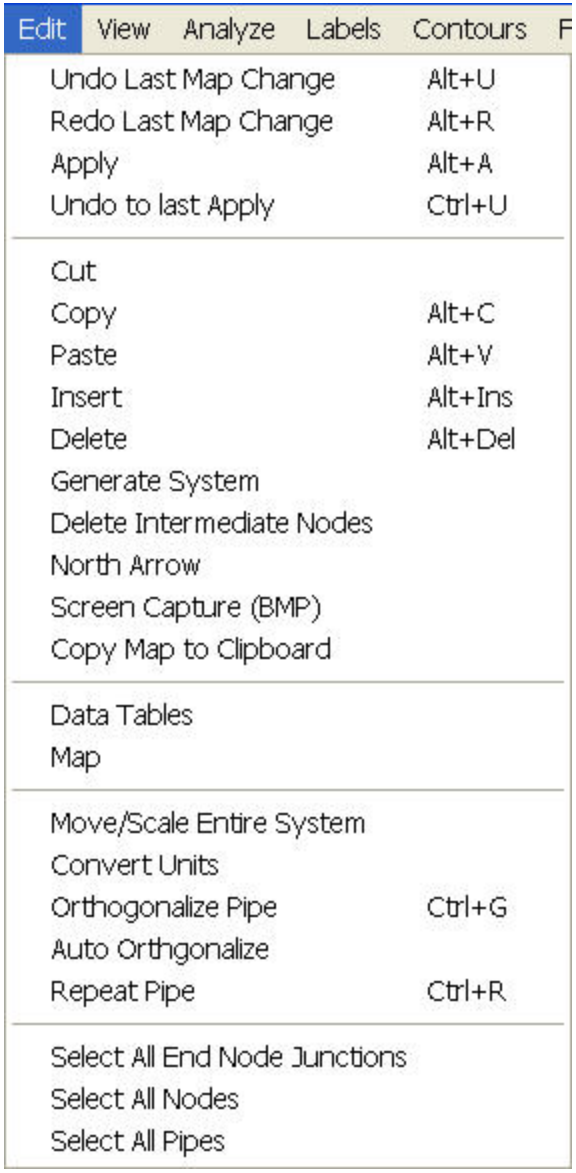
Edit

Cut/Copy/Paste - can copy paste all or part of a model. Close current model, open new existing model and paste in.
Apply/Undo - Will undo all map changes to last apply.
Repeat pipe - see Classic description below
Orthogonalize - pipe snaps to nearest vertical or horizontal
Auto Orthogonalize - automatically snap all pipes to nearest vertical or horizontal as they are laid out
Insert node
Delete Pipe
Delete Intermediate Nodes
Generate System
Move/Scale System - see Classic description below
Insert Internal nodes - Specify a minimum pipe length, then all pipes (or pipes selected in group mode) will be divded into length equal to or smaller than the minimum pipe length specified. Elevations are interpolated from existing data.

Group Edit

Group Edit Functions

**Classic Interface**:




Undo Last Map Change

**Undo map changes (up to three changes). Does not include input to data fields in the Information windows.**

Redo Last Map Change

**Redo map changes (up to three undone changes). Does not include input or deletions from to data fields in the Information windows.**

Apply

This selection causes the changes to the data file to be updated into the spreadsheets.

Undo to last Apply

This selection causes the data file to be restored to the state when the last Apply was performed.

Cut

This selection removes the currently selected cells from the spreadsheet and places them onto the Windows clipboard. This is only applicable to the data tables.

Copy

This selection causes the currently selected cells from the spreadsheet to be copied onto the Windows clipboard. This is only applicable to the data tables.

Paste

This selection causes the spreadsheet cells on the clipboard to be pasted into the spreadsheet starting at the currently selected cell.

### Insert

Inserts an intermediate node into a pipe at the last selected location.

### Delete

Deletes the last selected node(s) or pipe(s).

### Generate System

For GoFlow users. Creates a fire sprinkler system based on various grid types (grid, tree, loop) with characteristics specified by user.

Delete Intermediate Node

**This allows the user to delete all or a portion of the intermediate nodes in the system. If the user declines to delete all of them, they will be prompted for a total number to delete. The individual nodes to be deleted are chosen based on pipe-link (or pipe segment) length. For instance, if the user specifies 20 intermediate nodes to be deleted, then Pipe2018 finds the shortest pipe-link in the system, deletes one of the intermediate nodes from that link, then looks for the next shortest and so on until 20 intermediates nodes have been deleted.**

North Arrow

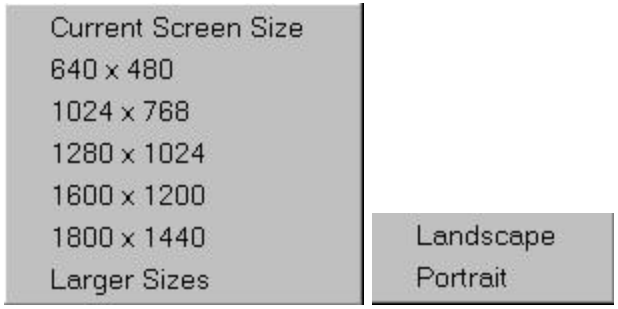
**Places a north arrow on the map for both viewing and printing.**

Pushpin notes

**see Pushpin Notes**

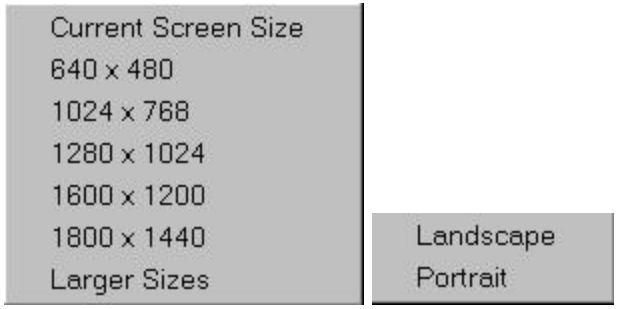
Screen Capture

**Allows the user to capture a bitmap of the map screen. The user is prompted with the specification choices below. Then a bitmap with the file name with a number (filename_1.bmp) will be saved in the file folder where the p2k model file is located.**




Copy Map to Clipboard

**Copies the current view to clipboard. Several size options are available. The current view will be cropped to fit within the selected size.**




Data Tables

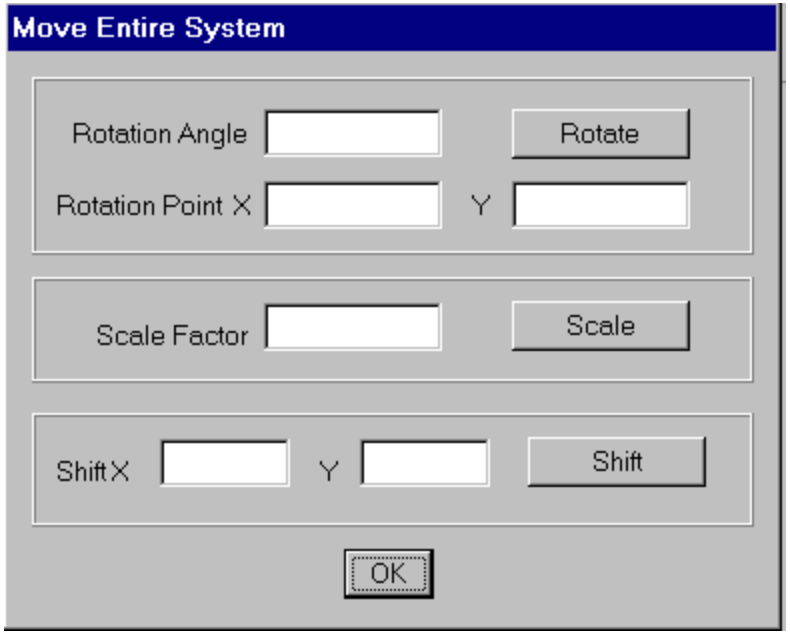
This selection causes the view to change to display the data tables (spreadsheets).

Map

This selection causes the view to change to display the system drawing.

Move/Scale Entire System

This selection causes the following dialog box to display which allows you to shift, scale, or rotate your piping system.




Orthogonalize Pipe

This feature causes the selected pipe to be orthogonalized to the nearest horizontal or vertical position. The node to be moved which is connected to the pipe must also be selected.

**Auto Orthogonalize**

All pipes created with a new node while this feature is on (shown with a ✔) will be orthogonalized to the nearest horizontal and vertical position

Repeat Pipe

**Select node and pipe. Starting at the selected node it will create a duplicate of the selected pipe in the same orientation. The selected node will be Node 1 of the new pipe and the new node will be Node 2. Useful when laying out grid-type systems.**

Select All End Node Junctions

**Selects with Group mode all junctions which occur at the end of a pipeline, i.e. connected to one pipe only.**

Select All Nodes

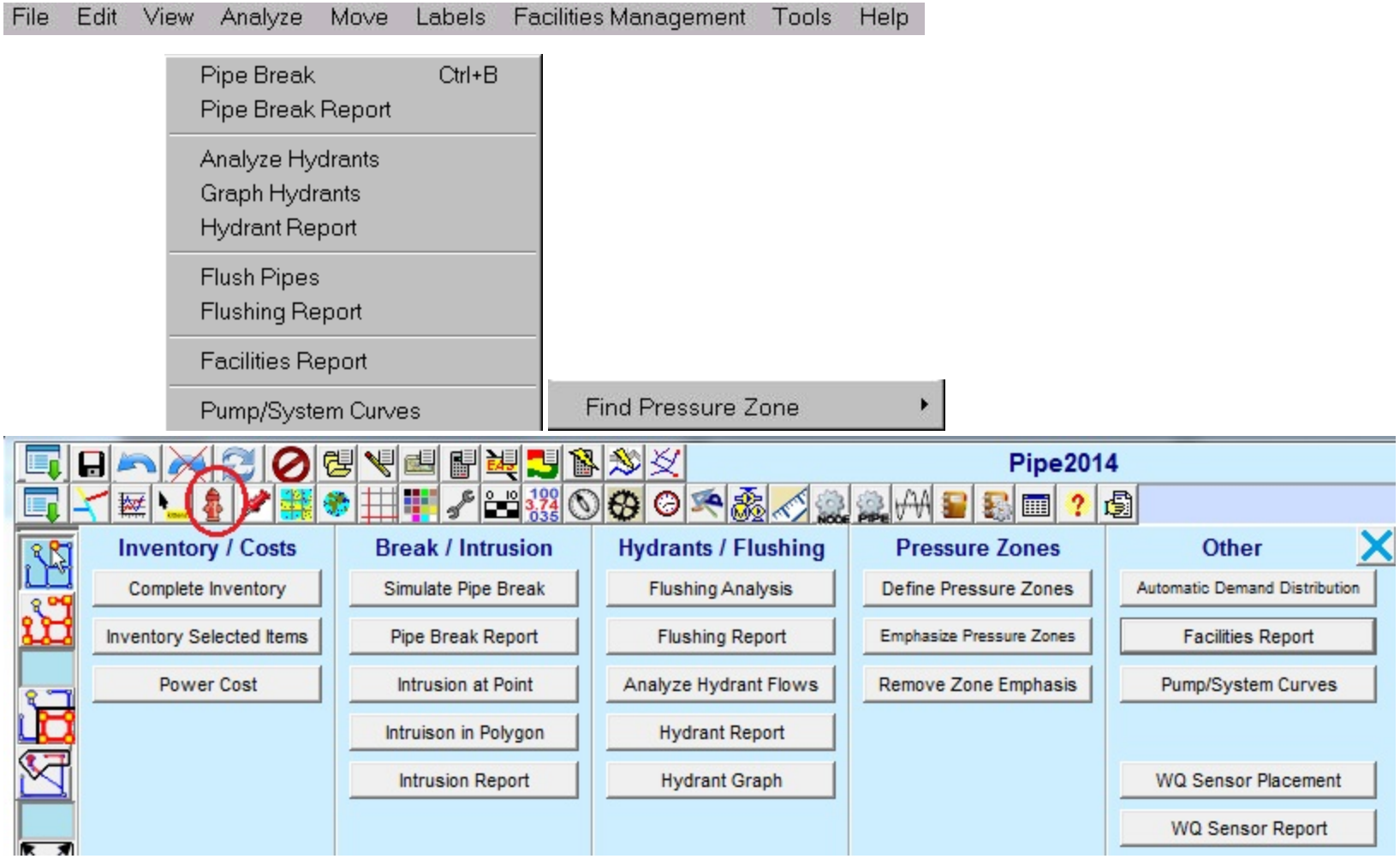
**Selects with Group mode all nodes in a model.**

Select All Pipes

**Selects with Group mode all pipes in a model.**

## 1.3.2.1.5 Facilities Management (Main Menu)

Facilities Management (Main Menu)



Pipe Break

**See Pipe Break. This selection allows you to click on a pipe to simulate a pipe break.**

**Pipe Break Report**
Provides a report of the valves that must be operated to contain the simulated break.

Analyze Hydrants

**See Hydrant Flows. This selection allows you to select hydrants and get calculated flow information for a set pressure.**

Graph Hydrants

**Provides a graph of all the hydrants which were selected and analyzed.**

Hydrant Report

**Provides a hydrant report for all of the hydrants which were selected and analyzed.**

Flush Pipes

**See Flushing Pipes**

Flushing Report

**See Flushing Pipes**

Facilities Report

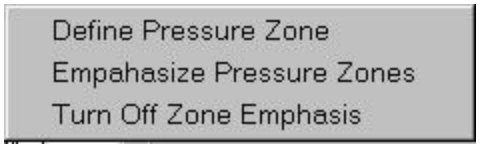
**Allows the user to click on a device and generate a detailed report.**

Pump/System Curves

**See Pump and System Curves. Details how to use pump curves to identify pumps in the system and how to create system curves.**

Find Pressure Zone

**Allows the user to define and emphasize pressure zone in the system. When Define Pressure Zone is selected, the user is prompted to click on a pipe within the pressure zone that is to be defined. A number is assigned to the zone and then user is given the option to emphasize the zone. This is done using the Pipe Emphasis feature.**



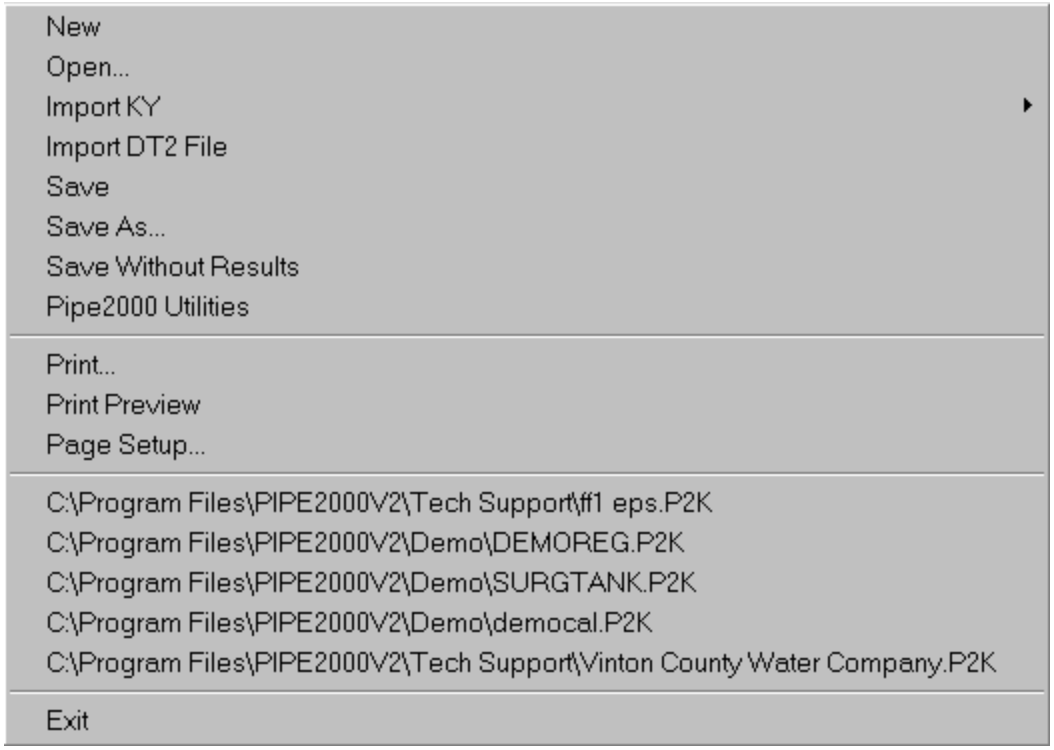

Inventory / Cost

Water Quality Sensor Placement

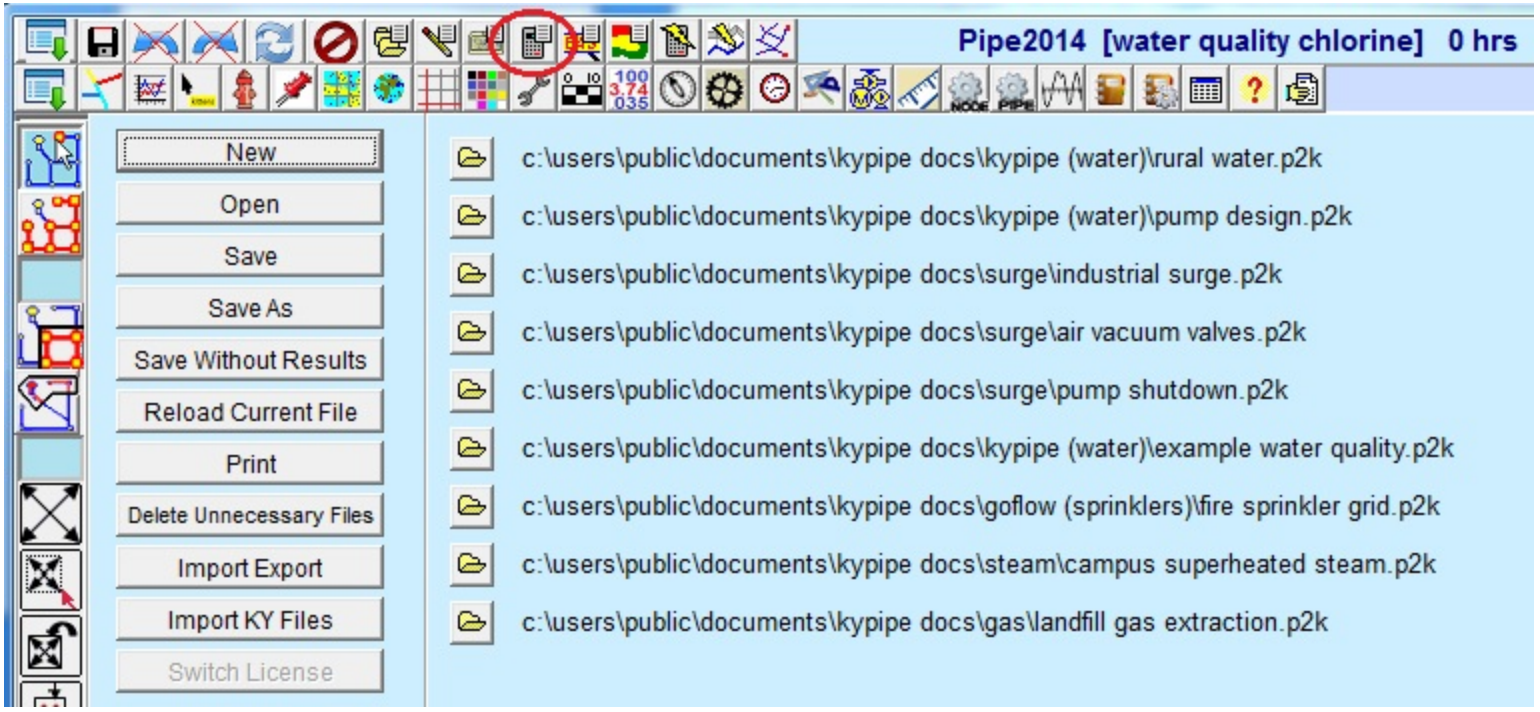
## 1.3.2.1.6 File (Main Menu)

File (Main Menu)

Classic File menu:



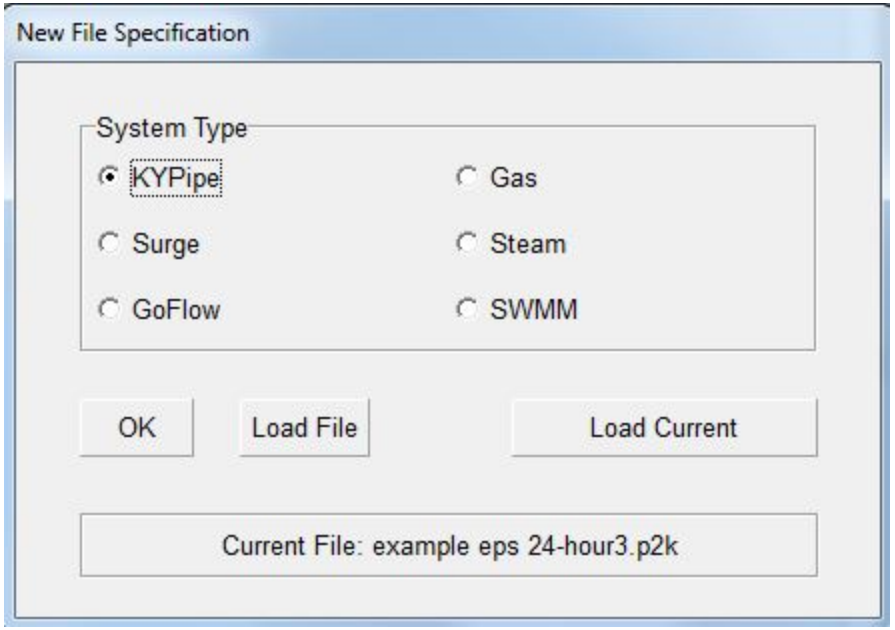

KYnetic File menu:




The file submenu controls all file interaction and printing.

New

This removes the currently loaded data file and prepares Pipe2018 to initiate development of a new system. Note the the default lists for all values (fittings, pipe types, etc) will be reloaded. The following menu is shown which allows you to setup some general system options. Options set using this menu can be changed later.



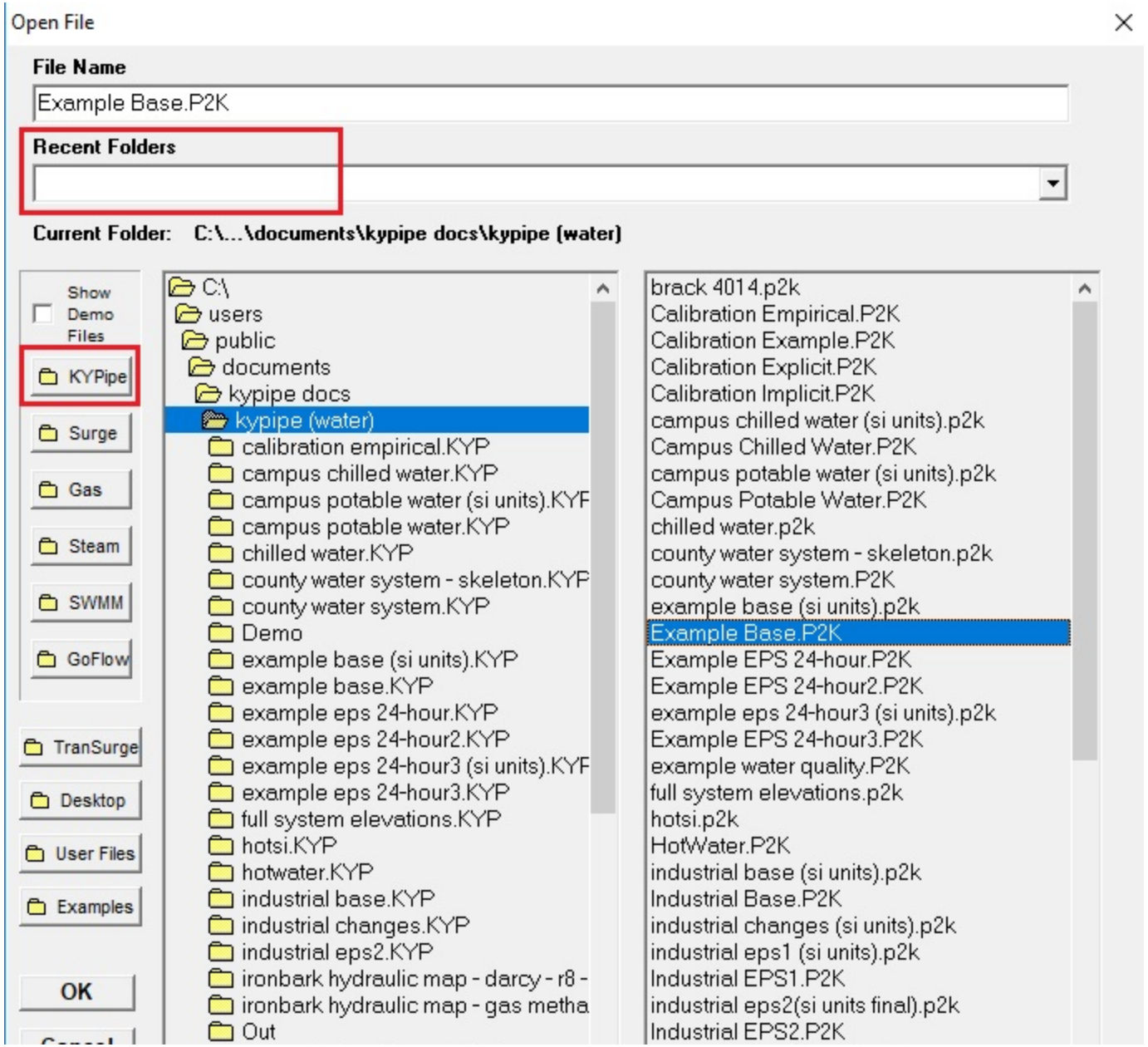

Open

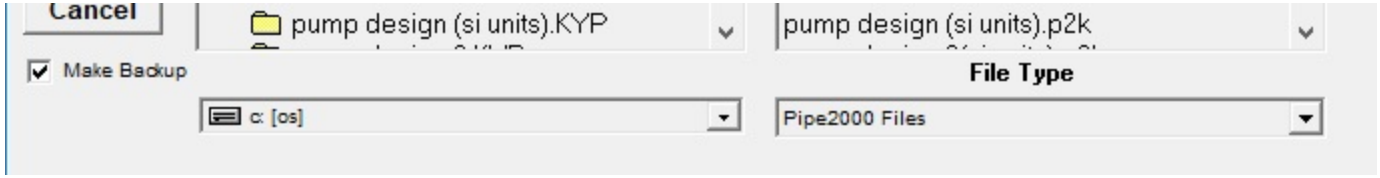
This brings up a file menu that allows you to reload a previously saved .p2k model file.

Two significant features of the File Open window are:

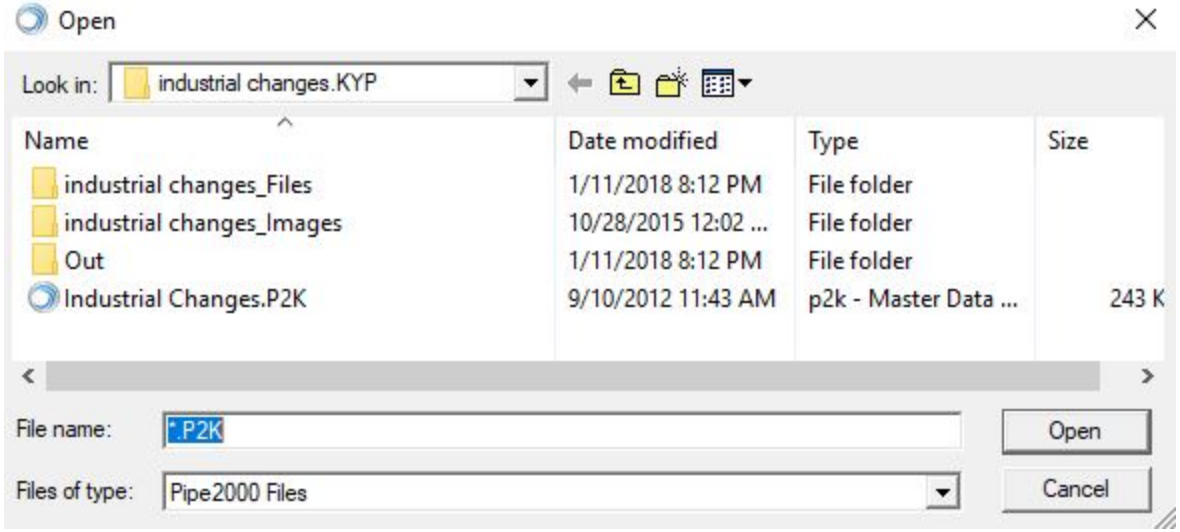
Recent Folders - added in Pipe2018, to locate folders that were previously used for loading and saving .p2k model files.

Demo buttons - KYPipe, Surge, etc. bring the user to a folder of model files which are installed along with the program. Many of these are described in the Examples manual which may be downloaded under Help.


Open File
File Name
Example Base.P2K
Recent Folders
Current Folder: C:\...\documents\kypipe docs\kypipe (water)
Show Demo Files
KYPipe
Surge
Gas
Steam
SWMM
GoFlow
TranSurge
Desktop
User Files
Examples
OK
C:\
users
public
documents
kypipe docs
kypipe (water)
calibration empirical.KYP
campus chilled water.KYP
campus potable water (si units).KYP
campus potable water.KYP
chilled water.KYP
county water system - skeleton.KYP
county water system.KYP
Demo
example base (si units).KYP
example base.KYP
example eps 24-hour.KYP
example eps 24-hour2.KYP
example eps 24-hour3 (si units).KYP
example eps 24-hour3.KYP
full system elevations.KYP
hotsi.KYP
hotwater.KYP
industrial base.KYP
industrial changes.KYP
industrial eps2.KYP
ironbark hydraulic map - darcy - r8 -
ironbark hydraulic map - gas metha
Out
brack 4014.p2k
Calibration Empirical.P2K
Calibration Example.P2K
Calibration Explicit.P2K
Calibration Implicit.P2K
campus chilled water (si units).p2k
Campus Chilled Water.P2K
campus potable water (si units).p2k
Campus Potable Water.P2K
chilled water.p2k
county water system - skeleton.p2k
county water system.P2K
example base (si units).p2k
Example Base.P2K
Example EPS 24-hour.P2K
Example EPS 24-hour2.P2K
example eps 24-hour3 (si units).p2k
Example EPS 24-hour3.P2K
example water quality.P2K
full system elevations.p2k
hotsi.p2k
HotWater.P2K
industrial base (si units).p2k
Industrial Base.P2K
industrial changes (si units).p2k
industrial eps1 (si units).p2k
Industrial EPS1.P2K
industrial eps2(si units final).p2k
Industrial EPS2.P2K




In the Preferences Menu is an option to show a standard file open window as shown:




Import Export (KYnetic)

Brings up available data exchange options

Import KY

This selection brings up a file selector that allows you to import a KY data file from a previous version of KYPIPE directly into Pipe2018. See Import KY.

Import DT2 File (version 2)

**This may be used if p2k and BK1 (back-up) files are lost. Any time an analysis is done, a dt2 file is created. This file can be imported and baseline data can be retrieved. Changes and demand patterns and some Surge devices will not be able to be re-created. Several of the import**

**options, e.g. EPANET or Watercad, are two step, the second step being importing a DT2 file that has been created.**

Save

This selection saves your Pipe2018 data file using the current filename.

Save As

This selection brings up a file menu that allows you to save your Pipe2018 data file as a new file name.

Save Without Results

Allows user to save the system data without attaching the latest set of results

Pipe2000 Utilities (Classic)

This selection brings up a menu of utilities used to import and export data. See Pipe2000 Utilities / Data Exchange.

Print

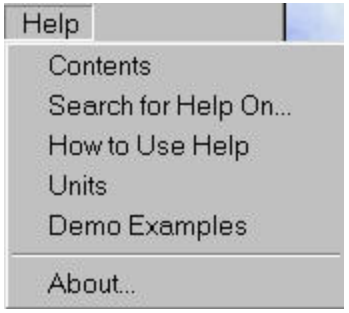
This selection causes your current view to be sent to the printer.

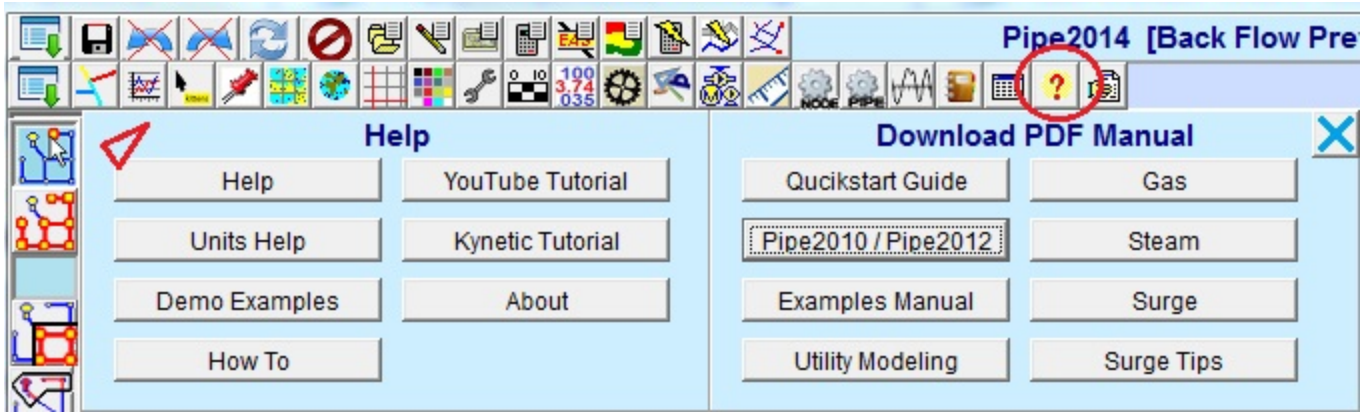
Exit

This selection exits the Pipe2018 program.

## 1.3.2.1.7 Help (Main Menu)

Help (Main Menu)




Contents

This selection brings up the table of contents for the Pipe2018 help file.

Search for Help On

This selection brings up an Index that allows users to search the Pipe2018 help system for a particular topic.

How to Use Help

This selection brings up information on how to use the Pipe2018 help system.

Units

**This selection brings up a table of units for Pipe2018. The units for a particular p2k file are based on the selected flow units for that system.**

Demo Examples

**This selection goes through the individual demonstration files. These demos are found by clicking the various model buttons in the File - Open box. Step-by-step procedures are given for various Pipe2018 features.**

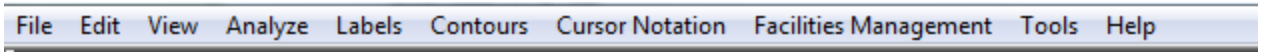
About

This selection brings up information about the Pipe2018 program, including the version number and the number of pipes for which the user is licensed..

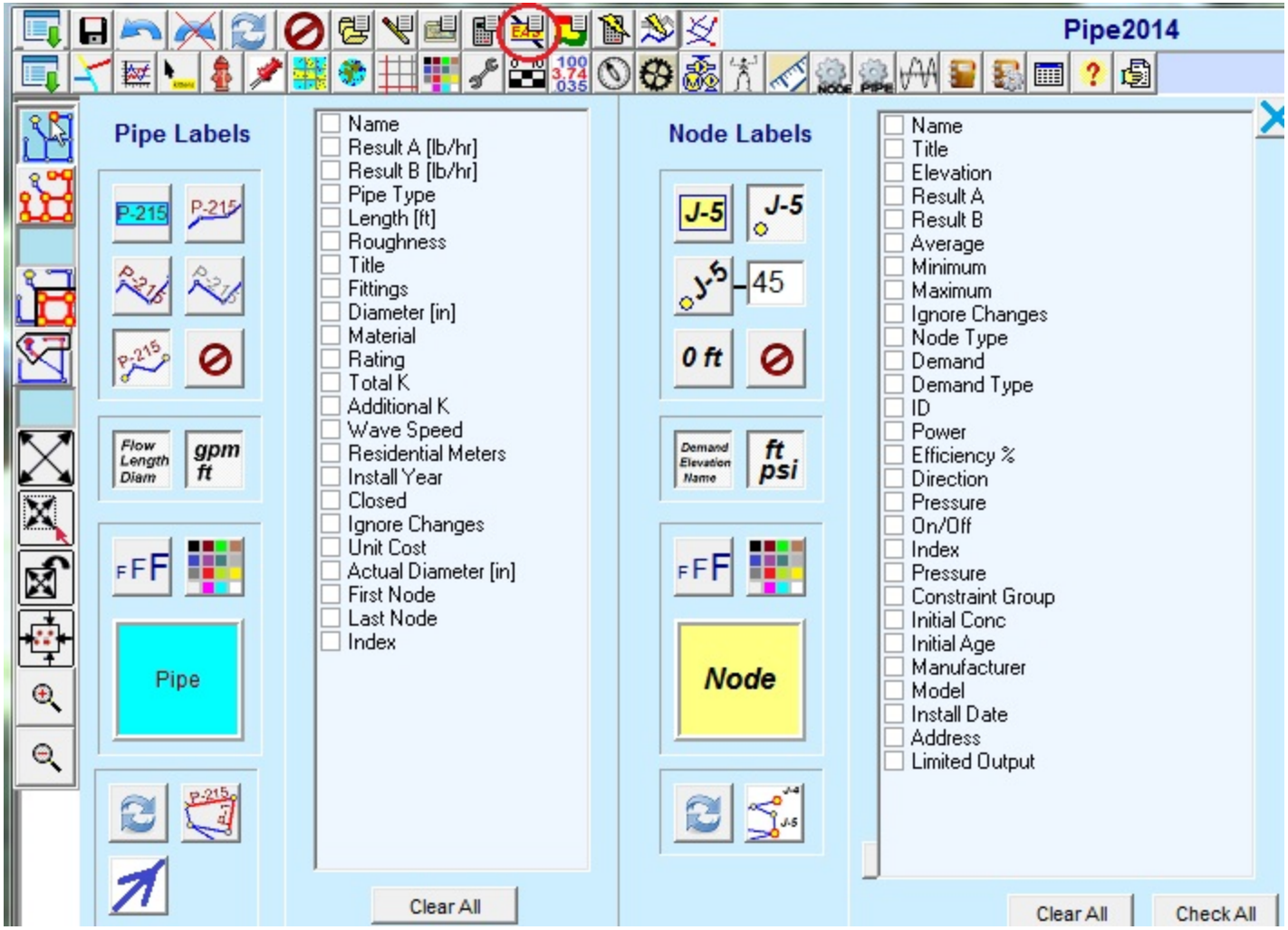
See also About the Pipe2018 Online Help

## 1.3.2.1.8 Labels (Main Menu)

Labels (Main Menu)



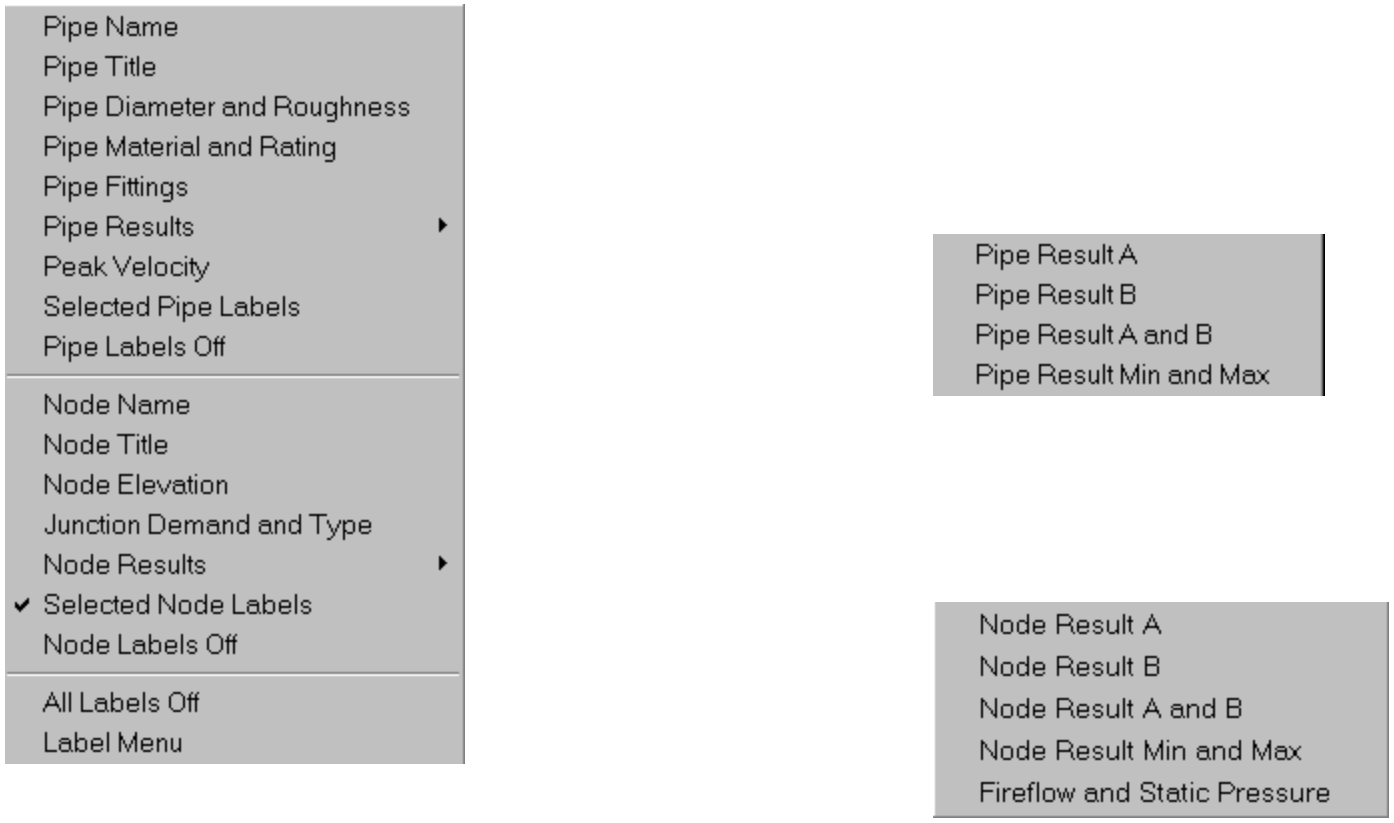

KYnetic Labels menu:




See the Labels video on the KYPipe web site.

---

These selections provide quick access to commonly used choices for information labels that are shown for the nodes and pipes on the map and on printouts. A much wider range of choices is available using the ***Other Pipe Labels*** and ***Other Node Labels*** and selections. Note for Results Selections the specific parameters to be displayed are set using the Results Selector Bar located at the bottom of the display.




**Pipe (Node) Name**

Displays the pipe (node) names

**Pipe (Node) Title**

Displays the pipe (node) title

**Pipe Diameter and Roughness**

Displays the pipe diameters and roughnesses

**Pipe Material and Rating**

Displays the pipe materials and ratings

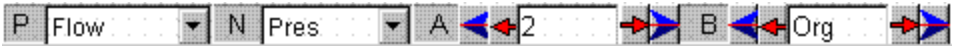
**Pipe Fittings**

Displays the fittings symbols

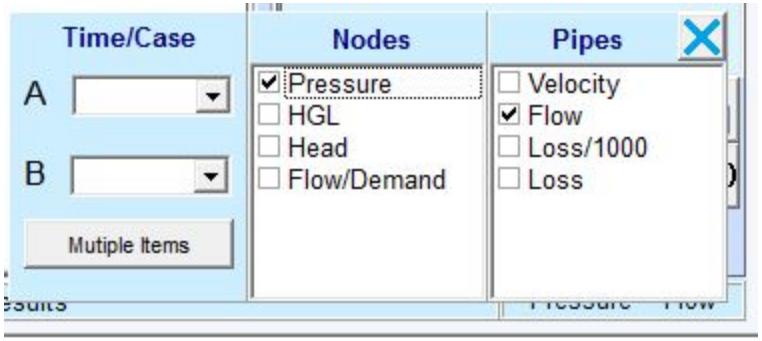
**Pipe (Node) Results**

Displays the selected results. The results are selected using the Result Selector boxes at the bottom of the Map screen.

Classic:




KYnetic:




See also Result Selector Bar.

The "P" drop-down box gives a list of pipe results from which the user may choose for display. The "N" drop-down box gives a list of node results. The "A" and "B" selectors are used to choose the simulation case (if simulation changes or an EPS simulation have been specified) for which the user would like to display the results. For nodes with both and inlet and an outlet result, the displayed result may be selected within the Node Result box.

**Peak Velocity**

Pans to the pipe with the highest velocity for the select case (as determined by the Results Selector Bar)

**Selected Pipe (Node) Labels**

Limits the currently displayed labels to those selected within Group Mode.

**Pipe (Node) Labels Off**

Turns all labels off

**Other Pipe (Node) Labels**

Takes the user to the Map Settings/Labels tab for advanced settings.

**Node Elevation**

Displays the node elevations

**Junction Demand and Type**

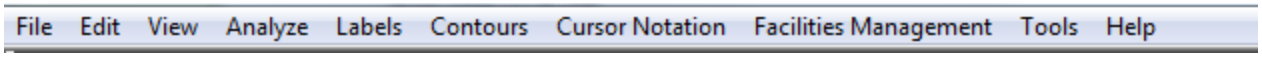
Displays the node demands and types

**Label Menu**

Takes the user to Map Settings | Labels tab for advanced labels settings

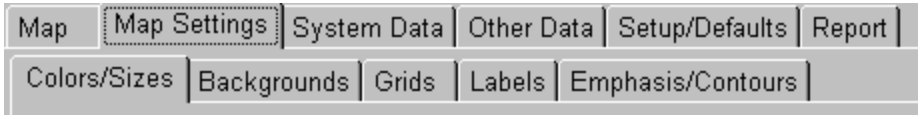
## 1.3.2.1.9 Main Menu

Classic:




The main menu at the top of the window provides access to many functions that control the operation of PIPE2018. Click on the menu items for more information.

## 1.3.2.1.10 MAP SETTINGS




**Classic interface: The Map Settings tab contains 5 screens that affect the appearance of the map.**

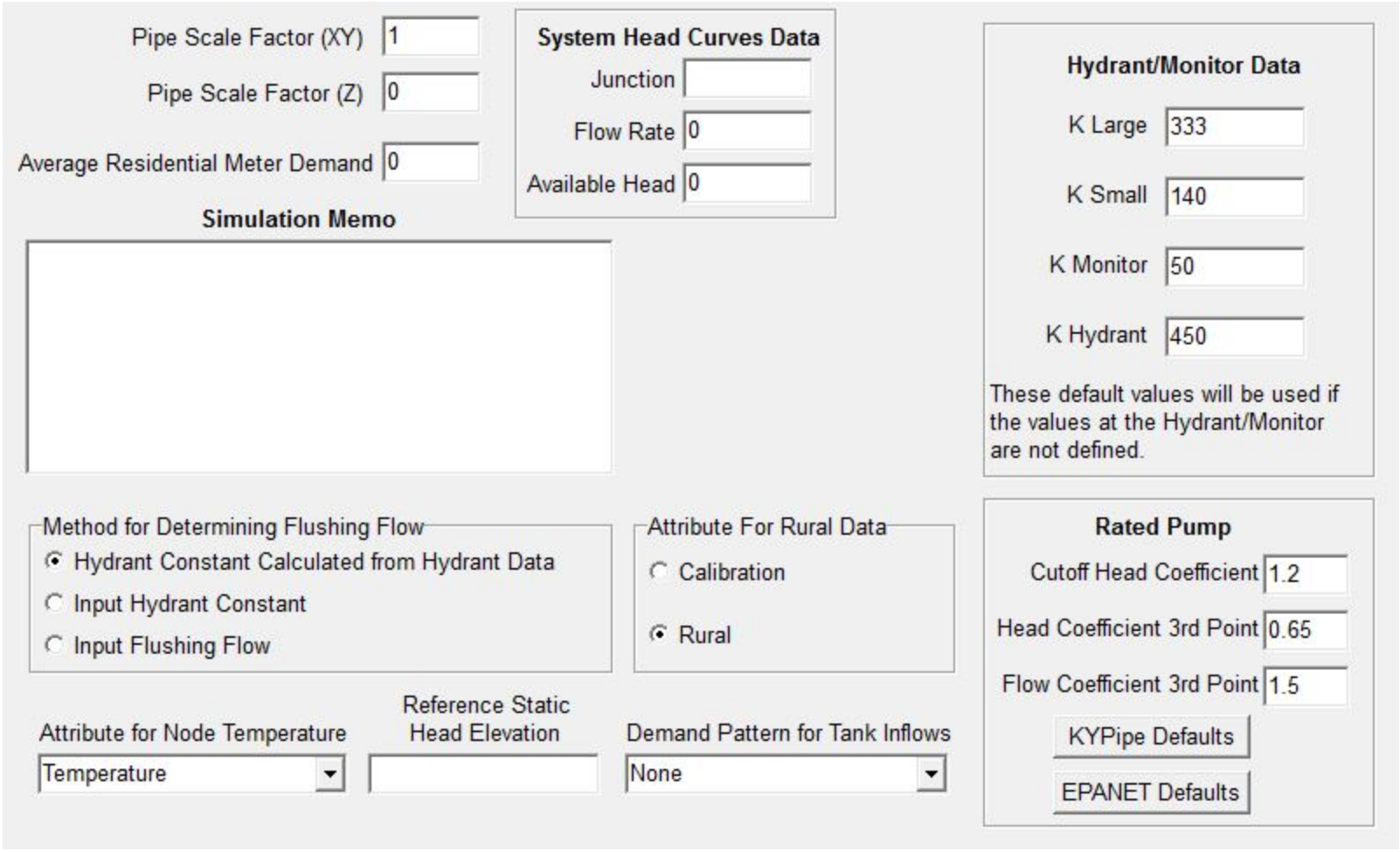
---

## 1.3.2.1.11 Other

Other

KYnetic Other menu icon:

Classic: System Data | Other




Pipe Scale Factor (XY)

This entry can be used to change the x,y or planar scaled lengths of the pipes by this factor. This may be useful, for example, to change length units (from meters to feet). It also may be used to scale an existing piping system to a newly added background map with a different scale.

Pipe Scale Factor (Z)

**This entry is used to set a scale factor for the z coordinate, thereby taking elevation into account when calculating the scaled lengths of the**

**pipes. (With the exception that this will not be done if the user indicates the pipe length is fixed.) The default value is zero, meaning the elevation is not taken into consideration when calculating pipe length. In general, this scale factor should be the same as the Pipe Scale x,y. However, it is useful to be able to be able to set the z scale independently of the x,y scale, for example, when a piping system is laid out using a vector background with a specialized coordinate system.**

Average Residential Meter Demand

This entry defines the demand you wish to impose for each residential meter (specified in the Pipe Data Boxes). This is normally the average daily demand per residence, in the specified flow units.

Simulation Memo

This box can be used to store any general information about the model or the analysis. Entries will be printed in the Output Report.

**System Head Curves Data**

See Pump and System Curves. This is where the junction node and flow are specified for the calculation of a system head curve

**Method for Determining Flushing Flow**

See Flushing. Choose one of these three options before running a flushing analysis.

**Attribute for Rural Data**

In order to handle loops in branched sections, selected pipes can be designated as using Calibration Group or Rural Group = 9 (User Data). These pipes will then be ignored (excluded pipes) when locating branch sections. The user data items, called either 'Rural' or 'Calibration' may be used to define this pipe group. See Sets and Group Mode and Rural Analysis.

**Attribute for Node Temperature**

See Temperature Dependant Liquid. When running a Temperature-Dependant analysis, the User Attribute used to assign the temperature must be specified.

**Reference Static Head Elevation**

Allows the user to define a reference elevation, and then display the static head based on that reference elevation as a map label or to select groups (using Group mode) based on the static head. For example, if a Tank has a level = 700' and based on that, 700 is used as the Reference Static Head Elevation. A node with an elevation of 650' would have a Static Head = 50' (700 - 650).

**Demand Pattern For Tank Inflows**

When an inflow is specified for a tank, the flow may be modified for an extended period simulation or a Surge simulation. This selector is used to identify a Demand Type which is used to modify tank inflows in the Demand Pattern Table.

**Hydrant Monitor**

Hydrant/Monitor element allows the monitor to operate while connecting a hose to one of the other connections. Hydrant may be analyzed with up to three valves open.

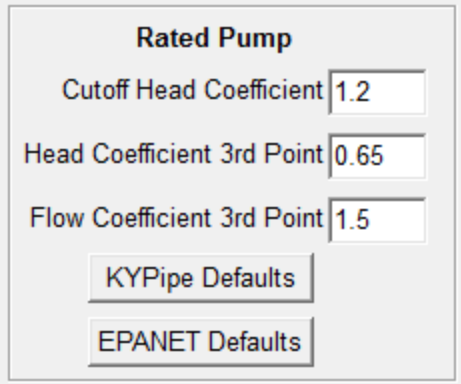
**Rated Pump**

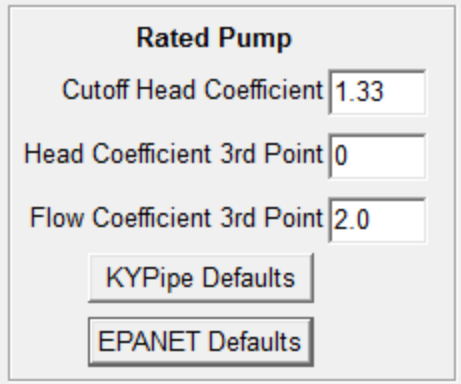
There are three Options for Rated Pump Curves:

Option 1 (KYPIPE Default)
1.) cutoff pressure = 1.2*(rated pressure), flow (cutoff) = 0
2.) rated pressure, rated flow
3.) 0.65*(rated pressure), 1.5*(rated flow

Option 2 (EPANET Default)
1.) cutoff pressure = 1.33*(rated pressure), flow (cutoff) = 0
2.) rated pressure, rated flow
3.) 0.0*(rated pressure), 2.0*(rated flow

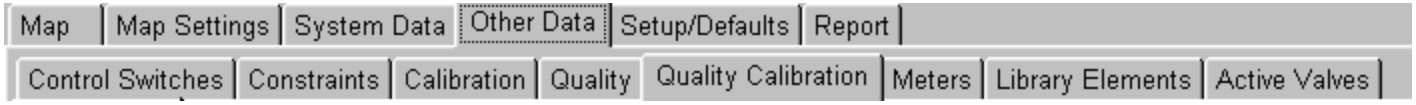
Option 3:
User enter data (shown below))




## 1.3.2.1.12 OTHER DATA

OTHER DATA (Classic Interface)




The OTHER DATA tab contains screens that set values for advanced simulations and devices.

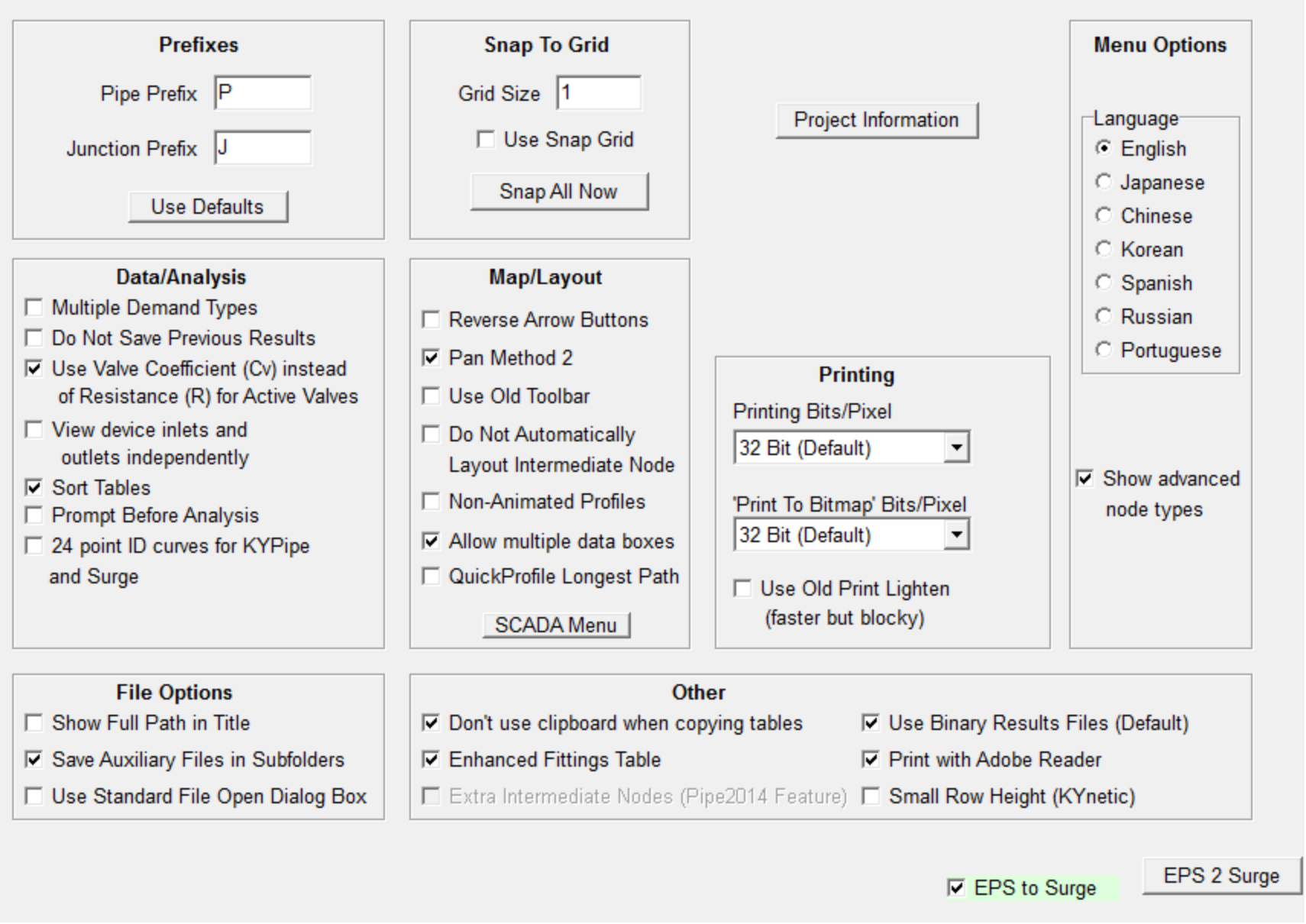
---

## 1.3.2.1.13 Preferences

Preferences settings are applied to the program interface and are not specific to a particular model file.

KYnetic Preferences menu icon:

Classic - System Data - Preferences.


Prefixes
Pipe Prefix P
Junction Prefix J
Use Defaults
Snap To Grid
Grid Size 1
Use Snap Grid
Snap All Now
Project Information
Menu Options
Language
English
Japanese
Chinese
Korean
Spanish
Russian
Portuguese
Data/Analysis
Multiple Demand Types
Do Not Save Previous Results
Use Valve Coefficient (Cv) instead of Resistance (R) for Active Valves
View device inlets and outlets independently
Sort Tables
Prompt Before Analysis
24 point ID curves for KYPipe and Surge
Map/Layout
Reverse Arrow Buttons
Pan Method 2
Use Old Toolbar
Do Not Automatically Layout Intermediate Node
Non-Animated Profiles
Allow multiple data boxes
QuickProfile Longest Path
SCADA Menu
Printing
Printing Bits/Pixel
32 Bit (Default)
'Print To Bitmap' Bits/Pixel
32 Bit (Default)
Use Old Print Lighten (faster but blocky)
Show advanced node types
File Options
Show Full Path in Title
Save Auxiliary Files in Subfolders
Use Standard File Open Dialog Box
Other
Don't use clipboard when copying tables
Enhanced Fittings Table
Extra Intermediate Nodes (Pipe2014 Feature)
Use Binary Results Files (Default)
Print with Adobe Reader
Small Row Height (KYnetic)
EPS to Surge
EPS 2 Surge

See also Surge - System Data \ Preferences

---

### Prefixes

You can choose a prefix for pipe or junction names which will be automatically used when these elements are added. The defaults P and J are recommended. To use numerical names, remove the prefix. Note that EPANET uses only numerical names.

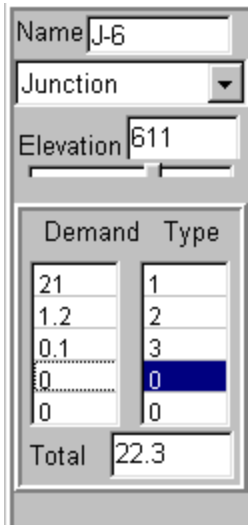
### Snap To Grid

When the Use Snap Grid box is checked, as the user lays out a pipe, the node will automatically snap to the nearest specified gridline intersection. The user specifies the Grid Size to which nodes will snap. If the user enters 100 in the Grid Size field, then each node created will snap to the nearest 100 ft (or m) gridlines. The user can go back and align an existing system by specifying the Grid Size and using the **Snap All Now** button.

## Data/Analysis

### Multiple Demand Types

Check this box to activate multiple demand types for junctions. This will allow up to five separate demands and types to be entered for each node. When a node is selected, the Node Information box appears as follows:




Once different demand types are specified, demand factors may be assigned to each type in the Demand Pattern table.

### Do Not Save Previous Results

Previous results are automatically saved unless this boxed is checked. Saving or not saving previous results effects file size. It may be desirable to check this box in particular with Surge files which have a large amount of results data (old method of analysis). If previous results are saved, they may be viewed in the Node/Pipe graphs or table. See Node Results Boxes or Pipe Results Boxes. Also can be important when a model approaches the limit of number of licensed pipes (current analysis method).

### Use Flow Coefficient (Cv) instead of Resistance (R) for Active Valves

When this box is checked a flow coefficient, normally provided by the manufacturer, may be used. See Active Valves.

### View Device Inlets and Outlets Independently

For directional nodes (pumps, active valves) if unchecked, the results selected in the Node Results box (Node Information window) will be the same for all nodes. Therefore if "Out" is selected, then on the Map Labels, all directional nodes will show outlet results. If this box is checked, then the inlet, outlet, or external results displayed on the map may be selected for each node independently.

### Sort Table

When this is checked, the Pipe and Node names in the report will be sorted numerically (as opposed to alpha-numerically).

### Prompt Before Analysis

A dialogue box appears which must be accepted before analysis can proceed.

### 24 point ID curves for KYPipe and Surge

By default, the Pump ID table has 12 rows of head and flow data input. This will allow 24 lines.

## Map / Layout

### Reverse Arrow Buttons

The red panning buttons used with the map view screen can be made to work in the reverse direction.

### Pan Method 2

Checking this box enables an alternative pan method where all pipes (but not the background) are completely visible during panning. The default method captures the entire map screen and this capture is what is shown during panning.

### Use Old Toolbar

Displays the left-hand column of buttons from Pipe2006 and earlier versions.

### Do Not Automatically Layout Intermediate Nodes
When this box is checked, junctions are created in place of the default intermediate nodes as pipes are laid out. See Layout of Piping System.

### Non-Animated Profile

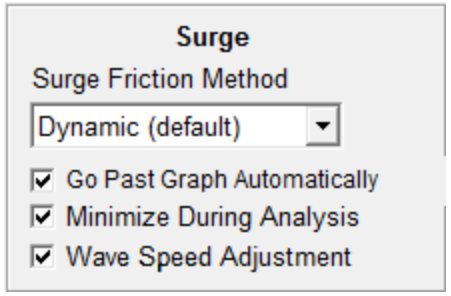
Use the Pipeline profile from Pipe2008 and earlier versions.

### Allow Multiple Data Boxes
Classic interface only. If this is unchecked, then the Information Window boxes Data, Chng, Rslt and User will only display one box or group at a time.

### Quick Profile Longest Path
See Create Profiles

### SCADA Menu
Settings and input for SCADA and URL feature (link)

## Surge




### Continue Past Surge Graph Automatically
When a Surge analysis is conducted a screen plot is drawn. When the plot is completed the user is by default prompted to 'Click here to continue" with analysis. Checking this box will bypass the user prompt, closing the screen plot and continuing on with analysis.

### Minimize During Surge Analysis
Certain graphics cards cause messages generated during a Surge analysis to be hidden behind the Pipe2018 window, giving the appearance that Pipe2018 has locked-up. Check this box to avoid this situation.

### Wave Speed Adjustment

With Surge analyses, each pipe length is adjusted so that is matches the length accuracy specified. If this features is used (checked) then the wave speed is adjusted in addition to the length. This helps quiet wave action in short pipes (less than length accuracy). It is checked by default and is the recommended setting.

### Analysis Display Option

Regular Display – program window stays visible (but behind screen plot) during analysis

Minimize During Analysis (default) –program window is minimized during analysis

Prompt Before Analysis – a screen prompt appears, pausing before running analysis until user clicks "OK".

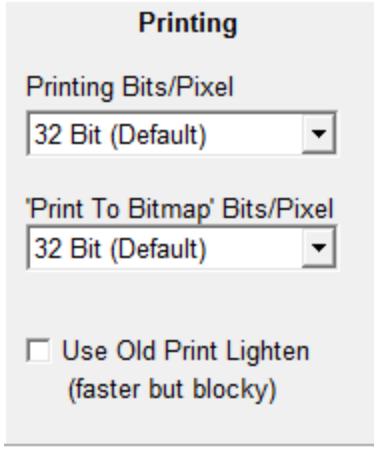
## Menu Options

### Language

KYnetic only. The hover hints (displayed as mouse pointer hovers over a menu item) may be displayed in a selection of languages.

### Show Advanced Node Types

This may be deselected to simplify the selection of available nodes.



# File Options

**Show Full Path in Title**
File path to p2k model file is shown at the top of the program window.

**Save Auxiliary Files in Subfolders**
Many secondary files are created during analysis and other operations. The default location for these is in a subfolder named with the filename and a .KYP suffix [filename].KYP. If this box is unchecked, all files will be stored in the same location as the .p2k model file.

**Use Standard Open File Dialogue Box**
Will use a Windows-type file browser to open and save files instead of the program default window.

# Other

**Don't Use Clipboard When Copying Tables**
Internally the clipboard is used regularly. If this causes a conflict with other applications, this option may be used to help work around the issue.

**Enhanced Fittings Table**
Classic only. If this box is deselected, the original Fitting Table format will appear with a limited selection of fittings and quantity check boxes instead of a scrollable table which includes all listed fittings.

**Use Binary Results File (default)**
Current analysis method only, effects the way results are internally stored. Leave this default setting checked unless advised otherwise by Tech Support.

**Print With Adobe Reader**
Adobe Reader is used as the print application when this option is checked to generate print jobs to the default printer. If this is unchecked, the driver for the default printer is used instead. Adobe Reader must be installed for this option to work.

**Small Row Height**
KYnetic only. Check this box to minimize row height in the Node and Pipe info window.

**EPS 2 Surge**
Used to convert EPS files automatically to Surge format.

## 1.3.2.1.14 REPORT

REPORT

KYnetic:

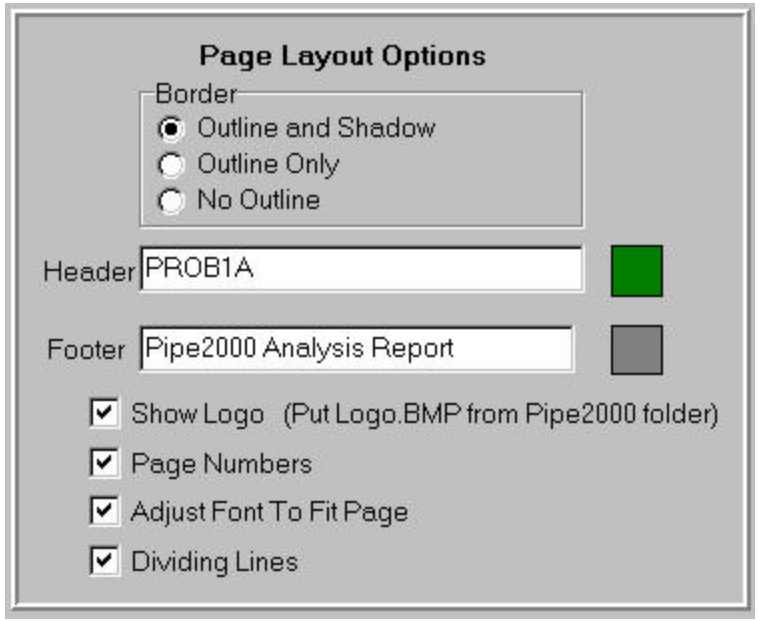
Classic:


---

This tab automatically accesses the tabulated output for the analysis for viewing or printing. Note that error messages may be displayed initially for the user to read The operation of the buttons is described below:

Edit/Copy - Use Edit | Copy to copy sections of the report to Clipboard.

This button allows the user to produce a printout of the Output Table. First the Print Configuration window will appear (see Printing). The number of copies, printer, margins, and orientation may be set in that utility. Click Print in the Print Configuration window and Print Report screen will appear. Several print setup options are available.




To display a **Logo,** save an image of the logo as a 'Logo.bmp' in your Pipe2018 folder.

To set margins and orientation, use the initial Print Configuration Utility window.

This button clears the display.

This button allows you to modify the font used for displaying and printing the Output Table.

This button allows you to load other .OT2 (Output Table) files.

This button connects to the Customized Reporting utility.

Save the Report and a Word .doc file.

This drop-down selector allows the user to display individual cases or times. Other output may be displayed such as a tank report, pump report, hydrant report, or other available reports depending on the type of analysis conducted. **All** will display all of the times or cases including the summary of original data and additional reports.

Note on Time/Cases

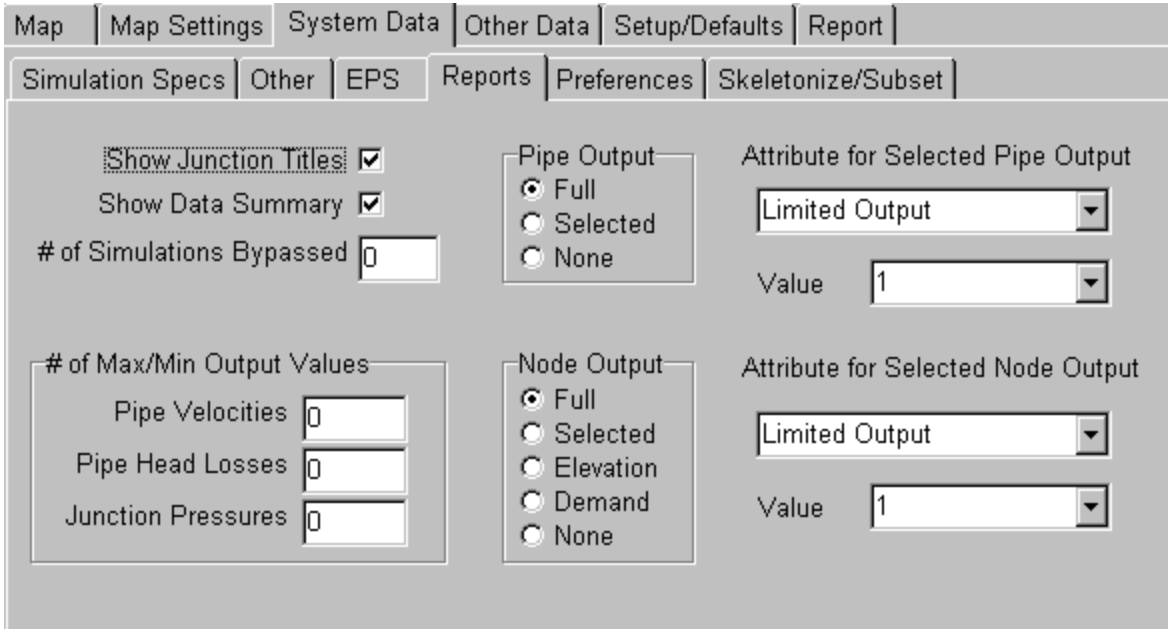
Steady State - case 0 is always the baseline case. If Changes are set up, the each subsequent case will reflect the changes.

EPS - case 0 is time 0. Each subsequent case is numbered with the computational period specified under System Data | EPS. If an intermediate time appears, then an event has occurred such as a tank is full or empty or a pump has turned on or off.

## 1.3.2.1.15 Reports (System Data)

Reports

KYnetic Report Setup menu icon:



**This window controls what information appears on the tabulated output for the analysis.**

Show Junction Titles

A check box to indicate Junction Titles in the tabulated output

Show Data Summary

A check box which is used to include (or to exclude) the tabulated data summary

# of Simulations Bypassed

If you have changes set up for a regular simulation this allows you to skip calculation of a set number of simulations. NOTE that the changes specified will still be implemented even if the simulation is bypassed.

# of Max / Min Output Values

If a value is provided for any of these three fields then an extra table of Max / Min values for that parameter is generated at the end of the tabulated results. The value entered corresponds to the number of items to be displayed and should not exceed half the total number of items (junctions or pipes). Therefore if 10 is entered beside Pipe Velocities, then a table of the 10 highest and 10 lowest velocities will be generated at the end of the report.

Pipe Output

Full - the output for all pipes will be included in the report.

Selected - only the output for the selected pipes will be shown in the report. The pipes are selected using the Attribute for Selected Pipe Output feature.

None - no output for pipes will be shown in the report.

Node Output

Full - the output for all nodes will be included in the report.

Selected - only the output for selected nodes will be included in the report. The nodes are selected using the Attribute for Selected Node Output feature.

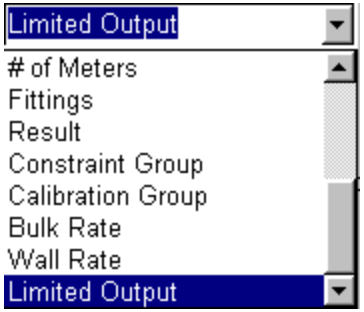
Elevation - only junction node results with non-zero elevation will be included in the report.

Demand - only junction node results with a demand assigned (non-zero) will be included in the report.

None - no output for nodes will be shown in the report.

Attribute for Selected Pipe Output

In the drop-down selector box, a number of pipe attributes are listed. This is a list of pipe User Data.




These attributes are assigned a value within Group Mode under the Pipe Information Window/Edit Pipe Set Box (See Sets and Groups). Some of the assigned Values are displayed and may be edited in the User Data Box (Pipe Information Window - click the User box at the top). If you don't have an attribute suitable for Node or Pipe Output or you would like to create a new attribute, the User Data Box is where attributes would be added.

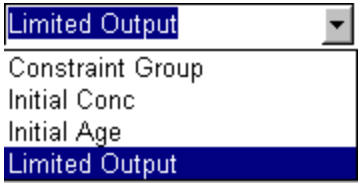
When using Selected Pipe Output, choose the attribute you would like to use to specify pipes. For example, if you want only the pipes with gate valves to appear in your report, select Fittings. Then in the Value box type in the symbol for gate valve (See the Pipe Data Box) which is G. When an analysis is run, the report will include only those pipes for which a gate valve has been specified in the Fittings section of the Pipe Data Box.

As another example, select Constraint Group. To use this attribute, a Constraint Group is specified using Group Mode selection, and the Edit Pipe Set Box (see Sets and Groups). When this attribute is selected, the number assigned to that constraint group will appear as an option in the Value drop-down box.

See Selected Output for an example of how to use this feature.

Attribute for Selected Node Output

In the drop-down selector box, a number of node attributes are listed. This is a list of node User Data.




These attributes are assigned a value within Group Mode under the Node Information Window/Edit Node Set Box (See Sets and Groups). The assigned Values are displayed and may be edited in the User Data Box (Node Information Window - click the User Box at the top). If you don't have an attribute suitable for Node or Pipe output or you would like to create a new attribute, the User Data Box is where attributes would be added.

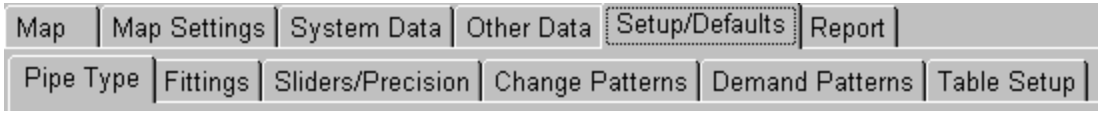
When using Selected Node Output, choose the attribute you would like to use to specify nodes. For example, to use the Constraint Group attribute, a Constraint Group is specified using Group Mode selection, and the Edit Node Set Box (see Sets and Groups). When this attribute is selected, the number assigned to that constraint group will appear as an option in the Value drop-down box.

See Selected Output for an example of how to use this feature.

## 1.3.2.1.16 SETUP / DEFAULTS

SETUP / DEFAULTS

Classic Interface




There are 5 sections under Setup / Defaults. Each section accesses a table or list of information which can be modified by the user. This information is used by Pipe2018 to provide various default data. Each of these lists are saved as files and it is possible to develop multiple versions. SAVE and LOAD commands are provided so that any of the available lists can be used.

---

## 1.3.2.1.17 SYSTEM DATA

SYSTEM DATA

KYnetic Interface, see System Data.

There are 5 sections under system data. These control the factors that affect the overall performance and output of the simulation. Click on any of the entries above for more information.

---

## 1.3.2.1.18 Tools (Main Menu)

Tools

10 Year HW Coefficient

Air Slam Pressure Surge

Air Valve Orifice Size

Bladder Precharge

Calculator

C Factor Calculator

Density and Viscosity Table

Extract Elevations from DEM Files

Force Calculations

Gas Properties

Generate Intermediate Pump File

Headloss Calculator

Hose and Nozzle Constants

Modulating (Regulating) Valve

Power (HP or KW) Calculations

Profile Import

Pump Characteristic Curves

Pump File Characteristics

Pump Selection

Residual Pressure Adjustment

Resistance Calculations

Select Pump File/Calculate Inertia
Spike Track
Sprinkler/Blowoff Constant
Turbine Data Convertor
Units Converter
Valve Stroking
Wave Speed

Tools - Surge only

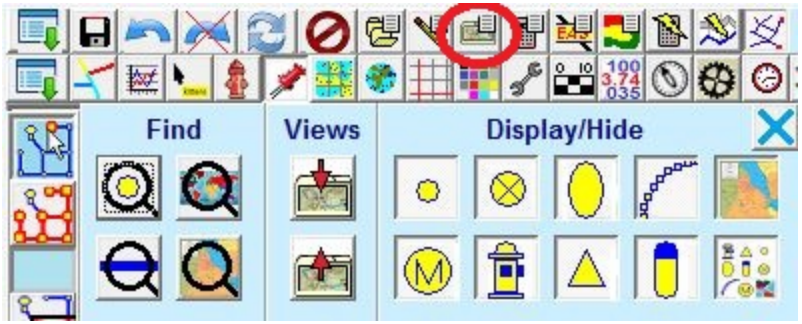
Equivalent Surge Tank Convertor
Export Air Valve Locations
Surge Tank Calculator

Tools - Gas only

Gas Cv Calculations
Gas K Calculations

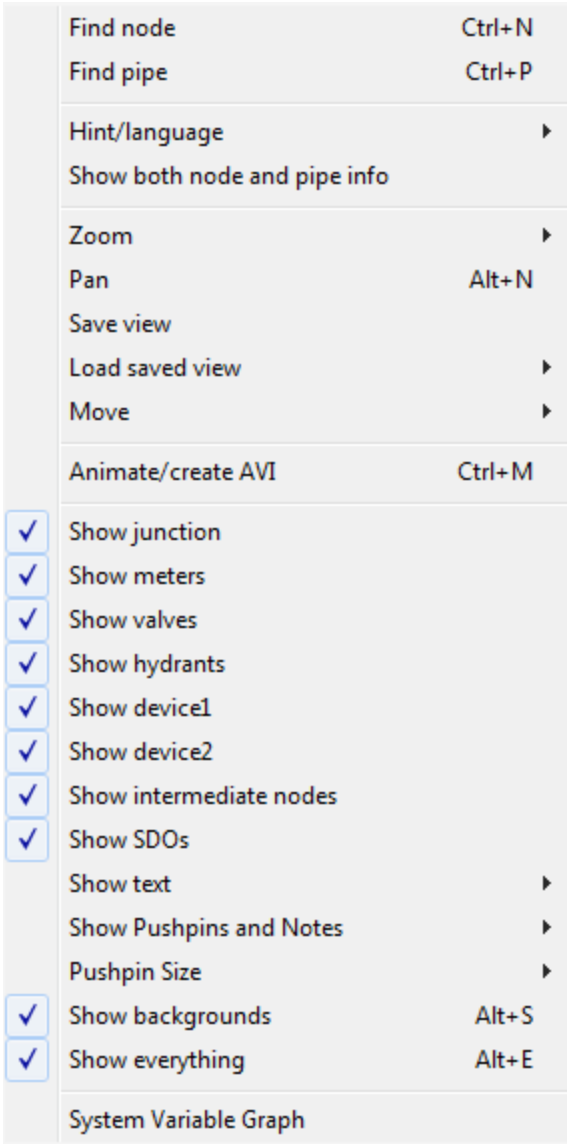
## 1.3.2.1.19 View (Main Menu)

View (Main Menu)




In KYnetic right click to access the menu format shown below:

File Edit View Analyze Move Labels Facilities Management Tools Help


Find node Ctrl+N
Find pipe Ctrl+P
Hint/language
Show both node and pipe info
Zoom
Pan Alt+N
Save view
Load saved view
Move
Animate/create AVI Ctrl+M
Show junction
Show meters
Show valves
Show hydrants
Show device1
Show device2
Show intermediate nodes
Show SDOs
Show text
Show Pushpins and Notes
Pushpin Size
Show backgrounds Alt+S
Show everything Alt+E
System Variable Graph

Find Node, Find Pipe - Type in the exact name of the node or pipe to select and zoom in to that element. The name is case, space, and symbol-sensitive.

Find World Map (KYnetic only) - opens a view of World Internet Map.

Find Place on Map (KYnetic only) - enter a city or landmark to find a location on an Internet Map.

Hints/Language (for KYnetic - see Preferences) - Choosing a language other than English translates the main menu, tabs, and mouse hover help (pops up when mouse hovers over an item)

Show Both Pipe and Node Info (for KYnetic - See Configuration Menu) - displays Node and Pipe data boxes simultaneously in the Information window to the right of the program window.

Zoom (for KYnetic under Edit and Map toolbar) - These six commands can also be performed by using buttons located on the left side of the display.

Zoom Out - This selection causes the map display to be reduced in scale.

Zoom In - This selection causes the map display to be enlarged in scale.

Zoom Selected - This selection causes the map display to be changed so that it maximally contains all of the selected nodes and pipes.

Zoom Previous - Zooms to the view just prior to the current view

Zoom Window - This selection allows you to drag out a window and then changes the display so the contents of the window are maximized.

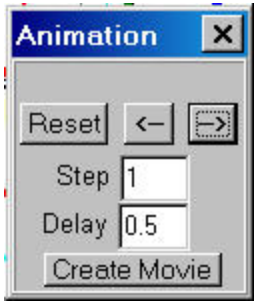
Zoom All - This selection causes the display to be changed so that the entire pipe system is displayed as large as possible while still fitting in the display window.

Pan - Allows user the move the display by holding down the left mouse button and moving the mouse

**Save View / Load Saved View** - Will save the location and zoom level currently in the map area. Once saved, this view can then be located again using the Load Saved View option which becomes available View.

**Number of available saved views was increased in Pipe2018**

Animate/ Create AVI (for KYnetic - Animate Controls bottom of Map window)- This brings up the animation menu shown below. If a file has multiple cases, the map can be made to display these cases one by one based on the Step and Delay defined in the Animation menu.




Step defines which cases are shown. If it is desired to show every other case, for instance, a Step of '2' may used.

Delay refers to the amount of time in seconds that lapses between each case display. The arrow can be used to change the direction of the animation. Reset stops the animation and brings it back to case 0.

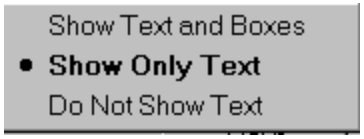
Create Movie will make an AVI which may be played on certain movie players. The animation may be used with contours and pipe emphasis enabled.

There is an option (under System Data/Preferences) to create bitmaps instead of an AVI file. The bitmaps will be in a folder called 'Movie' which you must create first within your Pipe2018 folder.

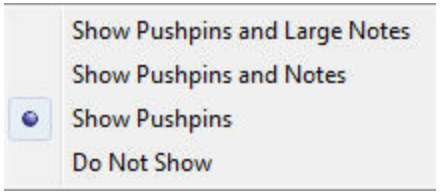
Note: AVI movies can be created faster if:
* You have more memory
* The step size is larger
* The Pipe2018 Window is made smaller

Show Junctions, Meters, Valves, Hydrants, Device 1, Device 2, Intermediate Nodes, SDOs - These selections toggle whether the specified items are or are not shown on the map.

Show Text (for KYnetic - see Text and Notes menu) - This selection brings up the following list which allows you to select the display of Text nodes.




Show Pushpin and Notes - This selection brings up the following list which allows you to select the display size of the Pushpin notes, limit display to pushpin only or turn off pushpins notes




Pushpin Size - Allows user to set the size of the pushpin icon accompanying the pushpin note.

Show Everything - checks all selections for elements to be shown on map.

System Variable Graph - For SWMM (Stormwater) applications

## 1.3.2.1.20 Hotkeys and Shortcuts

| | |
|---|---|
| List of Shortcuts | Ctrl-Alt-T |

**Editing**

| | |
|---|---|
| Apply | Alt-A |
| Undo to last apply | Ctrl-U |
| Copy | Alt-C |
| Paste | Alt-V |
| Delete | Alt-Del |
| Insert node | Alt-Ins |
| Error Check | Ctrl-E |
| Connectivity Check | Ctrl-Shift-E |
| Orthogonalize | Ctrl-G |
| Redo last map change | Alt-R |
| Undo last map change | Alt-U |
| Repeat pipe | Ctrl-R |
| Move down | Ctrl-down |
| Move left | Ctrl-left |
| Move right | Ctrl-right |
| Move up | Ctrl-up |
| *Toggle Fixed Mode 2 | Ctrl-F |
| *Select Everything - Nodes and Pipes | Ctrl-Alt-E |

**Map View**

| | |
|---|---|
| Show everything | Alt-E |
| Pan | Alt-N |
| Pan Arrows | Pans the map to the left (Ctrl+Left arrow), right (Ctrl+Right arrow), up (Ctrl+Up arrow), down (Ctrl+Down arrow). |
| Pan - Layout Mode | LC and drag. |
| Pan - Group Mode | Alt + LC and drag |

| | | |
|---|---|---|
| Cursor Notation Toggle | Alt-T | |
| Show background | Alt-S | |
| Hide labels | Ctrl-H | |
| Create contours | Ctrl-Alt-C | |
| Hide contours | Alt-H | |
| Animate | Ctrl-M | |
| Pump Status Emphasis Toggle | Ctrl-Alt-P | Not on Main Menu |
| NFPA Hydrant Color Coding | Ctrl-Alt-H | Not on Main Menu |
| Find Node | Ctrl-N | |
| Find pipe | Ctrl-P | |
| *Node Info Window | Ctrl-Shift-N | Only accesible by short-cut |
| *Pipe Info Window | Ctrl-Shift-P | Only accesible by short-cut |
| Zoom All | Ctrl-A | |
| *Zoom In | Ctrl-I | |
| *Zoom Out | Ctrl-O | |
| Zoom Previous | Ctrl-Z | |
| Zoom Selected | Ctrl-S | |
| Zoom Window | Ctrl-W | |
| * Saved Views | Ctrl-Alt-V | |
| * Toggle All Node Images | Ctrl-Alt-Z | |
| Toggle Intermediate Nodes | Ctrl-Shift-I | |

**Features**

| | |
|---|---|
| Add to presentation | Ctrl-Alt-A |
| Load optimal diameters | Ctrl-D |
| Show screen.txt file | Ctrl-F7 |
| Create Profile | Hold down the Ctrl key and select two nodes, release and type Alt-L |

| | | |
|---|---|---|
| Pipe Break | Ctrl-B | |
| *Settings Menu | Ctrl-Alt-S | Only accesible by short-cut |
| Surge Output Table | Ctrl-T | Not on Main Menu |
| Google Earth Export | Alt-G | Not on Main Menu |
| Show SCADA Status Charts | Ctrl-Shift-S | Not on Main Menu |
| Add to Presentation | Ctrl-Alt-A | Not on Main Menu |
| *Go from Classic to Kynetic | Shift-F3 | Only accesible by short-cut |
| *View Test.txt | Ctrl-F8 | Only accesible by short-cut |
| *Complete Inventory | Alt-I | |
| *Selected Inventory | Ctrl-Alt-I | |
| *Display Node Graph | Ctrl-Alt-G | Hidden (Kynetic Only) |
| *Surge Analysis | Ctrl-Shift-F7 | |
| *Surge Steady State | Ctrl-Shift-F8 | |

* Hidden

## 1.3.2.2 Information Windows and Data Boxes

## 1.3.2.2.1 Information Windows

Information Windows

---

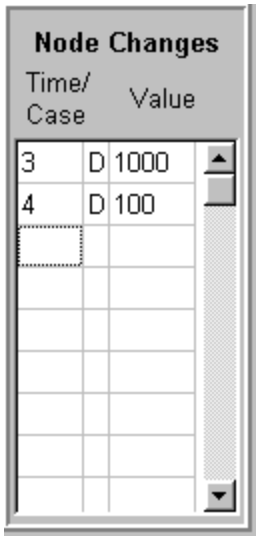
The right side of the Pipe2018 screen displays Information Windows for the most recently selected node or pipe link. These windows are used to enter and modify data and view results. The type of information shown and window size are controlled by the top buttons.

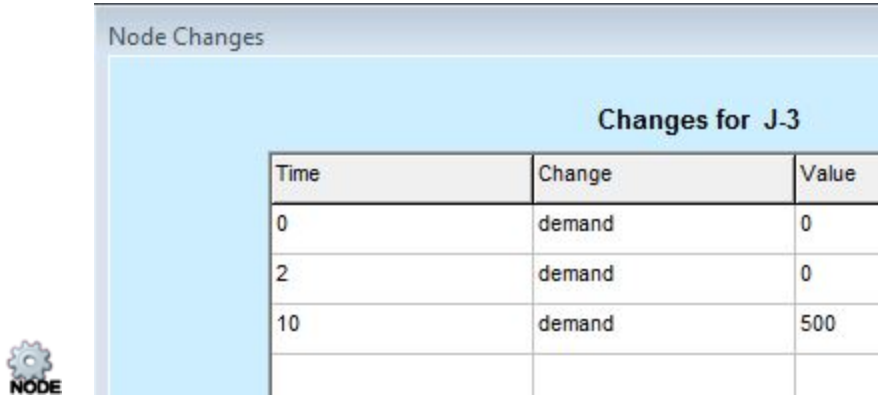
See Also:

- Pipe Information Window
- Node Information Window

## 1.3.2.2.2 Node Change Box

Node Change Box



*What is this*?

The Node Change Box allows you to edit, modify and view the changes to the selected node at a specified time (or case). To access this window, click on the CHNG button at the top of the Node Information window. The above box call for a demand of 1000 for case (time) = 3 and 100 for case (time) = 4 for the selected node. All changes are summarized in the Change Pattern window

Time / Case

Selects the time or case for the change to occur.

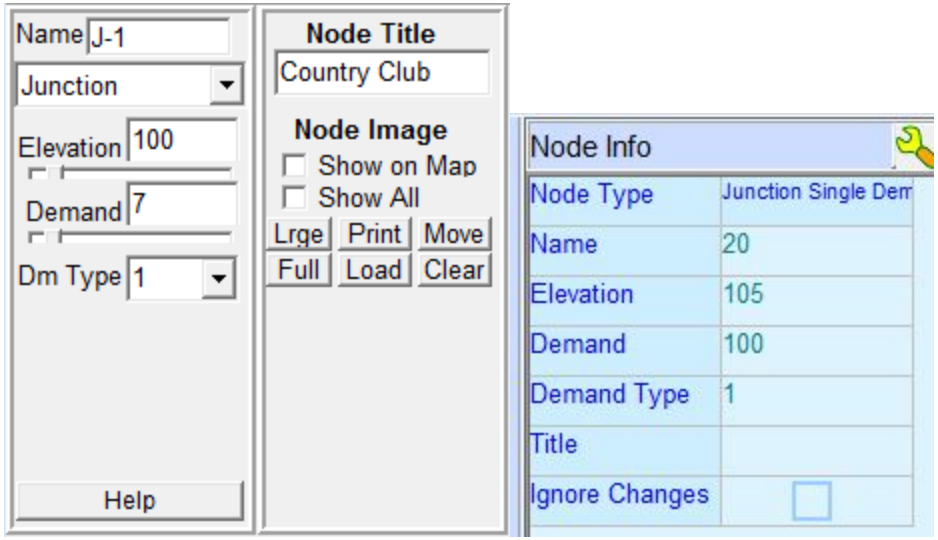
Clicking on the middle column

Pops down a parameter list to select the data to be changed (customized for node type).

Value

Selects the new value for the data item when change is implemented.

## 1.3.2.2.3 Node Data Boxes




- In KYnetic Interface use this icon to bring up the Configuration menu and change the appearance of the Node info window

Classic Interface:

To display both Pipe and Node data simultaneously, go to View and check "Show Both Node and Pipe Info"

Nodes have 2 (or 3) Data boxes. All have the boxes shown above. Some require additional data which is added to Box 1 box and others require an additional box.

These Node Data boxes are shown complete for the following nodes:

Intermediate Nodes

Valves

Hydrants

Check Valves

Inline Meters

Devices 1 and 2

Text Nodes

* * * Box 1 * * *

Node name

Alpha numeric name assigned by Pipe2018. This can be modified by the user. Option to use numeric or specific alpha prefix is available (System Data / Preferences).

Node type

Check here to select or modify node type. Note that certain conditions are required to change end node to internal node.

Node elevation [ft. (m)]

Elevation of the node based on the chosen datum.

* * * Box 2 * * *

Node Title

Alpha numeric information which can be displayed on screen or plots and incorporated in tables.

What is a Node Image?

An optional bitmap image for all nodes (including a Text node) which can be displayed on the map or expanded to show details on screen. The image can depict anything of interest - a photograph, detailed map, operation instructions or any other display. The following buttons are provided. See also Node Images and Text Nodes.

Show on Map

When selected this image will be displayed on the map.

Show All

When selected all images will be displayed on the map.

Lrge

When selected a larger image will be displayed in the Node Information Window.

Full

When selected the image will expand to full screen.

Print

Select to print the image.

Load

Select to load a bitmap (BMP) file of desired image for this node.

Move

Click here to move the image on map to different quadrant.

Clear

Click here to delete image for this node.

The following nodes have a customized Data Box 1 which handle additional information. The nodes marked with * also have an extra box for head/flow data.

Junction Data
Pump Data *
Loss Element Data *
Reservoir Data

Tank Data *
Pressure Supply Data *
Regulator Data
Sprinkler Data
Metered Connection Data

## 1.3.2.2.4 Node Information Window

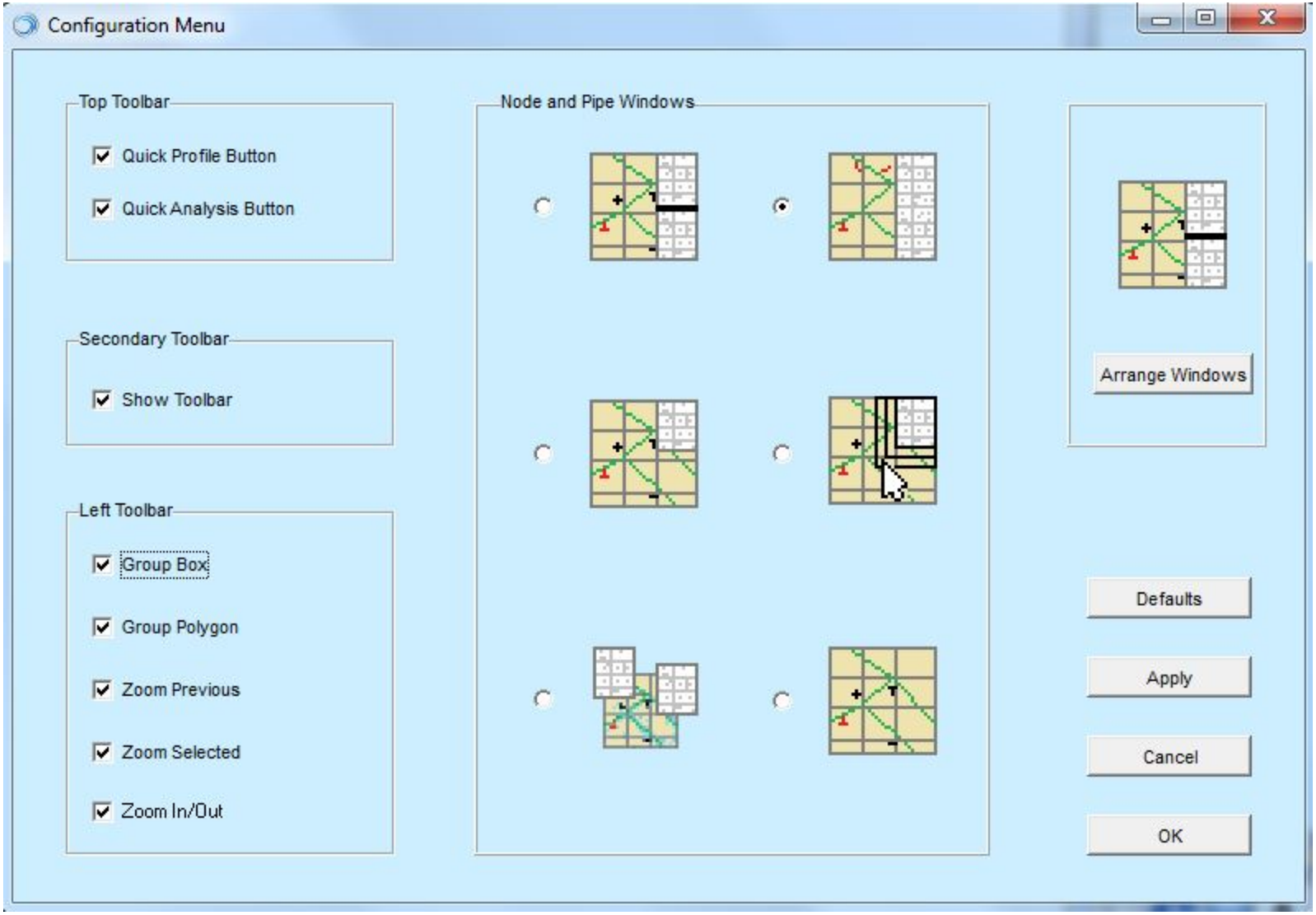
Node Information Window

---

**Node Information Window** automatically appears on the right side of the screen for the selected node.

**KYnetic Interface:**

- Opens the configuration menu shown below. There are six settings for how the Node/Pipe Info window appears.



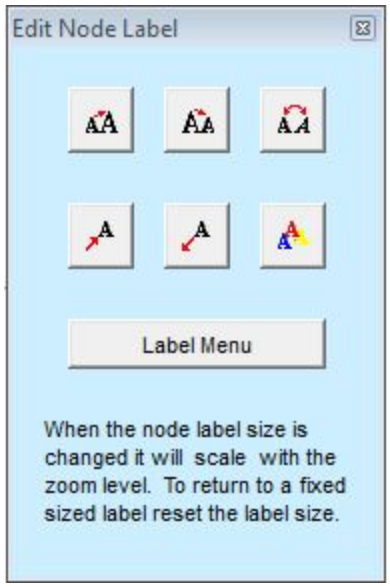

Icon functions:

- Results Graphs and Table

- Delete Node

- Open Node Changes menu.

- Bring up the following label menu. These buttons increase, decrease, italicize, set the position of and reset the text of the selected or selected pipes on the map.




Optional display items:

- Turn on User Data items, create a new User Data item

- Copy the current node, click on an existing node, then Paste to duplicated all the node data in that location (except elevation data if non-zero

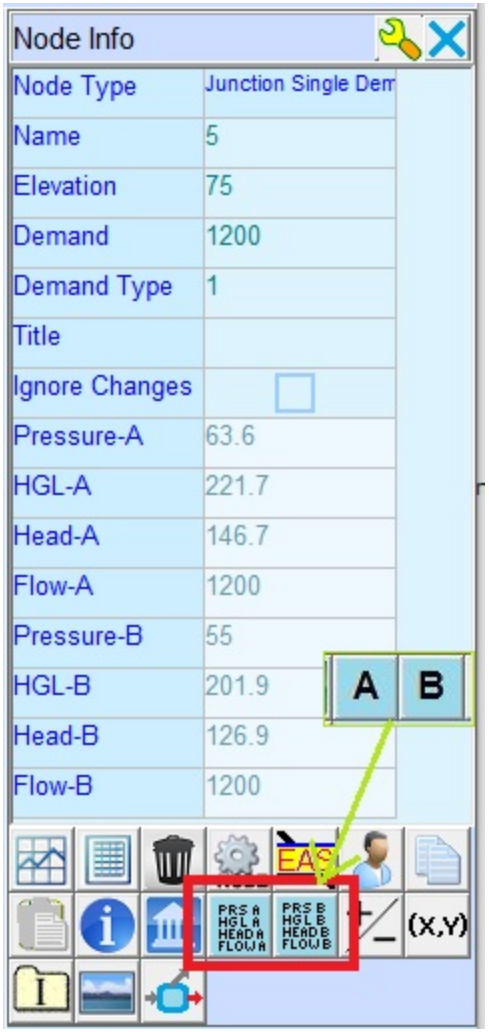
in target node data)

- Brings up detailed information about the selected node type.

- Used to bring up Library menu. See Creating Libraries.

- Display Results A, B and/or Max/Min/Avg in the Node Info table

All Results are displayed as of Pipe2018 as shown:



- Display coordinate values.

- Display Index value (internal reference data)

- Add, display, and manage images associated with a node.

- For directional nodes where results for inlet and outlet (and external for Surge devices) are different, this toggles between the various results to be displayed on the graph.

**Classic Interface:**

The window consists of four sections which can be individually selected using the appropriate button these are:

**Data (node data)** - Two boxes for entering and displaying node data.

**Rslt (results)** - Three boxes for showing a summary, a plot and a table of the results for the selected nodes (end nodes only).

**Chng (change)** - One box for entering and displaying specific changes at this node for the applicable change pattern (end nodes only).

**User** - One box for entering and displaying additional data which may be customized by the user (end and internal nodes).

**Node Information Window Controls** - The following control buttons appear at the top of the Node Information Window.

**Del (delete)** - This will delete the selected end node **and all connecting pipes**. If the selected node is an internal node it will delete just the internal node and combine the two connecting pipe segments into a single pipe segment.

**On (Off / On)** - This button will turn the node on or off ( not Junction nodes or Internal nodes).

**More** - This will provide space to display another column of information boxes.

**Less** - This will remove space for a column of information boxes if there is insufficient space to show all boxes.

, - If there is insufficient space to show all boxes these will cycle the last box shown to display the next (or previous) box.

**Data Rslt Chgn User** - These are on/off switches for displaying these data and information sections.

**Copy Paste -** See Copy and Paste. This allows the user to copy the node type and data of the selected node to other existing nodes. Does not apply to the node name or elevation.

- These buttons make the label text of the selected node or group of nodes larger, smaller or revert back to the default size (as set in Map Settings | Labels ) respectively. In Text mode, it will do the same to the text of the selected text node. If the size is changed from the default, then the label will be scaled as the map is zoomed in and zoomed out (similar to the labels in a dxf file).

- The first button allows the user to set the position of the label of the selected node. A "crosshairs" will appear. The user centers it on the desired location of the label and clicks. The second button reverts the label position back the the default setting. In Text mode, it will do the same to the text of the selected text node.

- This button allows the user to select the color of the text of either the selected node (Layout mode), selected group of nodes (Group modes), or the selected text node (Text mode).

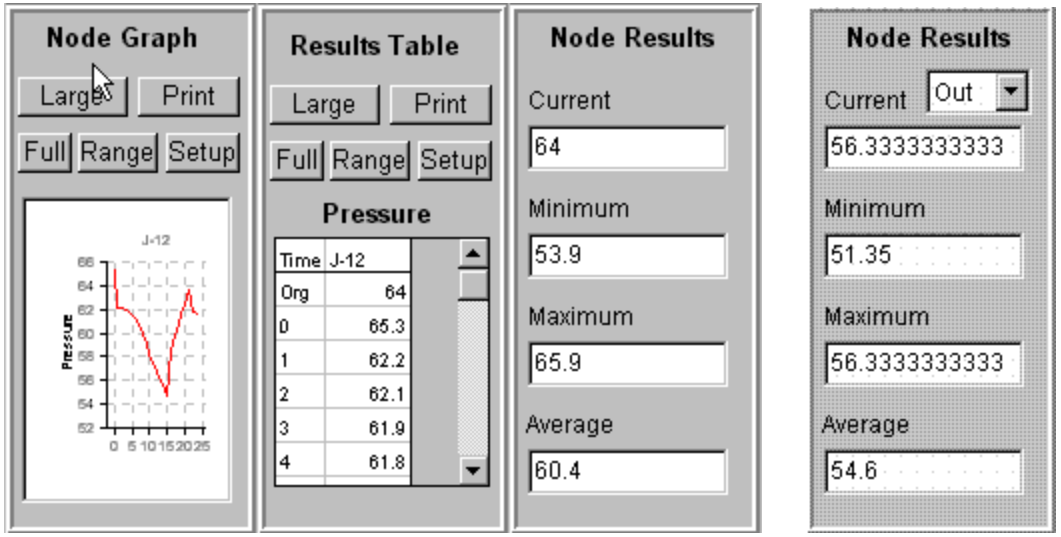
See Also:

Node Data Boxes
Node Results Boxes
Node Change Box
Node User Box
Node Images and Text Nodes

## 1.3.2.2.5 Node Results Boxes

Node Results

In KYnetic, open Graph and Table using these icons:




**Junction nodes** **Inlet/Outlet Nodes**

---

*What are these*?

**Node Graph** - A plot showing the selected results for this node or node set for all cases (times). The user may create a title and x and y labels, set the y scale, capture the image to a BMP, and paste the image to the clipboard. When a BMP is created, the file will be saved as NdGrf1.bmp (or NdGrf2, -3 , etc.) in the same folder as your p2k file. If the Previous Result box is checked, the last set of results will be graphed along with the current results. To view the buttons for these options, expand the graph view to Large or Full.

**Results Table** - A table showing the select result (pressure, head or HGL) for this node or node set (Group Mode) for all cases (times). The data may also be exported to Excel or ASCII formats. If the Previous Result box is checked, the last set of results will be tabulated along with the current results. To see the buttons for these functions, expand the table view to Large or Full.

**Node Results** - A summary of the node results for the selected node and selected time or case. Note that for nodes which have both an inlet and an outlet result, the result which is displayed on Map Labels, in the Node Results box, or in the Results Table box may be selected by the user. For Node Graphs both inlet and outlet results will be displayed. If multiple nodes are selected however the Node Graph will display only the selected (inlet or outlet) result. When toggling between inlet and outlet results, click the Refresh button to update the map display.

**Controls:**

Large

Expands the table or graph within the Node Information Window.

Full

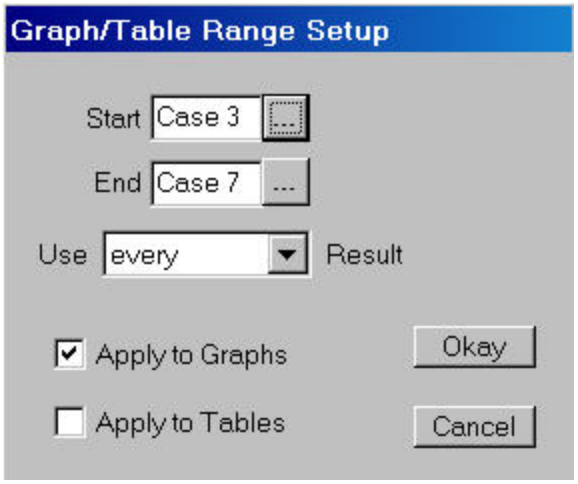
Expands the table or graph to full screen.

Print

Prints the table or graph.

Range

**Allows the range for cases to be set for tables and graphs.**



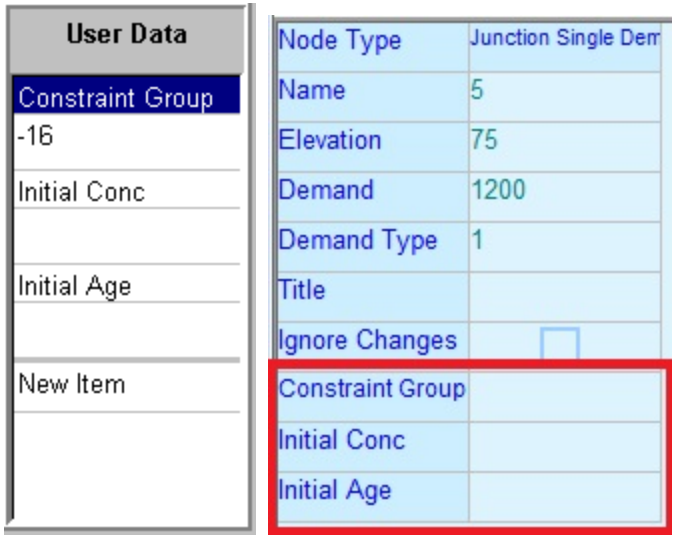

Setup

Accesses a menu to customize the table or plot.

Previous Result
If the Previous Result box is checked, the last set of results will be presented along with the current results. See System Data | Preferences

## 1.3.2.2.6 Node User Box

Node User Data




See Also: **User Data**

---

The User Data Box is accessed by clicking the User button at the top (Classic) or bottom (KYnetic) of the Node Information window (in Classic, make sure enough space is allowed for the box, if not, click More or scroll through the boxes with the pointers).

User Data is information about the node which is specified by the user. Typically, User Data is an attribute used to identify a group of nodes for a Constraint calculation (see Constraints Data) or a water quality simulation. User Data may also be used to define a group of nodes for Selected Output (see Selected Output and Reports (System Data)). New attributes or customized information may be added by clicking on **New Item** and editing the entry title. Other attributes may be edited or deleted this way also. User Data attributes added or edited in this box will be reflected in the Attribute for Selected Node Output under System Data/Reports.

A User Data group may be defined using the Group Mode (see Sets and Group Mode). User data may also be edited in the User Data Box for individual nodes by simply selecting the data item.

Back to Internal Node Data

## 1.3.2.2.7 Pipe Change Box

KYnetic Pipe Change icon:


The Pipe Change Box allows you to edit, modify and view the changes that are going to the selected pipe at a specified time (or case). Access this box by selecting the target pipe and clicking the CHNG box at the top of the Pipe Information window (you may need to click More if the display area is too small). The above box calls for the selected pipe to close for case (or hour) 5 and to open for case (or hour 8). All changes are summarized in the Change Pattern window.

Time / Case

Selects the time or case for the change to occur.

Clicking on the middle column

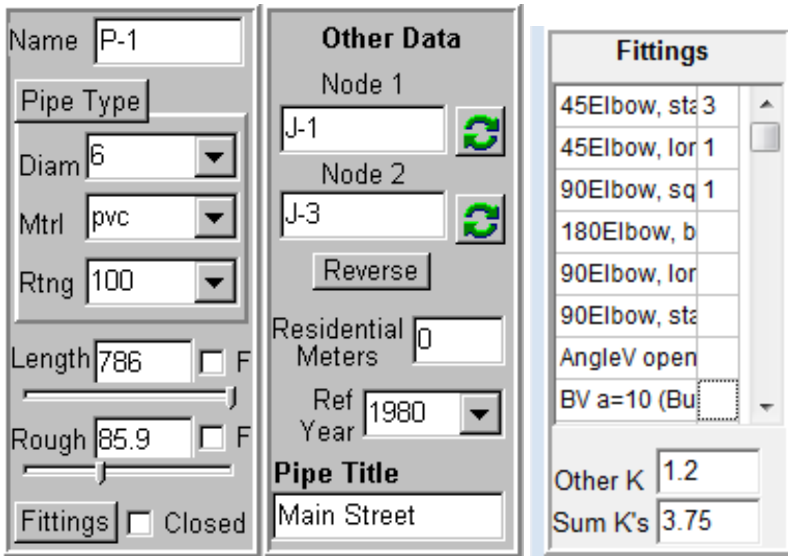
Pops down a parameter list to select the data to be changed.

Value

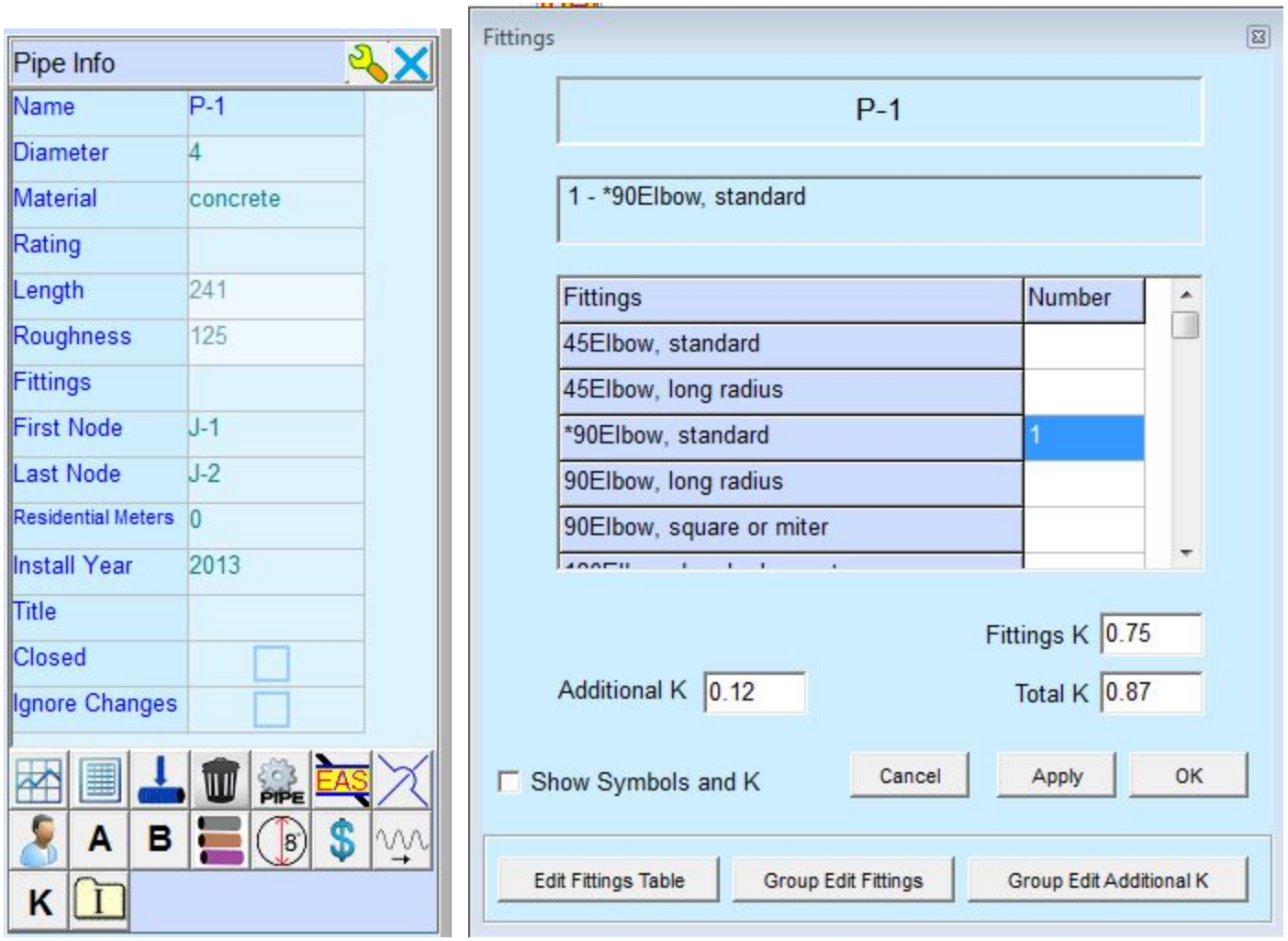
Selects the new value for the data item when change is implemented.

## 1.3.2.2.8 Pipe Data Boxes

Pipe Data Boxes




Box 1 Box 2 Box 3




To display both Pipe and Node data simultaneously:

Classic: go to View and check "Show Both Node and Pipe Info"

KYnetic - go to the Configuration menu

The Pipe Info window is used to enter or edit the pipe data. If you do not have non-zero Length, Diameter, and Roughness values assigned for every pipe in your system, an analysis cannot be performed.

Name - The Pipe name assigned when a pipe is added. This can be modified by the user.

Pipe Type - Click this to display a selection which includes the pipe diameter, material, and rating. A number of default data values are applied when pipe type is selected. You should provide data in the Pipe Type Table (Setups/Defaults - Pipe Type) for your system or use the default table. The diameter, material and rating can also be entered individually.

Pipe Diameter [in. (mm)] - Nominal diameter of the pipe. To define an Actual (inside) Diameter to be used in the analysis, use the the Pipe Type table.

Pipe Material - Material for the pipe wall.

Pipe Rating [psi (kpa)] - The pressure rating for the pipe.

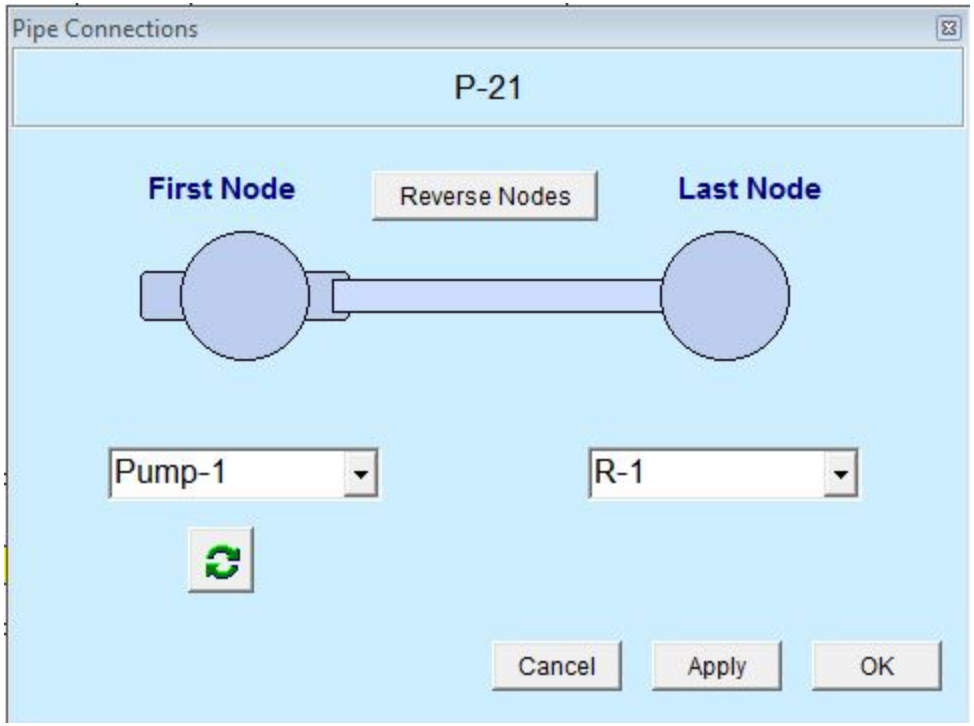
Pipe length [ft. (m)] - The total length of the pipe link which includes all pipe segments. To make this length fixed, check the box marked F. In this case, the length will not be altered as it is recognized in the analysis by any scaling or moving nodes (but the graphical appearance may change),

Pipe Roughness - This value depends on the head loss expression being used (noted - top bar of the Map screen). An age-based roughness calculation is made if you select a pipe type with a reference roughness and estimated 10 year roughness defined and provide a reference year (box 2). To make this roughness fixed, check the box marked F. In this case, the roughness is fixed as it is recognized in the analysis and is not altered during age-based roughness calculations or calibration.

Fittings - A button to access the Fittings Table to account for fittings such as elbows, T's, valves, etc. which cause pressure drop. The user may enter in any additional minor loss in the Other K box. The sum of the minor losses associated with all fittings checked plus any additional loss entered by the user appears in the box, Sum K's. Note that the head loss along the length of the pipe is accounted for in the analysis and there is no need for the user to enter an equivalent length loss to account for this.

Closed (pipe) - Check box to indicate closed pipe link. A pipe link should be designated closed if any valve within the link is closed.

First Node, Last Node / Node 1, Node 2 - End nodes for pipe (these can not be edited). In KYnetic, click one of these to bring up the menu shown below:




- Click to change the connection side (directional nodes only)

Reverse (nodes) - Click to reverse the node order. Pipe links with a check valve must list nodes in correct order (allowed flow direction).

Residential Meters - The total number of residential meters connected to a pipe link. Note the Average Residential Meter Demand (per meter) is entered with System Data / Other.

Reference Year - The year used for the age based roughness calculation (usually installation year).

Pipe Title - This is the title of pipe (optional). Titles can be 32 characters long and do not have to be unique for each pipe

Fittings - Click here to bring up the menu shown at the top of this topic. The loss coefficients are automatically tallied. Enter quantity for each fitting. At the top a symbol for each fitting selected appears. The fittings table may be customized by the user and is accessed under Components menu or (Classic) Settings / Defaults - Fittings. See Fittings.

Use Fitting icon to see additional information in the Pipe Info window

Other K - Entry under Fittings Table for additional loss coefficients (sum) to account for fittings not in the table.

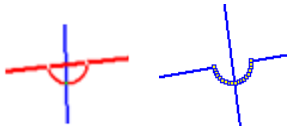
Sum K's - The sum of all the loss coefficients for selected fittings plus entry from Other K. This is calculated and used in the data file.

## 1.3.2.2.9 Pipe Information Window

Pipe Information Window automatically appears on the right side of the screen for the selected pipe.

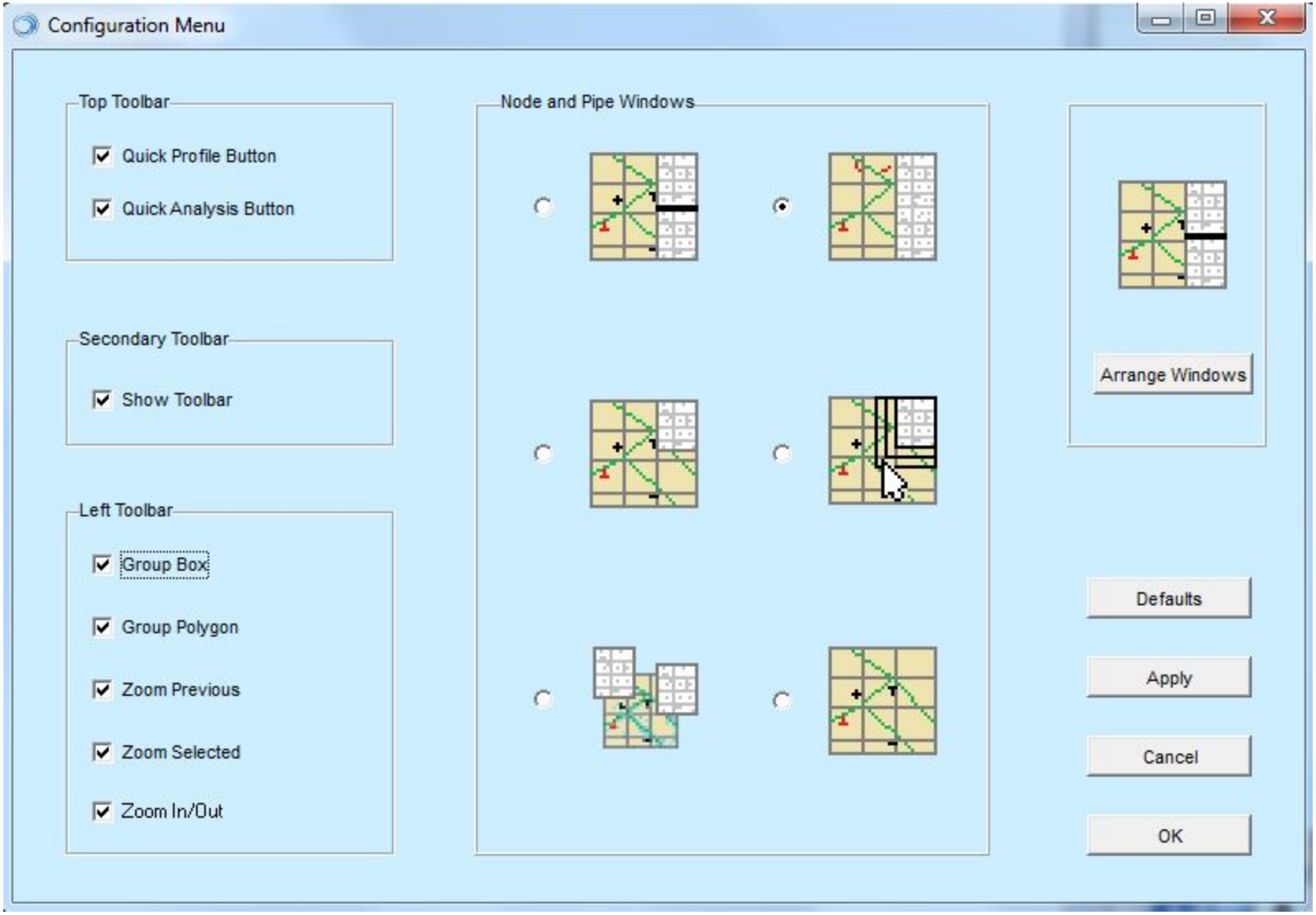
**Pipe Crossover (Pipe2012 and later versions)**

Click the Cross button. Cursor will become an arc which may be placed and sized on the currently-selected pipe.

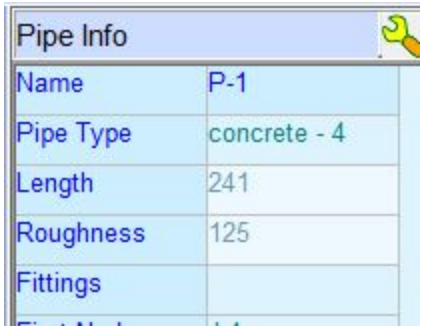


**KYnetic Interface**

- Opens the configuration menu shown below. There are six settings for how the Node/Pipe Info window appears.

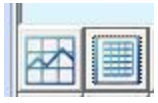



Pipe data is displayed as a Pipe Type by default.




Use the "Display Pipe Type..." icon to separate Pipe Type into individual parameters.  This is needed for direct editing of material, diameter, and rating.

Icon functions:

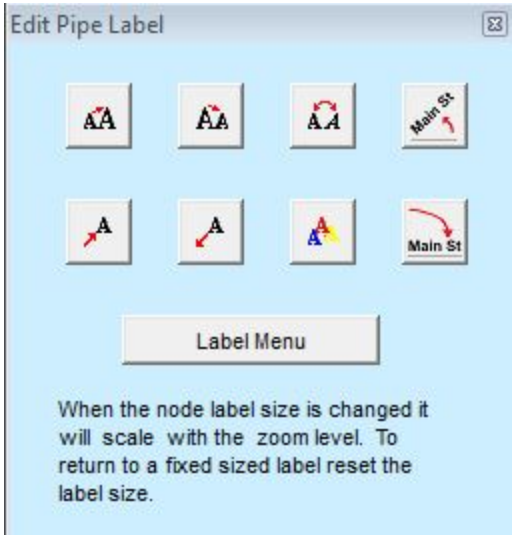
 - Results Graphs and Table

- Insert node in pipe at last location clicked for currently selected pipe.

- Delete Pipe

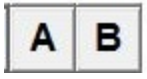
- Open Pipe Changes menu.

- Bring up the following label menu. These buttons increase, decrease, italicize, set the position of and reset the text of the selected or selected pipes on the map.

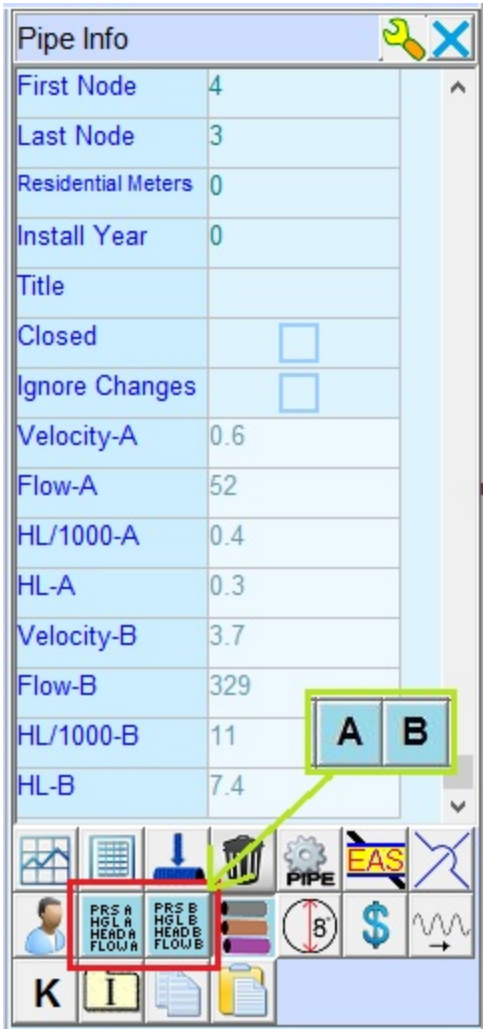



Optional display items:

- Turn on User Data items, create a new User Data item.

 - Display Results A and/or Results B

All Results are displayed as of Pipe2018 as shown:




- Display Actual Diameter

- Display Pipe Cost (if cost/unit length is defined)

- Display Wave Speed

- Display Minor Loss K values for pipe

- Display Index value (internal reference data)

**Classic Interface**

Data is input into boxes displayed or hidden by click buttons at the top of the Pipe Information window.

Data (Pipe data) - Three boxes for entering and displaying basic pipe data.

Rslt (results) - Three boxes for showing a summary, a plot and a table of the results for the selected pipe

Chng (change) - One box for entering and displaying specific changes for this pipe for the applicable change pattern

User - One box for entering and displaying additional data which may be customized by the user

Pipe Information Window Controls -

Del (delete) - This will delete the entire selected pipe link and all internal nodes within the link.

Insrt (insert) - This will insert an internal node at the location where the mouse was pointed to select the pipe. Any of the eight types of internal nodes can be selected from the pop up activated by the button.

More - This will provide space to display another row of information boxes.

Less - This will remove space for a row of information boxes.

, - If there is insufficient space to show all boxes these will cycle the last box shown to display the next (or previous) box.

Data Rslt Chgn User - These are on/off switches for displaying these data and information sections.

- These buttons increase, decrease, italicize, set the position of and reset the text of the selected or selected pipes on the map.

See Also:

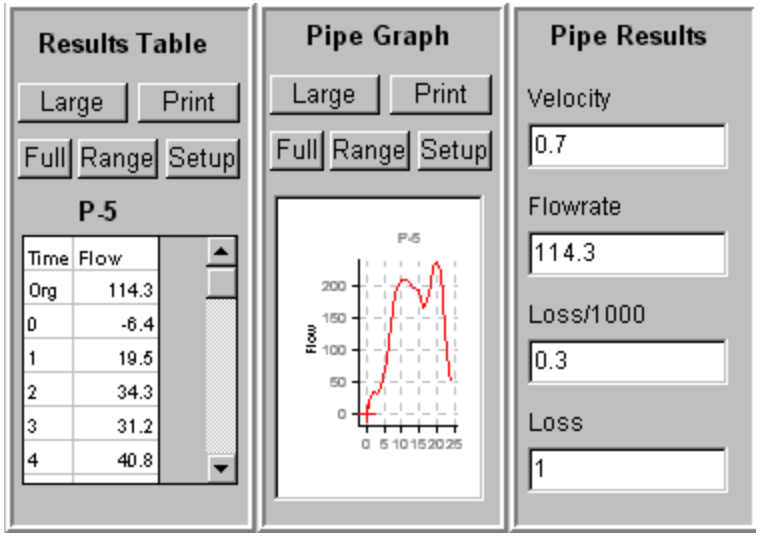
- Pipe Data Boxes
- Pipe Results Boxes
- Pipe Change Box
- Pipe User Box

## 1.3.2.2.10 Pipe Results Boxes

Pipe Results Boxes

KYnetic Graph and Table icons:

Classic Graph and Table boxes:




---

Results Table

A table showing the selected result type (velocity / flow / head loss or hl/1000) for this pipe or pipe set (Group Mode) for all cases (times). The data may also be exported to Excel or ASCII formats. If the Previous Result is selected, the last set of results will be tabulated along with the current results. To see the buttons for these functions, expand the table view to Large or Full.

Pipe Graph

A plot showing the selected result for all cases (times). The user may create a title and x and y labels, set the y scale, capture the image to a BMP, and paste the image to the clipboard. When a BMP is created, the file will be saved as PpGrf1.bmp (or PpGrf2, -3 , etc.) in the same folder as your p2k file. If the Previous Result is selected, the last set of results will be graphed along with the current results. To view the buttons for these options, expand the graph view to Large or Full.

Pipe Results

A summary of the pipe results for the selected pipe and selected time or case.

Controls:

Large

Expands the table or graph within the Pipe Information Window.

Full

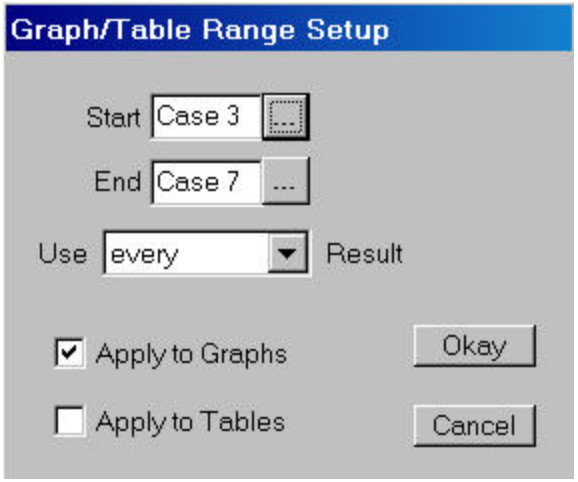
Expands the table or graph to full screen.

Print

Prints the table or graph.

Range

**Allow a range of cases to be set for the table or graph.**




Setup

Accesses a menu to customize the table or plot.

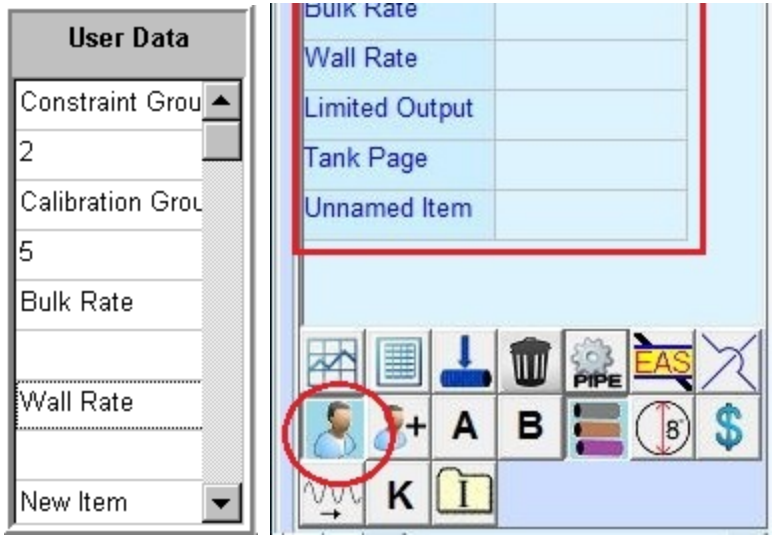
**Previous Result**
If the Previous Result box is checked, the last set of results will be presented along with the current results. See System Data | Preferences

Note: The result type and case (time) is selected using the Result Selection bar at the bottom of the display.

## 1.3.2.2.11 Pipe User Box

Pipe User Box

Pipe Info
Name
P-5
Pipe Type
di - 150 - 8
Length
3296
Roughness
118
Fittings
First Node
J-1
Last Node
J-6
Residential Meters
0
Install Year
2001
Title
Closed
Ignore Changes
Calibration Group



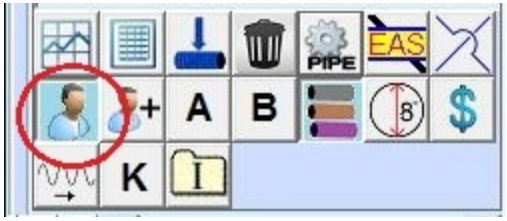

## User Data

---

The User Data Box is accessed by

Classic: clicking the User button at the top of the Pipe Information window (make sure enough space is allowed for the box, if not, click More or scroll through the boxes with the pointers).

KYnetic - clicking the User icon at the bottom of the Pipe Info window



User Data is information about the pipe which is specified by the user. Typically, User Data is and attribute used to identify a group of pipes for a Constraint calculation (see Constraints Data), a Calibration calculation, or a water quality simulation. User Data may also be used to define a group of pipes for Selected Output (see Selected Output and Reports (System Data)). New attributes or customized information may be added by clicking on **New Item** and editing the entry title. Other attributes may be edited or deleted this way also. User Data attributes added or edited in this box will be reflected in the Attribute for Selected Pipe Output under System Data/Reports.

A User Data group may be defined using the Group Mode (see Sets and Group Mode). User data may also be edited in the User Data Box for individual pipes by simply selecting the data item.

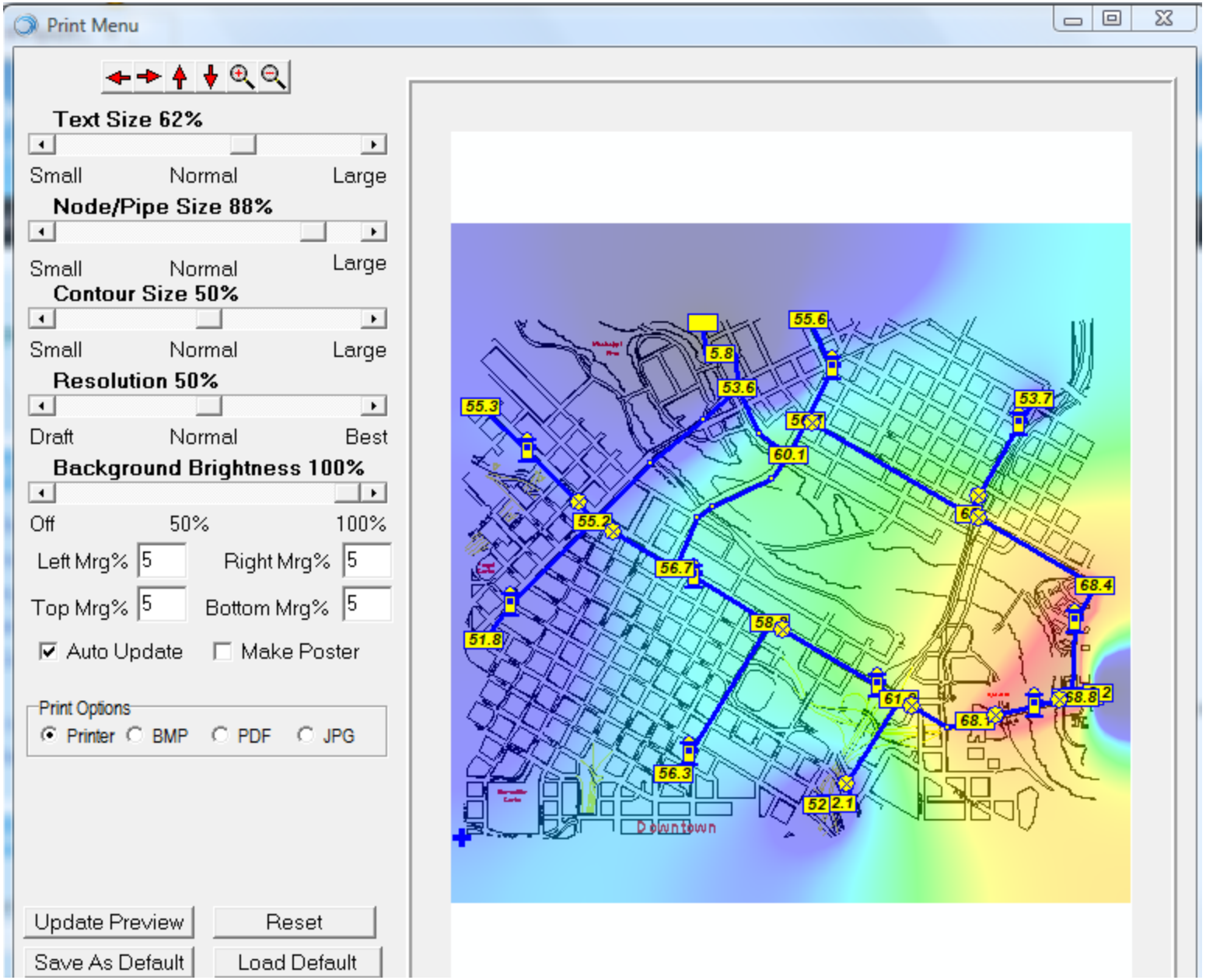
## 1.3.2.3 Printing

### Print

The File | Print command will bring up the window shown below. The print preview reflects what the final printout will contain (WYSIWYG). Several of the display attributes may be modified (Text Size, Node/Pipe Size, Contour Size, Resolution, Background Brightness) using the slider bars provided. If the preview is not automatically updated, use the Update Preview button to see the changes.

Please note that the Max Resolution option will produce very large temporary files and may be slow to print. Using the highest resolution is often appropriate when using large format printers.

The red arrows and magnifying glasses may be used to pan and zoom the print area before printing. The zoom buttons will not be available if a print scale is specified using the Scale button.


Print Menu
Text Size 62%
Small Normal Large
Node/Pipe Size 88%
Small Normal Large
Contour Size 50%
Small Normal Large
Resolution 50%
Draft Normal Best
Background Brightness 100%
Off 50% 100%
Left Mrg% 5
Right Mrg% 5
Top Mrg% 5
Bottom Mrg% 5
Auto Update
Make Poster
Print Options
Printer BMP PDF JPG
Update Preview
Reset
Save As Default
Load Default
55.6
5.8
53.6
55.3
53.7
60.1
55.2
56.7
68.4
51.8
58.8
61
68.1
56.3
52
2.1
Downtown

Left, Right, Top and Bottom margins may be set in terms of percent.

Auto Update - If this is checked, changes to the print options should be updated automatically in the preview window. If this is unchecked, use the Update Preview button to update.

Make Poster - When this option is checked the image will be printed in the specified number of sections/pages.

Print Options - When options other than "Printer" are specified, the paper size must also be specified.

Update Preview - applies changes made to the preview window.

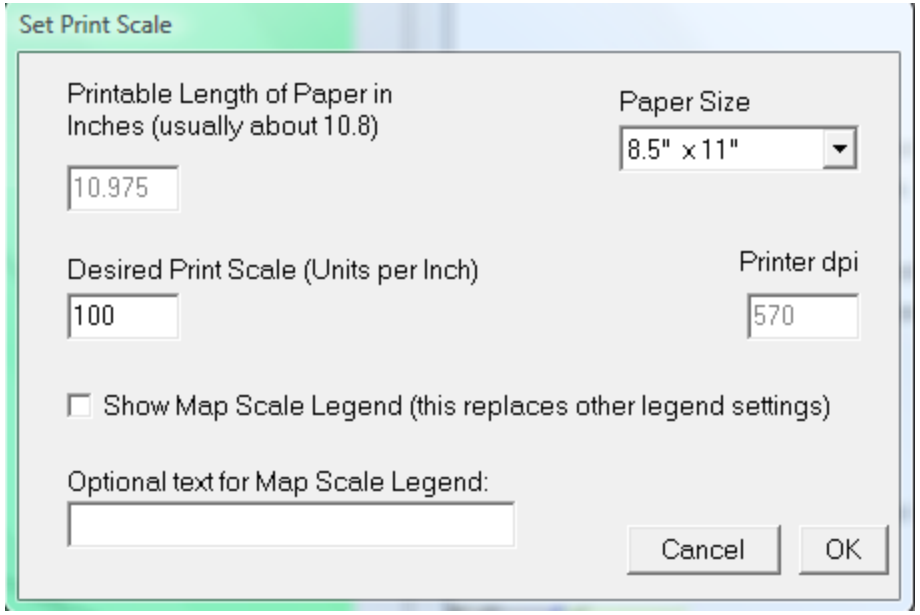
Reset - Resets the print menu settings to the overall default settings.

Save As Default - will save the current print menu settings as the default

Load Default - will load the previously saved set of default settings.

Printer Settings - May be used to select the printer and number of copies. Click Properties to set the orientation (Portrait/Landscape).

Scale - the following window appears:




Printable Length of Paper in inches (usually about 10.8) - used in conjunction with Desired Print Scale and a Custom paper size. If this value is specified, then the Desired Print Scale may be defined in units per inch. When values are entered for both fields, the print image will zoom to reflect this setting. The zoom buttons will no longer be available.

Desired Print Scale (Units per Inch) - Ignoring individual fixed pipe length data, these are the units of your system, as reflected by the x,y coordinates of your map screen (ft or m), per inch of printed paper.

Show Map Scale Legend - if you have a legend set up to show on printouts (under Map Settings | Legend), then checking this box will replace that legend with a new distance scale.

Optional text for Map Scale Legend - text entered here will be included in the Map Scale Legend.

Paper Size - The typical printable length will be shown when the paper is selected here. If it is a custom length, the Printable Length must be entered.

Printer dpi - The typical dots per inch is shown when the paper size is selected, or the Printable Length is specified for a custom paper size.

Optional Text for Map Scale Legend - when a legend is requested (see above), a title may be added in this field.

Print - sends the image to the printer, bmp, pdf or jpg. If you choose bmp, pdf, or jpg you will be prompted to supply a filename and location for the image file.

Cancel - exits the print menu

Note: Pipe2018 lighten background settings have been improved. The old print method may still be used. Go to System Data | Preferences | Printing and check the "Use Old Print Lighten" box.

File Size - Under System Data/Preferences, there is an option to create bmps at a lower quality setting of 16 bits per pixel. This results in a BMP file which is half the file size of the standard setting.

See also Report - Printing.

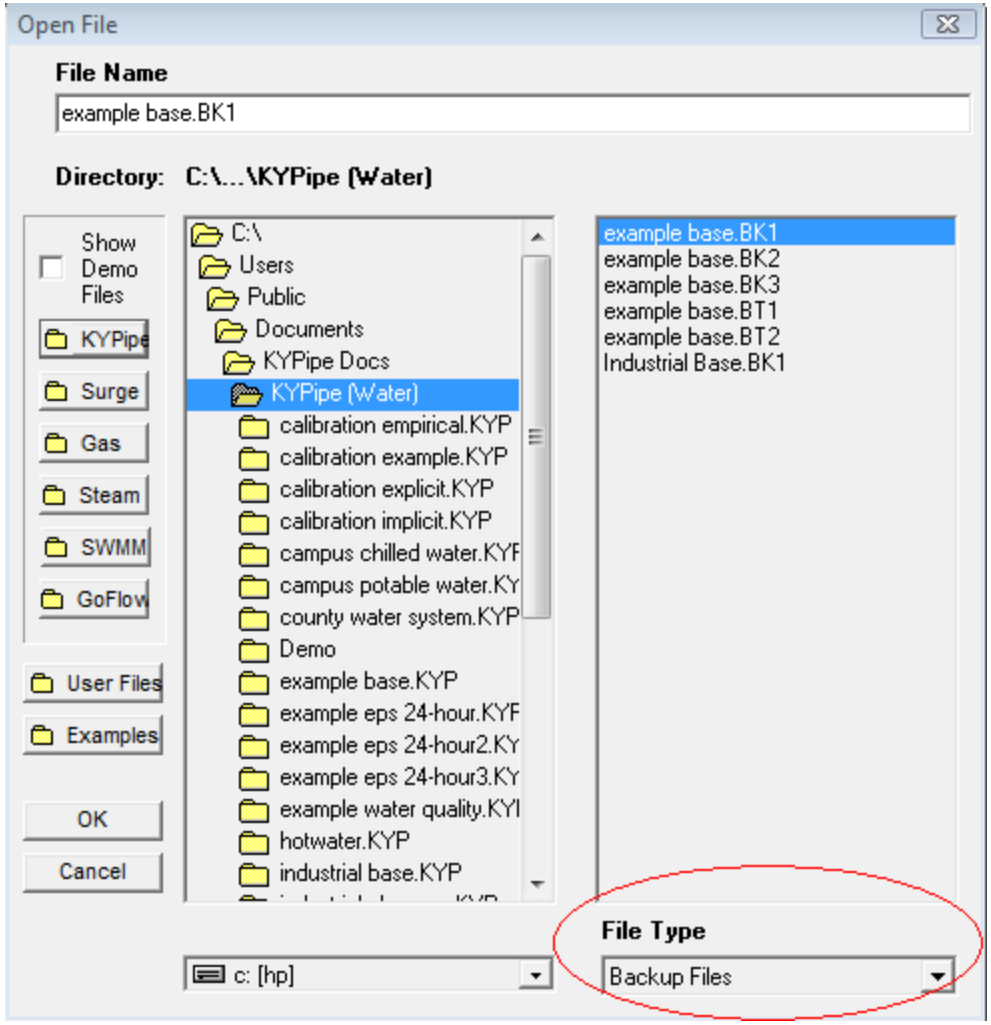
## 1.3.2.4 Files

### 1.3.2.4.1 Backup Files

Backup Files

---

Backup files are automatically saved each time a file is saved in Pipe2018. With each save, the previous version of your Pipe2018 file is kept and given the file extension BK1. With each subsequent save, that file is renamed BK2, then BK3, etc. Four backup copies of each Pipe2018 file are kept at any one time. These files may be loaded by selecting Backup Files in the File Type drop-down selector in the Open File Box.




## 1.3.2.4.2 Data Files

Data Files / Scenario Management

The data file consists of the following sections:

1. Baseline Data
2. Demand Pattern
3. Change Pattern

Baseline Data- The Baseline Data includes all the pipe and node data associated with the distribution system. It also includes all data items in System Data and Other Data appropriate to your system and the type of simulation you wish to run.

Demand Pattern- This data defines the multipliers for each demand type for one or more times (or cases). It also includes a power cost for each time (or case). This data is used with the demand data (Baseline Demand Data) and meter data (if applicable) to calculate the demands for each simulation performed.

Change Pattern- This data defines a pattern of changes for pipe and node data associated with a time (or case) for the simulation. This includes changes in the on/off status, reservoir levels, valve settings, and a variety of other pipe and node data. This data can be set up graphically by selecting the desired pipe or node and providing the specific times (or cases) and the new data in the Node Changes or Pipe Changes.

When a data file is saved, all three sections are incorporated into that data file and when it is subsequently accessed, this same data applies. If desired, however, the Demand Pattern and Change Pattern Data can be also saved as separate data files using a unique name. These data files can be subsequently accessed and placed into the currently loaded model. This provides a powerful capability for scenario management.

Using existing data files

A simulation may be performed using the current Baseline Data file and a designated Demand Pattern data file (or none) and a designated Change Pattern data file (or none). A Demand Pattern (or Change Pattern) file is designated by accessing the Demand Pattern (or Change Pattern) screens

(Setups / Defaults tab Classic), clicking Load and selecting the desired file. Each of these data section files has a name which is displayed in the upper left corner.

Any combination of Demand Pattern and Change Pattern data files may be selected as long as they are compatible with the Baseline Data file and each other. This means that the times (cases) referenced are compatible and the specific pipes and nodes incorporated in the Change Pattern are included in the Baseline Data. This approach provides maximum flexibility for managing the simulations.

## 1.3.2.4.3 Data Records

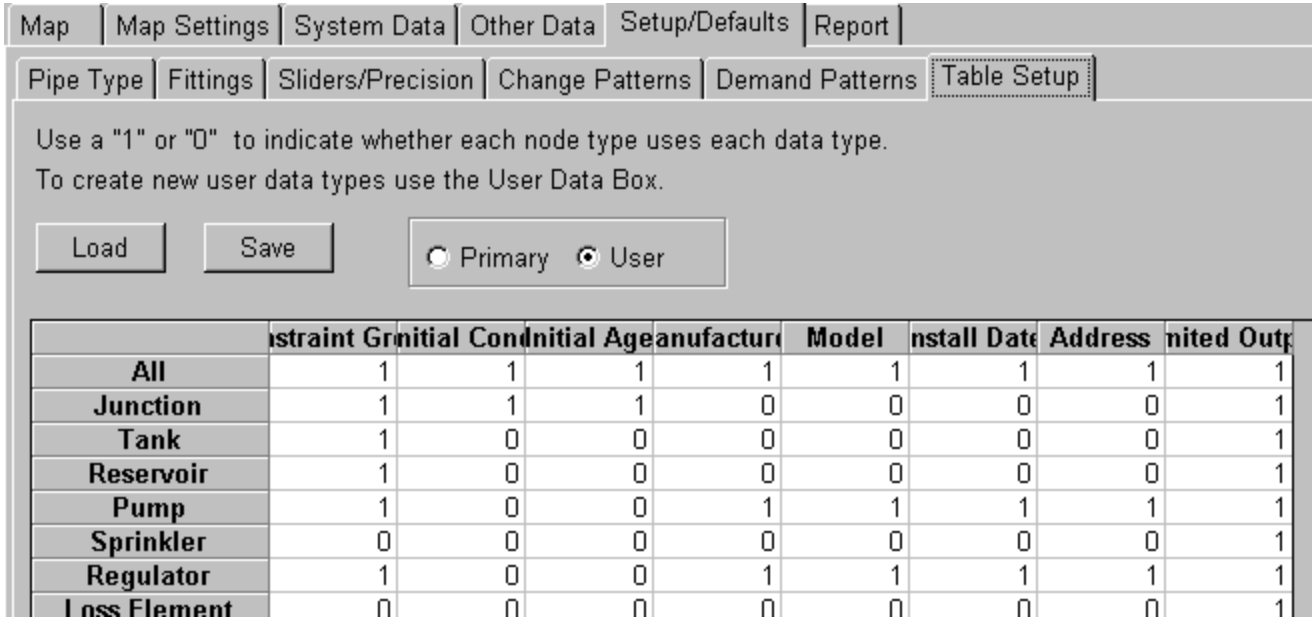
Data Records

---

The data for all of the nodes and pipe links is stored in EXCEL compatible tables that can be customized to include any desired data. Therefore, **Pipe2018** may be used to maintain complete inventory and maintenance records for all distribution system devices. In addition to the wide range of standard devices handled, two additional devices can be displayed and connected to a customized data table. In this manner, **Pipe2018** can serve a dual purpose of maintaining comprehensive inventory and maintenance records and using this information to produce current model data files that are referenced to a scaled map of the distribution system. This provides a basic AM/FM (automated mapping/facilities management) capability with an integrated hydraulic/water quality modeling capability. See Excel Import.

## 1.3.2.4.4 Table Setup

Table Setup




---

Classic Interface only.

Table Setup is used for two main purposes. One is to specify the data items appearing in the Data Tables and the other is to specify the data items appearing in the User Data box in the Node Information window.

The data items for each node are turned "on" or "off" by entering a 1 or 0 in the data field respectively. When Primary data is being viewed and set up, specifying a 1 or 0 in the data field determines whether or not that data item will appear in the Data Table for that node type. When User data is being viewed and set up, it is being determined whether or not the data item will be applied to that node type as User Data and whether or not it will appear in the data table.

The node type, All, refers to the option in the Data Tables to display all node types.

See Data Tables for information on Data Table options.

## 1.3.3 Creating and Editing a Model

### 1.3.3.1 Pipe System Layout and Modification

Laying Out a Pipe System

See the Layout and Model videos on the KYPipe web site.

See also Copy and Paste System.

See also Input and Editing Shortcuts

---

**Pipe2018 is designed to provide a very simple, intuitive user interface for 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.**

1. **Select a Node or Pipe Link** - Point mouse 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). **NOTE** - Left click AGAIN and the newly created node will NOT automatically be made into an intermediate node if it will subsequently be made an in-line node, it will remain as a junction.

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 and drag to the new location.

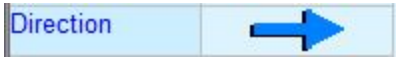
5. **Insert Node** - Point mouse to desired location in pipe link and LC (left click). Click Insert icon at the bottom of the Pipe Info window (Classic - 'Insrt' top of Pipe Information Window) and select node type from pop-up list.

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

7. **Delete Internal Node** - Select internal node and click on Delete icon (Classic - click 'Del' (delete) at top of 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.

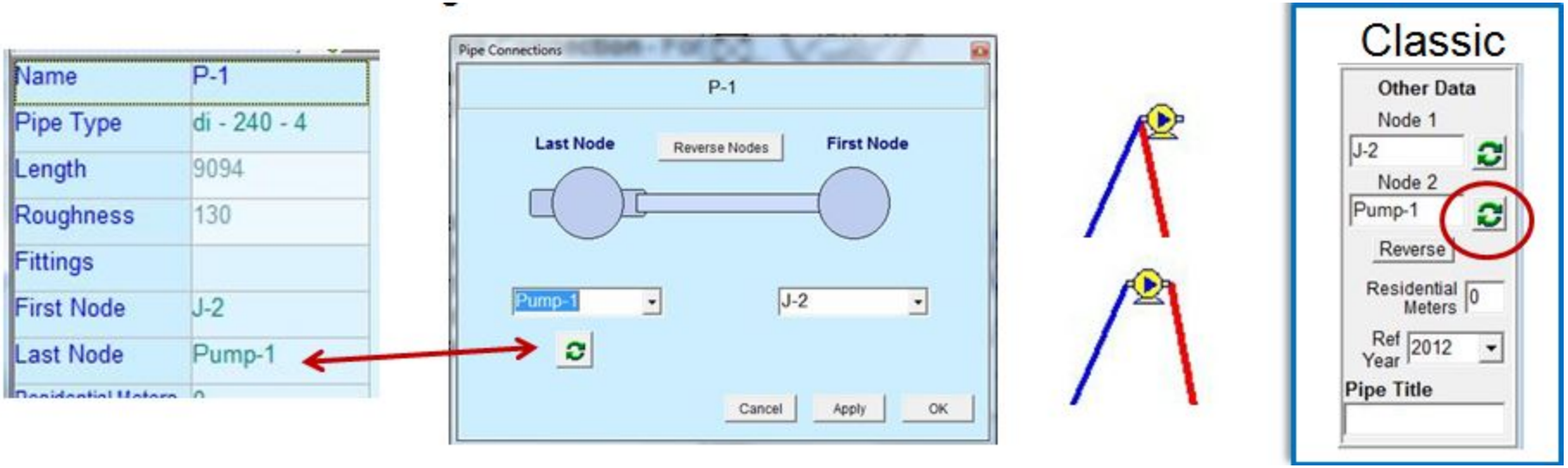
8. **Delete End Node** - Select end node and click on Delete icon (Classic - click 'Del' (delete) at top of Node Information Window). **** Note that this will also delete ALL the pipe links connecting the node. If you do not wish to do this, change the node type to a junction.

9. **Delete Pipe Link** - Select pipe and click on Delete icon (Classic - click 'Del' (delete) at top of Pipe Information Window).

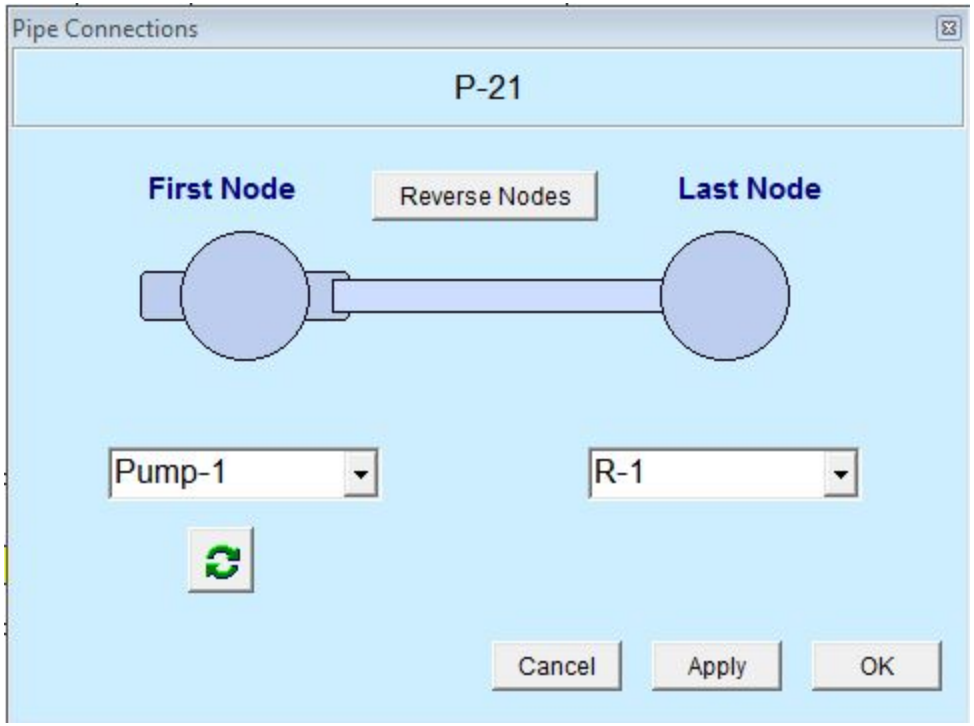
10. **Change Node Direction** - For directional end nodes (pumps, loss elements and regulators), select node and click on 

or (Classic: ) 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). The menu below is brought up by clicking the "First Node" or "Last Node" cell or in Classic it is in the Other Data box. It is necessary to ensure pipes with check valves are in the correct direction.




12. **Change Pipe Link Connection** - For pipe link connections to directional nodes, in the Pipe Information window, open the menu shown below by clicking the "First Node" cell or in Classic, this is in the Other Data box. Then click the symbol adjacent to the directional node. You will see the link connection change to the other side of the directional node.




As you lay out pipes, intermediate nodes are inserted at all changes in alignment. These are automatically changed to junction nodes if only one or more than two pipes are connected or if the properties of the two connecting segments differ. To automatically layout juctions instead of intermediate nodes, go to System Data | Preferences and click on "Do Not Automatically Layout Intermeditate Nodes".

## 1.3.3.2 Scenarios, Changes, Demand Patterns

### 1.3.3.2.1 Change Data

Change Data

---

The baseline data is used for the first analysis for both regular and EPS runs. Change data defines data changes for additional cases (regular simulations) or at

specifies times (EPS). The change data is normally entered by selecting the appropriate node or pipe and using the

icons at the bottom of the Node Info window or CHNG button at the top (Classic) ndow to access the associated Node Change Box or Pipe Change Box to enter data. All change data is saved in the

Change Pattern file and this data can be reviewed and edited by accessing the Change Pattern window under the menu icons

or Setups/Defaults / Change Pattern (Classic).

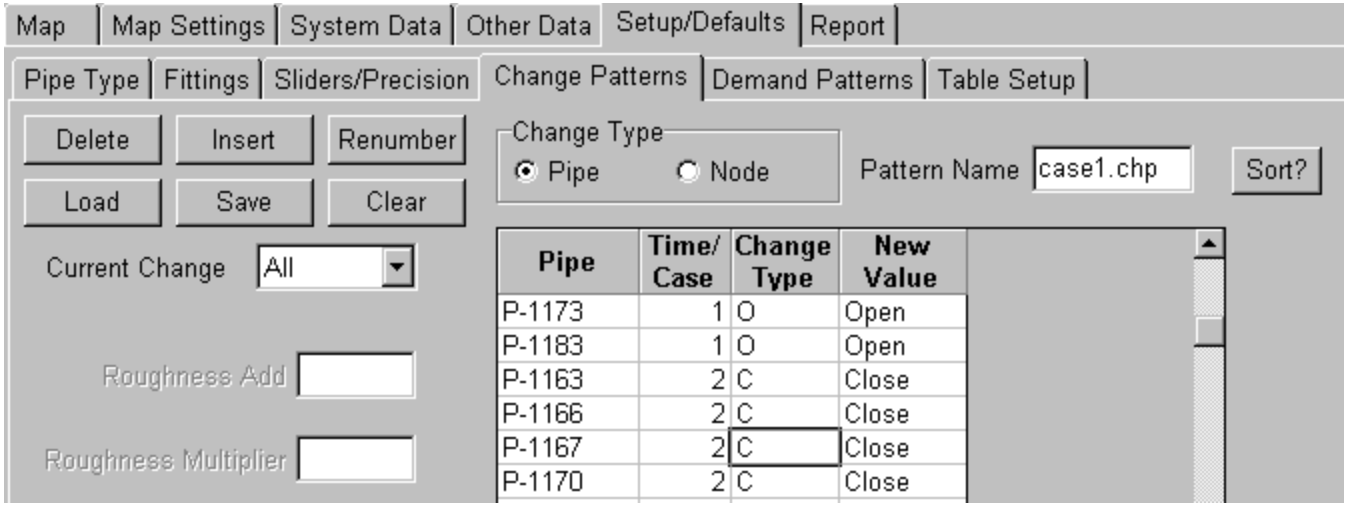
See also Change Patterns

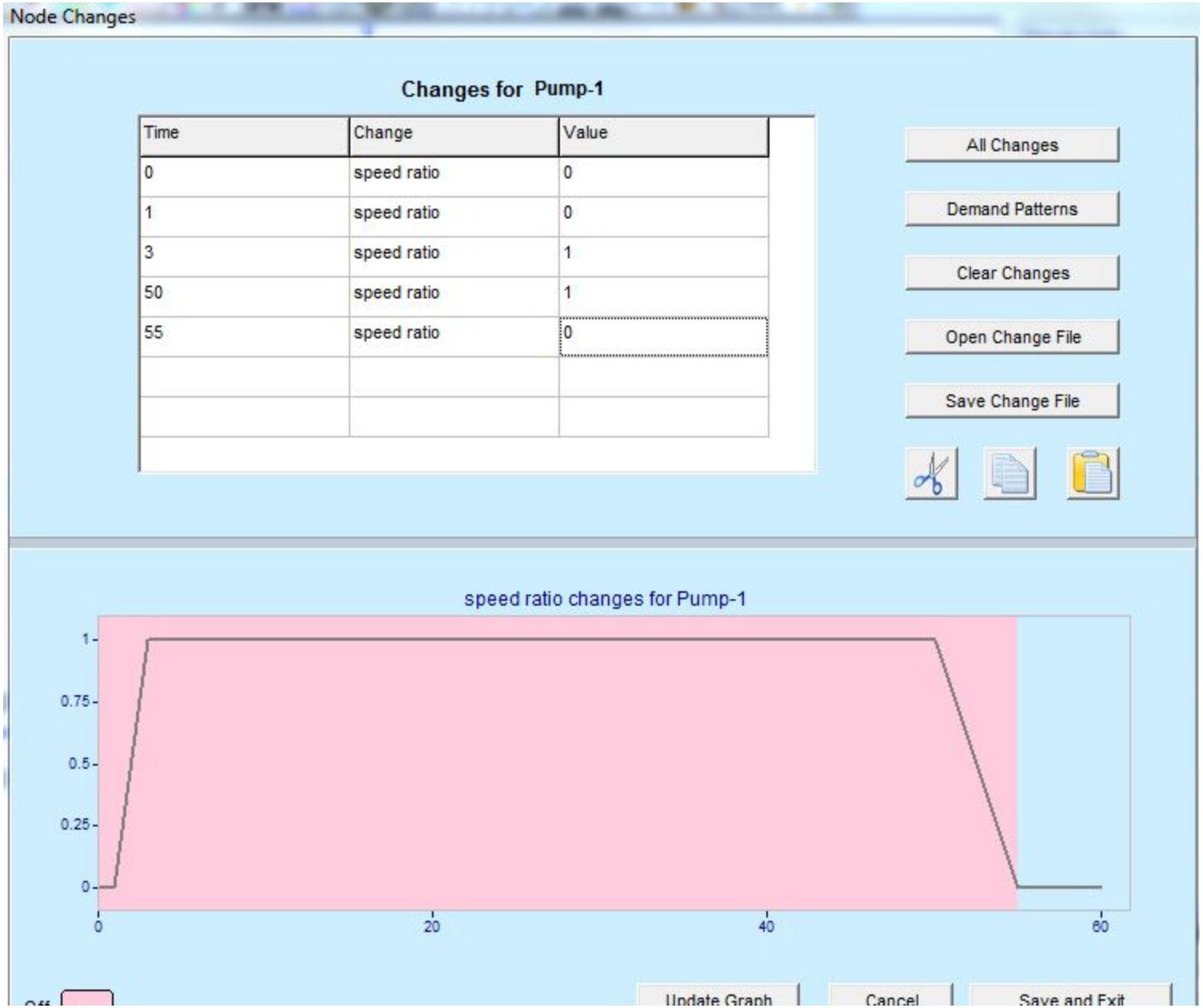
Data Files/Scenario Management

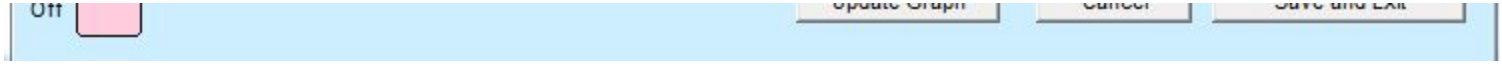
## 1.3.3.2.2 Change Patterns

Change Patterns

KYnetic icons


Node Changes
Changes for Pump-1
Time
Change
Value
0
speed ratio
0
1
speed ratio
0
3
speed ratio
1
50
speed ratio
1
55
speed ratio
0
All Changes
Demand Patterns
Clear Changes
Open Change File
Save Change File
speed ratio changes for Pump-1
1
0.75
0.5
0.25
0
0
20
40
60
Update Graph
Cancel
Save and Exit




This window can be accessed under or under Setups and Defaults / Change Patterns (Classic). The Change Pattern window defines the various node and pipe changes set up for different cases (regular simulation) or times (EPS). The Change Pattern can be viewed as shown above by selecting either Pipe or Node under Change Type. A Change Pattern can be saved and imported for use with a compatible baseline data file.

The Change Pattern is normally set up by selecting individual nodes or pipes and entering data into the Pipe Change Box or Node Change Box displayed in the Information Window.

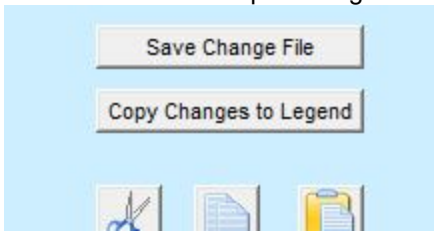
The Change Pattern also can be input or edited using the Change Patterns window shown above.

The user may assign any time case number (0 and up, does not have to be an integer), and the changes will be calculated in numerical order regardless of the order entered. The Sort button may be used to place changes in numerical order. See Demand Patterns, Important Notes

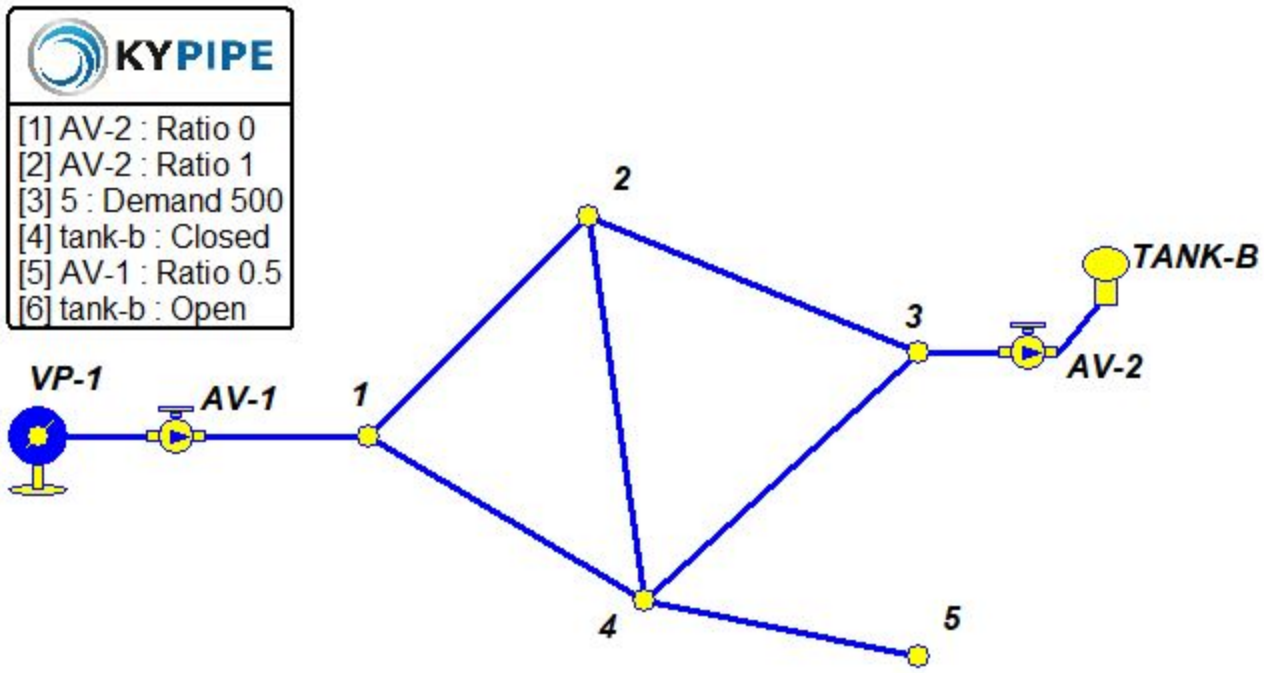
See also Data Files / Scenario Management

**Copy Changes to Legend**

Added to the Node/Pipe Changes table as shown:




A list of the changes will be added. To change the appearance or location, go to Legend menu

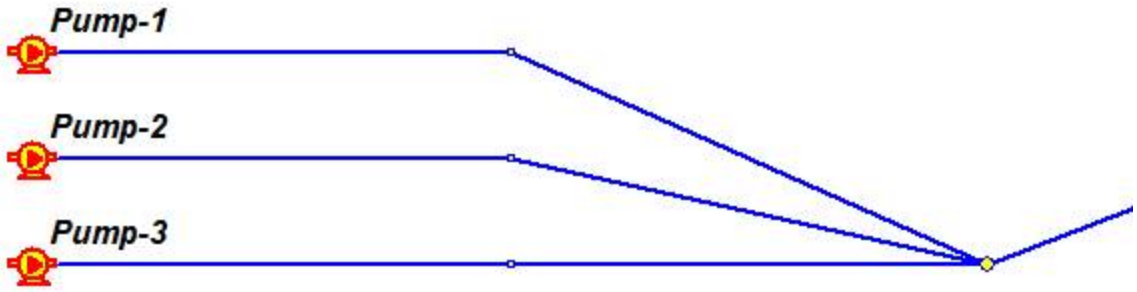



## 1.3.3.2.3 Generate Change Pattern

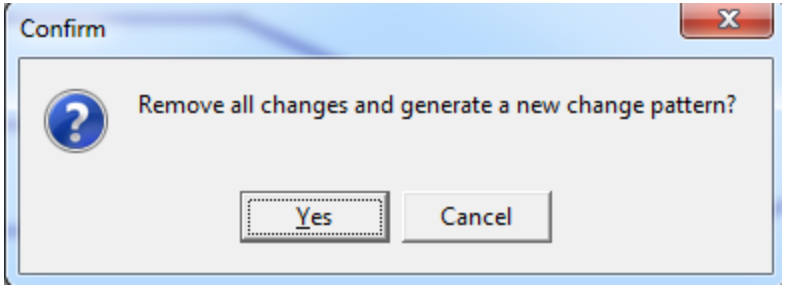
### Generate Change Pattern

KYnetic interface only.

A group of nodes may be selected in Group Mode.

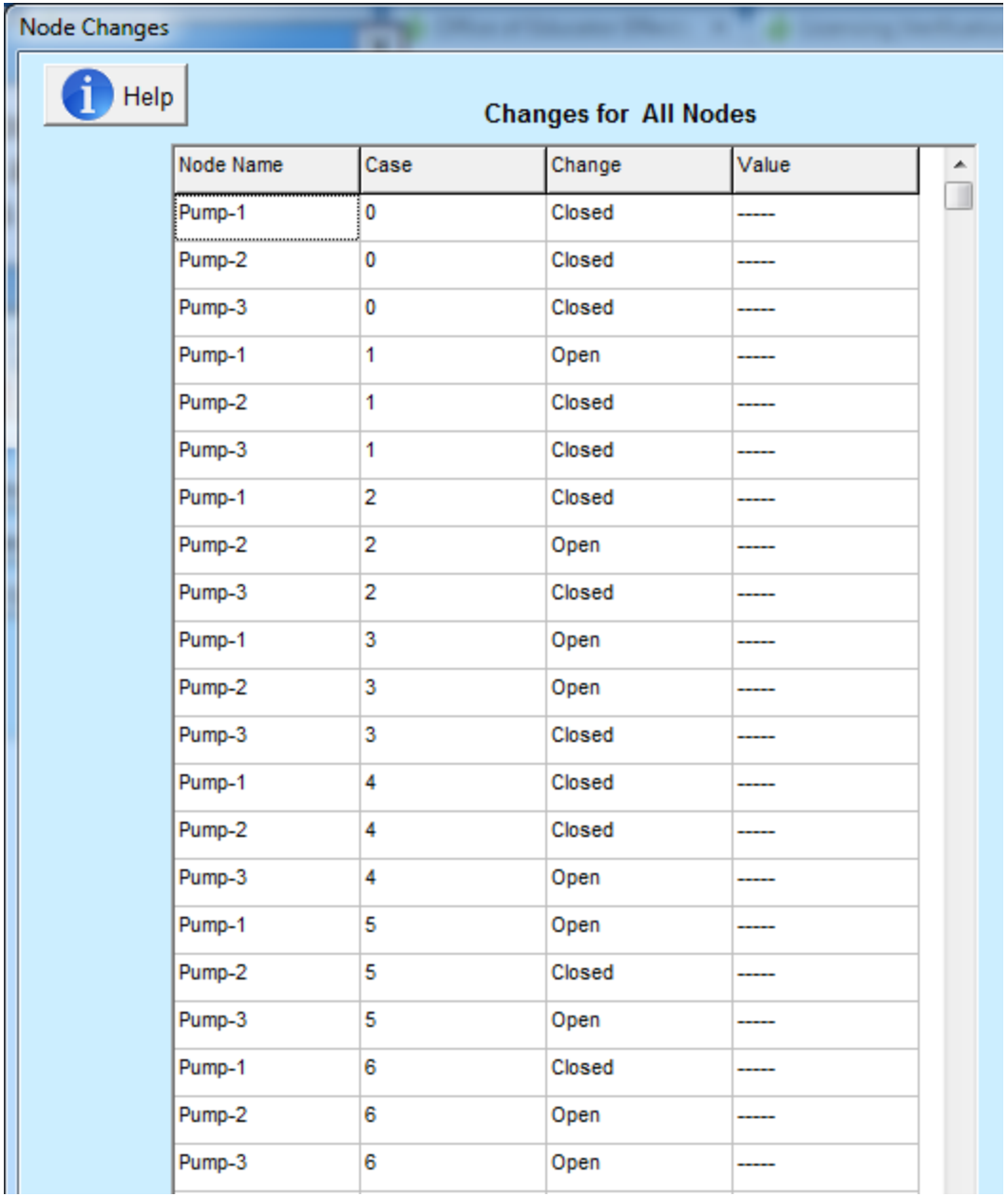



Go to Facilities Management and click Generate Change Pattern.

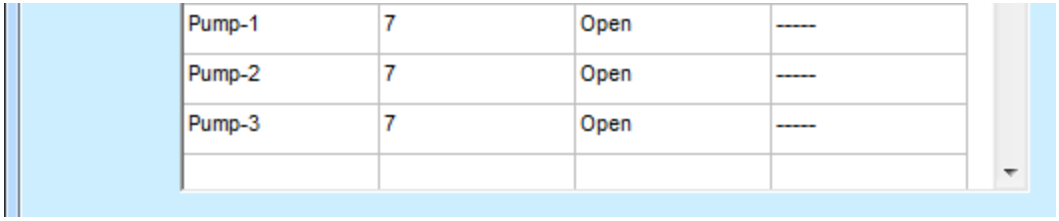



Click Yes. If you do not wish to lose the current change pattern, click Cancel. Open the Change Pattern window and Save the change pattern, then run Generate Change Pattern again.

All possible combinations of of/on status settings for the selected nodes will be automatically generated in the change table.




| Node Name | Case | Change | Value |
|---|---|---|---|
| Pump-1 | 0 | Closed | ----- |
| Pump-2 | 0 | Closed | ----- |
| Pump-3 | 0 | Closed | ----- |
| Pump-1 | 1 | Open | ----- |
| Pump-2 | 1 | Closed | ----- |
| Pump-3 | 1 | Closed | ----- |
| Pump-1 | 2 | Closed | ----- |
| Pump-2 | 2 | Open | ----- |
| Pump-3 | 2 | Closed | ----- |
| Pump-1 | 3 | Open | ----- |
| Pump-2 | 3 | Open | ----- |
| Pump-3 | 3 | Closed | ----- |
| Pump-1 | 4 | Closed | ----- |
| Pump-2 | 4 | Closed | ----- |
| Pump-3 | 4 | Open | ----- |
| Pump-1 | 5 | Open | ----- |
| Pump-2 | 5 | Closed | ----- |
| Pump-3 | 5 | Open | ----- |
| Pump-1 | 6 | Closed | ----- |
| Pump-2 | 6 | Open | ----- |
| Pump-3 | 6 | Open | ----- |



| | | | |
|---|---|---|---|
| Pump-1 | 7 | Open | ----- |
| Pump-2 | 7 | Open | ----- |
| Pump-3 | 7 | Open | ----- |
| | | | |

## 1.3.3.2.4 Demand Allocations/Meters

Demand Allocations / Meters

---

Pipe2018 provides a variety of advanced capabilities for handling, allocating, and varying demands. You can assign the demands directly to the junction nodes or you can incorporate meters into your model and Pipe2018 will assign the demands to the adjacent junctions. See Meters. The following definitions apply.

**Demand** - A baseline flow requirement associated with a junction node or metered connection and specified in the Junction Data or Metered Connection Data.

**Demand Type** - An integer (or R for Residential) is used to associate the demand with a particular user type (residential, commercial, industrial, etc.). Specified in the Junction Data or Metered Connection Data.

**Demand Pattern** - A matrix of multipliers for each demand type and case or time (EPS) to be analyzed. The baseline flow is multiplied by the associated multiplier for each case (time). See Demand Patterns.

### There are three way which Pipe2018 can handle assigned demands:

1. **No Specified Type** - All demands default to Demand Type = 1 and the Demand Pattern can include only one row of multipliers (type = 1). Users will not enter demand type data.
2. **Single Demand Types** - Each junction can be assigned one baseline demand and one demand type. As many different types as desired can be used in the system. The Demand Pattern will include a row for each demand type.
3. **Multiple Demand Types** - Each junction can be assigned up to five different baseline demands and types. As many different types as desired can be used for the system and the Demand Pattern will contain a row for each type specified.

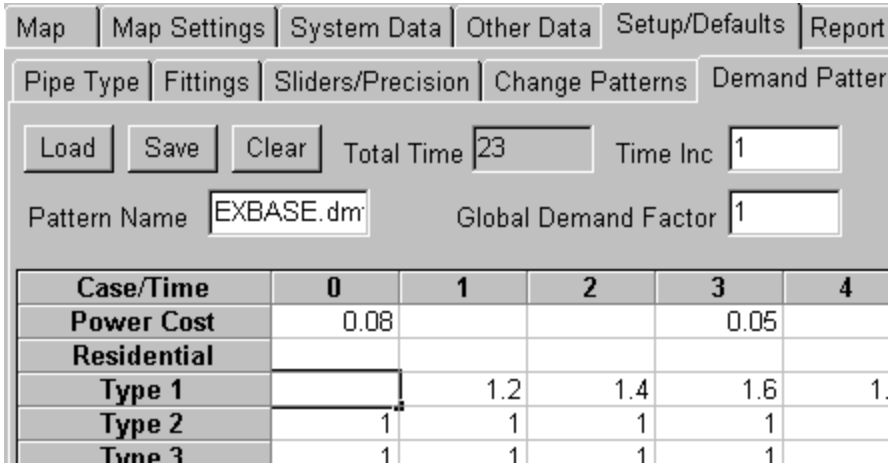
For Multiple Demand Types select the box labeled Multiple Demand Types. See Preferences menu or System Data / Preferences (Classic).

For other types this box is not checked.

## 1.3.3.2.5 Demand Patterns

Demand Patterns

KYnetic Demand Patterns menu icon:




---

This table defines demand multipliers for each demand type (Type 1, in this case) and time / case covered by the simulation. For EPS the multipliers are provided at constant intervals which is normally set to the Computational Period (System Data - EPS).

Note: Zero Demand - If the cell is blank or 0, the demand factor from the last non-zero input will be used. This will carry over until it is changed by another non-zero value. If it is desired to set a Demand Type to zero, use a very small non-zero number such as 0.001 so that the resulting very small demands are effectively zero in the simulation.

The Power Cost is the cost of electricity (dollars / kwh) and is used to compute the cost of electricity for pump operation for an EPS. A default value may be

defined in the System Data - EPS and entries in this table will override the value and allow you to define a variable rate over the simulation period. In this example the cost will be 0.08 cents/kwh for hours 0, 1, and 2 and will be 0.05 cents/kwh for hours 3, 4, 5, etc. until a new value is specified.

Entries left blank in the table will default to the last value entered. If the first entry is blank, the multiplier will default to 1.0.

**Important Notes for Steady State Simulations:**

When viewing the results in the Report or in the Node/Pipe Results tables or graphs for non-EPS simulations, Case 0 is ALWAYS a baseline case (no changes or demand multipliers apply), regardless of whether a demand factor is entered in column 0 in this table. If a demand factor is entered in column zero, then this factor will be calculated and reported as Case 1. Regardless of the Time/Case number assigned by the user in this table or in the Change Data or Change Pattern, the cases will be numbered with integers in numerical order. In other words for instance, if the user creates three changes, numbered 1, 1.5, and 2 and then enters demand factors into columns 0, 2 and 3 the results will be reported as follows:

Case 0: Baseline case

Case 1: Demand factor in column 0

Case 2: Change 1

Case 3: Change 1.5

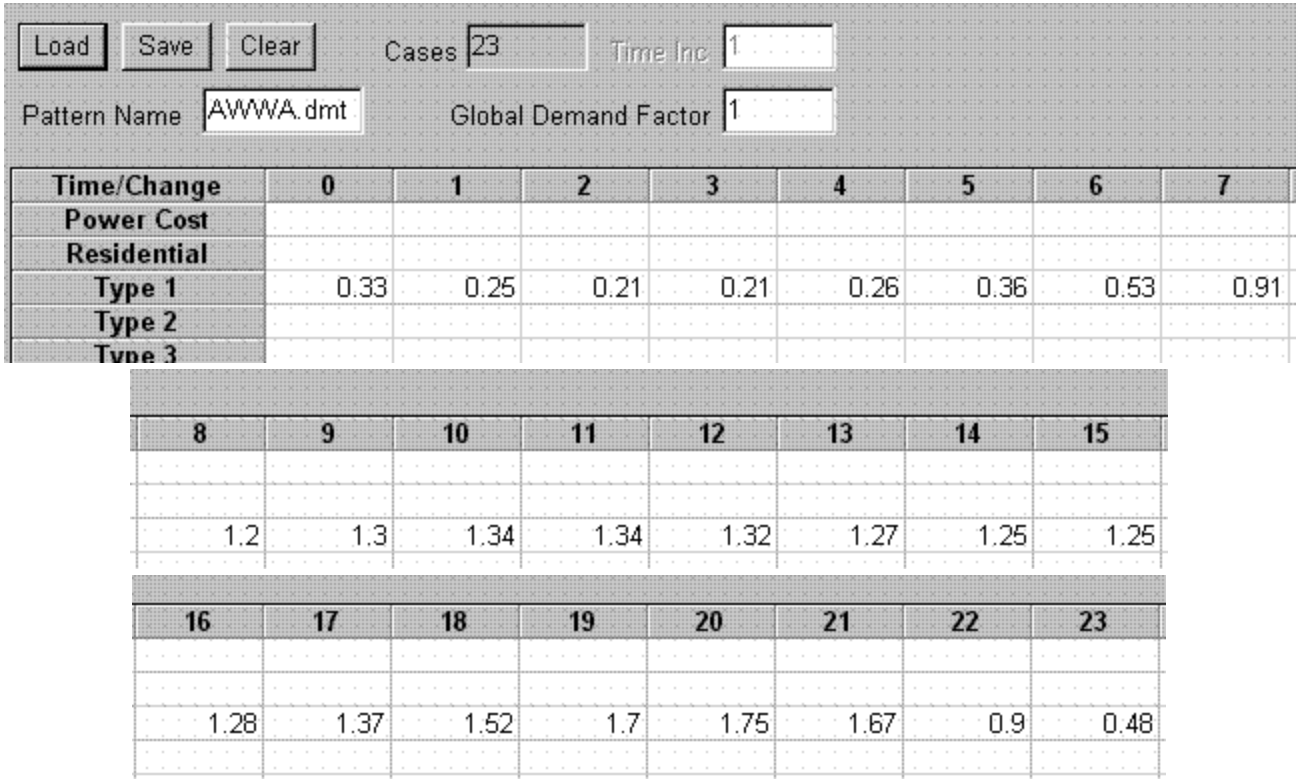
Case 4: Change 2 and demand Factor 2

Case 5: Demand factor 3

If an EPS is being conducted, there is NO BASELINE CASE. Therefore, Case 0 is time 0 and will use the demand factor entered into column 0.

The American Water Works Association (AWWA) provides a typical example of a 24-hr demand curve. This demand pattern, named AWWA.dmt, is available for use in your Pipe2018 folder. The multipliers are depicted in the Pipe2018 Demand Pattern table below.

The demand pattern is set here for Extended Period Simulations or for multiple case scenarios. Important Note: The zero time case in this table is equivalent to case 1 in the map labels and the Report. The demand table begins with case zero to correlate with time zero (usually equals midnight) for EPS simulations.




| Time/Change | 0 | 1 | 2 | 3 | 4 | 5 | 6 | 7 |
|---|---|---|---|---|---|---|---|---|
| Power Cost | | | | | | | | |
| Residential | | | | | | | | |
| Type 1 | 0.33 | 0.25 | 0.21 | 0.21 | 0.26 | 0.36 | 0.53 | 0.91 |
| Type 2 | | | | | | | | |
| Type 3 | | | | | | | | |

| 8 | 9 | 10 | 11 | 12 | 13 | 14 | 15 |
|---|---|---|---|---|---|---|---|
| | | | | | | | |
| | | | | | | | |
| 1.2 | 1.3 | 1.34 | 1.34 | 1.32 | 1.27 | 1.25 | 1.25 |

| 16 | 17 | 18 | 19 | 20 | 21 | 22 | 23 |
|---|---|---|---|---|---|---|---|
| | | | | | | | |
| | | | | | | | |
| 1.28 | 1.37 | 1.52 | 1.7 | 1.75 | 1.67 | 0.9 | 0.48 |
| | | | | | | | |

See also Data Files/Scenario Management

## 1.3.3.2.6 Demand Specification

Demand Specification - Overview

Certain data are required to describe boundary pressure and flow specifications. The most important of these are the flows entering or leaving the distribution system at the junction nodes (demands). For some systems, analyses are carried out with no inflows or outflows (demands) specified. For most systems, however, demand requirements are specified at designated junction nodes and the pressure and flow distribution is determined for this situation. At any junction node, the external inflow (negative) or outflow (positive) demand may be

specified. For each different case or time (EPS) any change in these demands from the initial specifications must be input.

Variations in demands represent very important data. Pipe2018 allows multiple global demand factors associated with up to ten junction demand types to enable you to easily create multiple demand patterns. In this manner the demands at junctions which may represent residential, commercial or industrial users can be changed using different demand factors to represent different types of demand variations which occur for regular simulation changes or throughout an EPS.

The elevations of junction nodes must be specified if the pressures (or pressure heads) are to be calculated. Values for the elevation of junction nodes are not required to compute the flow distribution and only affect the pressure calculation at the junction nodes. Thus, elevations need only be specified where calculated values of pressure are desired. Elevations are required if an accurate representation of pressure contours are to be displayed.

At each FGN, including variable level storage tanks for (EPS only), the initial HGL (pressure head + elevation) is an operating condition which must be specified. This means that the elevation of surface levels in reservoirs and the initial levels for storage tanks must be specified for regular simulations. Also, if there are pressure requirements at fixed grade nodes, these are incorporated into the value specified for the HGL maintained by the FGN. If there are pressure regulating valves or pressure sustaining valves in the system HGL representing the setting must be specified. The regulated pressure is incorporated into the calculation of the HGL representing the valve setting (pressure head + elevation).

## 1.3.3.2.7 Scenario Management

Scenario Management

---

Pipe2018 has a variety of tools to aid the user in Scenario Management. Under the Change Pattern and Demand Pattern tabs are drop-down menus from with the user may select a number of saved patterns. The Pipe2018 program installation includes a variety of default patterns which the user may add to the directory of their file for easy access. Or patterns may be created by the user and saved for Scenario Management purposes.

## 1.3.3.3 Copy and Paste

Copy and Paste

## Copy and Paste Pipes

You may select a group of pipes and nodes and copy them with all assigned attributes to another system (another .p2k file). Using the GBox or individually selecting nodes and pipes in Group Mode, select the portion of the system which you want to copy. Under Edit in the Main menu, select copy. Close the file and open the file to which you want to paste. Under Edit select paste. This will paste the nodes and pipes at the same coordinates at which there were located in the original file. However, if you enter Layout mode and select a node and then paste, the paste will occur at that node. The pasted system will not be connected to or overwrite any part of the system to which it was pasted, even if two nodes are at the same coordinate location. After pasting the system you can click near the pasted pipes and nodes and drag them to the desired position. It may be necessary to de-select and re-select one of the nodes if the click and drag doesn't work initially.

## Excel – Merging Pipe2018 Data Files using Excel (For Version 1 Users)

**The main purpose of merging the Pipe2018 datafiles is so the user can work on a simple subsystem and later merge the subsystem datafile into an existing main system datafile. There are a few restrictions that must be adhered to concerning the subsystem file before the merger can be successful.**

1. No Duplicate Names. The Subsystem Datafile cannot have any pipe, node or component names identical within the Subsystem file itself or identical to an existing name within the Main System Datafile.

2. No Change Data or EPS Data can be merged from the Subsystem Datafile into the Main System Datafile. Any of this data for the Subsystem file will need to be added through the final merged datafile.

3. The Subsystem Datafile cannot have any Intermediate nodes or components in a pipe between end nodes such as check valves, hydrants, control valves, etc. Only those components that can be deemed as end nodes may be included in the Subsystem Datafile such as tanks and pumps. However, using the Edit Group feature of Pipe2018 the user can change the intermediate data into end node data, rather than lose it, and change it back to intermediate data in the merged file.

### MERGING DATAFILES USING EXCEL

### IN Pipe2018:

**1. LOAD Pipe2018.**

**2. OPEN the Main System File** and make any necessary changes before opening EXCEL.

**3. Click FILE | Import Export (Pipe2000 Utilities in Classic interface) | EXPORT EXCEL FILE.**

**4.** On the Save the file? Information box, **Click Yes**.

**5.** On the Saving C:\pathname\systemfilename.XLS Pipe2018 box, **Click OK**.

**6. OPEN the Subsystem File** and make any necessary changes to comply with the rules defined above before opening EXCEL.

**7. Click FILE | Import Export (Pipe2000 Utilities in Classic interface) | EXPORT EXCEL FILE.**.

**8.** On the Save the file? Information box, **Click Yes**.

**9.** On the Saving C:\pathname\subsystemfilename.XLS Pipe2018 box, **Click OK.**

## IN MICROSOFT EXCEL:

**1. LOAD Microsoft Excel.**

**2. OPEN the System Excel File.**

**3. Align the System Excel File Window** so the top of the System file is at the top of the Excel file window and the bottom of the System file is below the halfway point of the Excel file window. See figure below.

**4. Open the Subsystem Excel File. Do Not** close the System file opened in step 2.

**5. Align the Subsystem Excel File Window** so the bottom of the Subsystem file is at the bottom of the Excel file window and the top of the Subsystem file overlaps the bottom of the System file. See figure below.

**MERGE NODES:**

**1. Click on the NODES tab** at the bottom of the Subsystem file.

2. Starting at Column A, Row 2 **hold left click down and drag mouse down and to the right** to highlight all rows of nodes and all columns from Column A to Column AA.

**3. Click Edit | Copy.**

**4. Click anywhere on System file.**

**5. Click on the NODES tab** at the bottom of the System file.

**6. Click on Column A on the first row after all node data.** The cell becomes highlighted.

**7. Right click highlighted cell and Paste.** Node data is now merged.

**MERGE PIPES:**

**1. Click anywhere on Subsystem file.**

**2. Click on the PIPES tab** at the bottom of the Subsystem file.

**3.** Starting at Column A, Row 2 **hold left click down and drag mouse down and to the right** to highlight all rows of pipes and all columns from Column A to Column Z.

**4. Click Edit | Copy.**

**5. Click anywhere on System file.**

**6. Click on the PIPES tab** at the bottom of the System file.

**7. Click on Column A on the first row after all pipe data.** The cell becomes highlighted.

**8. Right click highlighted cell and Paste.** Pipe data is now merged.

**MERGE PUMP IDs:**

**1. Click anywhere on Subsystem file.**

**2. Click on the Pump ID Data tab** at the bottom of the Subsystem file.

3. Starting at Column A, Row 2 **hold left click down and drag mouse down and to the right** to highlight all rows of pump ID s and all columns from Column A to Column AM.

**4. Click Edit | Copy.**

**5. Click anywhere on System file.**

**6. Click on Column A on the first row after all pump ID data.** The cell becomes highlighted.

**7. Right click highlighted cell and Paste.** Pump ID data now merged.

**SAVING MERGED FILE:**

**1. Click anywhere on Subsystem file.**

**2. Click File | Close.**

**3. Click File | Save As.**

**4.** Give the merged file a new filename and **click Save**.

**5. Click File | Exit.**

**IN Pipe2018:**

1. **Reopen the Pipe2018 window.** If Pipe2018 is not still loaded then **RELOAD Pipe2018**.
2. **Click FILE | Import Export (Pipe2000 Utilities in Classic interface) | IMPORT EXCEL FILE.**
3. On the Save the File? Information box, **Click No.**
4. On the Opening File box **select the merged file and click OK.**
5. **Correct the Pump ID s for Subsystem pumps.** The pumps from the Subsystem file will have new ID numbers assigned in the new merged file. The correct pump curve data is included in the merged file but the assigned ID numbers are now different and needs to be corrected.

For copy and paste of node data, see Node Information Window

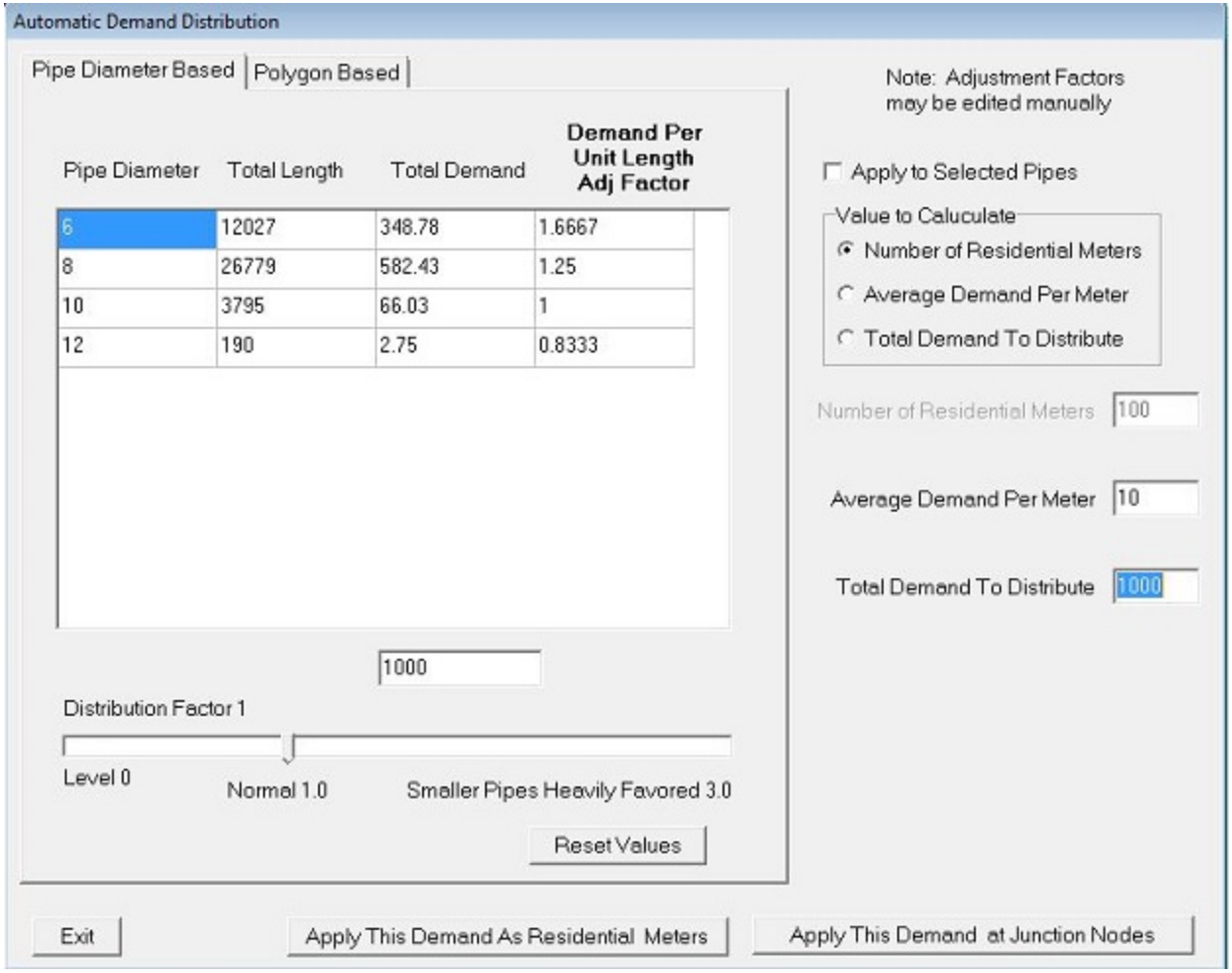
## 1.3.3.4 Automatic Demand Distribution

Two approaches for demand distribution are available: Pipe Diameter Based and Polygon Based

**Polygon Based Demands** (Pipe2014 and later versions) - input a Total Demand. The more densely clustered an area of piping is, the less demand per pipe is assigned. More isolated pipes receive more demand.

**Pipe Diameter Based Demands** - Demands are distributed based on pipe diameter. The default is for smaller pipes to be favored but the user has some control over this relationship using the Distribution Factor slider

In the Main Toolbar, under Faciliites Management / Other is the Automatic Demand Distribution option. The following menu appears:




## Pipe Diameter-Based

All of the pipes in the system will appear in the list above.

**Apply Demands Only to Selected Pipes** - If pipes are selected on the map view using Group Mode, this option may be used to apply demands only to those pipes.

**Value to Calculate** - depending on this selection, the user fills out two of the following data fields.

**Number of Residential Meters** - Enter **Average Demand Per Meter** and **Total Demand to Distribute** both in system flow units.

**Average Demand Per Meter** - Enter **Number of Residential Meters** and **Total Demand to Distribute** in system flow units

**Total Demand To Distibute** - Enter **Number of Residential Meters** and **Average Demand Per Meter** in system flow units

You'll notice as these fields are filled, the **Total Demand** for each pipe is automatically updated. Also the field below the pipes represents the total number of meters.

**Distribution Factor** - to change the amount of demand placed on smaller or larger pipes, use this slider. As you move the slider to the right, more demand is assigned to smaller diameter pipes, to the left will favor larger diameter pipes. **Reset** will place the slider in the orginal position. These may be edited manually also. Just click on the **Demand Per Unit Length Adj. Factor** column and enter the factor. The formula for this is:

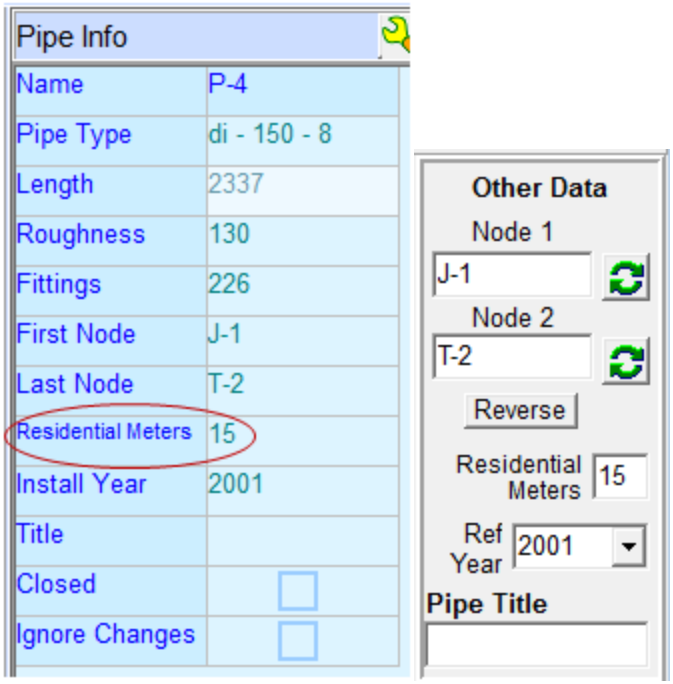
**Demand per Unit Adj Factor = (10/Diam)^(Distrubution Factor).**

**Apply This Demand as Residential Meters** - demand will be distributed as a number of Residential Meters for each pipe.

**Apply This Demand at Junction Nodes** - demands will be distributed as Demands at junctions nodes in system flow units

**OK** - exit

The demands are applied as Residential Meters with an Average Residential Meter Demand. The number of Residential Meters assigned to each pipe is shown in the Pipe Info window. In the Classic interface, this is in the Other Data pipe box.




Then under System Data | Other you'll see the Average Residential Meter Demand

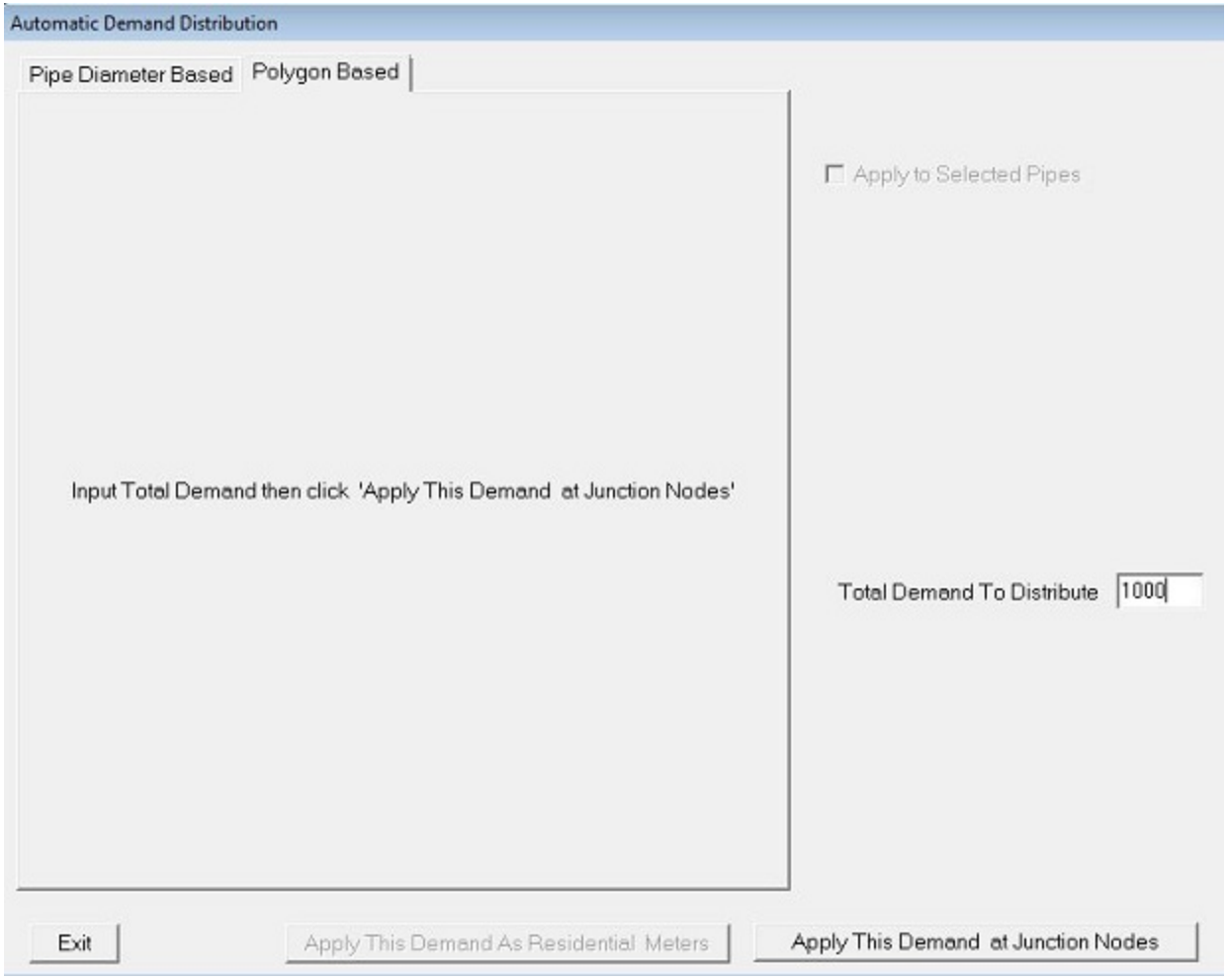
Note that demands and meters may be edited further in the map after automatic demands have been applied.

### Polygon-Based

Only total demand is input. The more densely clustered an area of piping is, the less demand per pipe is assigned. More isolated pipes receive more demand.

The demand will be applied at junction nodes.

Theissen polygon method is used to distribute demands.

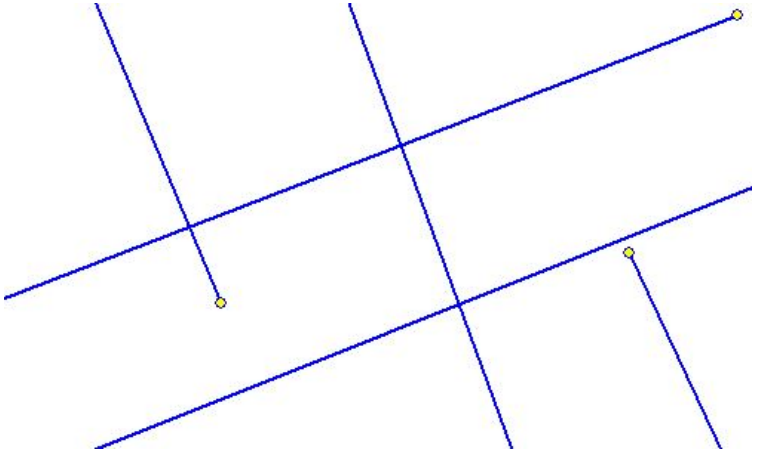



1.3.3.5 Connect Crisscross Pipes

**Connect Crisscross Pipes**

This feature connects pipes which cross over or come very close to other pipes but are not connected. It adds a junction node at the intersection and connects crisscrossed or near pipes to the node. The pipes are divided into multiple separate pipes.

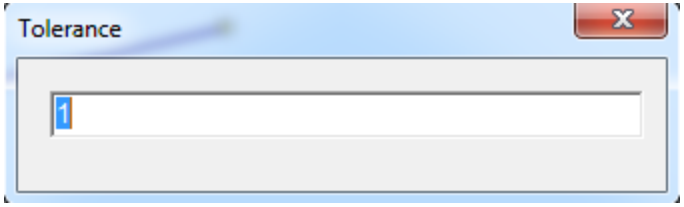
Crisscross pipes and near pipes:



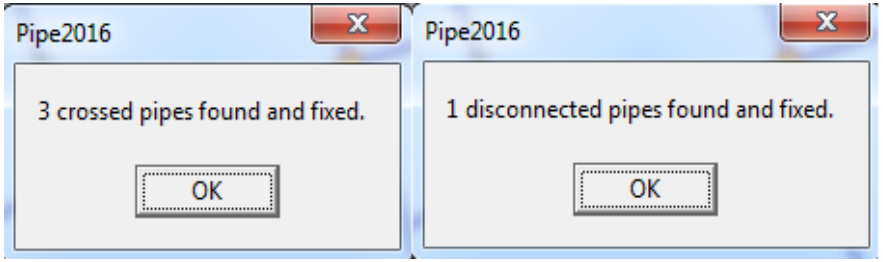
Go to Edit

Click Connect Crisscross Pipes

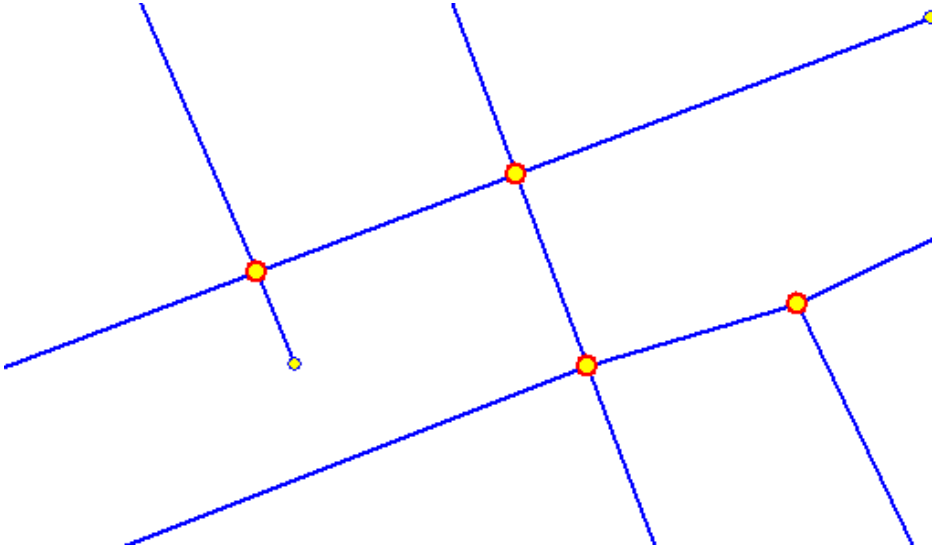

A tolerance value may be set. The default is 1 (ft or m). For pipes which are close but not crossed the tolerance refers to the distance from the pipe end to the nearest pipe:




Confirmation of connected pipes:




Pipes are connected, if they cross or fall within the specified tolerance:



## 1.3.3.6 Data Requirements

Data Requirements

---

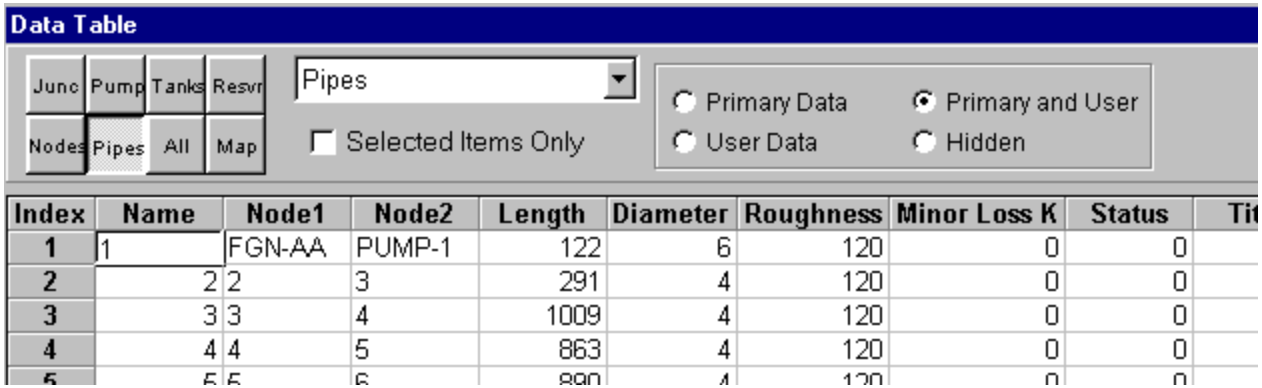
System Data
Pipe Data
End Node Data
- all nodes
- junction data
- pump data
- tank data
- reservoir data
- regulator data
- pressure supply data
- loss element data
- sprinkler data
- active valve

Internal Node Data
- all nodes
- metered connections
- hydrants

Text Node Data
Change Data
Demand Pattern Data

## 1.3.3.7 Data Tables

Data Tables



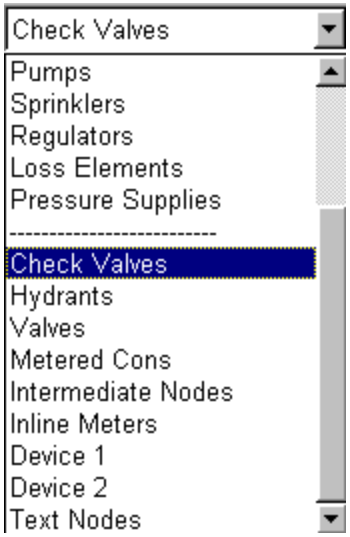

| Index | Name | Node1 | Node2 | Length | Diameter | Roughness | Minor Loss K | Status | Ti |
|---|---|---|---|---|---|---|---|---|---|
| 1 | 1 | FGN-AA | PUMP-1 | 122 | 6 | 120 | 0 | 0 | |
| 2 | 2 | 2 | 3 | 291 | 4 | 120 | 0 | 0 | |
| 3 | 3 | 3 | 4 | 1009 | 4 | 120 | 0 | 0 | |
| 4 | 4 | 4 | 5 | 863 | 4 | 120 | 0 | 0 | |
| 5 | 5 | 5 | 6 | 890 | 4 | 120 | 0 | 0 | |

See also Data Tables - Quickstart Example

---

The Data Table is the spreadsheet-style, writable format which can be used for entering, editing, and manipulating (see Excel Import) data for a Pipe2018 file. In general this is not the recommended method for handling data, but the use of data tables can provide some additional options and capabilities. Data Tables are accessed by clicking the Table button in the left-hand, vertical tool bar on the Map screen. Editing of data tables may be done directly with the data tables (for existing elements), through the map screen or in the advanced spreadsheet editor, accessed by clicking the ALL button. Note that new pipes and nodes may only be added in the advanced editor. See Data Tables - Quickstart Example for and example of how to enter a system using the data table editor.

There are six main tables; Junctions, Pumps, Tanks, Reservoirs, Nodes, and Pipes. Click on the corresponding button or on the drop-down box to access these. Click on All to access the advanced editor. Click on Map to return to the Map screen. Other network elements may also be summarized in a data table. These items are all listed in the drop-down selector box.




## Junctions - summary of all Junction Nodes. See also Junction Data

| Index | Name | Elv. | Demand 1 | Demand 1 Type | Demand 2 | Demand 2 Type | Demand 3 | Title | Demand 3 Type |
|---|---|---|---|---|---|---|---|---|---|
| 1 | 1 | 109 | 0 | 1 | 0 | 0 | 0 | | 20 |
| 2 | 2 | 100 | 0 | 1 | 0 | 0 | 0 | | 20 |

| Demand 4 | Demand 4 Type | Demand 5 | Demand 5 Type | Constraint Group | Initial Conc | Initial Age | Limited Output |
|---|---|---|---|---|---|---|---|
| 0 | 0 | 0 | 0 | -16 | | | |
| 0 | 0 | 0 | 0 | -16 | | | |

Demand 1, 2, etc. - These columns contain the baseline value (in specified flow units, gpm, cfs, etc.) for each demand type at that node. Many users will not choose to use Multiple Demand Types (see Junction Data) and will have only one demand per node. The default column for this case is Demand1. Each node may have up to five different demand types, if multiple demand types are specified..

Demand Type 1, 2, etc. - The Demand Type columns contain the integer identifier for each demand type group. While each node may only have 5 different demands associated with it, the system as a whole may have as many demand types as desired. A zero always refers to a Residential demand, but all other integers are user-defined groups (e.g. for the user a 1 may identify light industrial demands, a 2 may be heavy industry, etc.). Assigning a Demand Type to each demand at a node allows the user to manipulate a group of demands for such applications as an Extended Period Simulation.

## Pumps - summary of all Pumps Nodes

See also Pump Data

| Index | Name | Elv. | Speed/Power | Efficieny | Type | Title | ID | Off Status | Constraint Group |
|---|---|---|---|---|---|---|---|---|---|
| 22 | PUMP-1 | 95 | 0 | 0 | 0 | | 1 | | -16 |

| Initial Conc | Initial Age | Limited Output |
|---|---|---|
| | | |

Speed/Power - For a pump with an ID (meaning a pump curve is associated with the pump) this is where the speed goes (a multiplier, 1 being the normal speed). For a constant power pump (no ID) this is the power in horsepower or kilowatts.

Efficiency - In units of 0-100%, for constant power pumps (hp or kw).

Type - There are two types of pumps. Either a 1 is entered in this column for constant power pumps, or a 0 is entered for a pump with a pump curve.

ID - When a pump is identified with a pump curve, the curve data is assigned an ID number. That ID (an integer, 1 - 250) is entered in this column. The column is blank for a constant power pump. To edit a pump curve, do so in Node Information Window on the Map screen.

## Tanks - summary of all Tank Nodes

See also Tank Data

| Index | Elv. | Max Level | Min Level | Initial | Inflow | Volume(Diameter) | Title | Shape ID | Off Status |
|---|---|---|---|---|---|---|---|---|---|
| 23 | 120 | 0 | 0 | 250 | 0 | 0 | | 0 | |

| Constraint Group | Initial Conc | Initial Age | Limited Output |
|---|---|---|---|
| -16 | | | |

Max Level -The maximum level (elevation in ft. or m) to which a tank may fill.

Min Level - The minimum level (elevation in ft. or m) to which a tank may drain.

Initial - The elevation of water in the tank at the beginning of the simulation.

Inflow - This is the amount (if any) of flow (in specified flow units, mgd, cfs, etc.) into the tank from an external source.

Volume (Diameter) - If the tank is a constant diameter tank, the diameter (in ft or m) is entered in this column as a negative number. Otherwise enter the volume of the tank (ft3 or m3)..

Shape ID - For tanks which are not constant diameter tanks, shape data to describe the change in volume as the tank fills and empties is entered and edited in the Node Information Window in the Map screen. This data is assigned a Shape ID. This ID is entered in this column.

## Reservoirs - summary of all Reservoir Nodes

See also Reservoir Data

| Index | Name | Elv. | grade | Title | Off Status | Constraint Group | Initial Conc | Initial Age | Limited Output |
|---|---|---|---|---|---|---|---|---|---|
| 21 | FGN-AA | 94 | 120 | | | -16 | | | |

Grade - elevation of water surface (or elevation plus pressure head of a pressure supply) in ft or m.

## Nodes - summary of all Nodes (including pumps, tanks, reservoirs, etc)

| Index | Name | Elv. | Item1 | Item2 | Item3 | Item4 | Item5 | Title | ~Type | ~Reference | Item6 (ID) | Item7 | Item8 |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 19 | 19 | 109 | 0 | 1 | 0 | 0 | 0 | | 1 | 11 | 20 | 0 | 0 |
| 20 | 20 | 105 | 100 | 1 | 0 | 0 | 0 | | 1 | 13 | 20 | 0 | 0 |
| 21 | FGN-AA | 94 | 120 | 0 | 0 | 0 | 0 | | 3 | 21 | 0 | 0 | 0 |
| 22 | PUMP-1 | 95 | 0 | 0 | 0 | 0 | 0 | | 4 | 22 | 1 | 0 | 0 |

| Item9 | Item10 | ~Not Used | ~Result Index | Result | Image File | Off Status | Constraint Group | Initial Conc | Initial Age | Limited Output |
|---|---|---|---|---|---|---|---|---|---|---|
| 0 | 0 | | | | | | -16 | | | |
| 0 | 0 | | | | | | -16 | | | |
| 0 | 0 | | | | | | -16 | | | |
| 0 | 0 | | | | | | -16 | | | |

Item1 - 5, etc. - Because this screen is a summary of several types of nodes, Items 1 - 5 contain node-specific data. See individual tables to identify the data.

~Type - The data in this column are Node type identifiers. Node types are identified with an integer as follows: 1 - junction, 2 - tank, 3 - reservoir, 4 - pumps, 8 - sprinkler, 9 - regulator, 11 - loss element, 12 - active valve, 14 - pressure supply, 17 - rack sprinkler. Other node types are intermediate nodes and should be entered and edited in the map screen. When adding a Node in the editor, a Node Type must be entered.

~Reference - Internal use only.

Item6 - 10 - Node-specific data. See individual data tables to identify the data.

~Not Used - Internal use only

~Result Index - Internal use only

Result - Internal use only

Image File - Nodes may have a bitmap image file associated with them for display on the Map screen. This column contains the full path and filename for this node image including the .bmp extension.

## Pipes - summary of all Pipe Links

See also Pipe Data

| Index | Name | Node1 | ~Index1 | Node2 | ~Index2 | Length | Diameter | Roughness | Minor Loss K | Status | ~Length Fixed |
|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | 1 | FGN-AA | 21 | PUMP-1 | -22 | 122 | 6 | 120 | 0 | 0 | 1 |
| 2 | 2 | 2 | 2 | 3 | 3 | 291 | 4 | 120 | 0 | 0 | 1 |
| 3 | 3 | 3 | 3 | 4 | 4 | 1009 | 4 | 120 | 0 | 0 | 1 |

| ~Selected | Title | ~Roughness Fixed | Material\|Rating | Reference Year | # of Meters | Fittings | ~Reference | ~Result Index |
|---|---|---|---|---|---|---|---|---|
| 0 | | 1 | | 0 | 0 | | 1 | |
| 0 | | 1 | | 0 | 0 | | 2 | |
| 0 | | 1 | | 0 | 0 | | 3 | |

| Result | Constraint Group | Calibration Group | Bulk Rate | Wall Rate | Limited Output |
|---|---|---|---|---|---|
| | -16 | 1 | | | |
| | -16 | | | | |
| | 16 | | | | |

Node1 - The name of the first connecting End Node. The Node order should be in the allowed flow direction for a pipe containing a check valve. New pipes may only be added in the advanced editor (All button). When adding a new pipe in the editor be sure to specify node 1 and 2 and leave ~index 1 and 2 blank. In addition, when adding pipes in the editor, the nodes entered must be end nodes (not intermediate nodes).

~Index1 - Do not edit

Node2 - This is the name of the next connecting Node along the pipe link (may be an Intermediate Node and not necessarily the next Junction Node) to which flow in the pipe goes. When using the advanced editor to add new pipes, always enter the end node (as opposed to an intermediate node). Even though an intermediate node may be listed in this column, Pipe2018 recognizes the End Node for the pipe link in it's analysis. To view a pipe link list, select Hidden Data and view the Nodes screen. For pipes which list an Intermediate Node in the Node2 Column, the end nodes will be listed in the column called Links.

~Index2 - Do not edit

Length - Length of pipe in ft or m.

Diameter - Diameter of pipe in inches or mm.

Roughness - Roughness of the pipe according to the specified method of analysis (Hazen-Williams, etc.)

Minor Loss K - Sum of minor loss coefficients for Fittings (length is taken into consideration in the analysis)

~Length Fixed - A 1 in this column means the length will be fixed. Otherwise, the length will be scaled as it appears in the Map screen.

~Selected - Internal use only

~Roughness Fixed - A 1 in this column means the roughness will be fixed. This precludes the pipe from inclusion in any roughness calibration or age-based calculation.

Material/Rating - Material and rating entered as "pvc|200" (for example). Keep the same same format when entering the material, separate the material and rating by a "|" (symbol above the backslash), and then enter the rating.

Reference Year - This is the reference year for roughness calculations, usually the installation year.

# of Meters - Number of residential meters connected to the pipe link.

Fittings - The symbols for each fitting is entered. There are default symbols, but these may be user defined.

~Reference- Internal use only

~Result Index - Internal use only

Result - Internal use only

Calibration Group - user data item

Bulk Rate - Water quality (EPANET) data

Wall Rate - Water Quality (EPANET) data

## All

This is the advanced spreadsheet editor. It may be used to add pipes, nodes, etc. Additional columns may be added in this mode. The format is identical to the individual data tables (includes User and Hidden data). Use the advanced editor to import data from another spreadsheet. See Excel Import. To edit a table, click on All, and then the tab at the bottom of the screen to access the table of interest. Files may be saved and loaded within the advanced editor.

## Map

Returns to the Map Screen

## Selected Items Only

This is an option to show only selected rows. These rows are first selected in Group Mode in the Map screen and then will appear in the data table.

## Primary Data

This selection will display only the primary data associated with each data table.

## User Data

This selection will display only the User Data for each data table. User Data refers to the data used to specify groups within the system to be used for Selected Output or for such functions as Calibration or Constraint calculation (see also Reports (System Data), Sets and Group Mode, Pipe User Box, Node User Box ). For the Junction, Pump, Tanks, Reservoir, and Nodes data tables, columns 26 and higher are reserved for User Data. For the Pipes data table, columns 22 and higher are reserved..

## Primary and User

This selection will display Primary and User Data. In the above explanations of the data tables, Primary and User data is displayed for all data tables except for pipes, which includes Hidden Data also.

## Hidden

This selection will display all of the columns in the data table. This includes Primary Data, User Data, and some additional items such as node coordinates which, in general, will not need to be edited using the data tables. An exception to this would be a user entering all data through the data tables, instead of graphically, where coordinate data entry would be necessary.

## Common column headings:

Index - In all data boxes the Index is the spreadsheet row number.

Name - the alpha-numeric reference for the network element. A default is assigned by Pipe2018, but may be edited by the user. Do not change the names of elements in the editor, use the map!

Elevation - this is the elevation of the network element. For End Nodes such as Tanks, Reservoirs, and Pumps this is the elevation of the Pipe Link at the point where it connects to the End Node. The units are in ft (or m)

Title - This is a user-assigned reference of up to 255 characters. This alpha numeric information can be displayed on the Map screen (in the Main Menu, click on Labels and either Pipe or Node Title), and in printouts.

Constraint Group - This is an integer identifier used when setting up a constraint to define a parameter for a group of pipes. See Sets and Group Mode or the video entitled Group Editing on the Pipe2018 tutorial program for instructions on how to set up a Constraint Group. Constraint groups may be edited in the data table, but it is recommended to create such groups in Group Mode in the Map screen.

Initial Concentration - For EPANET analyses, this is the constituent concentration of Chlorine or other chemical additive in ppm.

Initial Age - For EPANET analyses, this is the age of water at beginning of an EPANET simulation. This is user defined. May use more than one Initial Age column, e.g. Initial Age 1, Initial Age 2.

Limited Output - Limited Output is considered an Attribute for Selected Pipe or Node Output (see Reports (System Data)). This column is simply reserved as an additional way to specify a group of network elements for use with the Selected Output feature. Selected Output is a feature used to generate an output report displaying a group of results of interest to the user. This is especially useful for large systems where output reports can be lengthy . Unless otherwise specified when creating a new file (appears as a check box option in the New File Specifications Screen), Limited Output columns will appear in each data table by default. As with other group types, Limited Output groups may be created using Group Mode in the Map Screen (See Sets and Group Mode). A group may also be created or edited within the data table. Elements to be included in a Selected Output report are identified by an integer value. A different integer is used for each separate output group. Select the group to be used for each analysis in the System Data / Reports screen. See also Selected Output for an example.

Off Status - a 1 in this column means the network element is turned off, a 0 means on.

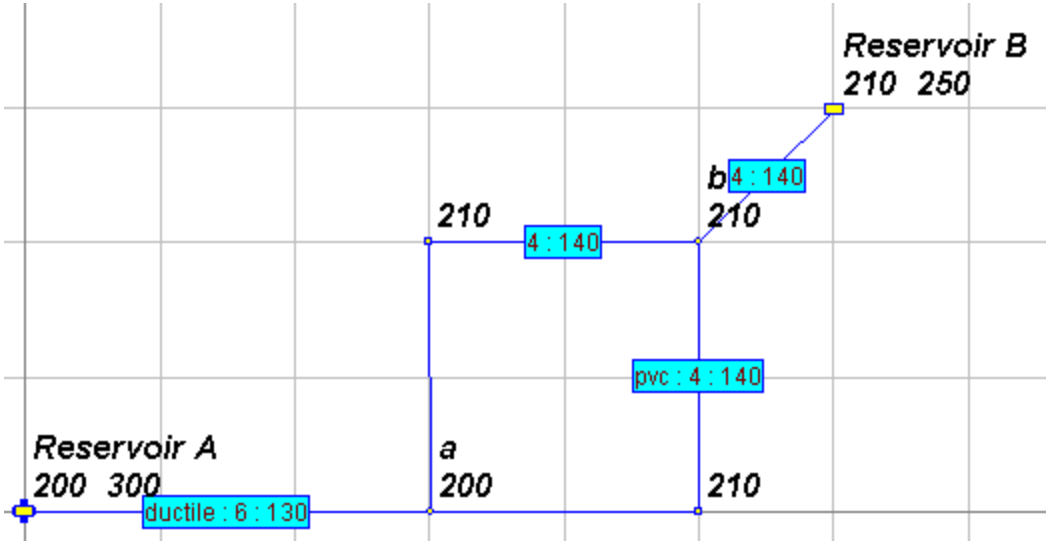
## 1.3.3.8 Data Table - Quickstart Example

Data Table - Quickstart Example

See also Data Tables

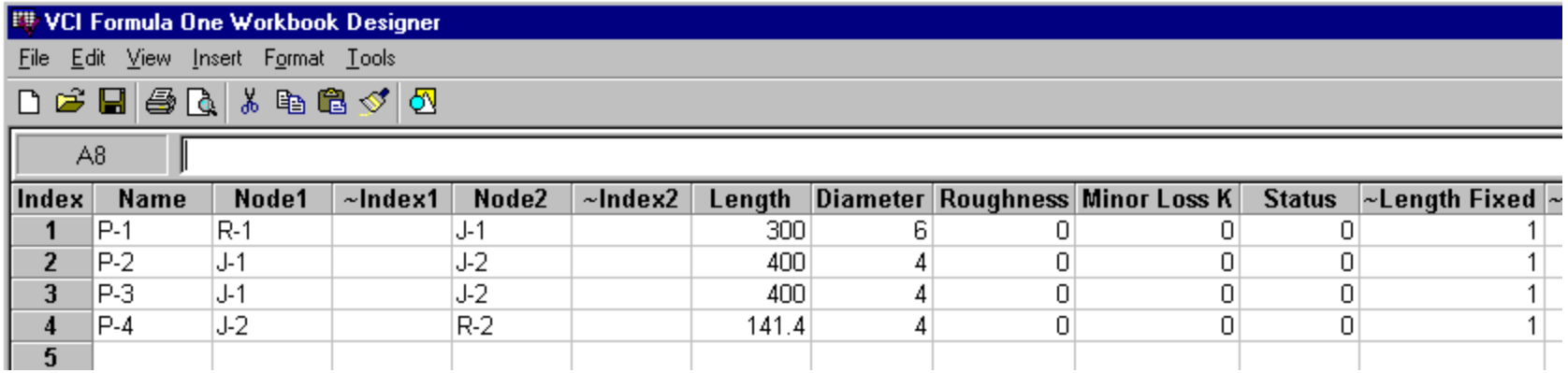
---

This is an example of how to use the data tables to enter a simple system. For this example we will use the data in the Quickstart Example found in the Quickstart Example Pipe2018 Quickstart Guide. The initial data to be entered is as follows:




This system has been placed on a 100-foot grid and the first node, Reservoir A is placed at coordinates (0,0). In addition, the following ratings data has been provided: for the pipe from Reservoir A use ductile:250, for the rest of the pipes use pvc:150.

To enter the piping data, click on Data Table icon in the Map screen to access the Data Table. Select Hidden data, so that the coordinates columns will appear on the Nodes table as well as some other important data items. Click on the Pipes table and then on All to enter the advanced editor. The data will be entered under the Pipes tab in this screen. It is a good idea to use the Pipe2018 naming convection and to call the pipes P-1, P-2 etc., the Junction Nodes J-1, J-2, etc., and the Reservoirs R-1, R-2, etc.. Most importantly, however, remain consistent when entering this type of alpha-numerical data into the editor. Your data should look as follows:

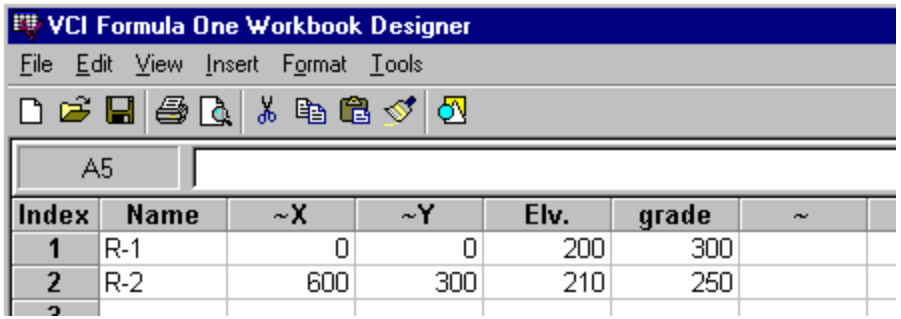



| Index | Name | Node1 | ~Index1 | Node2 | ~Index2 | Length | Diameter | Roughness | Minor Loss K | Status | ~Length Fixed | ~ |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | P-1 | R-1 | | J-1 | | 300 | 6 | 0 | 0 | 0 | 1 | |
| 2 | P-2 | J-1 | | J-2 | | 400 | 4 | 0 | 0 | 0 | 1 | |
| 3 | P-3 | J-1 | | J-2 | | 400 | 4 | 0 | 0 | 0 | 1 | |
| 4 | P-4 | J-2 | | R-2 | | 141.4 | 4 | 0 | 0 | 0 | 1 | |
| 5 | | | | | | | | | | | | |

| ss Fixed | Material\|Rating | Refere |
|---|---|---|
| 0 | ductile\|250 | |
| 0 | pvc\|150 | |
| 0 | pvc\|150 | |
| 0 | pvc\|150 | |
| | | |

Note the ~Index columns are blank and the Intermediate Nodes (alignment changes) in Pipes 2 and 3 are NOT accounted for at this point. When entering a new pipe in the editor, only enter the End Nodes in the Node1 and Node2 columns, never Intermediate Nodes. Placing the Intermediate Nodes must be done on the Map screen after data entry is complete. Pipe2018 will automatically add the necessary data after those Intermediate Nodes are placed. Note the convention used to enter the material and rating. A '1' in the ~Length Fixed makes the length entered into the Length column fixed, that is, it will not be changed or scaled if a connecting Node is moved in the Map screen.

Next the Node data must be entered. We will start with the Reservoir data. Exit the editor, select Reservoir and then click on All to return to the editor. This step is important because it lets the advanced editor know the type of data to be entered and the column headings under Nodes will be labeled accordingly, thus avoiding any confusion as to the type of data to be entered for each Node Type. If this is not done then the Node editing table will be generalized for all Node Types, i.e. the columns will be called Item1, Item2, etc, instead of Grade, Demand, etc. Click on the Nodes tab. The entered data for Reservoirs will look like this:




| Index | Name | ~X | ~Y | Elv. | grade | ~ | ~Type | ~Refe |
|---|---|---|---|---|---|---|---|---|
| 1 | R-1 | 0 | 0 | 200 | 300 | | 3 | |
| 2 | R-2 | 600 | 300 | 210 | 250 | | 3 | |

Note that the coordinates correlate with the distance in feet, making the coordinates straightforward to calculate in the Quickstart example. A very important item is the Junction Type. In the column ~Type, an integer identifier is entered which will differentiate the Node Types for the analysis. These are the types of End Nodes which may be entered by the user (other numbers are reserved for intermediate nodes and will not be entered in the data tables by the user):

1 - Junction

2 - Tank

3 - Reservoir

4 - Pumps

8 - Sprinkler

9 - Regulator

11 - Loss element

12 - Active valve

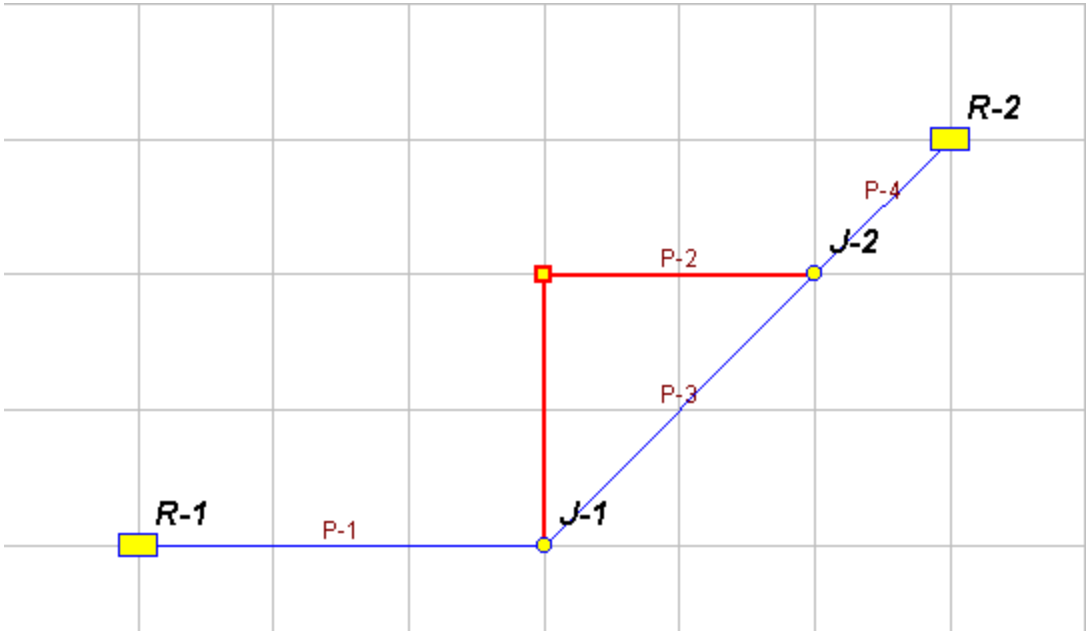
14 - Pressure supply

17 - Rack sprinkler

Next enter the Junction data. Exit the editor, click on Junctions and then All to bring up the Junction-specific column titles in the advanced editor. Enter the remaining data after the Reservoir data as follows (note the differing column headings for Junctions):

| Index | Name | ~X | ~Y | Elv. | Demand 1 | Demand | ~Type | ~Refere |
|---|---|---|---|---|---|---|---|---|
| 1 | R-1 | 0 | 0 | 200 | 300 | | 3 | |
| 2 | R-2 | 600 | 300 | 210 | 250 | | 3 | |
| 3 | J-1 | 300 | 0 | 200 | | | 1 | |
| 4 | J-2 | 500 | 200 | 210 | | | 1 | |

Now, when the above data is viewed in the Map screen, the layout will be similar to that of the Quickstart example, excepting the placement of the two intermediate nodes in Pipes 2 and 3. In the Map screen, select one of the two pipes (at this point you will not be able to visually differentiate between them as they fall in the same location on the screen), Click Insert on the Pipe Information window and add an Intermediate Node. Click and drag the node which appears to the appropriate location.




Repeat this for pipe P-3 to complete the system layout. If you view the data tables again after this layout is complete, you will see the data that has been entered to account for this alignment change.

Finally, let's add a pump into the system using the advanced editor. Click on Pumps and then All and go the the Nodes tab in the advanced editor. The Quickstart Example calls for a constant power pump at 40 hp to be placed 100 ft from Reservoir A (R-1) in pipe P-1 with an elevation of 210 feet. Enter this data after the other Nodes as follows:

| Index | Name | ~X | ~Y | Elv. | Speed/Power | Efficieny |
|---|---|---|---|---|---|---|
| 1 | J-1 | 300 | 0 | 200 | 0 | 0 |
| 2 | J-2 | 500 | 200 | 210 | 0 | 0 |
| 3 | R-1 | 0 | 0 | 200 | 300 | 0 |
| 4 | R-2 | 600 | 300 | 210 | 250 | 0 |
| 5 | | 300.1331588 | 200.0571131 | 0 | 0 | 0 |
| 6 | | 499.6402767 | 0.5528242111 | 0 | 0 | 0 |
| 7 | Pump-1 | 100 | 0 | 210 | 40 | |

| | Type | ~Link | ~Status | ~Selected | Title | ~Type |
|---|---|---|---|---|---|---|
| 0 | 0 | 0 | 0 | 0 | | 1 |
| 0 | 0 | 0 | 0 | 0 | | 1 |
| 0 | 0 | 0 | 0 | 0 | | 3 |
| 0 | 0 | 0 | 0 | 0 | | 3 |
| 0 | 0 | -2 | 1 | 0 | | 15 |
| 0 | 0 | -2 | 1 | 0 | | 15 |
| | 1 | | | | | 4 |

It is important to enter a 1 in the Type column, signifying a constant power pump (a 0 indicates a pump identified by a pump curve) and a 4 in the ~Type column indicating this Node Type is a Pump. The layout including the pump may now be viewed in the map screen and the example is ready for an analysis.

For more detailed explanations of the data items, see Data Tables.

## 1.3.3.9 Deleting Intermediate Nodes

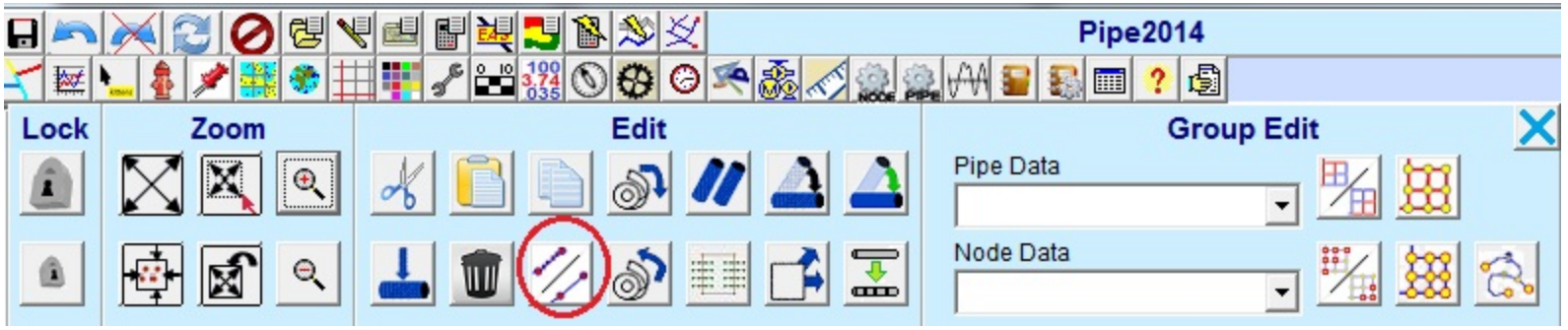
Deleting Intermediate Nodes

---

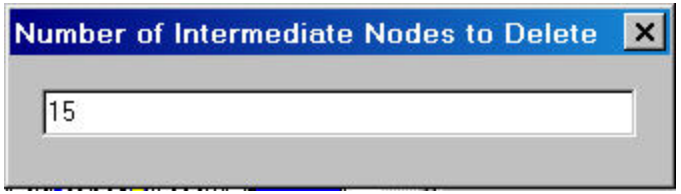
Pipe2018 has the ability to delete all or a portion of the intermediate nodes in a system. This may be particularly useful when a model is created from GIS or AutoCAD data which occasionally will import with a large number of intermediate nodes.

- In the main menu, select Edit, and Delete Intermediate Nodes.

In KYnetic:




- You may choose to delete all of intermediate nodes by clicking Yes.
- If you choose No, the following will appear:




- Enter the number of intermediate nodes to be deleted. Hit return.
- In the example 15 was entered. Pipe2018 will look at all pipe segments, choose the 15 smallest ones and delete an intermediate node from each.

  This process can be repeated until the user is satisfied with the appearance of the model.

## 1.3.3.10 Fittings

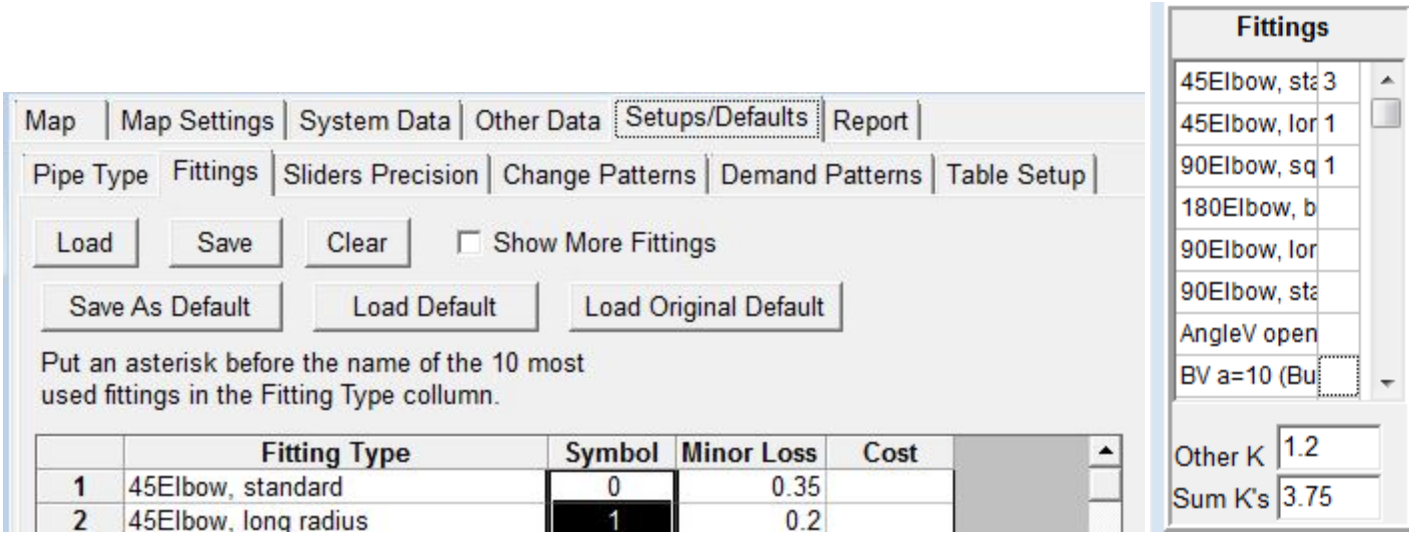
Fittings

See the Fittings video on the kypipe.com web site.

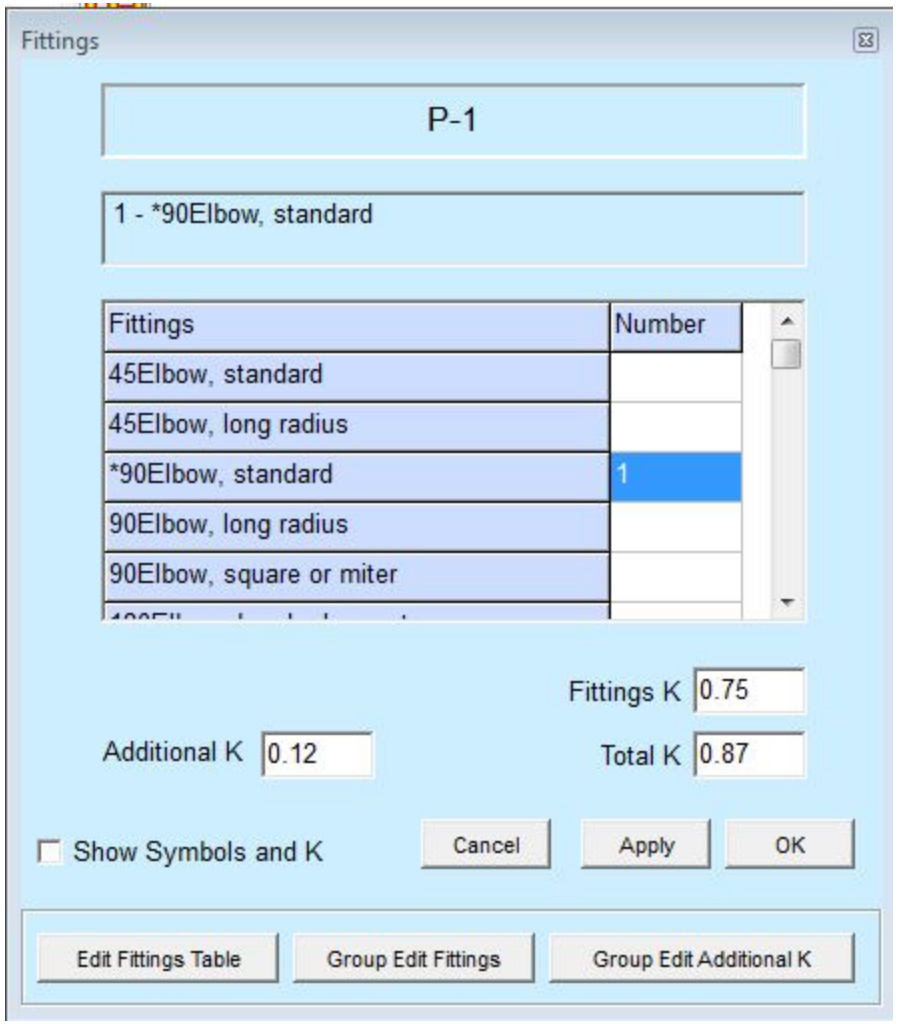
See also the Minor Loss Table for Fittings.

KYnetic Components menu icon:

To enable entering more than 75 fittings, check Show More Fittings box (Pipe2014 and later versions).




KYnetic Fittings menu (access by clicking on Fittings input field in Pipe Info window):




Fittings Data Table Fittings box

Pipe system models require as input data the sum of the loss coefficients (K's) for all the fittings associated with pipe links. The process of looking these up and summing them for each pipe can be quite time consuming. A Fittings box is provided where you can select appropriate fittings from a list of common fittings and the loss coefficients are automatically applied and tallied. A comprehensive list of fittings and associated loss coefficients is provided in the Fittings Table (Setup / Defaults - Fittings) and you can add to or modify this list. All the fittings in this table appear in the Fittings box. A table of Minor Loss Coefficients for fittings is provided for reference. See also Pipe Data Boxes.

The Fittings Data Table is a user prepared table of up to 75 fittings with the name and the minor loss coefficient entered. Symbols (numbers, letters, and characters) for 75 fittings are provided and users may enter a fitting at any location of the list to utilize the appropriate symbol.

Different tables can be prepared, saved, and loaded to be used for data preparation. One list will be designated as the default which will be used automatically for new files. A default Fittings Setup Table is provided for Pipe2018 users.

This button brings up a file selector box to load a previously saved list of fittings.

This button brings up a file selection window to save the current fittings to a file.

This button clears all the entries from the fittings table

This button causes the current fittings table to be saved as the default table which is used for new systems.

This button reloads the default fittings into the fittings table.

Note that if the original default file is lost and needs to be recovered a backup copy is included in the program installation as fittings.bak.

Table entries include:

Fitting Type

A description of the fitting (standard elbow, for example)

Symbol

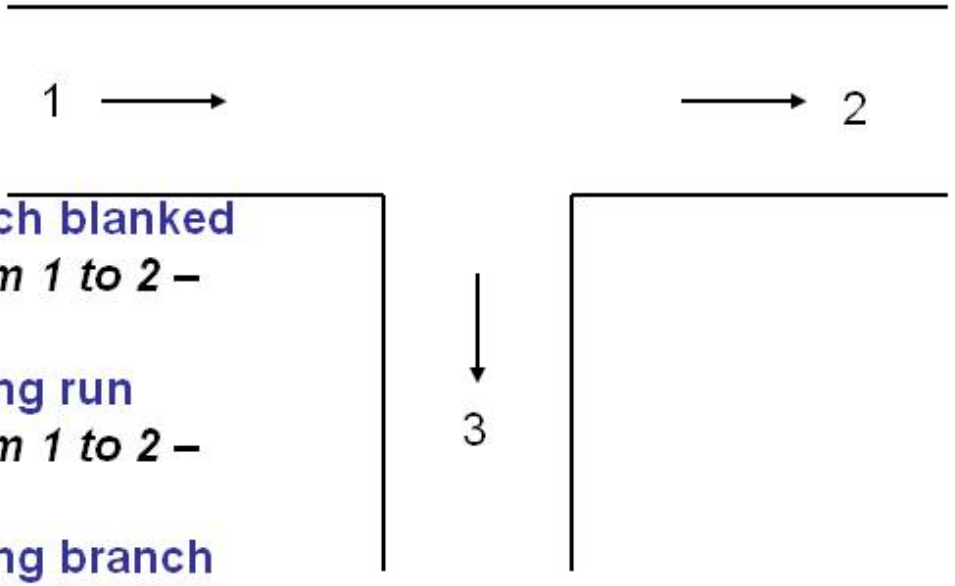
A non-editable single character (number, letter, etc.) associated with the fitting and used to label the fittings in the pipe link.

Minor Loss

The loss coefficient (K) for this fitting.

# T Losses

## (note the K for a T is applied as and Entrance Loss for pipes connecting the T)

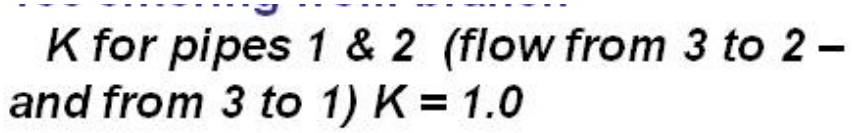



**Tee std, along run, branch blanked**
*K for pipe 2 (flow from 1 to 2 – no flow in 3) K = 0.4*

**Tee elbow usage, entering run**
*K for pipe 2 (flow from 1 to 2 – and from 1 to 3) K = 1.3)*

**Tee elbow usage, entering branch**
*K for pipe 3 (flow from 1 to 2 – and from 1 to 3) K = 1.5*

**Tee entering from branch**




## 1.3.3.11 Group Operations

Group Operations

See the Groups video on the KYPipe web site.

---

Pipe2018 provides some advanced capabilities to select sets (of pipes or nodes) and selectively label these sets, to edit the associated data or change the node type of the set. These operations are done in Group Mode. Some examples of operations you can do include:

1. Show all pipes with velocity above 8 ft/s.
2. Show all 6 inch pvc pipes, which were installed, between 1970 and 1980.
3. Show all nodes with a pressure between 30 and 40 psi.

See also: Sets and Groups

## 1.3.3.12 Groups, Subsets, Minimizing

Groups, Subsets, Minimizing

There are several related functions in Pipe2018.

Basic Grouping - This is using Group mode (Group button on the left side of the map screen) to put together user-defined groups for various purposes, such as building a constraint, calibration, user reference, creating subsets for analysis or limited output, etc. See Sets and Group Mode

Creating Subsets for Analysis - Using group mode, a subset may be created which can then be used for analysis, excluding the rest of the model. In contrast to Skeletonization (described below), a subset used for analysis does not incorporate demands, supply elements or other aspects of the excluded portion of the model for the analysis. See Subsets.

Limited Output - Also refered to as Selected Output. This group is created within group mode and is used to define which nodes and pipes are to be included in the output report. Useful for large systems. See Selected Output.

Limited Output for Surge -

Skeletonizing - This is a specialized Module (premium version only). It allows the user to minimize the size of a model by reducing the total number of pipes but at the same time maintaining the total system demand and supply nodes (if desired). This is especially useful for Surge analyses. Does not utilize group mode in creating the skeletonized system. See Skeletonize.

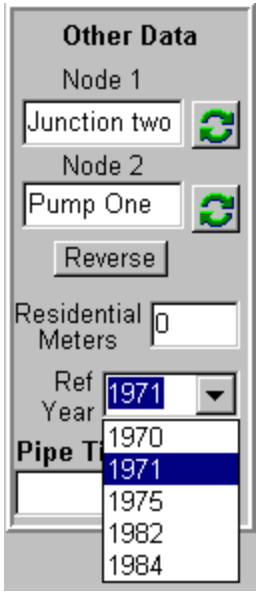
## 1.3.3.13 Input and Editing Shortcuts

Input and Editing Shortcuts

---

**dynamic drop down data list**

Most drop down data lists (which are available if a down arrow button is displayed on the right end of the data box) are dynamic. This means that when new values are entered they are added to the drop down list and can be selected for future entries. For example, if the data you need for the pipe Reference Year (installation year) does not appear in the drop down list, key it in and it will be added to the list the next time you access this data list.




series data entry

When you select a new pipe or node the active entry location remains unchanged. This allows you to enter a specific data parameter with a minimum of effort. For example, if you wish to enter node elevations you can select a node and enter the elevations. You can then select other nodes and directly key in the desired elevation values.

data sliders

Certain data entry boxes are accompanied by a slider below the box, which can be used to select a data value. The range and increments, which appear in each slider, may be customized by the user (Setup/Default - Units). The arrow keys will also move an activated slider to the next smallest (largest) value. Below is an example of a slider which is used to set the pipe length.

orthogonalize Pipe

Under Edit in the main menu, with the target pipe selected, this feature causes the pipe to orthogonalize to the nearest horizontal or vertical position.

**auto orthogonalize**
Under Edit in the main menu, all pipes created with a new node while this feature is on will be orthogonalized to the nearest horizontal and vertical position.

repeat pipe

**Select node and pipe. Starting at the selected node it will create a duplicate of the selected in the same orientation. The selected node will be Node 1 of the new pipe and the new node will be Node 2. Useful when laying out grid-type systems.**

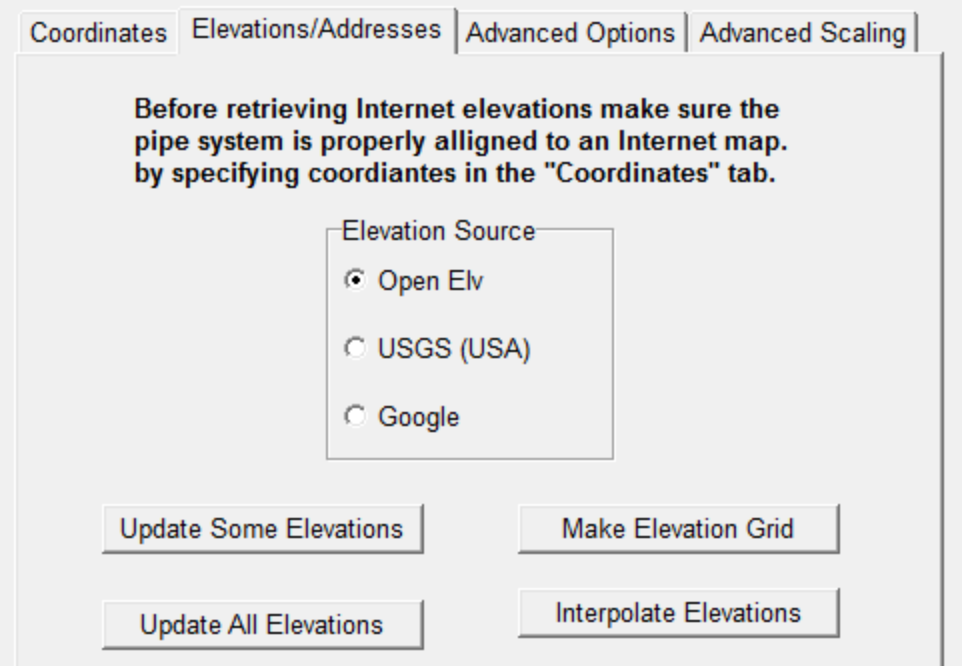
copy and paste pipes

**You may select a group of pipes and nodes and copy them with all assigned attributes to another system (another .p2k file) or within the same system. Using the GBox or individually selecting nodes and pipes in Group Mode, select the portion of the system which you want to copy. Under Edit in the Main menu, select copy. Close the file and open the file to which you want to paste (or click on the desired location in the existing file). Under Edit select paste. This will paste the nodes and pipes at the same cooridinates at which they were located in the original file. However, if you enter Layout mode and select a node and then paste, the paste will occur at that node. The pasted system will not be connected to or overwrite any part of the system to which it was pasted, even if two nodes are at the same coordinate location. After pasting the system you can click near the pasted pipes and nodes and drag them to the desired position. It may be necessary to de-select and re-select one of the nodes if the click and drag doesn't work intially.**

## 1.3.3.14 Interpolate Elevations

The Interpolate Elevations feature is found in the Internet Maps menu under Elevations/Addresses tab.

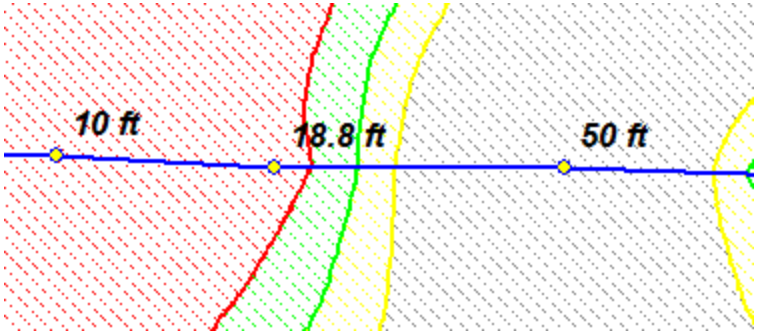
The feature uses existing elevation data to update nodes with missing elevation data.




It is NOT necessary to have an Internet map loaded in the model to use this feature.

If there are nodes with Elevation =0 (no data has been input) in a model, elevation values may be interpolated from nearby node elevations.

After a new elevation value is added, elevation contours will be applied. Note: If the model has contours currently applied, these will be replaced by elevation contours.




## 1.3.3.15 Logical Text Inputs

Logical Text Inputs

---

To simplify the use of two of the more advanced modeling features Logical Text Input data setups are provided. With these, you need only to fill in some information, which is integrated into a logical description of the setup and function. This approach will result in a clearer understanding and more effective use of these powerful capabilities. Examples of Logical Text Inputs are shown below. The information in italics is entered or selected from drop down lists by the user.

Control Switch Setup:

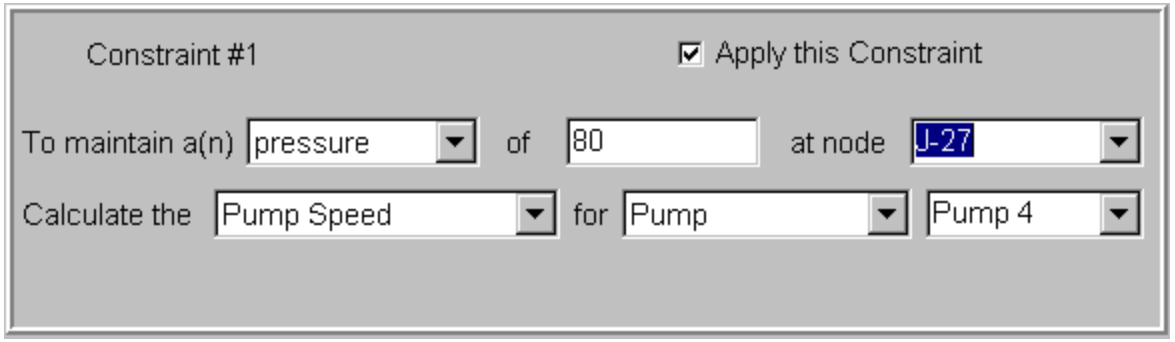
***Pump B*** is on when ***Tank B*** is below 200 and off when above 250

Switch Units: Head (level) (ft or m)

| controlled element | is | on/ off | when | sensing node | is | below/ between | low level | and | on/ off | when | above | high level |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| Pump B | is | on | when | Tank B | is | below | 200 | and | off | when | above | 250 |

Constraint (Parameter Calculations) Setup:

To maintain a pressure of ***80*** at node ***J-27*** calculate the ***pump speed*** for pump ***Pump 4***.




## 1.3.3.16 Materials and Rating

Materials and Rating

---

Materials refers the material of which the pipe is made. See also Material Cost.

Rating refers to the maximum pressure for which a pipe is rated.

Material, Rating and Roughness defaults may be set up in the Pipe Type data table under Setups/Defaults in order to have a selection of pipes to choose from as a piping system is laid out (to eliminate the step of typing in this data for every pipe). See Pipe Type (under Setups/Defaults)

See also

- Pipe Data Boxes
- Hazen-Williams Table
- Darcy-Weisbach Table

## 1.3.3.17 Meter Based Demands

Meter Based Demands

See the Meter video on the kypipe.com web site.

---

Rather than using the traditional time-consuming approach for manually distributing user demands at nodes, Pipe2018 includes the option to graphically represent meter connections that are associated with usage records. In this manner, the usage (demands) are automatically allocated to adjacent nodes for the model calculations relieving the user of this tedious operation. This also allows the model to be readily updated at any time using current meter readings. Of course additional demands may be specified at nodes in the traditional manner. This feature provides the two following options:

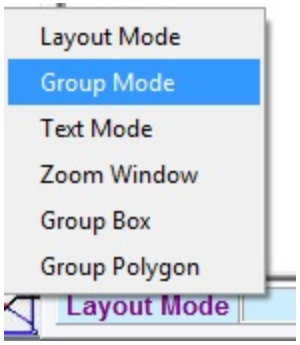
- All metered connections can be individually represented. Each can be associated with a unique graphical symbol at a desired location or one graphical symbol (metered connection node) may represent a collection of meters.

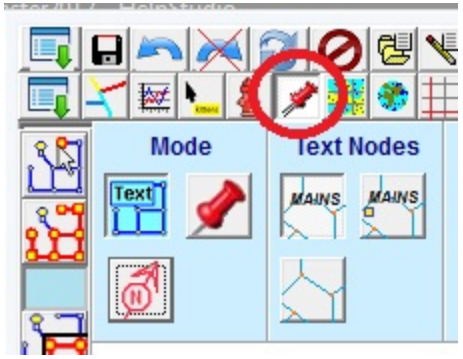
- The number of residential meter connections is input for each pipe link along with individual metered connections for larger users. An average user value for a residential meter connection is used to account for residential usage, while the additional input (System Data) and metered usage is tied to the non-residential meter records.

## 1.3.3.18 Operating Modes

**KYnetic:**

In KYnetic, click the bottom left corner of the program window to change modes.






Access to Layout, Group, Text and Fixed Modes are shown above.

Other Operating modes include Undo, Redo, Refresh and Clear buttons shown above. Read Classic entry below for more detail.

**Classic Interface:**

There are four Operating Modes for Pipe2018 and you can select the appropriate one by clicking on the desired button on the upper left edge of the display.

Hover over each button in the Pipe2018 Classic to see it's name.

**Layout** - In layout mode you can add nodes and pipes and move individual nodes. You should operate in this mode only when you wish to add pipes and or nodes.

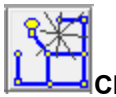
**Fixed** - In this mode you can not add or move nodes or add pipes. You can change node types and input and edit data. You should always operate in this mode when appropriate so you will not inadvertently modify your system.

**Fixed 2** - Nodes and pipes may be added but existing nodes may not be moved.

**Text** - This mode is used to add, modify or delete Text nodes.

**Group** - This mode is used for Group Operations such as set selection and editing. You can also move a group of nodes and delete groups of nodes and pipes. Within Group Mode the following commands may be used:

**G Box** - Within Group Mode, this allows the user to select all network elements within a box drawn with the cursor.

 **Clear** - Within Group Mode, click this to clear all selected elements. Also releases pan lock and other modes. See Pan Controls.

Other functions in this toolbar are as follows:

**Refresh** - This button is used to update the display. You may want to do this when changing zoom levels with contours on (to recalculate the contours) or to update labels.

**Table** - Enter the Data Tables.

Push Pin - add notes to the map screen. More.

North Arrow - add to map. The angle of rotation may be specified. Enter TEXT mode to edit, move, rotate, or delete.

Saved Views - set up saved zoom areas. Right click to name and save a view. Left click to select a saved view. See also Views.

Undo/Redo - can undo and redo up to 3 MAP CHANGES (not data changes). Note the Apply and Undo to Last Apply Command.

See also Panning Controls.

## 1.3.3.19 Pipe Diameter

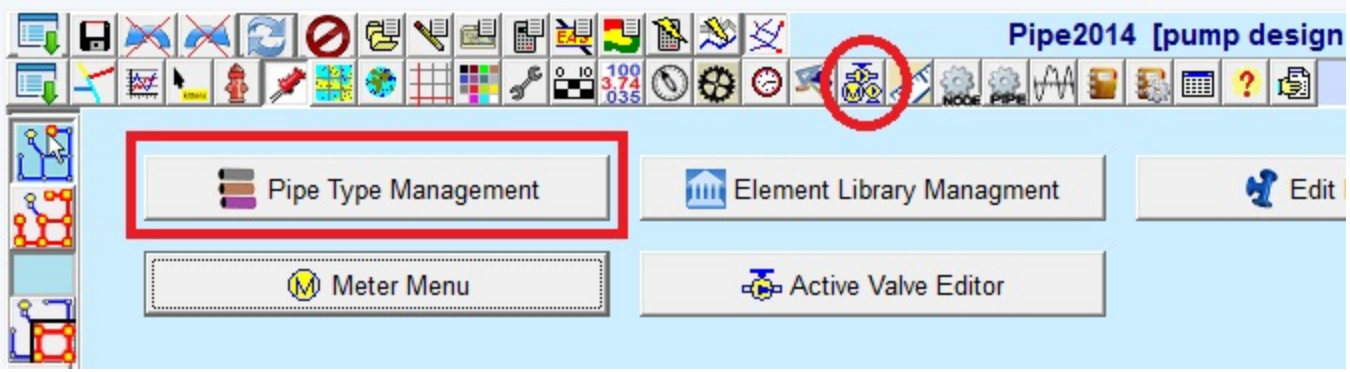
**Nominal Diameter** - this is the diameter entered into the Pipe Data Box in the Pipe Information window.

**Actual Diameter** - Used in analysis, the actual inside diameter of the pipe. If no actual diameter is specified then the analysis defaults to nominal diameter. This value is entered in the Pipe Type table. If an actual diameter is specified, this will appear in the Data Summary of the Report. The actual diameter may be displayed as a pipe label under (Classic - Map Settings | Labels) or in the Pipe Info table by using the Actual Diameter icon at the bottom of the Pipe Info window (KYnetic only).

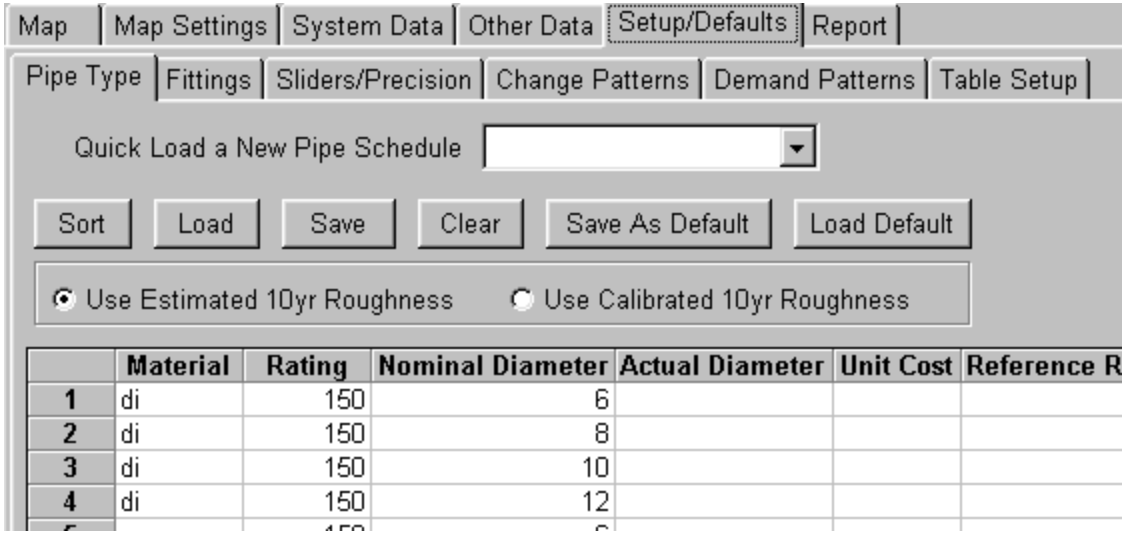
## 1.3.3.20 Pipe Type Data

Pipe Type Data

KYnetic:




Classic:



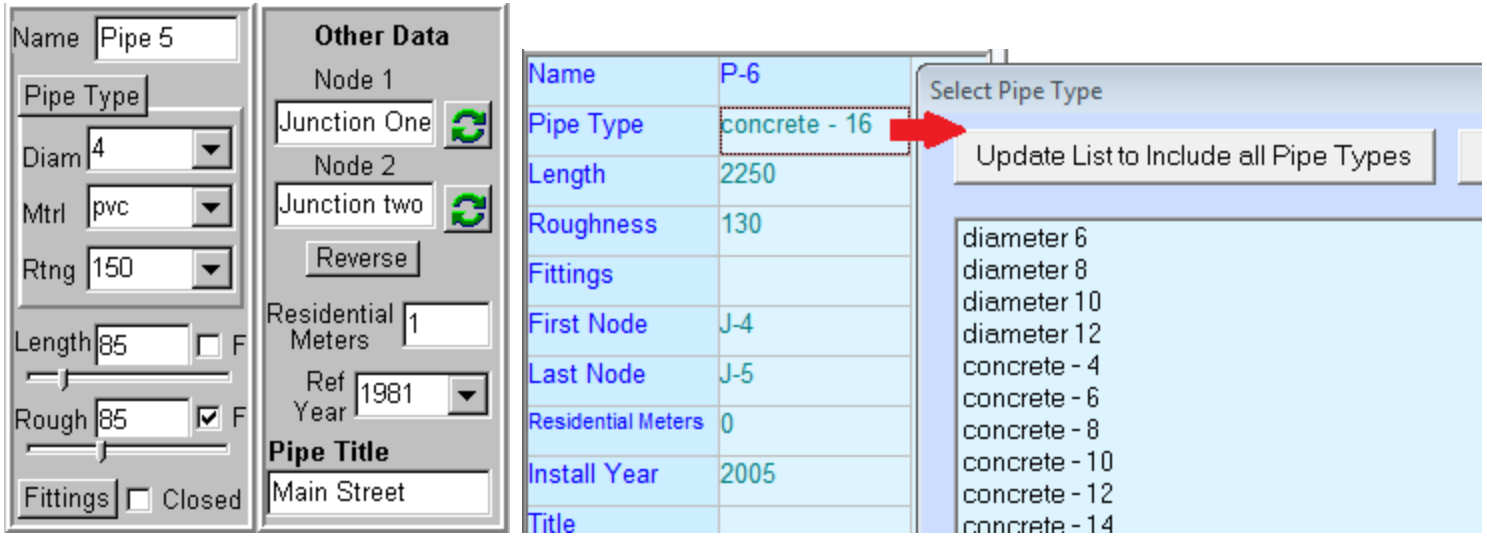

See the Pipe Type video on the KYPipe web site.

---

**The Pipe Type Table (shown above, under Setup/Defaults / Pipe Type) provides some very important capabilities which can save time for data entry. Once the different Pipe Types to be used in the system have been set (or the default used) in the Pipe Type Table, a single Pipe Type selection in the Pipe Information window will set the material, rating, and diameter.**

**Quick Load a New Pipe Schedule - Several default pipe schedules may be loaded. When a schedule is loaded for an existing system, the schedule pipes along with any pipes which are already entered into the system will appear in the table.**

**Note on Diameters: The analysis of a system considers the Actual (inside) Diameter entered in this table. If no Actual Diameter is specified, then the analysis defaults to the Nominal Diameter. The Nominal Diameter is the value read from the Pipe Information window.**
With the exception of Fittings Data the pipe characteristic for a selected pipe can be fully set in the Pipe Information window (below) by entering a Reference Year (usually installation year) and then clicking on Pipe Type and selecting from the list which appears.




This sets the pipe material, rating, diameter, and roughness and the length is scaled. The roughness is calculated based on the age of the pipe. To effectively utilize this feature the Pipe Type Table should include all the selections (material, rating, and diameter) in your system. Therefore, you should first edit the current Pipe Type Table (above) or load in a previous one so that your selections are available. The roughness calculations are based on values in the table for new pipe and either an estimate of the value for a 10 year old pipe or a calculated 10 year value based on calibration. Age based roughness will be assigned to each pipe if the required data (Reference roughness and 10 year roughness) is entered into the table and the Reference Year is entered for the pipe (Pipe Information Window). A radio button is provided to select whether the 10 year roughness will be based on estimated values (no calibration) or values computed through calibration.

The Pipe Type Table contains the following items:

Material

The pipe wall material (such as pvc, ductile, etc.)

Rating

The pressure rating of the pipe.

Nominal Diameter

The rounded off inside diameter of the pipe (6" for example).

Actual Diameter

The actual inside diameter of the pipe (6.078" for example).

Unit Cost

The cost per unit length of the pipe.

Reference Roughness

The initial roughness (normally for new pipe) used for age based roughness calculations (applies at age = 0).

Estimated 10-Year Roughness

The estimated pipe roughness at age 10 years for the age based roughness calculations. See Age-Based Roughness - Estimating the 10-yr roughness and Tools for calcuting the 10 year roughess.

Calibrated 10-Year Roughness

The pipe roughness at age 10 years based on calibration data.

Calibration Group

An integer which is used to identify a group of pipes to be used for calibration or other grouping applications.

Wall Reaction Rate

The rate at which a constituent decreases due to a chemical reaction with material along the pipe wall. See Quality.

Bulk Reaction Rate

Rate at which a constituent within the bulk flow decreases. See Quality.

## 1.3.3.21 Sets and Group Mode

Sets and Group Mode

Selecting a set

Editing a set

Using sets for presentations

See the Groups video on the KYPipe web site.

---

Pipe2018 provides some advanced capabilities for defining and utilizing pipe and node groups. These features will allow you to do some complex operations very quickly.

Examples of these operations could include:

Select all pipes with a diameter of less than 6 inches and flowrates below 100 gpm.

Add 20 feet to the elevations of all nodes.

What is Group Mode?

Group mode is a mode for set selection, set editing, and set results (tables and graphs) presentations.

What is a Set?

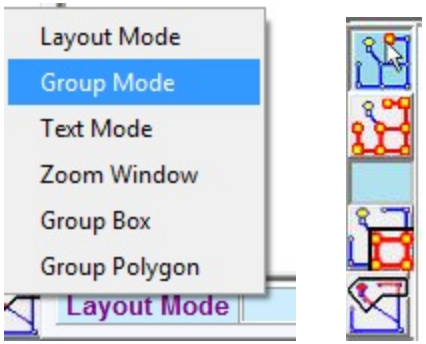
A set is a collection of nodes or pipes specified in Group Mode.

How do you select a Set?

**Set selection (from Map)**

First enter Group Mode. There are a few ways to do this:

1. (KYnetic only) Select Group option by clicking bottom left corner of program screen
2. Click the Group button then proceed to click on individual nodes or pipe. Click again to deselect.
3. Click Group Box and draw a box around pipes and nodes to be selected.
4. (KYnetic only) Click Group Polygon and left click to draw polygon points, right click to finish polygon.






**Deselect (from Map)**

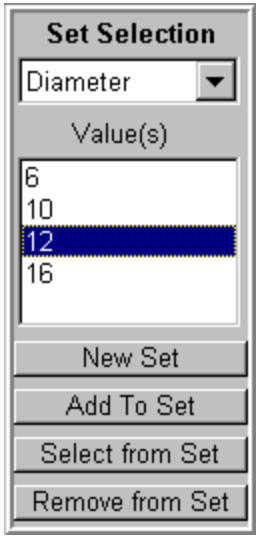
Click Clear button to deselect all, or click individual pipe/node with mouse.

KYnetic: Classic:

**Set selection (based on pipe/node data)**

Classic:

In group mode this Information Window (Set Selection) allows pipes or nodes to be selected or deselected by parameter or attribute.




For example: all 12 inch PVC pipe with a rating of 150psi may be selected.

1) Select Diameter (from drop down parameter list) , select 12" from the listed values, and click on ***New Set***

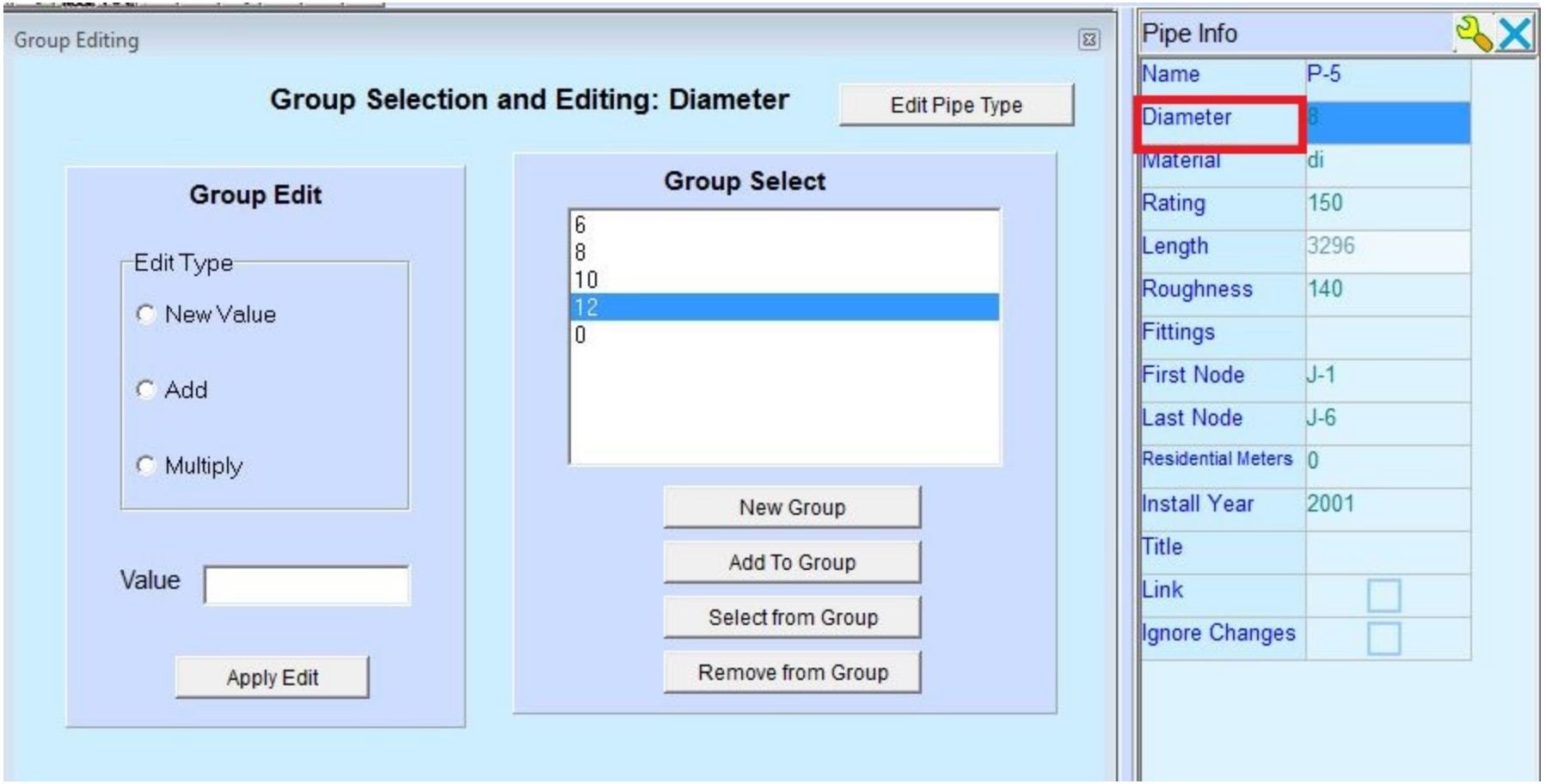
2) next select Material, choose PVC, and click on ***Select from Set***

3) select Rating, choose 150, then click on ***Select from Set***.

KYnetic:

Click the row heading of any parameter in the node or pipe window.

Note, to edit diameter (not Pipe Type) as shown, click the "Display Pipe Type..." icon at the bottom of the Pipe info window, shown below:



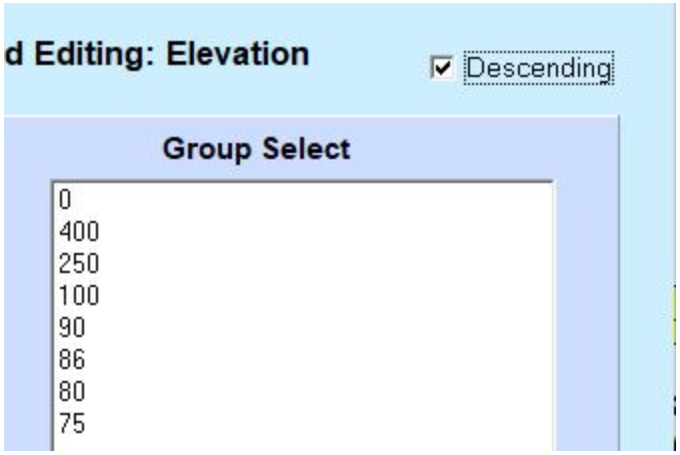

For example: all 12 inch PVC pipe with a rating of 150psi may be selected.

1) Click the diameter row heading. In the box that appears, select "12" and click New Group button

2) Next click Material row heading, choose PVC, and click on ***Select from Group***

3) Click on Rating, choose 150, then click on ***Select from Set***.

**Added in Pipe2018: Ability to sort the values in Descending order as shown:**




The following limitations apply to Classic Interface only. All parameters may be Group Mode selected in KYnetic interface.

For Nodes, here is a list of all the possible attributes that can be used to select nodes:

| | | | |
|---|---|---|---|
| Result A | Tank Volume | Reg Setting | Limited Output |
| Elevation | Tank Diameter | Constraint Gro | Previous Resul |
| Junc Demand | Tank Inflow | Initial Conc | Static Pressure |
| Junc Dm Type | Sprink Const | !last maintaine | Fireflow |
| Res Grade | Pump Power | Manufacturer | User Notes |
| Tank Init Level | Pump ID | Model | Node Type |
| Tank Min | Pump Speed | !Install Date | Node Name |
| Tank Max | Pmp Efficiency | Address | Static Head |

Note: Static Head select is based on the static head reference elevation defined in System Data | Preferences. For pressure results, use Result A.

For Pipes, here is a list of all the possible attributes that can be used to select nodes:

| | |
|---|---|
| Result A | Fittings |
| Length | Rating |
| Diameter | wave speed |
| Roughness | Calibration Gro |
| MinorLoss | Bulk Rate |
| Material | Wall Rate |
| Install Year | Limited Output |
| # of Meters | Previous Resul |

How do you edit a set?

Classic:

In Group Mode this Edit Pipe (or Node) Set window allows you to group edit the selected node or pipe set.

Three types of changes are possible:

1) Turn selected set on (open) or off. (closed)

2) Change selected set of nodes to another node type (if the requirements for the change are fulfilled). Select operation from item to edit drop down list.

3) Change selected node or pipe values using one of the following operations:

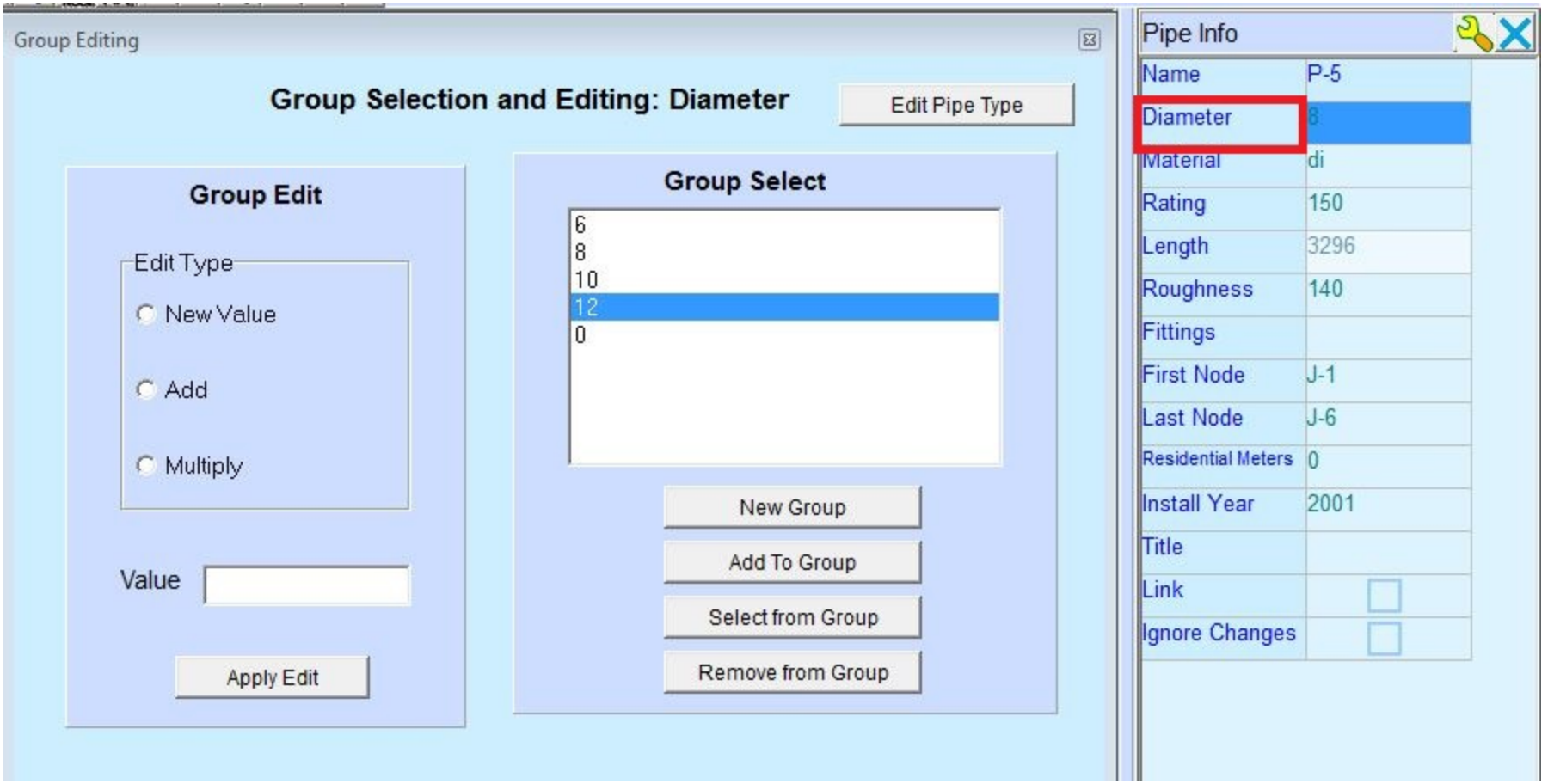
   a) New value

   b) Add / Subtract a designated value

   c) Multiply / Divide a designated value

   d) Specify an Exclusive Value (this will assign the chosen value to the selected set and change any element outside of that set which shares the same value to 0).

KYnetic Interface:




The available editing options will appear. The options will depend on whether pipes or nodes are selected and which node type.

Three types of changes are possible:

1) Turn selected set on (open) or off (closed)
2) Change selected set of nodes to another node type (if the requirements for the change are fulfilled). Select operation from item to edit drop down list.
3) Change selected node or pipe values using one of the following operations:
   a) New value
   b) Add a designated value

c) Multiply a designated value

The following limitation apply to Classic Interface only. All parameters may be Group Mode edited in KYnetic interface.

For Nodes, here is a list of all the possible attributes that can be edited and nodes types that may be changed to:

| | | | | | | |
|---|---|---|---|---|---|---|
| Elevation | Tank Diameter | Constraint Grc | Previous Resu | to reservoir | to meter con | |
| Junc Demand | Tank Inflow | Initial Conc | Static Pressur | to pump | to loss elemer | |
| Junc Dm Type | Sprink Const | !last maintaine | Fireflow | to check valve | to pres supply | to vacuum bre |
| Res Grade | Pump Power | Manufacturer | User Notes | to hydrant | to intermediate | to LPS tank |
| Tank Init Level | Pump ID | Model | junc to intr | to valve | to device 1 | to 1/2 stg air v |
| Tank Min | Pump Speed | !Install Date | intr to junc | to sprinkler | to device 2 | add chk valves |
| Tank Max | Pmp Efficienc | Address | to junction | to regulator | to blowoff/rack | remove chk vls |
| Tank Volume | Reg Setting | Limited Outpu | to tank | to inline meter | to active valve | Name/Prefix |

For Pipes, here is a list of all the possible attributes that can be edited:

| | | |
|---|---|---|
| Length | Rating | |
| Diameter | wave speed | |
| Roughness | Calibration Gr | |
| Other K | Bulk Rate | |
| Material | Wall Rate | |
| Install Year | Limited Outpu | |
| # of Meters | Previous Resu | |
| Fittings | Name/Prefix | Pipe Type |

How to use sets for selected presentations

**Selected Labels Only (Map Setting / Labels Tab)**

When this check box is activated only labels for the selected set will be displayed (Group Mode)

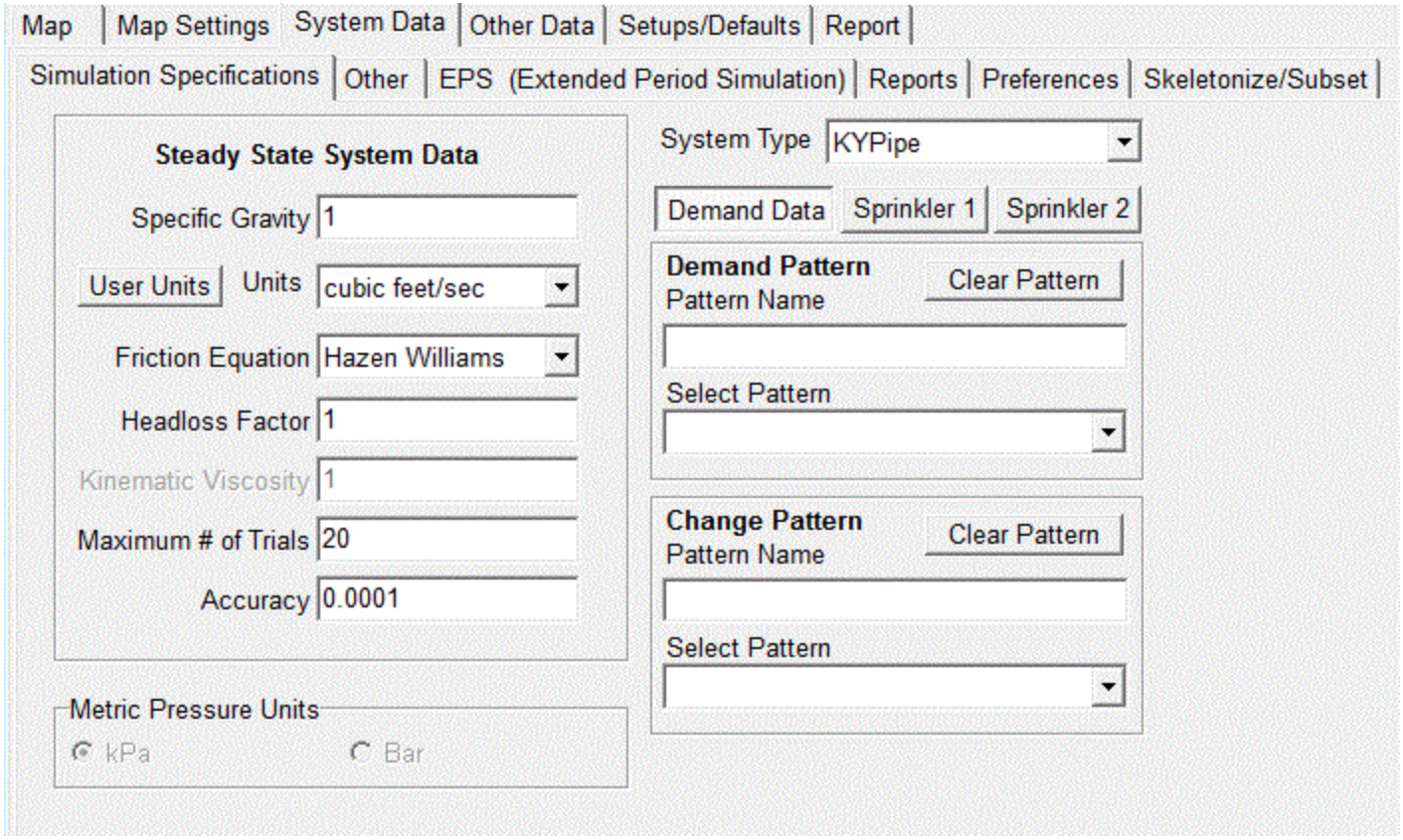
**Selected Results**

In Group Mode the Node (Pipe) Graph and Results Table will include the specified node or pipe set (up to 10 plots for a graph and 50 columns for the results table)

## 1.3.3.22 Simulation Specs

Simulation Specs

In KYnetic use System Data icon:




---

This window controls some of the primary information about the analysis to be performed.

Specific Gravity

Unless otherwise defined, water (specific gravity = 1) is assumed to be the liquid being transported. Other liquids are considered by inserting a non zero entry. This number is the specific gravity of the liquid being considered (ratio of liquid density to water density). Note the use of liquids other than water requires utilizing an appropriate head loss equation (usually Darcy Weisbach).

| Specific Gravities of various liquids | | | |
|---|---|---|---|
| **Liquid** | **S.G.** | **Liquid** | **S.G.** |
| Acetic Acid | 1.050 | Glycerine | 1.260 |
| Acetone | 0.790 | Glycol | 1.125 |
| Acetaldehyde | 0.782 | Isopropyl Alcohol | 0.780 |
| Alcohol, Ethyl | 0.790 | Kersosene | 0.820 |
| Alcohol, Methyl | 0.800 | Mercury | 13.620 |
| Aniline | 1.020 | Nitric Acid | 1.502 |
| Benzol | 0.878 | Oil, Crude | .81 to .97 |
| Carbolic Acid | 1.081 | Oil, Vegetable | 0.925 |
| Ether | 0.741 | Sulfuric Acid | 1.831 |
| Formic Acid | 1.229 | Turpentine | 0.870 |
| Gasoline | 0.750 | Water | 1.000 |

Units

See Units/User Units. Selects English or Metric units for the flowrate and associated input data and results output, or allows the user to create units. See also the individual system types:

- Surge
- Fire (GoFlow) or Units GoFlow
- Gas or Units Gas and Steam
- Steam or Units Gas and Steam
- Stormwater

Friction Equations

Hazen-Williams - For water and wastewater only. Note: some limitation exists in the Hazen Williams method of calculation for pipes < 3 inches and velocity > 3 m/s (10 ft/s).

Darcy Weisbach - For water and other liquids

Manning - When working with storm sewer models, generally storm sewer systems flow in open channel conditions (which is not supported in KYPipe or Surge models) but during peak flow conditions they may flow as surcharged (full flow) conditions. When this occurs, KYPIPE may be used to model storm sewer network models and the Manning equation may be used.

Modified Hazen Williams - This equation is commonly used in India and is recommended by the government agency (Central Public Health and Environmental Engineering Organization - CPHEEO).

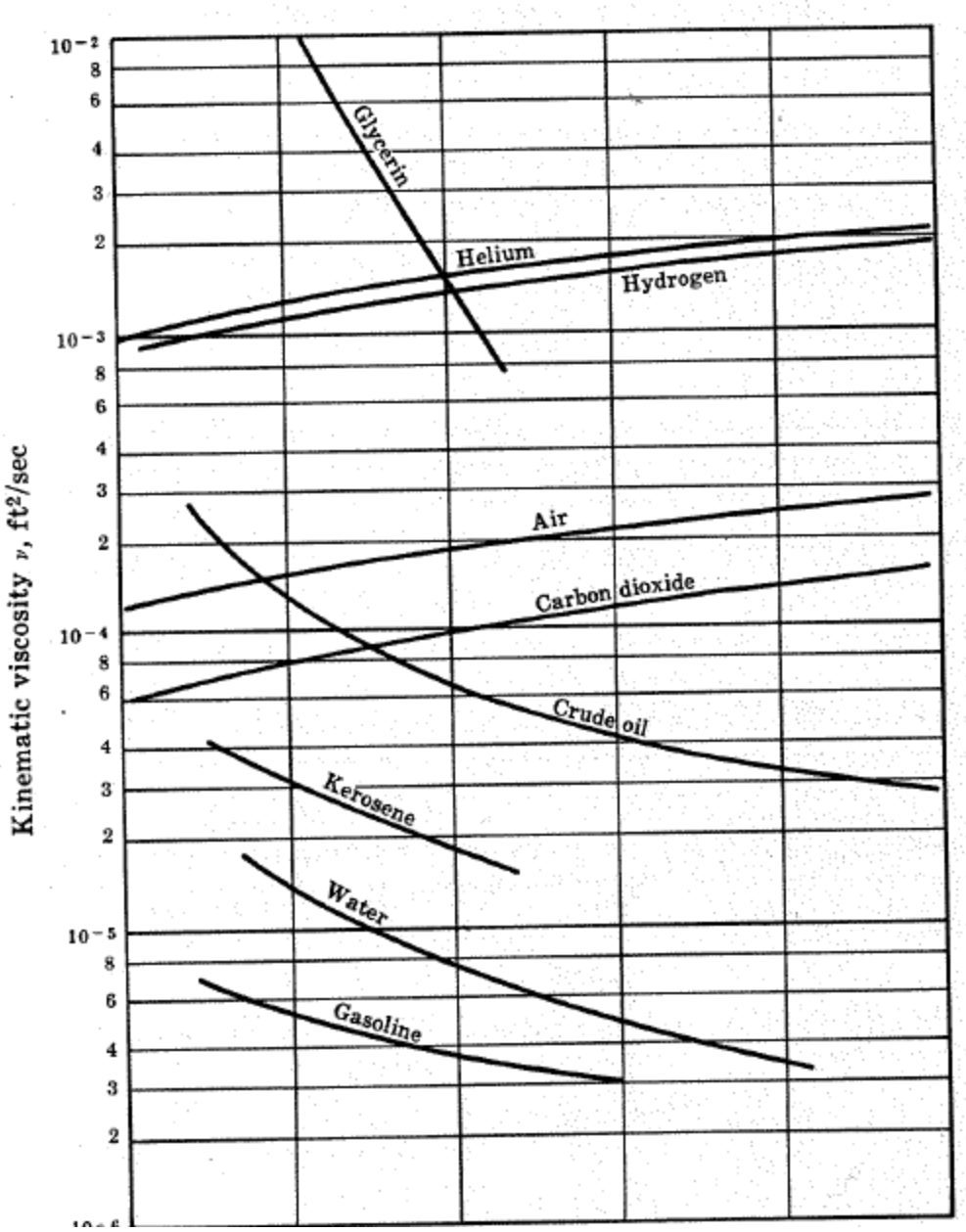
*Note Roughness values will need to be input according to friction equation used. For suggested roughness values see Hazen Williams and Darcy Weisbach

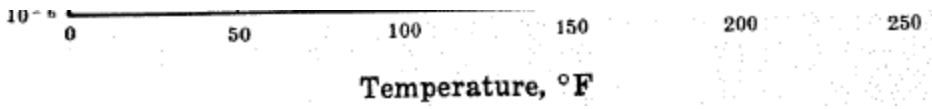
Headloss Factor

This factor will be applied to the friction loss equation and will increase/decrease computed frictional headloss in a pipeline by Headloss factor. Many modelers use this method for handling minor losses (equivalent pipe length method) when doing hand or spreadsheet calculations and this makes it available in the Pipe program.

Kinematic Viscosity

When the Darcy-Weisbach equation is specified for head loss calculations, the kinematic viscosity, which is needed to employ this relationship, is input in this field. For other liquids (and for water, if desired) the Darcy-Weisbach equation must be used and this option requires inputting the value for the kinematic viscosity (in ft*ft/s or m*m/s). If this option is used, the pipe roughness must be input for use with the Darcy-Weisbach equation in the units of millifeet or mm. If kinematic viscosity is given in Centistokes (cSt), one centistoke = 10-6 m2/s = 1.076 x 10-5 ft2/s


Kinematic viscosity ν, ft²/sec
10⁻²
8
6
4
3
2
10⁻³
8
6
4
3
2
10⁻⁴
8
6
4
3
2
10⁻⁵
8
6
4
3
2
Glycerin
Helium
Hydrogen
Air
Carbon dioxide
Crude oil
Kerosene
Water
Gasoline




Maximum # of Trials

This limit is set at 20 unless a different limit is specified here. It is unlikely that this limit will ever be reached, but it is imposed to guard against an unforeseen convergence problem (this conceivably could be caused by poor data or a check valve or a pump operating extremely close to its boundary condition). Also attempting to analyze a non-feasible situation involving parameter calculations may result in convergence not occurring. This option will also allow a smaller number of trials to be run if desired. .

Accuracy

This parameter determines when the solution is accepted. It is defined as the total (absolute) change in flowrate in the pipes from the previous trial divided by the total (absolute) flowrate. If this field is left blank the default value of 0.0001 is used which normally provides an extremely accurate result.

System Type

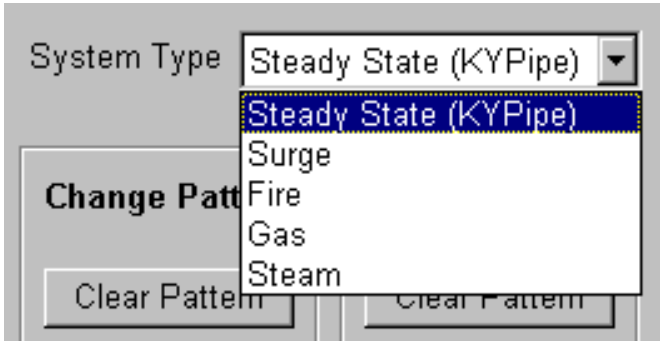
See Surge

Fire (GoFlow)

Gas

Steam

Stormwater




The user may choose the type of system being analyzed. Based on the chosen option, the calculated parameters and hence the output will reflect the system type. For instance, for Gas and Steam, no hydraulic grade or head is calculated, but density and pressure are.

**Change Pattern and Demand Pattern**

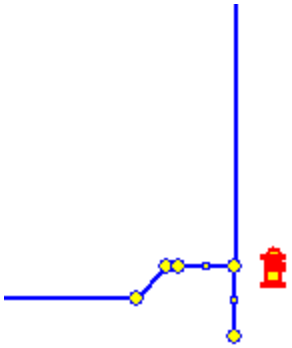
**Use the Select Pattern drop down box to choose from a selection of patterns created and saved by the user. Once selected, the active pattern file name will be displayed in the Pattern Name box. To clear the pattern, use the Clear Pattern button or select a new pattern. For information on creating and using patterns, see Change Patterns or Demand Patterns under Setup/Defaults.**

## 1.3.3.23 Snap Hydrants Valves and Meters

# Snap Hydrants Valves and Meters

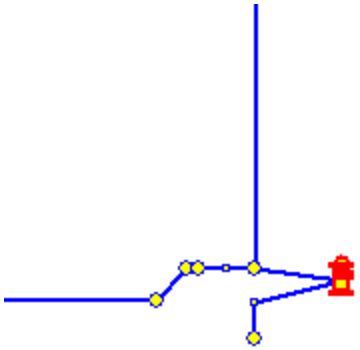
Available in both KYnetic and Classic interfaces.

It allows disconnected hydrants, valves and meters to be automatically inserted into the nearest pipe.



Go to Facilities Management – Snap Hydrants, Valves, Meters.

A tolerance setting appears with a default value.



If some nodes are still disconnected, the feature may be launched again with a different tolerance value.

Edit – Undo will undo the entire process – to three changes or steps.

**Use with GIS Import**

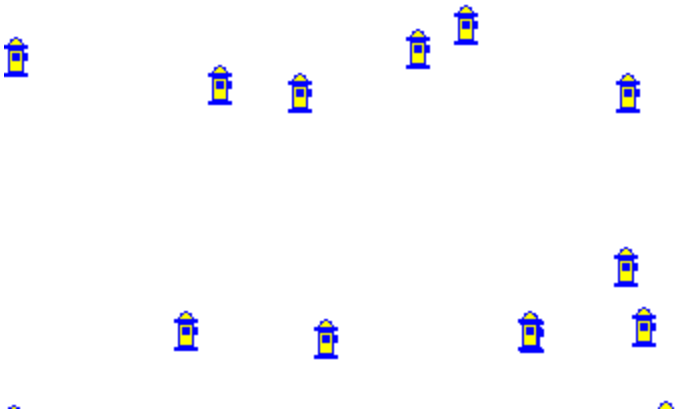
A context for this specialized feature is found in the GIS import process. To simplify the import of GIS data using the ArcGIS Import utility, especially with large systems, a possible approach is to begin by importing only pipes and foregoing any other node files such as hydrants.

Columns of data for hydrants (for example) are copied from a spreadsheet type format into the Data Table columns. Name, X and Y coordinates and ~ Type are the minimum required data.

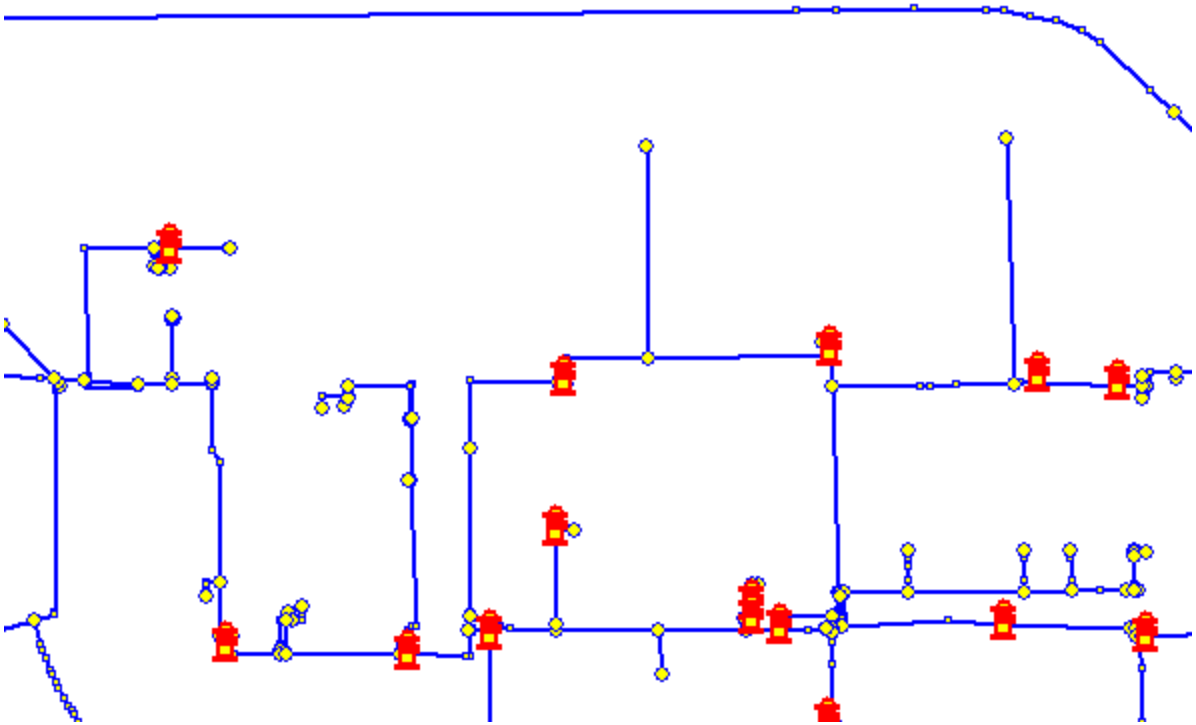
~ Type values: Hydrant = 6, Valve = 7, Meter = 10

| | Name | ~X | ~Y | ~Type | ~R |
|---|---|---|---|---|---|
| 7 | FHY006 | 495447.6134 | 1283254.309 | 6 | |
| 8 | FHY007 | 495770.9857 | 1283212.288 | 6 | |
| 9 | FHY008 | 495896.0664 | 1283199.862 | 6 | |
| 10 | FHY009 | 496159.3369 | 1283307.338 | 6 | |
| 11 | FHY010 | 487319.9184 | 1275047.195 | 6 | |
| 12 | FHY011 | 495369.3535 | 1282821.785 | 6 | |
| 13 | FHY012 | 495717.0458 | 1282822.823 | 6 | |
| 14 | FHY013 | 495937.1569 | 1282810.101 | 6 | |
| 15 | FHY014 | 496263.9854 | 1282821.176 | 6 | |
| 16 | FHY015 | 495443.7442 | 1282674.79 | 6 | |
| 17 | FHY016 | 495465.4395 | 1282436.118 | 6 | |
| 18 | FHY017 | 495709.6002 | 1282427.492 | 6 | |
| 19 | FHY018 | 495939.9163 | 1282447.928 | 6 | |
| 20 | FHY019 | 496243.7448 | 1282439.716 | 6 | |
| 21 | FHY020 | 495168.5988 | 1282443.56 | 6 | |

This results in a model file with “floating” hydrants and no pipes:



Save the model. Group select all the hydrants, go to Edit – Copy. Close the model, the open the file with the imported piping. Go to Edit – Paste. The model now looks like this, but the hydrants are not connected:



Now the system is ready for the Snap Hydrant, Valves, Meters process.

## 1.3.3.24 User Data

**What is User Data?**
**Defining User Data and Pipe Type List**
**Inputting User Data.**
Adding User Data Items.
Using User Data to Maintain Records.
Labeling User Attributes.

Group Selections by User Attributes.

---

**What is User Data?**

User Data provides the means to set up and access customized data records for nodes and pipes. The User Data is secondary data which is normally not required to carry out the hydraulic calculations. Once defined, User Data can be used to label maps, select groups, color emphasize, and for other Pipe2018 functions. There are several important default User Data items which may be required for certain analyses.

Bulk Rate - water quality pipe parameter required for water quality analysis.

Wall Rate - same as above

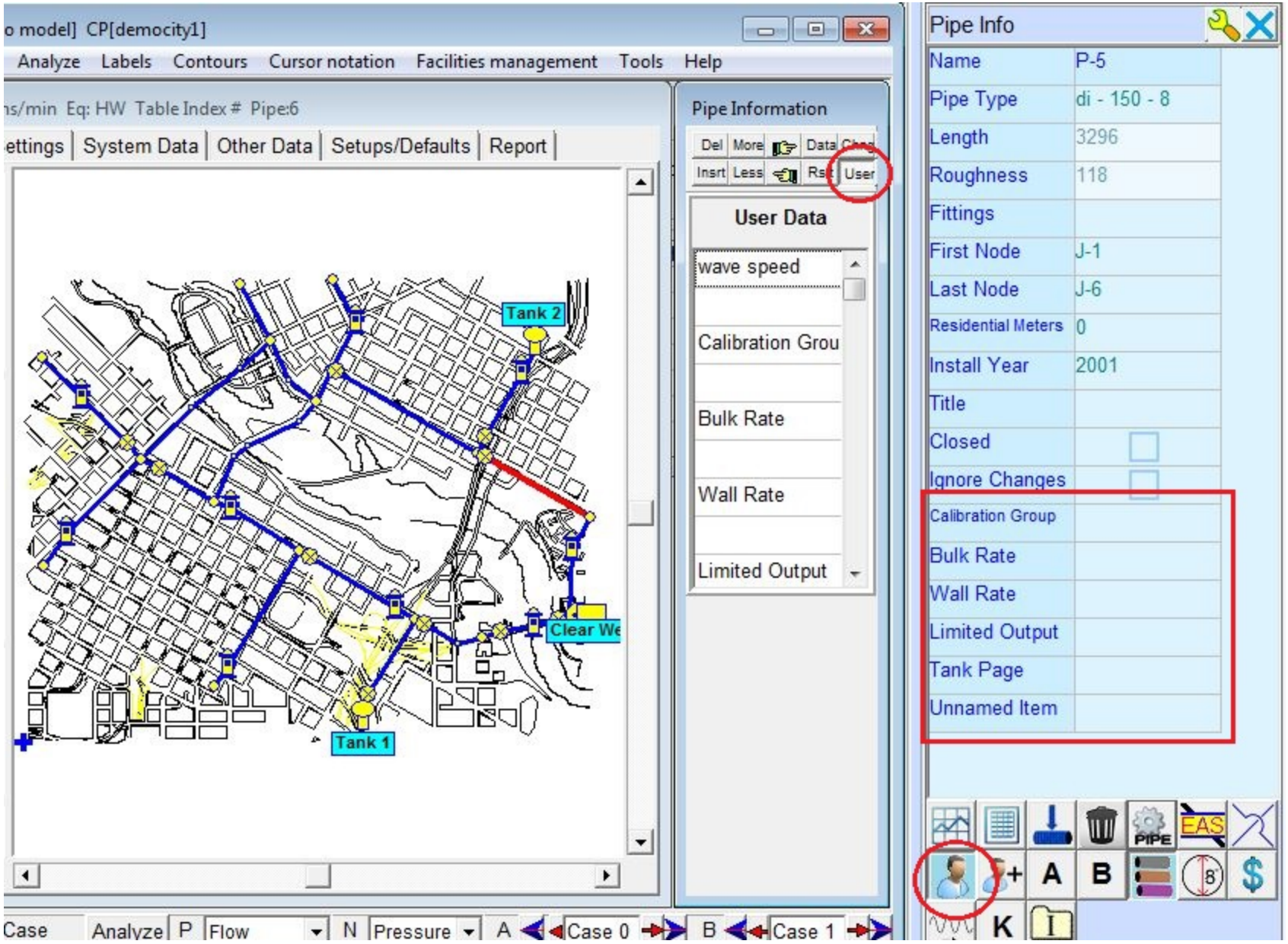
Limited Output - a code used by the various modules to designate pipes and nodes for output tables and files. Usually the code is set to 1 to designate output. See Selected Output.

Wave Speed - the sonic wave speed for a pipe - required for Surge transient analysis (see Surge - Pipe Data)

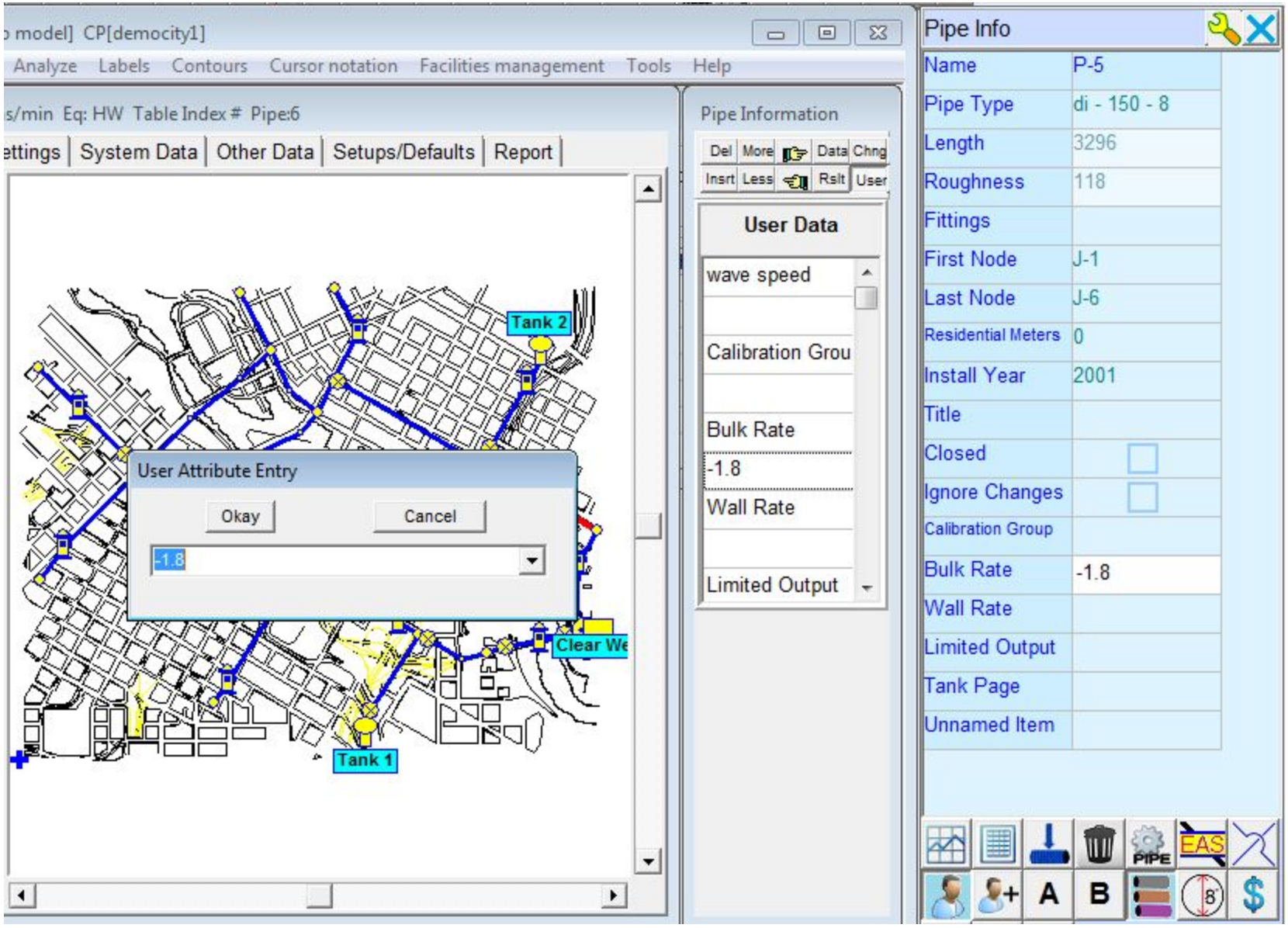
Calibration Group - a designation for a group of pipes (0-9) which is used for the Optimized Calibration to designate pipe groups for roughness adjustments (see Calibration Data).

**Defining User Data and Pipe Type List.** With the exception of Limited Output, each of the above parameters can be defined in the Pipe Type Table. If this is done, considerable time and effort can be saved if the data is required for an analysis. (For example, defining the Wave Speed in the Pipe Type data will cause the values to be entered automatically when the Pipe Type is selected.)

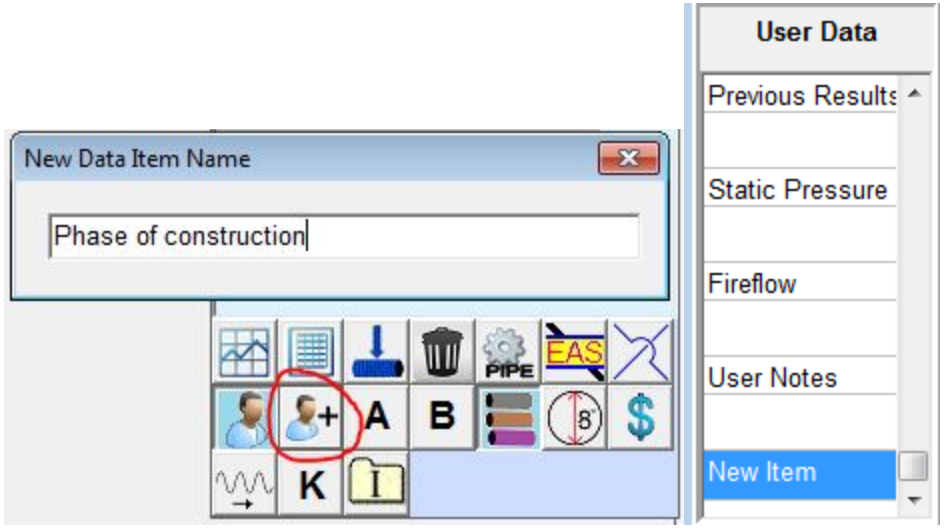
**Inputting User Data.** The User Data is accessed by clicking on the User button in the Node (or Pipe) Information window as shown below.

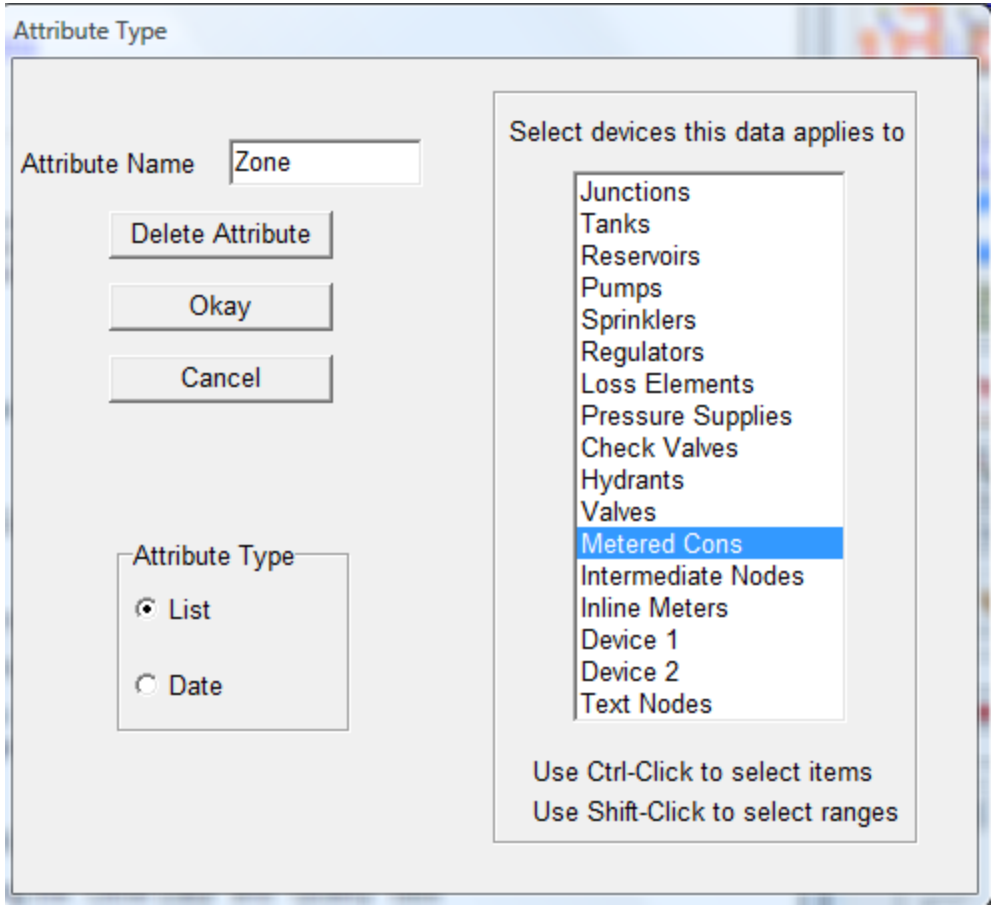

o model] CP[democity1]
Analyze Labels Contours Cursor notation Facilities management Tools Help
ns/min Eq: HW Table Index # Pipe:6
ettings System Data Other Data Setups/Defaults Report
Tank 2
Tank 1
Clear We
Pipe Information
Del More Data
Insrt Less Rst User
User Data
wave speed
Calibration Grou
Bulk Rate
Wall Rate
Limited Output
Case Analyze P Flow N Pressure A Case 0 B Case 1
Pipe Info
Name P-5
Pipe Type di - 150 - 8
Length 3296
Roughness 118
Fittings
First Node J-1
Last Node J-6
Residential Meters 0
Install Year 2001
Title
Closed
Ignore Changes
Calibration Group
Bulk Rate
Wall Rate
Limited Output
Tank Page
Unnamed Item
PIPE
EAS
A
B
K

A value for the User Data item is entered by clicking on the data field next to or below the data item. If the data has been previously defined, the value will appear in this box.

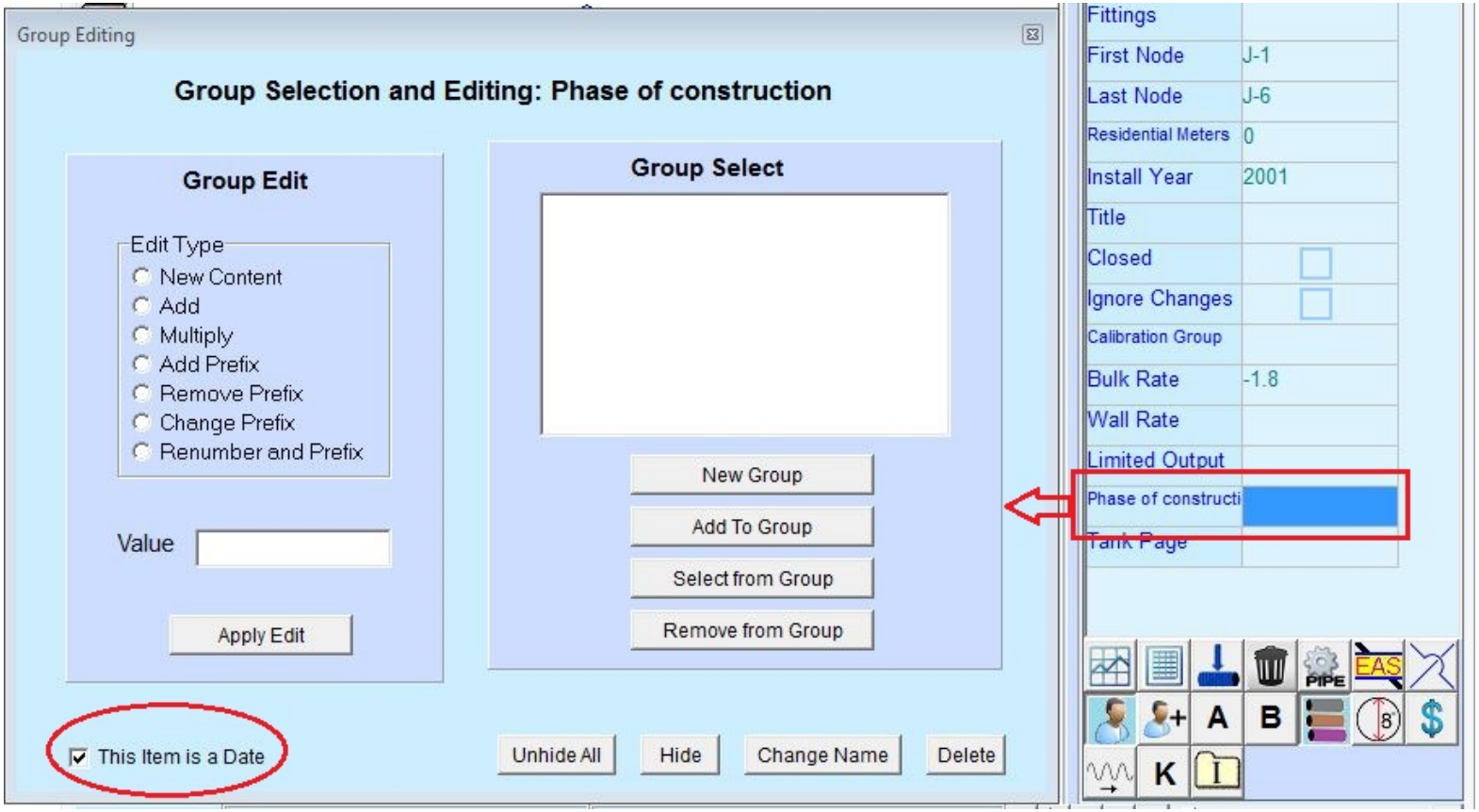

model] CP[democity1]
Analyze Labels Contours Cursor notation Facilities management Tools Help
s/min Eq: HW Table Index # Pipe:6
ettings System Data Other Data Setups/Defaults Report
Tank 2
Tank 1
Clear We
User Attribute Entry
Okay
Cancel
-1.8
Pipe Information
Del More Data Chng
Insrt Less Rslt User
User Data
wave speed
Calibration Grou
Bulk Rate
-1.8
Wall Rate
Limited Output
Pipe Info
Name P-5
Pipe Type di - 150 - 8
Length 3296
Roughness 118
Fittings
First Node J-1
Last Node J-6
Residential Meters 0
Install Year 2001
Title
Closed
Ignore Changes
Calibration Group
Bulk Rate -1.8
Wall Rate
Limited Output
Tank Page
Unnamed Item
PIPE
EAS
A
B

**Adding New User Data Items.** You can add any desired User Data attribute by clicking directly on New Item (not the data value box below) which brings up the Attribute Type box shown below.

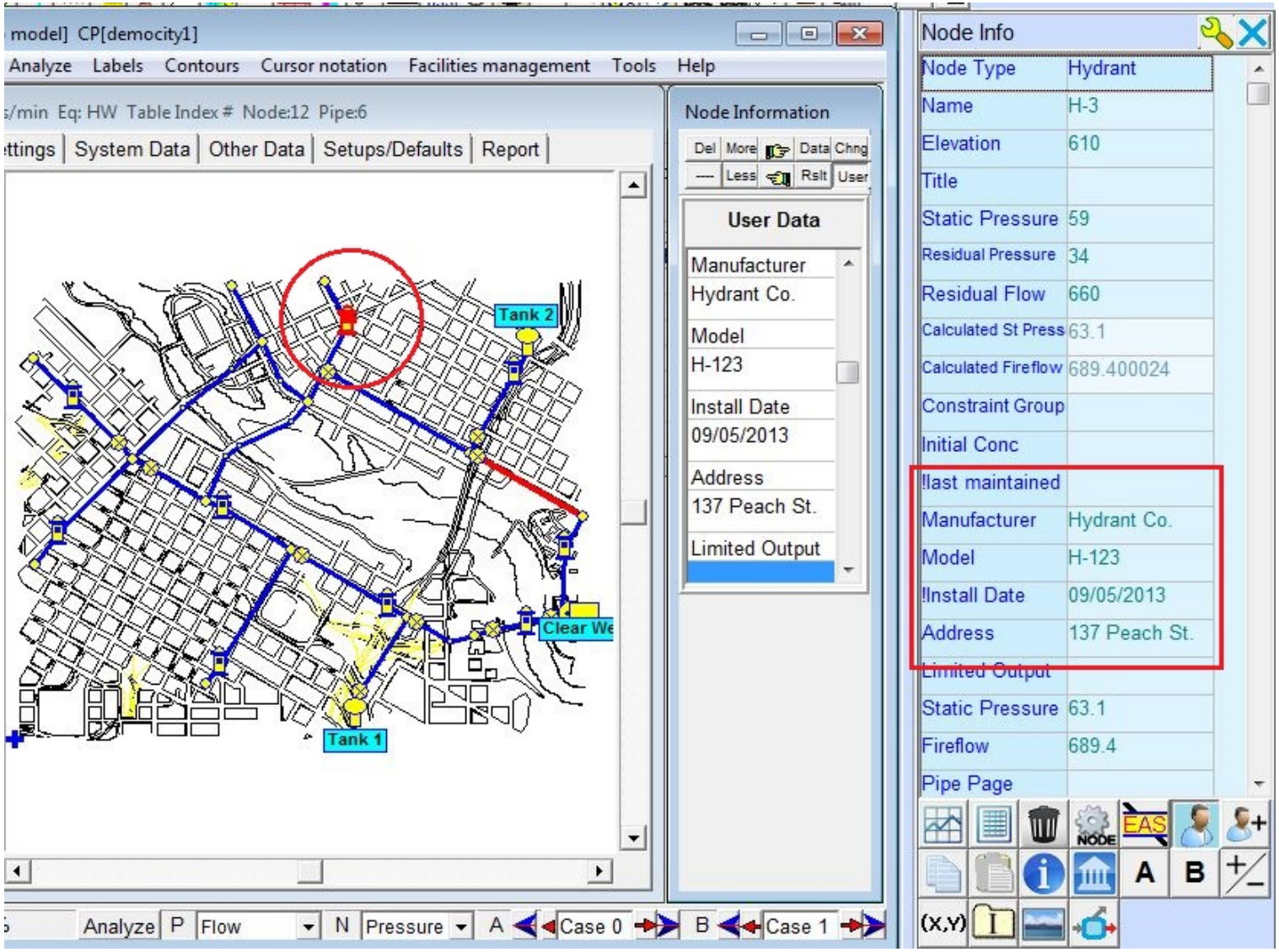





You can enter the Attribute Name (for example Zone or Phase of construction) as shown. In KYnetic, the attribute may be changed to a Date format by clicking the new item name and checking "This is a Date" in the Edit box that appears (see below). In Classic, select Attribute Type (Date for a date and List for all others), if applicable, select the node type to which it will apply, and click OK. Now a new User Data item called Zone or Phase of construction will appear as User Data. Note you can use the same procedure to change the name of an existing User Data item.

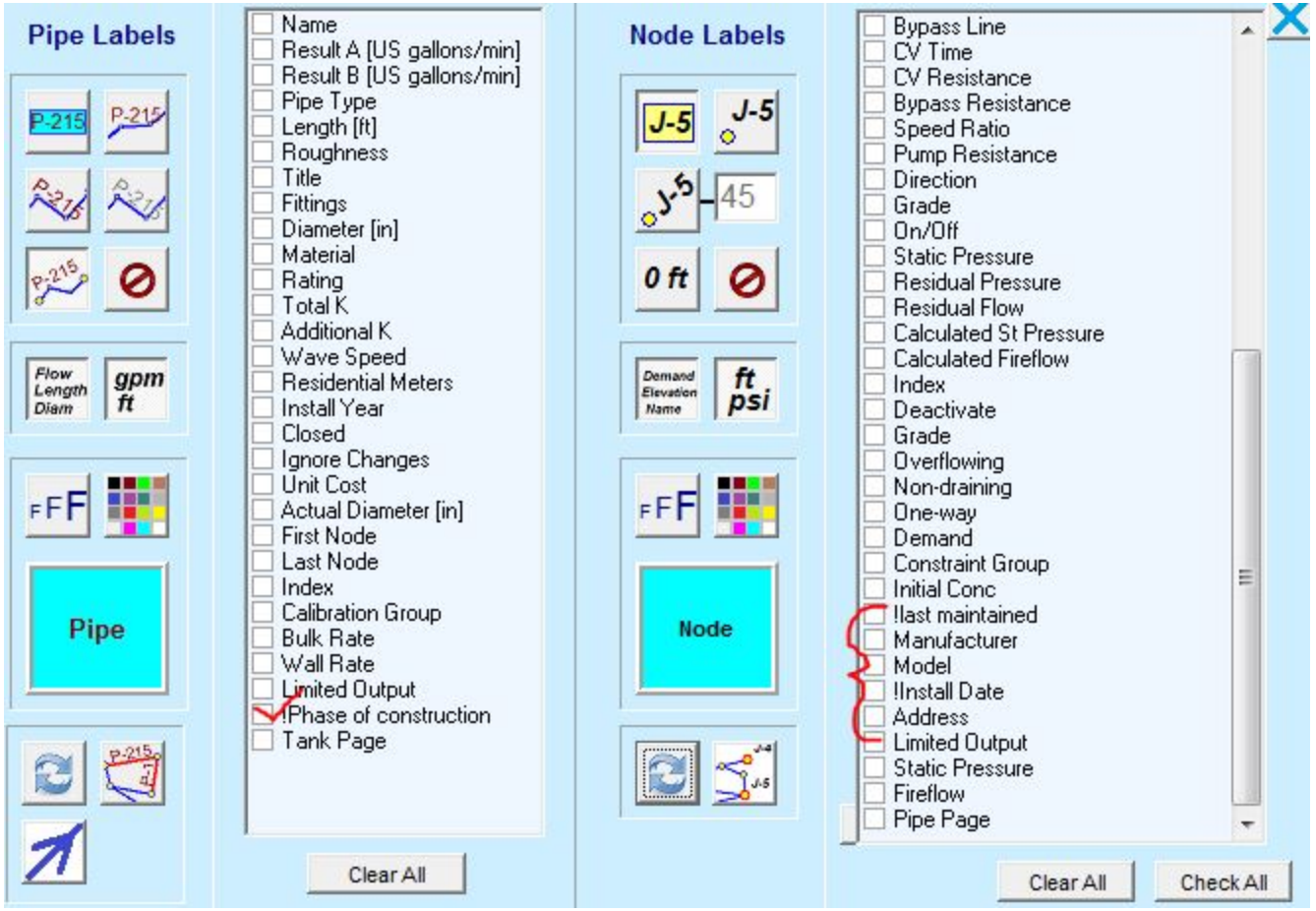


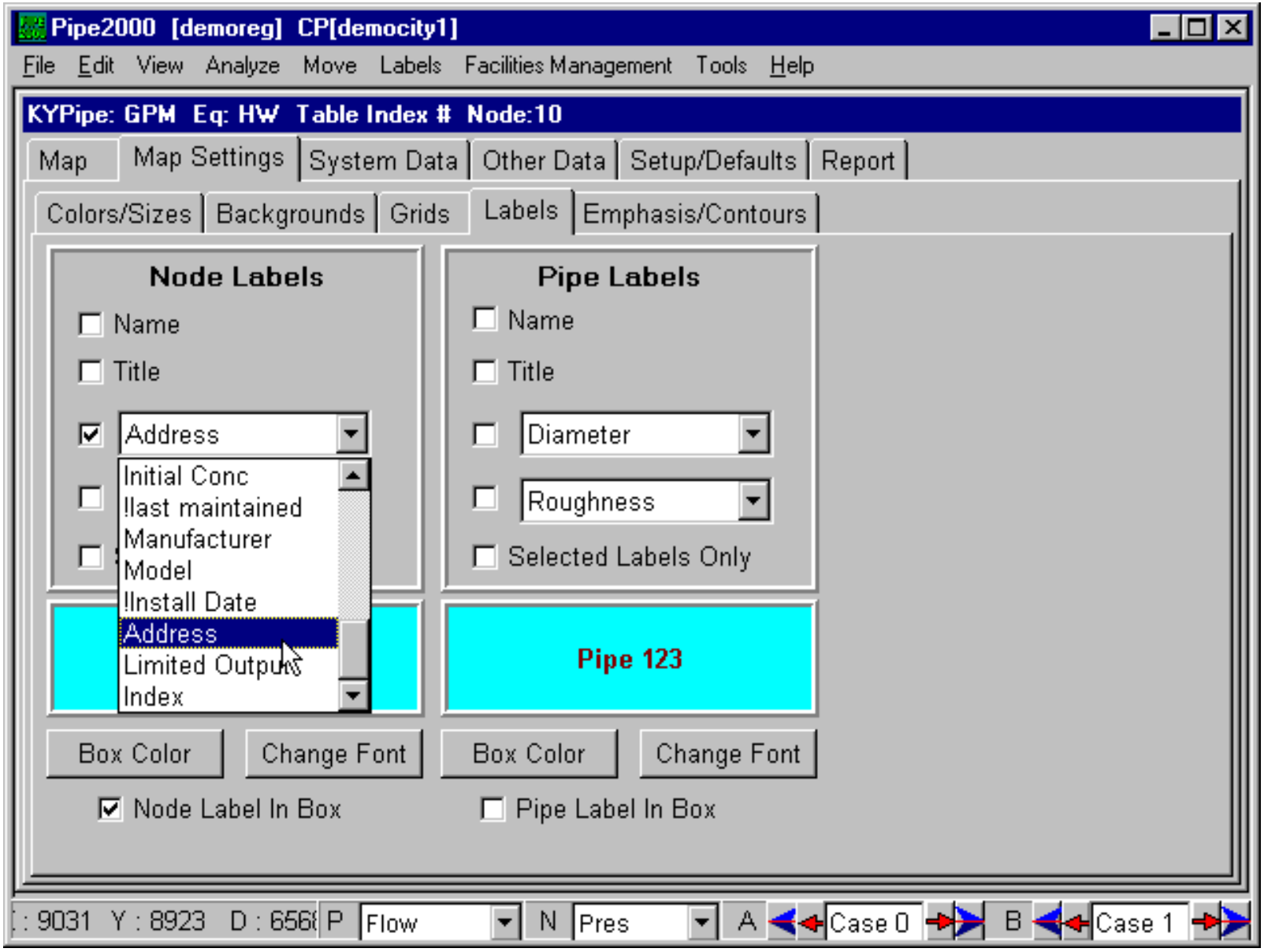

**Using User Data to Maintain Records.** Records for devices in the pipe system can be entered using User Data as shown below.



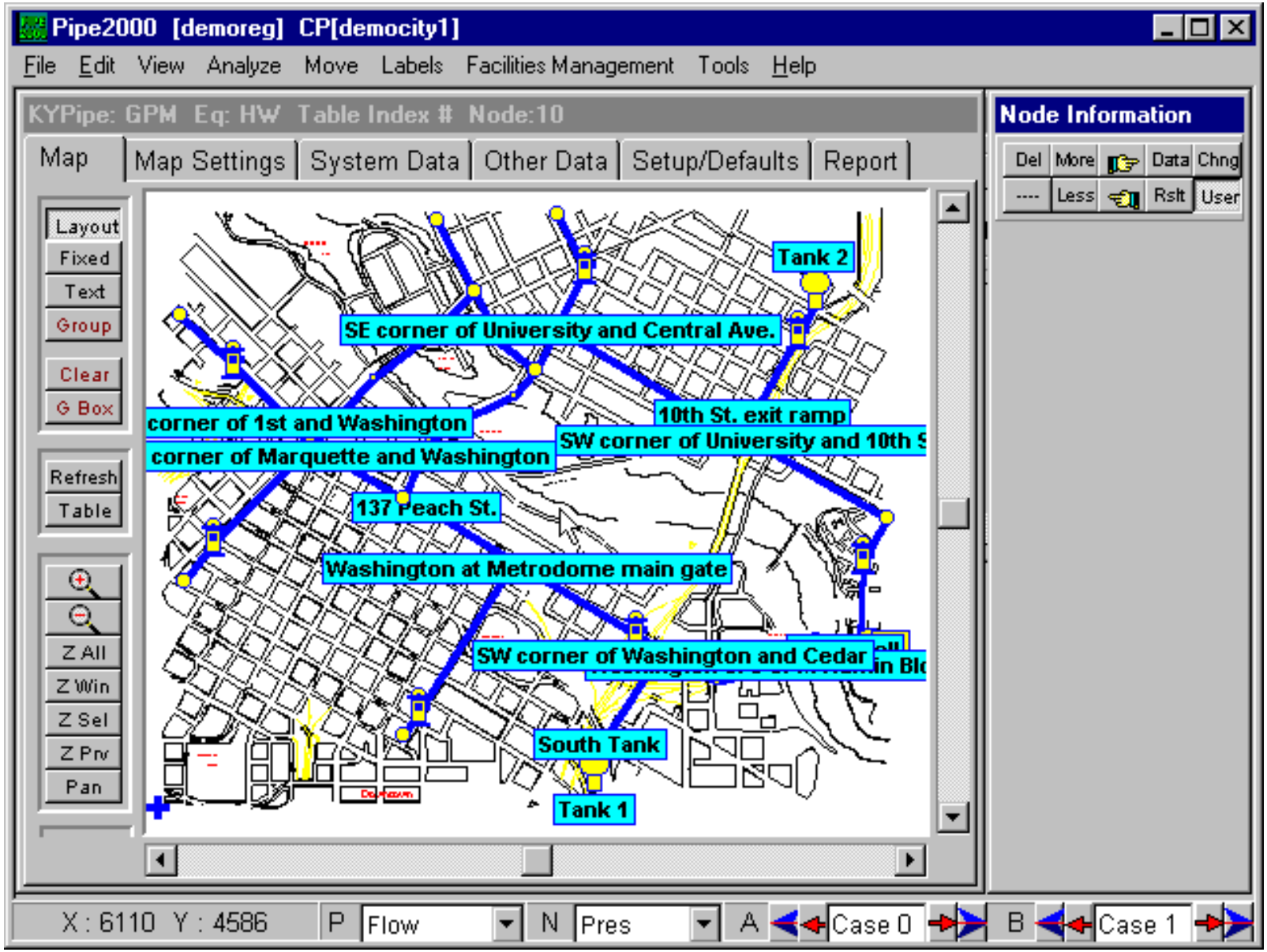

This represents some user data for the selected hydrant.

**Labeling User Attributes.** Once a User Data item is added, this attribute becomes available for other operations such as map labels. If, for example, you wish to display the address of certain devices you can select that attribute from the Node Label drop down list as shown below.

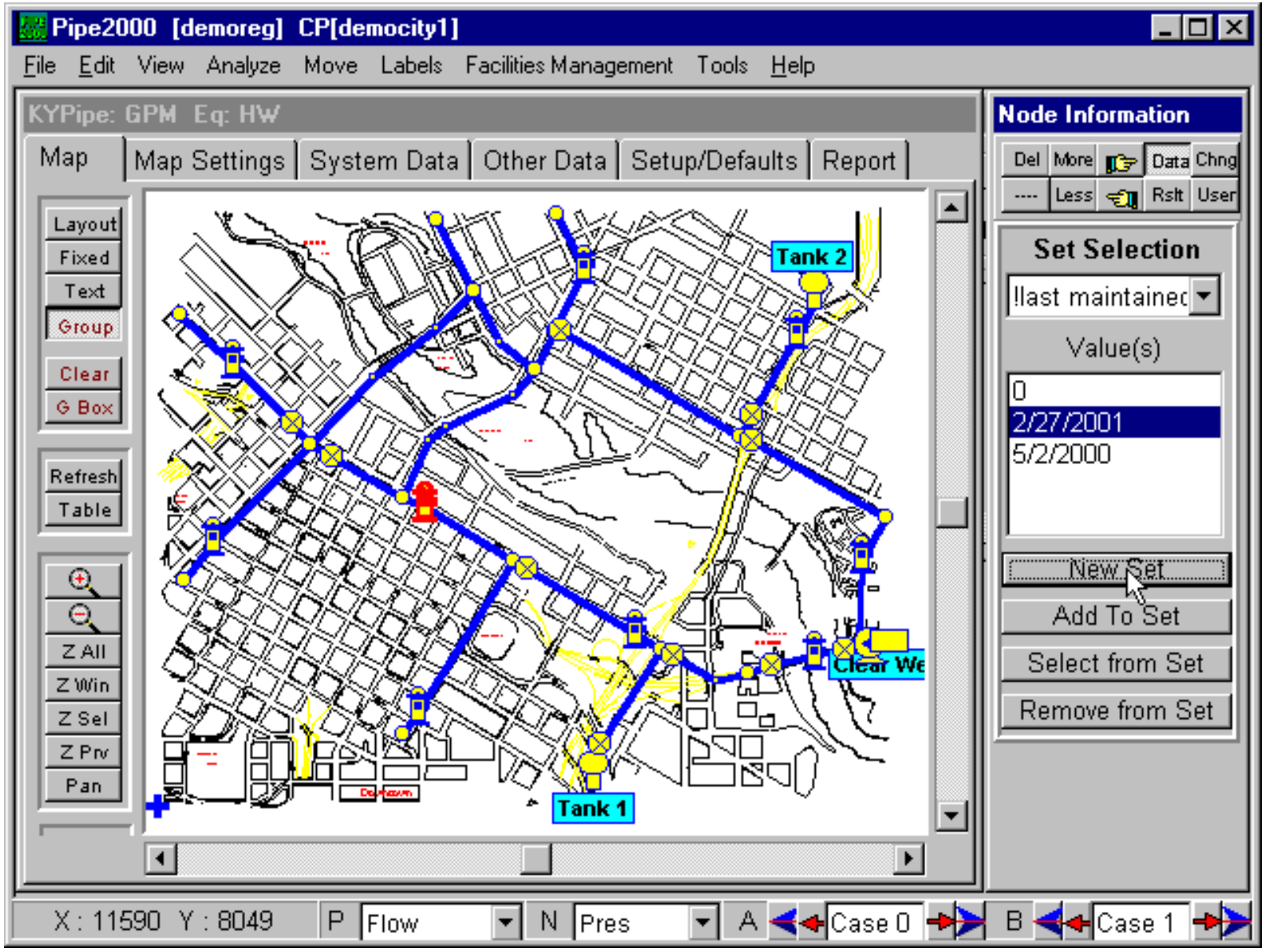

Pipe Labels
P-215
P-215
Flow Length Diam
gpm ft
FFF
Pipe
Name
Result A [US gallons/min]
Result B [US gallons/min]
Pipe Type
Length [ft]
Roughness
Title
Fittings
Diameter [in]
Material
Rating
Total K
Additional K
Wave Speed
Residential Meters
Install Year
Closed
Ignore Changes
Unit Cost
Actual Diameter [in]
First Node
Last Node
Index
Calibration Group
Bulk Rate
Wall Rate
Limited Output
!Phase of construction
Tank Page
Clear All
Node Labels
J-5
J-5
J-5
45
0 ft
Demand Elevation Name
ft psi
FFF
Node
Bypass Line
CV Time
CV Resistance
Bypass Resistance
Speed Ratio
Pump Resistance
Direction
Grade
On/Off
Static Pressure
Residual Pressure
Residual Flow
Calculated St Pressure
Calculated Fireflow
Index
Deactivate
Grade
Overflowing
Non-draining
One-way
Demand
Constraint Group
Initial Conc
!last maintained
Manufacturer
Model
!Install Date
Address
Limited Output
Static Pressure
Fireflow
Pipe Page
Clear All
Check All

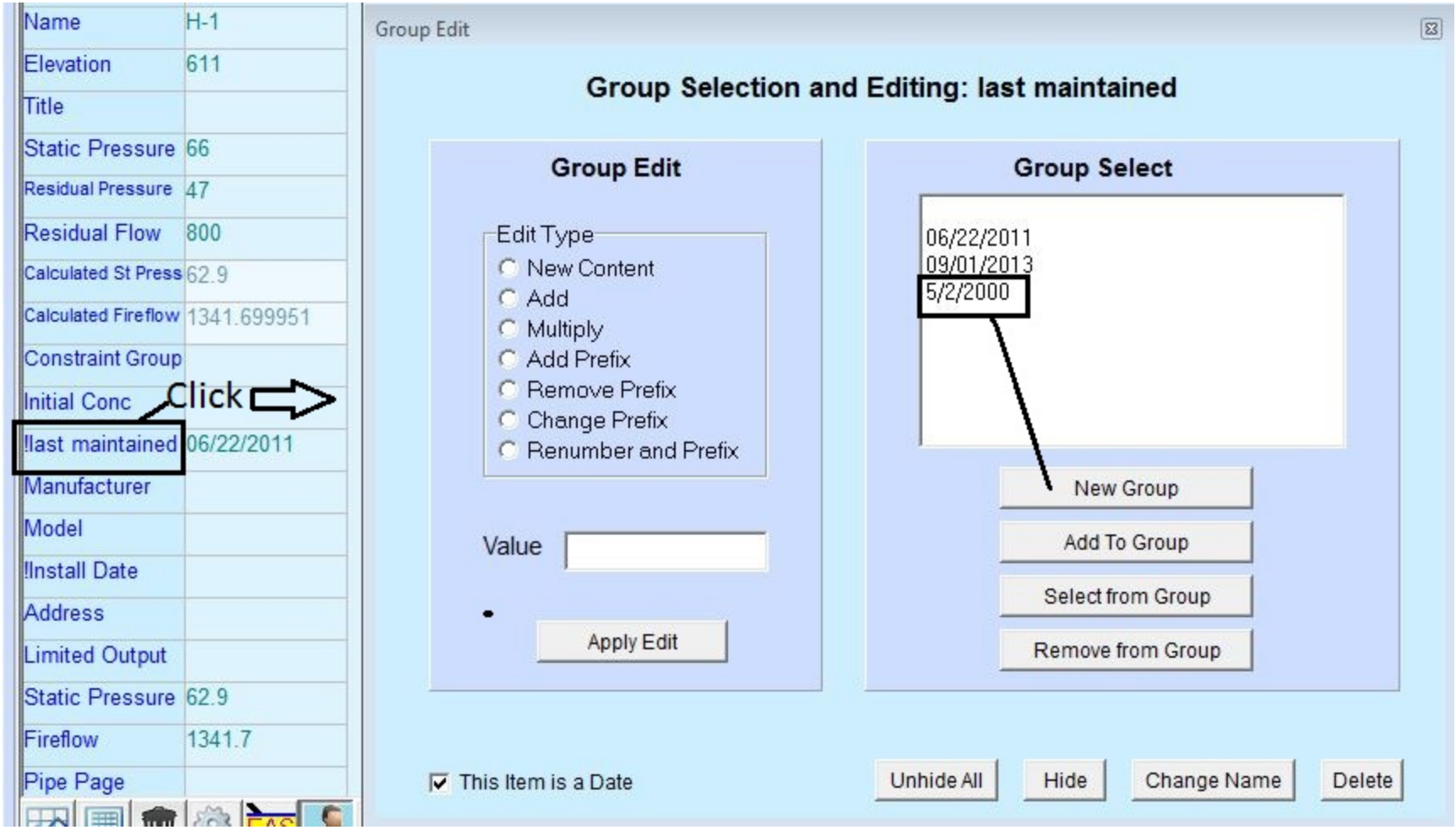



You will get a display with the selected labels as shown below.


Pipe2000 [demoreg] CP[democity1]
File Edit View Analyze Move Labels Facilities Management Tools Help
KYPipe: GPM Eq: HW Table Index # Node:10
Map
Map Settings
System Data
Other Data
Setup/Defaults
Report
Layout
Fixed
Text
Group
Clear
G Box
Refresh
Table
Z All
Z Win
Z Sel
Z Prv
Pan
Tank 2
SE corner of University and Central Ave.
10th St. exit ramp
corner of 1st and Washington
SW corner of University and 10th
corner of Marquette and Washington
137 Peach St.
Washington at Metrodome main gate
SW corner of Washington and Cedar
South Tank
Tank 1
Node Information
Del
More
Data
Chng
Less
Rslt
User
X : 6110 Y : 4586
P
Flow
N
Pres
A
Case 0
B
Case 1

**Group Selections by User Attributes.** The attributes are also available for Group Set Selections as shown below. Since the Set selection can select ranges and can filter sets by several layers of selection this provides the opportunity to use Pipe2018 to select and display groups of items based on several criterion. For example, one could select all valves manufactured by ABC and installed from years 1980-1999 which have not been maintained in the last two years.


Pipe2000 [demoreg] CP[democity1]
File Edit View Analyze Move Labels Facilities Management Tools Help
KYPipe: GPM Eq: HW
Map
Map Settings
System Data
Other Data
Setup/Defaults
Report
Layout
Fixed
Text
Group
Clear
G Box
Refresh
Table
Z All
Z Win
Z Sel
Z Prv
Pan
Tank 2
Tank 1
Node Information
Del
More
Data
Chng
Less
Rslt
User
Set Selection
!last maintainec
Value(s)
0
2/27/2001
5/2/2000
New Set
Add To Set
Select from Set
Remove from Set
X : 11590 Y : 8049
P
Flow
N
Pres
A
Case 0
B
Case 1




User Data provides an advanced data maintenance and selection capability.

## 1.3.4 Units and Reference Tables

### 1.3.4.1 Analyzing for liquids other than water

Analyzing for liquids other than water

---

For a Pipe2018 analyses with any liquid, the specific gravity and viscosity are the two attributes required to be specified. Darcy Weisbach friction loss equation is used.

### 1.3.4.2 Darcy-Weisbach Table

**Important Note:** Values of ε are entered in Pipe Roughness field in units of millifeet or millimeters. Convert the data below into the appropriate units.

---

VALUES OF ROUGHNESS FOR THE DARCY WEISBACH EQUATION

| Surface | Absolute Roughness Coefficient | |
|---|---|---|
| | *(mm)* | *(millifeet)* |
| Copper, Lead, Brass, Aluminum (new) | 0.001 - 0.002 | 0.0033 – 0.0067 |

| | | |
|---|---|---|
| PVC and Plastic Pipes | 0.0015 - 0.007 | 0.005 – 0.023 |
| Stainless steel | 0.015 | 0.05 |
| Steel commercial pipe | 0.045 - 0.09 | 0.15 – 0.3 |
| Stretched steel | 0.015 | 0.05 |
| Weld steel | 0.045 | 0.15 |
| Galvanized steel | 0.15 | 0.5 |
| Rusted steel (corrosion) | 0.15 - 4 | 0.5 – 13.3 |
| New cast iron | 0.25 - 0.8 | 0.8 – 2.7 |
| Worn cast iron | 0.8 - 1.5 | 2.7 - 5 |
| Rusty cast iron | 1.5 - 2.5 | 5 - 8.3 |
| Sheet or asphalted cast iron | 0.01 - 0.015 | 0.033 – 0.05 |
| Smoothed cement | 0.3 | 1 |
| Ordinary concrete | 0.3 - 1 | 1 - 3.33 |
| Coarse concrete | 0.3 - 5 | 1 - 16.7 |
| Well planed wood | 0.18 - 0,9 | 0.6 – 3.0 |
| Ordinary wood | 5 | 16.7 |

$$h_{LP} = \frac{f L V^2}{2 g D}$$

Darcy-Weisbach Equation

$$f = \frac{0.25}{\left[\log_{10}\left(\frac{\varepsilon}{3.7D} + \frac{5.74}{\mathrm{Re}^{0.9}}\right)\right]^2}$$

Jain - friction factor Equation

where *f* is a function of:

- Roughness height, ε (m, ft)
- Pipe diameter, *D* (m, ft)
- Reynolds number, *Re* (unitless).

## 1.3.4.3 Hazen-Williams Table

Hazen-Williams Table

---

VALUES OF C IN HAZEN WILLIAMS EQUATION

| PIPE MATERIAL | PIPE AGE | PIPE SIZE | C |
|---|---|---|---|
| Cast Iron | New | All Sizes | 130 |
| | 5 years old | 12" and Over | 120 |
| | | 8" | 119 |
| | | 4" | 118 |
| | 10 years old | 24" and Over | 113 |
| | | 12" | 111 |
| | | 4" | 107 |

| | | | |
|---|---|---|---|
| | 20 years old | 24" and Over | 100 |
| | | 12" | 96 |
| | | 4" | 89 |
| | 30 years old | 30" and Over | 90 |
| | | 16" | 87 |
| | | 4" | 75 |
| | 40 years old | 30" and Over | 83 |
| | | 16" | 80 |
| | | 4" | 64 |
| | 50 years old | 40" and Over | 77 |
| | | 24" | 74 |
| | | 4" | 55 |
| Welded Steel | Values of C the same as for cast-iron pipes, 5 years older | | |
| Riveted Steel | Values of C the same as for cast-iron pipes, 10 years older | | |
| Wood Stave | Average value, | | 120 |

| | | |
|---|---|---|
| | regardless of age | |
| Concrete or Concrete Lined | Large sizes, good workmanship, steel forms | 140 |
| | Large sizes, good workmanship, wooden forms | 120 |
| | Centrifugally spun | 135 |
| Vitrified | In good condition | 110 |
| Plastic or Drawn Tubing | | 150 |

$$h_{LP} = \frac{4.73\, L\, Q^{1.852}}{C^{1.852}\, D^{4.87}}$$

Hazen Williams Equation in English units:

L-ft, Q-cfs, D-ft

$$h_{LP} = \frac{10.675\, L\, Q^{1.852}}{C^{1.852}\, D^{4.87}}$$

Hazen Williams Equation in SI units:

L-m, Q-cms, D-m

Note: some limitation exists in the Hazen Williams method of calculation for pipes < 3 inches and velocity > 3 m/s (10 ft/s).

-Liou, C.P. "limitations and proper use of the HW equation", Journal of Hydraulic Engineering, Vol 124, #9, American Society of Civil Engineering, NY 1998.

-Diskin, M.H."the limits of applicability of the HW formula", La Houille Blanche, #6, Nov 1960.

## 1.3.4.4 Modified Hazen Williams Formula

This equation is commonly used in India and is recommended by the government agency (Central Public Health and Environmental Engineering Organization - CPHEEO).

**Modified Hazen – Williams Formula**

$$V = 143.534\ CR\ r^{0.6575}\ S^{0.5525}$$

$$h = \frac{[L(Q / CR)\ 1.81]}{994.62D^{4.81}}$$

V = velocity of flow in m/s
CR = pipe roughness coefficient, ( 1 for smooth pipes; < 1 for rough pipes)
r = hydraulic radius in m
s = friction slope
D = internal diameter of pipe in m
h = friction head loss in m
L = length of pipe in m
Q = flow in pipe in m3 / s

## 1.3.4.5 Minor Loss Coefficients Table

Minor Loss Coefficients Table

---

**MINOR LOSS COEFFICIENTS FOR COMMON FITTINGS**

| TYPE OF FITTING | MINOR LOSS COEFFICIENT(M) |
|---|---|
| **Elbow:** *45â standard* | **0.35** |
| *45â long radius* | **0.20** |
| *90â standard* | **0.75** |

| | |
|---|---|
| *90â long radius* | **0.45** |
| *90â square or miter* | **1.30** |
| *180â bend, close return* | **1.50** |
| | |
| **Tee:** *standard, along run, branch blanked* | **0.40** |
| *used as elbow, entering run* | **1.30** |
| *used as elbow, entering branch* | **1.50** |
| *branching flow* | **1.00** |
| | |
| **Coupling** | **0.04** |
| | |
| **Union** | **0.04** |
| | |
| **Gate Valve:** *Full Open* | **0.17** |
| *¾ Open* | **0.90** |
| *½ Open* | **4.50** |
| *¼ Open* | **24.00** |
| | |
| **Diaphragm Valve:** *Full Open* | **2.30** |

| | |
|---|---|
| *¾ Open* | **2.60** |
| *½ Open* | **4.30** |
| *¼ Open* | **21.00** |
| **Globe Valve:** *Bevel seat: Full Open* | **6.40** |
| *½ Open* | **9.50** |
| *Composition seat: Full Open* | **6.00** |
| *½ Open* | **8.50** |
| *Plug disk: Full Open* | **9.00** |
| *¾ Open* | **13.00** |
| *½ Open* | **36.00** |
| *¼ Open* | **112.00** |
| **Angle Valve:** *Full Open* | **3.00** |
| **Y or blowoff valve:** *Full Open* | **3.00** |
| **Plug cock:** *α = 5â* | **0.05** |
| *= 10â* | **0.29** |

| | |
|---|---|
| *= 20â* | **1.56** |
| *= 40â* | **17.30** |
| *= 60â* | **206.00** |
| **Butterfly valve:** *α = 5â* | **0.24** |
| *= 10â* | **0.52** |
| *= 40â* | **10.80** |
| *= 60â* | **118.00** |
| **Check Valve:** *Swing* | **2.00** |
| *Disk* | **10.00** |
| *Ball* | **70.00** |
| **Foot Valve** | **15.00** |
| **Water Meter:** *Disk* | **7.00** |
| *Piston* | **15.00** |
| *Rotary (start-shaped disk)* | **10.00** |
| *Trubine wheel* | **6.00** |

$$h_{LM} = M\frac{V^2}{2g}$$

## 1.3.4.6 Tools (Main Menu)

Tools

10 Year HW Coefficient

Air Slam Pressure Surge

Air Valve Orifice Size

Bladder Precharge

Calculator

C Factor Calculator

Density and Viscosity Table

Extract Elevations from DEM Files

Force Calculations

Gas Properties

Generate Intermediate Pump File

Headloss Calculator

Hose and Nozzle Constants

Modulating (Regulating) Valve

Power (HP or KW) Calculations

Profile Import

Pump Characteristic Curves

Pump File Characteristics

Pump Selection

Residual Pressure Adjustment

Resistance Calculations
Select Pump File/Calculate Inertia
Spike Track
Sprinkler/Blowoff Constant
Turbine Data Convertor
Units Converter
Valve Stroking
Wave Speed

Tools - Surge only

Equivalent Surge Tank Convertor
Export Air Valve Locations
Surge Tank Calculator

Tools - Gas only

Gas Cv Calculations
Gas K Calculations

## 1.3.4.7 Units

Units for Simulation Specs

See also Units KYPipe and Surge

Units GoFlow

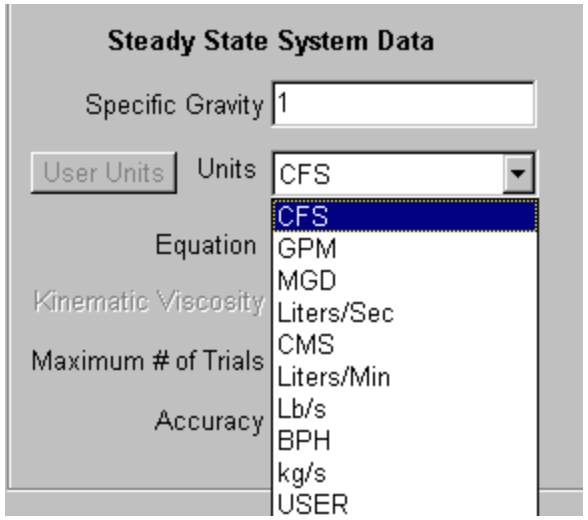
Units Gas and Steam

Units Help - Hover Buttons and Device Help

---

### Units

For KyPipe and Surge there are 10 options for system flow units under System Data | Simulation Specs.



**Units - Flow Units** CFS (cubic feet/second)

GPM (gallons/minute)

MGD (million gallons/day)

Liters/Sec (liters/second)

CMS (cubic meters/second)

Liters/Min (liters/minute)

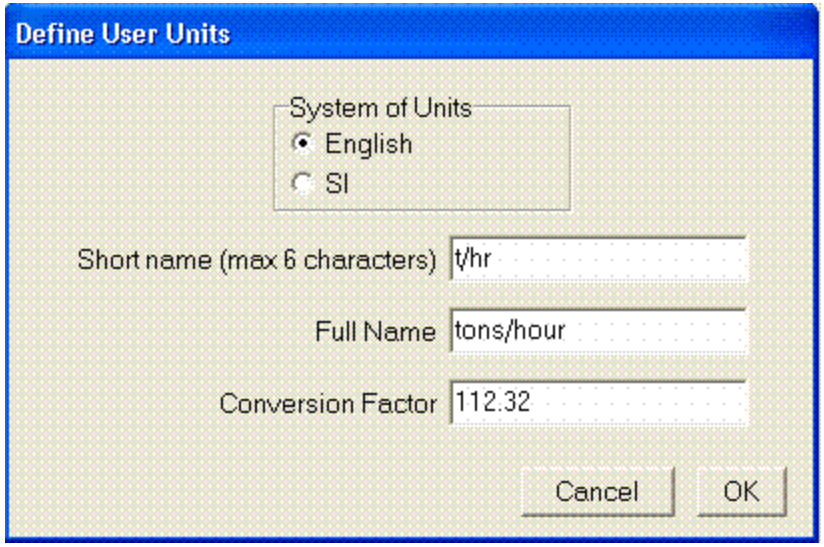
Lb/s (pounds/second)

BPH (barrels/hour)

kg/s (kilograms/second)

USER (user defined units)

User Flow Units One of these options is USER. If USER is selected, then click on the User Units button and the following window will appear. The user may name the flow units however they choose and then provide the conversion factor; cubic feet per second for English or cubic meters per second for SI to the unit chosen. In the example below, we have chosen tons/hour and have provided the conversion factor of 112.32 tons/hr/cfs. All other units remain the same based on the English or SI selection. See Units.



## 1.3.4.8 Units Help - Hover Buttons

Units Help - Hover Buttons and Device Help

---

**Hover Buttons** - Point to (hover, do not click on) any data item title in the Pipe or Node Information window and a hint with units for that item will appear, both as a pop-up box and as a note in the bar at the bottom center of the map screen.

- For more complicated elements, especially Surge Devices, there is a button in the Node Information window which says Device Help. Click on this button for a detailed diagram of the device to which it is associated.

## 1.3.4.9 Units Pipe2000 : KYPipe / Surge

Units KYPipe / Surge

---

| Flow | length | diameter | roughness (Darcy Weisbach) | kinematic viscosity (Darcy Weisbach) | demand | elevations, grades, tank level* | pressure | velocity | head loss | pump head | pump flow | pump power |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| cms | m | mm | mm | m*m/s | cms | m | kpa (gauge) | m/s | m | m | cms | kW |
| l/s | m | mm | mm | m*m/s | l/s | m | kpa (gauge) | m/s | m | m | l/s | kW |

| | | | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| cfs | ft | in | mft | ft*ft/s | cfs | ft | psi (gauge) | ft/s | ft | ft | cfs | useful HP |
| mgd | ft | in | mft | ft*ft/s | mgd | ft | psi (gauge) | ft/s | ft | ft | mgd | useful HP |
| gpm | ft | in | mft | ft*ft/s | gpm | ft | psi (gauge) | ft/s | ft | ft | gpm | useful HP |

* Remember that elevations, grades, and tank levels are all measured from the same datum (elevation datum). If the elevation of the pipe connection to a reservoir (or tank) is 100 ft (m) and the reservoir (initial tank level) is 20 ft (m) above that then the grade for the reservoir (or initial tank level) is 120 ft (m) (NOT 20 ft (m)).

**Loss Element** - table of pressure drop (ft or m) versus flow in defined units.

**Sprinkler Constants** - (flow rate in gpm (l/s))/(sprinkler pressure drop in psi (kpa))^0.5.

Example: flow = 10 gpm, pressure drop = 4 psi

K = 10 gpm/(4psi)^0.5 = 5

- See the Sprinkler/Blowoff Constant tool under Tools in the Pipe2018 main menu

**Valve Resistance** - (head drop in ft (m))/(flow in cfs (cms))^2

Example: flow = 1000 gpm (2.228 cfs), pressure drop = 15 ft

R = 15ft/(2.228cfs)^2 = 3.022

- See the Resistance Calcluation tool, under Tools in the Pipe2018 main menu.

For Cv, flow coefficient, the units are number of gpm per 1 psi pressure drop.

**Regulator Setting** - PRV, PSV - same as pressure (psi or kpa)

FCV - flow in defined units

**Additional Units for Surge Devices:**

**Surge Protection Devices Inflow and Outflow Resistances** - (head drop in ft (m))/(flow in cfs (cms))^2

For all types of Surge Tanks, Pressure Relief Valves, Surge Anticipation Valve, Rupture Disk, see Resistance Tool - dropdown options.

**Surge Tank Diameter** - ft. or m

**Surge Tank Maximum Level (Spilling Surge Tank)** - ft. or m. above pipe center line (not above Elevation Datum)

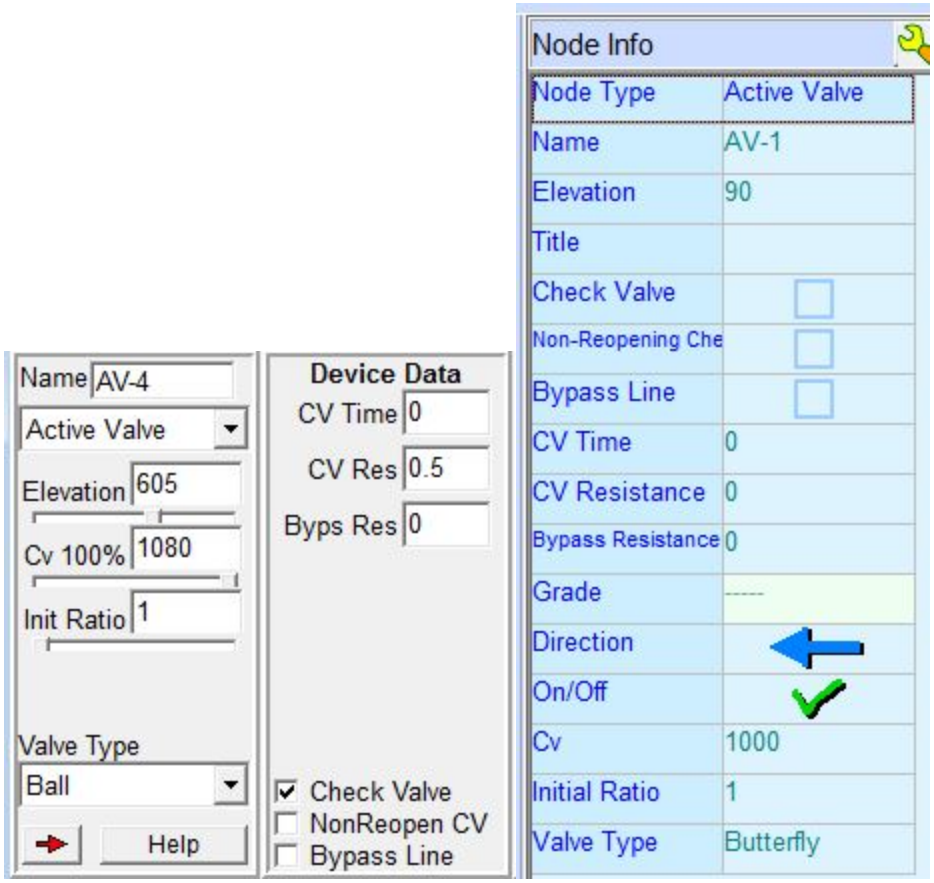
**Air/Vacuum Valve Diameters** - inches or mm

**Air Vacuum Valve Initial Air Volume** - cubic feet or cubic meters

For more units information, see also 'Surge Elements and Data Requirements' in the Pipe2018 Help file.

## 1.3.5 Elements and Devices

### 1.3.5.1 Active Valve




An Active Valve is a valve which may be opened, throttled, or closed for modeling purposes. Unlike other on/off valves, the minor loss for an Active Valve is based on the valve type and the open ratio is calculated and incorporated into the analysis. The open ratio can be changed by the user during a simulation using change data.

### Active Valve Data

Elevation - This is the elevation (ft. or m) of the valve.

Cv 100% (R 100%) There are two options to define loss across a valve, one is resistance (R) and another is flow coefficient (Cv).

R 100% - This is the resistance of the valve while it is 100% open. The resistance is the head drop (in ft. or m) over the flow squared (in cfs or cms). For example, a wide open valve which has a head drop of 1.2 ft at 500 gpm (1.114 cfs) has a wide open resistance of (1.2 / (1.114^2) ) = 0.97.

Cv 100% - Under Preference menu or System Data/Preferences (Classic), a check box is provided to allow the use of a flow coefficient, Cv, usually provided by the

manufacturer, in lieu of the calculated resistance. Cv 100% must be a non-zero number. Cv is the flowrate in gpm required to cause a 1 psi drop in pressure (units gpm/psi^0.5). In SI units, Kv is used (m3/h at sqr1bar). The conversion is made internally (Cv= Kv*1.156).

Initial Ratio - the is the ratio of the initial valve stem position to the fully opened stem position (0 = closed and 1 = wide open). The minor loss coefficient based on this number is calculated and used in the analysis.

Grade - This appears only if no pipe links are attached to one side of the active valve so that the valve is at the end of a pipe. It is the HGL [ft (m)] of the connecting reservoir. To model a discharge to atmospheric pressure, make this value equal to the elevation.

Valve Type - The type of valve is chosen from the drop-down selections. For all valve types (except **Other**), the open ratio refers to the ratio of the stem position to the fully open position. A ratio of 0.4 means the stem has moved 40% of the range from fully closed to fully open. For the **Other** designation, the open ratio means the ratio of the open area to the fully open area for the valve.

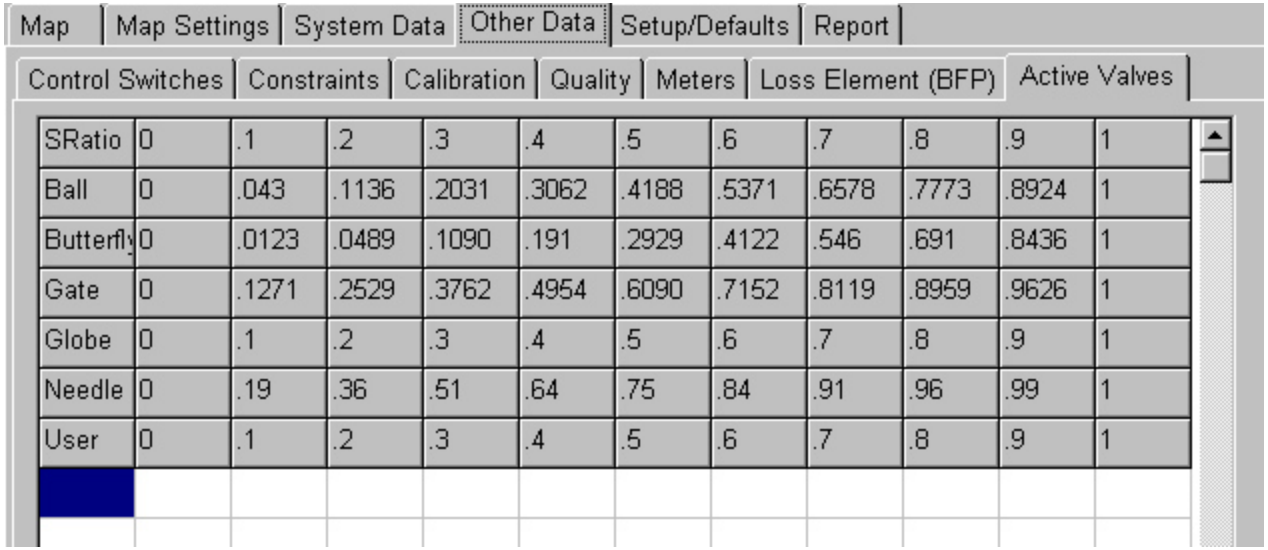
Check Valves - Pumps, active valves, and loss elements all can be equipped with a check (non-return) valve to prevent backflow. Either a normal check valve which will close or open depending on conditions or a non-reopening check valve which will close only once and remain closed can be designated. The closing time (CV Time) is the time it takes for the check valve to close once closing is initiated. The check valve resistance (CV Res) is the resistance (head/flow^2) when the valve is fully open.

Bypass Lines - (Surge feature) A bypass line is one which will open when the suction (upstream) head exceeds the discharge (downstream) head. The bypass line resistance (Byps Res) is the resistance of this pipe. A valve bypass line is used to define a different resistance when flow reverses and is assumed to flow in the bypass line.

*Active Valve Table*

Under Component menu or Other Data | Active Valves (Classic), the following table appears. These values represent the open areas vs. open ratios (valve stem positions) for various active valves. The user may enter data for user-defined valves as well. These additional valves will be included in the Valve Type drop-down selector box in the Node Information Data window.

KYnetic Components menu icon:




| SRatio | 0 | .1 | .2 | .3 | .4 | .5 | .6 | .7 | .8 | .9 | 1 |
|---|---|---|---|---|---|---|---|---|---|---|---|
| Ball | 0 | .043 | .1136 | .2031 | .3062 | .4188 | .5371 | .6578 | .7773 | .8924 | 1 |
| Butterfly | 0 | .0123 | .0489 | .1090 | .191 | .2929 | .4122 | .546 | .691 | .8436 | 1 |
| Gate | 0 | .1271 | .2529 | .3762 | .4954 | .6090 | .7152 | .8119 | .8959 | .9626 | 1 |
| Globe | 0 | .1 | .2 | .3 | .4 | .5 | .6 | .7 | .8 | .9 | 1 |
| Needle | 0 | .19 | .36 | .51 | .64 | .75 | .84 | .91 | .96 | .99 | 1 |
| User | 0 | .1 | .2 | .3 | .4 | .5 | .6 | .7 | .8 | .9 | 1 |

When creating a valve description, Effective area should be used for the area ratio vs stem position so that change in coefficient of discharge will be accounted for.

## 1.3.5.2 All About Pipes

**Pipe Sections** - The total length, inside diameter and roughness of each pipe section must be input as data. The designation of pipe roughness depends on the head loss equation used. Users primarily interested in water distribution systems may prefer the Hazen Williams Equation which was developed to compute line losses for pipes carrying water. If this expression is to be employed, the Hazen-Williams factor must be input as data for each pipe. This roughness coefficient depends on the type and condition of each pipe and the Hazen-Williams Table gives the Hazen Williams Equation along with some representative values for this coefficient. However, the variation with age depends somewhat on the location of the water distribution system and sometimes field tests are required to obtain reliable values of the Hazen-Williams roughness coefficient for old pipes.

You may prefer or be required to employ the Darcy-Weisbach Equation for computing head loss. This expression can be applied to

systems transporting water and is required for fluids other than water. If this option is employed, the roughness for each pipe section corresponding to the Darcy-Weisbach expressions must be input as data as well as the kinematic viscosity of the fluid for that system. The Darcy-Weisbach Table gives the Darcy-Weisbach Equation along with the explicit relationship for calculating the friction factor employed by KYPIPE and some typical values for roughness for new pipes. Again, the roughness depends on type and condition of the pipe.

**Normal Flow Directions** - Flow directions for lines with pumps, check valves, and pressure regulating valves must be correctly specified in the data input and this is done by the order which the connecting nodes for the pipe section are input. The normal flow direction is assumed to be from the first node input to the second node input. If the calculated flow is in the opposite direction it will be tabulated with a negative sign. An option is available to report all flows as positive regardless of flow directions. This is available under the Analysis Setup window.

See Also: Model Calibration

## 1.3.5.3 All about nodes

All about nodes

See also End 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 (for licensing purposes) while internal and text nodes do not.

### *End Nodes*

**End Node Properties** - 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 are accepted.

**End Node Types** - These include the following node types:

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 (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.

## *Internal Nodes*

**Internal Node Properties** - 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.

**Internal Node Types** - These include the following node types that are situated at a connection of two pipe segments with identical basic pipe properties:

1. **Intermediate Node** - No device at this location - usually represents a change of alignment.

2. **Valve** - Indicates location of cut-off valves.

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).

***Additional Information***

**Text Nodes** - Text nodes are used to provide information and can be located anywhere on your map. They are added and deleted in the '**TEXT**' mode and several options are available for displaying this information.

**All Nodes** - All nodes, including text nodes, have a location ('**x**' and '**y**' coordinate) and an elevation ('z' coordinate). A text title can be defined and displayed. Also, a bitmap image can be imported and displayed. Except for text nodes and intermediate nodes, all nodes have additional attributes (data). They are defined in the data information boxes that are displayed when the node is selected and in the data tables that can be accessed for all nodes.

Check Valve

Active Valve

BFP (Back Flow Preventer)

Hydrant

 Intermediate Node

Junction

Loss Element

Meter

Pump

Rack Sprinkler ON

Rack Sprinkler OFF

Reservoir

Sprinkler ON

Sprinkler OFF

Tank

Regulating Valves

Variable Pressure Supply

Pressure Switch

Flow Meter

See Also: Minor Loss Components

Demand Specification

## 1.3.5.4 Annulus Pipe Flow

KYPIPE uses DW equation which was derived based on the assumption that the flow takes place in the entire inner space of the pipeline.

Hazen-Poiseuille equation is derived and generalized into DW equation by replacing viscosity term (vD/n) with 64/f, etc. For flows in annular regions of pipelines we can derive a similar equation starting from fundamentals taking into account the friction term associated with inner friction surface. However, KYPipe currently does not do this.

The best estimation for non circular pipes is that the hydraulic radius is calculated and used for the pressure drop calculation.

**For an annulus**

An annulus is formed by placing a smaller diameter pipe inside a larger diameter pipe. The cross sectional area of flow will be the cross sectional area of the larger pipe calculated using the inner pipe diameter minus the cross sectional area of the smaller pipe calculated using the outer pipe diameter. The wetted perimeter will be the inner circumference of the larger pipe plus the outer circumference of the smaller pipe.

$D_h = 4 \times (\pi \times (d1^2 - d2^2) / 4) / (\pi \times d1 + d2)$

where d1 = inner diameter of larger pipe, d2 = outer diameter of smaller pipe"

Another equation used by the Petroleum industry:

$EffectiveDiameter=((ID_{outer}+OD_{Inner})^2 * (ID_{outer}-OD_{Inner})^3)^{(1/5)}$

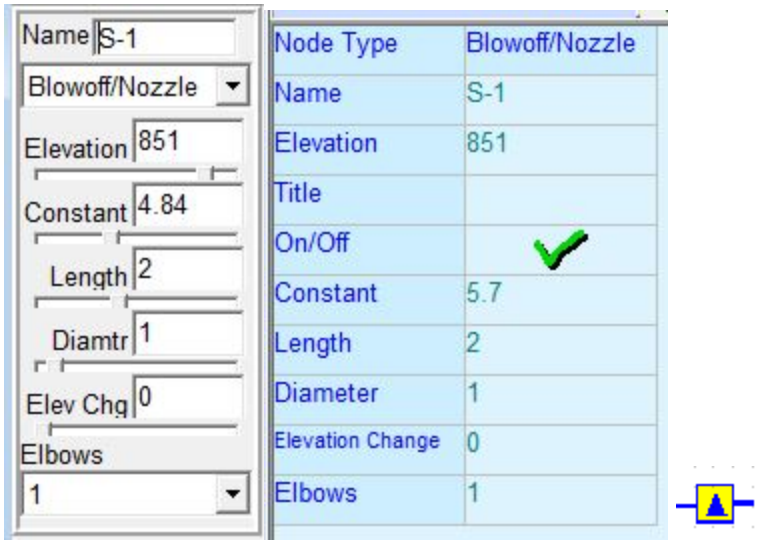
This effective diameter is used in substitution for ID (inner diameter) in any friction equations.

## 1.3.5.5 Blowoff Hydrant

Blowoff Hydrant

---

Blowoff / Hydrant Data




This element models a section of piping which discharges through an orifice to the atmosphere. It models a blowoff or hydrant. The element will normally be closed but is opened to flush the system or otherwise provide flow. The constant is defined by the relation:

$$Q = C \sqrt{P}$$

where Q is the flowrate in gpm (or l/s) and P is the pressure inside the opening in psi (or kPa). Using this definition the constant equals the flow in gpm for a 1 psi pressure drop.

The constant (C) must be provided. Tools in the main menu contains a Sprinkler/Blowoff Constant calculator.

**Blowoff/Hydrant Connection -** A connecting pipe to the sprinkler can be defined with the following entries:

Length (ft. or m)
Diameter (in. or mm)
Elev Chg. (ft. or m)

Elevation Change

A positive entry means the blowoff orifice is above the connecting pipe. For no connecting pipe, ignore these data items (entries = 0).

A blowoff is not considered in the Fireflow and Hydrant Analysis.

## 1.3.5.6 Check Valve

Check Valve

---

These valves allow flow only in the specified direction. If conditions exist for flow reversal, the valve closes and the line carries no flow. Check valve locations and allowed flow directions are specified in the input data. There are some restrictions on the placement of check valves which are noted as various components are discussed. The use of check valves can result in additional trials to converge. Therefore, it is recommended that you utilize them only when necessary. Check valves are automatically incorporated in pumps to prevent flow reversal through the pump.

Simple Check Valves in KYPipe
Check Valves in Surge

## 1.3.5.7 Closed Loop Systems

Even if a loop is closed, all models require a supply, such as a Reservoir or Tank element, which sets the pressure in the system. No flow needs to go into or out of this supply. It usually modeled as a branch off the loop.

When doing a closed loop system, a beginning and terminal end may be modeled as reservoirs. Regulators controlling flow, FCV-1s, or Loss Elements are often

used to model heat exchangers.  Short pipes with high resistance is another options.

There is a slight difference between chilled water system and a cooling water system.  The chilled water system uses the same reservoir Grade for beginning and terminal ends.  In a cooling water system, the cooling is represented by two reservoirs (beginning and terminal) at different grades.

## 1.3.5.8 Compressors

Compressors

---

Modeled with the Pump element in Pipe2018.  Compressors may be described by useful power or by operating data. Useful power refers to the actual power which is transformed into an increase in the pressure of the gas as it passes through the compressor. This may be computed from the compressor pressure ratio p2/p1, the flow W in lb/sec, and the ratio of specific heats k from hp = (WRT/550)(k/(k-1))[(p2/p1)^((k-1)/k)-1]. Alternately, a compressor may be described by operating data. If the compressor cutoff pressure P1 and two additional points of operating data (pressure and flow) are entered as an ID (see Pump Data), the program will fit a curve to this data. Note, if flow reversal occurs with a compressor described by operating data, that compressor will operate at the cutoff pressure. This would indicate that the compressor cutoff pressure is not adequate to overcome the system pressure.

If the compressor discharge pressure is to be specified or is known, then a compressor may be taken to be a fixed pressure node. If the compressor is on an external line, this fixed pressure node is simply a supply reservoir. If the compressor is on an internal line (a booster compressor), then this may be modeled as a pressure regulator with the regulated pressure being the discharge pressure of the compressor.

If a compressor of unknown size is to be selected to input a specified amount of gas into the system, the compressor location can be represented as a junction node. The desired inflow can be specified at this node and the program will compute the compressor discharge pressure which would be required to produce the specified inflow.

## 1.3.5.9 End Nodes

End Nodes

See also Data Requirements

---

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 are accepted.

End Node Types - These include the following node types:

1. Junction -

A connection of one (dead end junction) or more pipe links. See also Junction Data.

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. See also Reservoir Data.

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. See also Tank Data.

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 (either static pressure, residual pressure and flow, or multiple pressure-flow data points). See also Pressure Supply Data.

5. Sprinkler (Pressure Dependent Outflow) -

A connection of one or more pipe links to a point where flow is discharged into the atmosphere 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. See also Sprinkler Data.

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. The pumps characteristics may be defined by constant power, a rated condition, head-flow data, or a pump file. See also Pump Data.

7. Loss Element (Directional) -

An element identical to a pump except instead of a head gain, a head loss occurs. See also Loss Element Data.

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. See also Regulator Data.

**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.

**9. Active Valve -**
A valve for which the open ratio may be specified and for which the head loss based on the valve opening is included in the network analysis. SeeActive Valve Data.

## 1.3.5.10 Flow Control Valve (FCV)

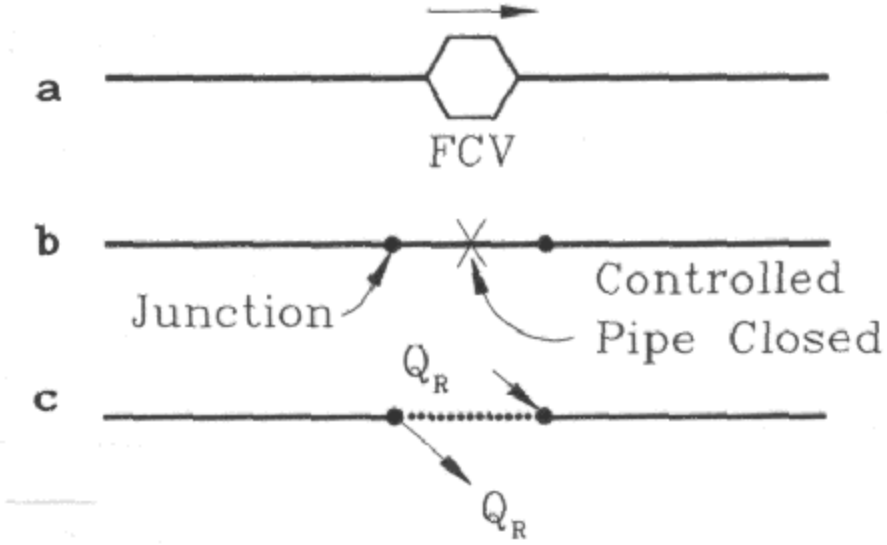
Flow Control Valve (FCV)

See also Regulator Data

---

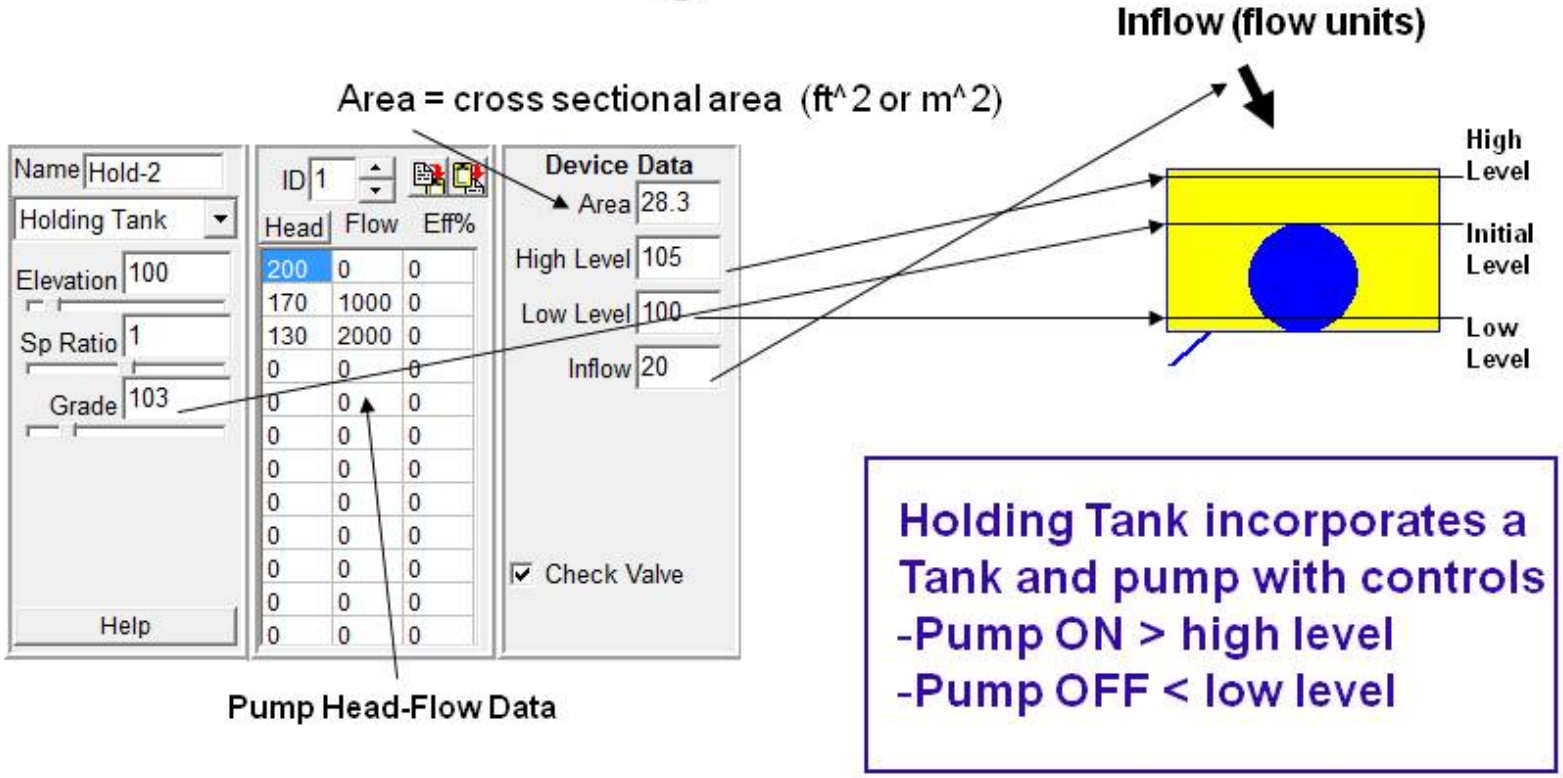
A flow control valve, designed to produce a designated flowrate, is modeled as depicted in line "b" below, by locating a closed line segment at the valve location with a junction node positioned on both sides of the flow control valve. Within KYPIPE the set flowrate, QR, is imposed as a demand at the upstream junction and an inflow (negative demand) at the downstream junction (line "c"). The analysis will calculate the pressure at both nodes and, therefore, the pressure drop required to produce the designated flow. If the upstream HGL falls below the downstream value the designated flow demands

are removed and the line segment opened. This will model the situation where the flow control valve is unable to maintain the designated flow. You also may choose to model an abnormal situation where the set flow is maintained under all conditions.




## 1.3.5.11 Holding Tank

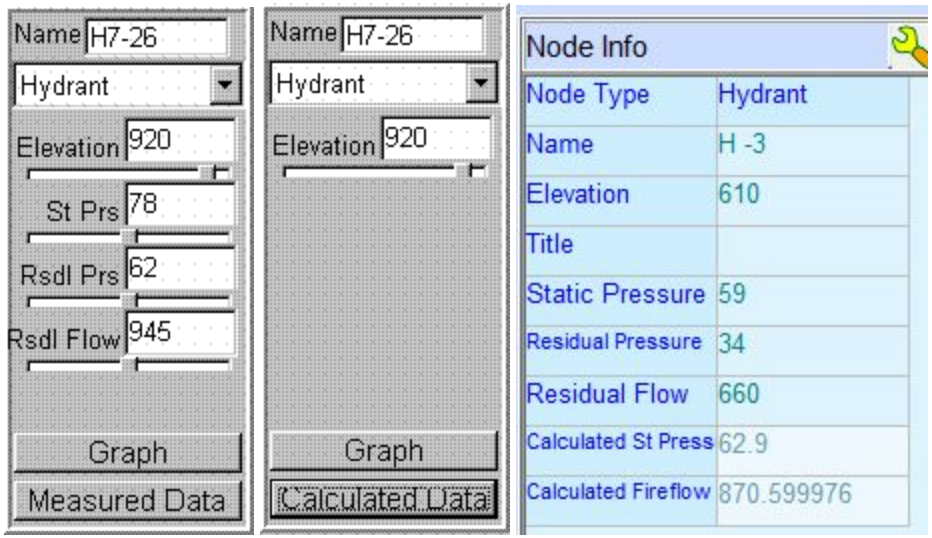
# Holding Tank




Elevation = elevation of pipe connection (ft or m)
Sp Ratio = pump speed ratio (actual rpm/rated rpm)

## 1.3.5.12 Hydrant Data

Hydrant Test Data and Fire Flow Plots




See also Hydrants, Fireflow and Flushing

---

***What is a Hydrant?***

A hydrant is an internal node which models a fire hydrant. Test data can be provided and plots of the test data of one or multipe hydrants can be obtained.

**Elevation -** Elevation (ft or m) of the hydrant

**St Prs (Static Pressure) -** Static pressure measured in a field test. (User reference data, not used in hydraulic calculation).

**Rsdl Prs (Residual Pressure) -** Residual Pressure measured in the field during a flow test of this hydrant. (User reference data, not used in hydraulic calculation).

**Rsdl Flow - Residual Flow -** Residual Flow measured in the field during a flow test of this hydrant. (User reference data, not used in hydraulic calculation).

**Graph -** This will create a graph of hydrant test or calculated flow data. See Fire Flow Graphs for detailed information.

**Measured Data -** This allows the user to input field test data for reference or for graphing the fire flow based on test data. This option shows the fire flow based on test data in the Node Results box.

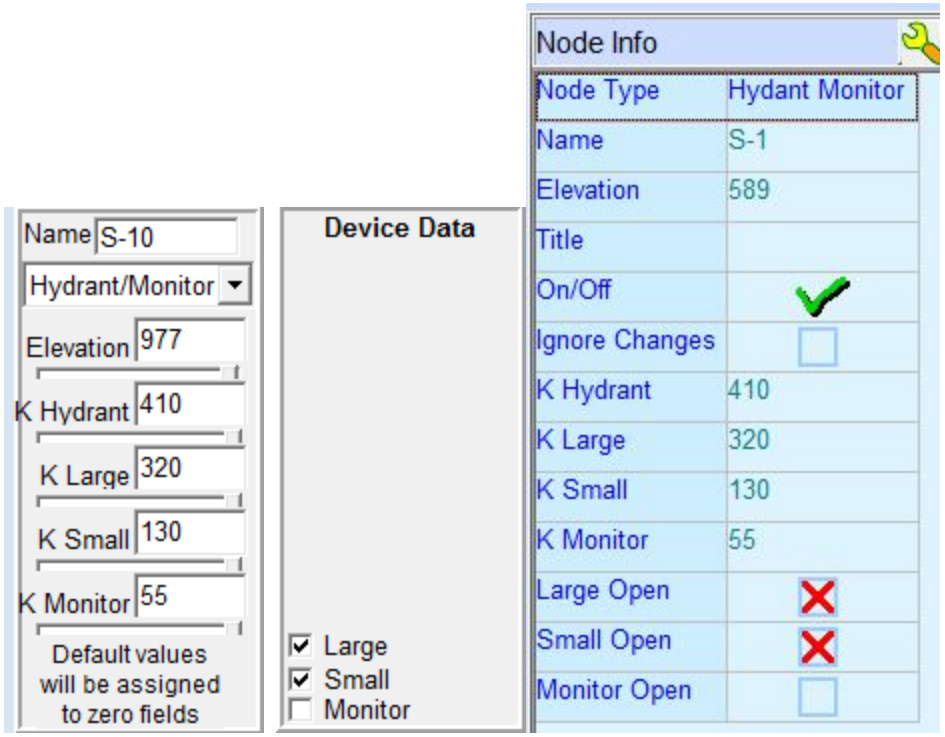
**Calculated Data -** This option removes the test data fields so Pipe2018 model fire flow calculations are graphed and displayed in the Node Results box.

## 1.3.5.13 Hydrant Monitor

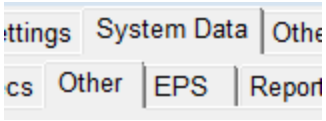
The Hydrant/Monitor models a device that has a monitor which is a rotating nozzle. The nozzle sprays water in a circular pattern - used primarily in refineries or chemical plants. In addition to the nozzle is a large and small orifice for connecting a fire hose. As shown in the data input shown below, up to 3 flows can be provided (monitor, small and large orifice) .

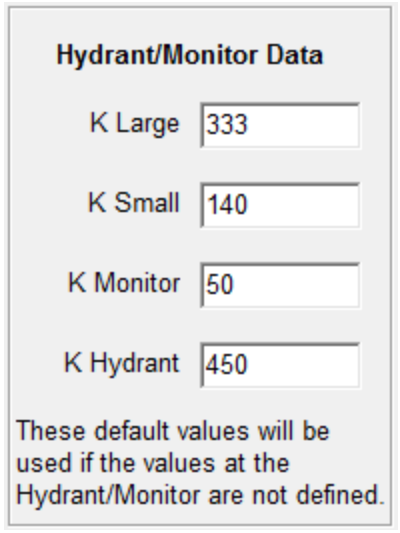
Hydrant K - accounts for losses due to the hydrant piping. This K value is combined with each of the other constants to determine the flow through the various orifices.

Hydrant/Monitor element allows the monitor to operate while connecting a hose to one of the other connections. The Hydrant/Monitor may be analyzed with up to three valves open simultaneously. The K values for the Hydrant/Monitor may be set at each individual hydrant using the Node Information Data box. Then the valves that are open may be selected in the Device Data box as shown:




Or go to Other Data (Classic - System Data | Other) to set default values as shown below. Any K values left blank in the Node Data box will default to the values set in this screen:






The hydrant monitor operates as open, discharging to atmosphere, in the KYPipe analysis if one or more of the boxes, Large, Small, or Monitor, is checked.

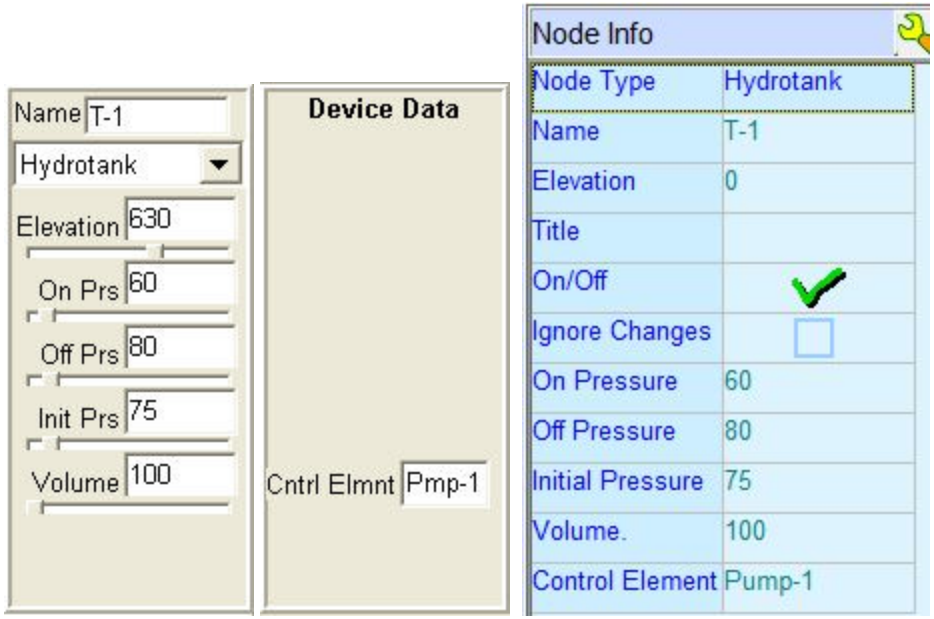
The Hydrant Monitor is NOT treated like a hydrant in the FireFlow and Hydrant Analysis.

Example: Assuming the Hydrant Montor shown below uses the above-referenced default K values. Only the Monitor is open. The flow out of the monitor is shown.

The flow out the monitor is Q = K(p).5 where K is the combined hydrant & monitor K. Here Q = 628 gpm and K = around 50. The Hydrant K of 450 is added to that of the Monitor, but it doesn't have much effect.

## 1.3.5.14 Hydrotank

Hydrotank




---

Hydropnuematic or pressurized tank. Uses air pressure to supply pressure to move water out of tanks. When low, water is added to build up pressure. For this element an external pump is specified which operates to fill the tank when the pressure falls below a certain level and turn off when above a certain pressure level.

Elevation - Elevation of the tank

On Prs - Pressure (psi or kPa) at which the Control Element (e.g. pump) turns on to begin filling hydrotank

Off Prs - Pressure (psi or kPa) at which the Control Element turns off

Init Prs - Pressure (psi or kPa) in the tank at the beginning of the simulation

Volume - Units depend on Flow units for the model.

| **Flow Units** | **Volume Units** |
|---|---|
| CFS | cubic feet |

| | |
|---|---|
| GPM | gallons |
| MGD | gallons |
| l/s | liters |
| CMS | cubic meters |
| l/min | liters |
| lb/s | cubic feet |
| BPH | cubic feet |
| kg/s | cubic meters |
| USER English | cubic feet |
| USER SI | cubic meters |

Cntrl Elmnt - Name of the element used to fill the tank, e.g. pump (pressure supply, reservoir, tank).

For EPS it is best to use a very small time step for hydrotanks which cycle often.

## 1.3.5.15 Hydrotank Using Tank Element

Hydrotank using Tank element

---

Hydropneumatic Tank

How to model a hydropneumatic tank in Pipe2018.

For a regular simulation this tank is just a fixed grade node (FGN) with the grade equal to the water level plus the pressure head.

For an EPS this tank should be modeled as an equivalent standpipe with the lowest level at the low water level plus the low pressure head. The high level is the high water level plus the high pressure head. The tank diameter is determined using the total water volume from high water to low water level and determining the equivalent standpipe diameter with this amount of water over the overall high to low level.

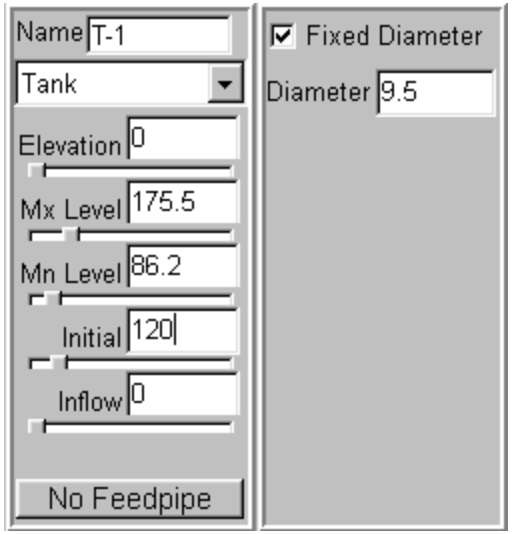
Example

20 foot diameter tank.

low water level = 40 feet, low pressure = 20 psi. Therefore low overall level = 40 + 20(2.31) = 86.2
high water level = 60 feet, high pressure = 50 psi. Therefore high overall level = 60 + 50(2.31) = 175.5

water volume = (60-40)*area = 6280 cubic feet
equivalent standpipe height = 175.5-86.2 = 88.3 feet
equivalent standpipe diameter = 6280/88.3 = area. Thus the equivalent standpipe diameter = 9.5 feet

The data would be entered as follows:




| Node Type | Tank Fixed Diamete |
|---|---|
| Name | T-1 |
| Elevation | 0 |
| Max Level | 175.5 |
| Min Level | 86.2 |
| Initial Level | 120 |
| Inflow | 0 |
| Title | |
| Diameter | 9.5 |

## 1.3.5.16 Inline Meter

Inline Meter

---

An Inline Meter is an Internal Node which can be designated for any pipe. For an EPS simulation, these meters produce a tabulation (report) of the total volume of flow passing through that pipe during the EPS.

See Internal Nodes

## 1.3.5.17 Internal Nodes

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.

When adding pipe segment and a node a starting node (existing) is selected and the mouse is RIGHT clicked on the ending node location. IF the starting node is now in-line (no more than two pipe segments attached), it is automatically turned into an intermediate node.

**NOTE** - Right or left click AGAIN and the newly created node will NOT automatically be made into an intermediate node if it will subsequently be made an in-line node, it will remain as a junction.

Internal Node Types - These include the following node types that are situated at a connection of two pipe segments with identical basic 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. On/Off Valve -

Indicates location of on/off valves. To close a line, these may be selected and set to 'off' by clicking the on/off button at the top of the Node Information window. 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 use the active valve element.

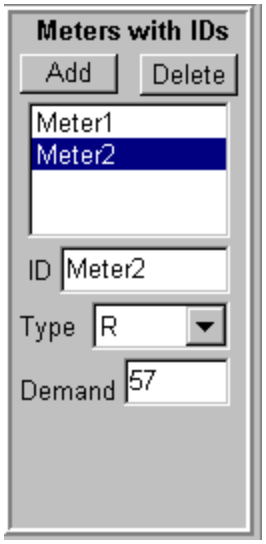
3. Hydrant -

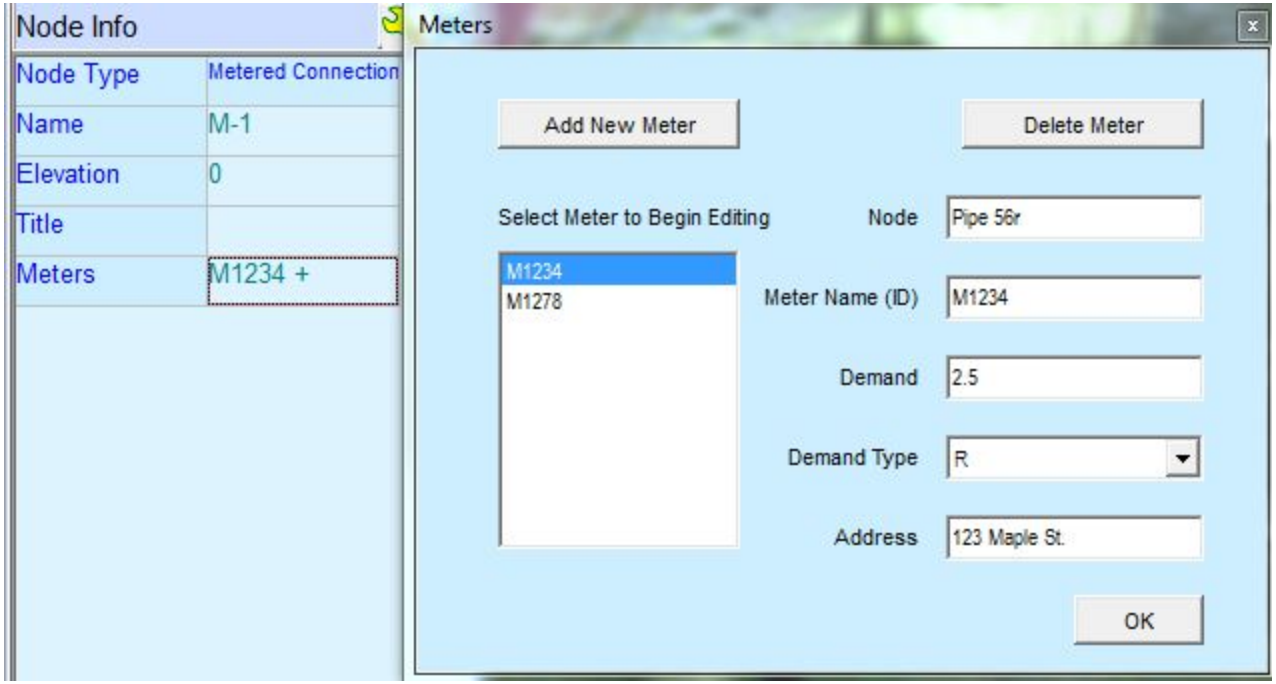
Indicates location of fire hydrants. See Hydrants.

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. See also Metered Connection Data.

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.

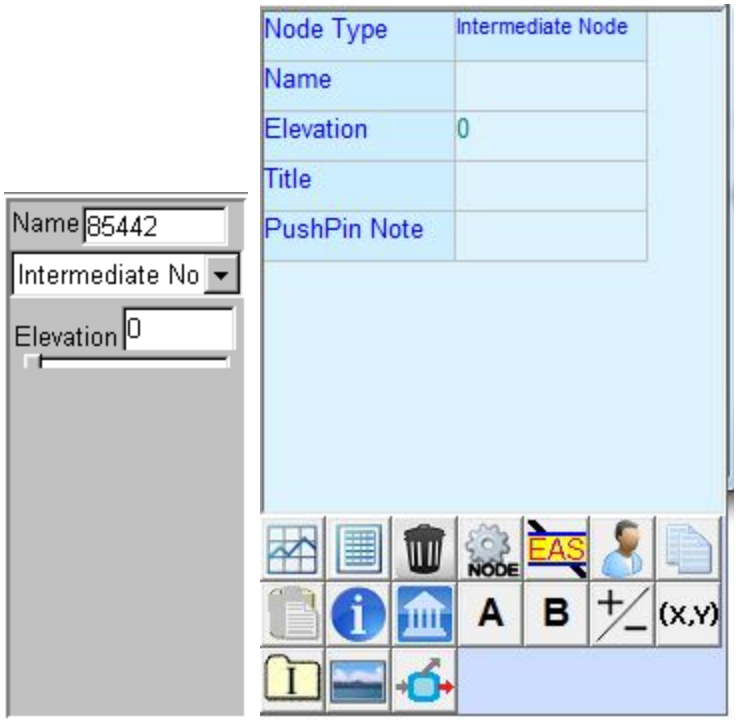
7. Customized Device -

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

## 1.3.5.18 Internal Node Basic Data

Internal Node Basic Data

---




Back to Internal Node Data

## 1.3.5.19 Internal Node Data

Internal Node Data

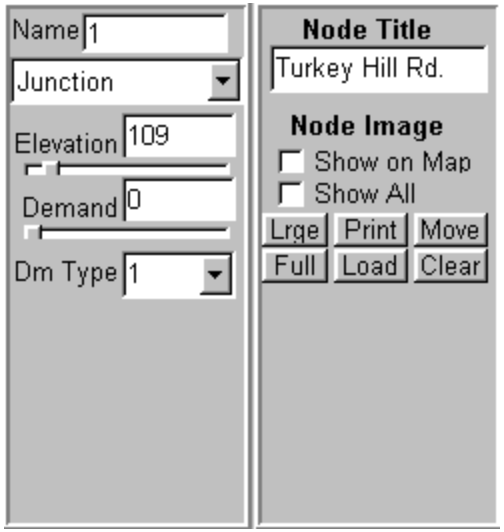
---

There are 2 data categories with a total of 3 windows common for all internal node types. There is one additional INFO window for Metered Connections. The specific window groups are accessed using the buttons indicated.

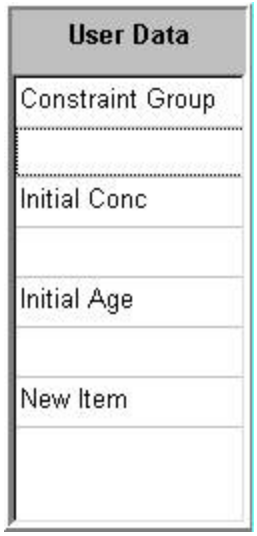
COMMON WINDOWS

INFO **Basic Data**

Node Title / Image




USER User Data



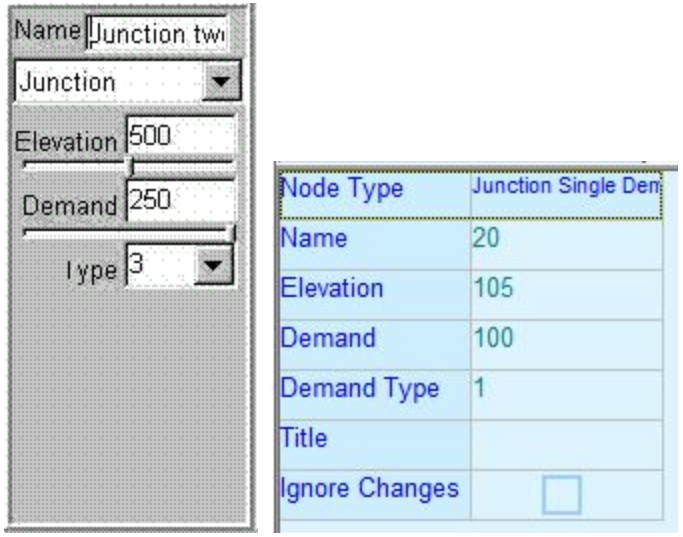

INDIVIDUAL WINDOWS

INFO Metered Connection Data

In KYnetic, the data is either already visible in the Node Info window or can all be accessed by one of the icons at the bottom.

## 1.3.5.20 Junction Data

Junction Data

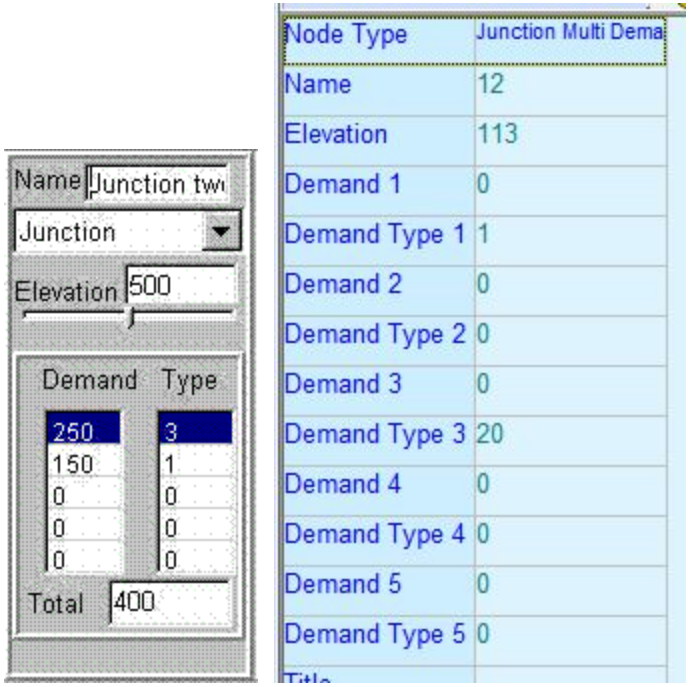



## Single Demand Type

***What is a Junction Node?*** **-** A junction node is an end node where there is a connection of one or more pipe links. For connections of two pipe links a junction node is required if the diameter or roughness changes or a demand is imposed.

**Junction Demand [specified units]** - The demand (consumption) imposed at this junction node in the specified units (noted top bar) for the demand type noted.

A single or multiple (up to ten) demand types can be utilized and this choice is user specified (under Preferences icon or System Data / Preferences (Classic) check the Multiple Demand types box for multiple demands as shown below).




**Multiple Demand Type**

**Demand Type** - An integer designation to group demands with identical patterns. Demand types generally classify the type of user (residential, commercial, industrial, etc.) but can represent any common property. The Demand Pattern is used to define multipliers for each demand type for the times (cases) covered by the simulation.

See Also:

- Demand Patterns
- Node Data Boxes
- Preferences

## 1.3.5.21 Libraries

Libraries

---

See also Loss Element Library

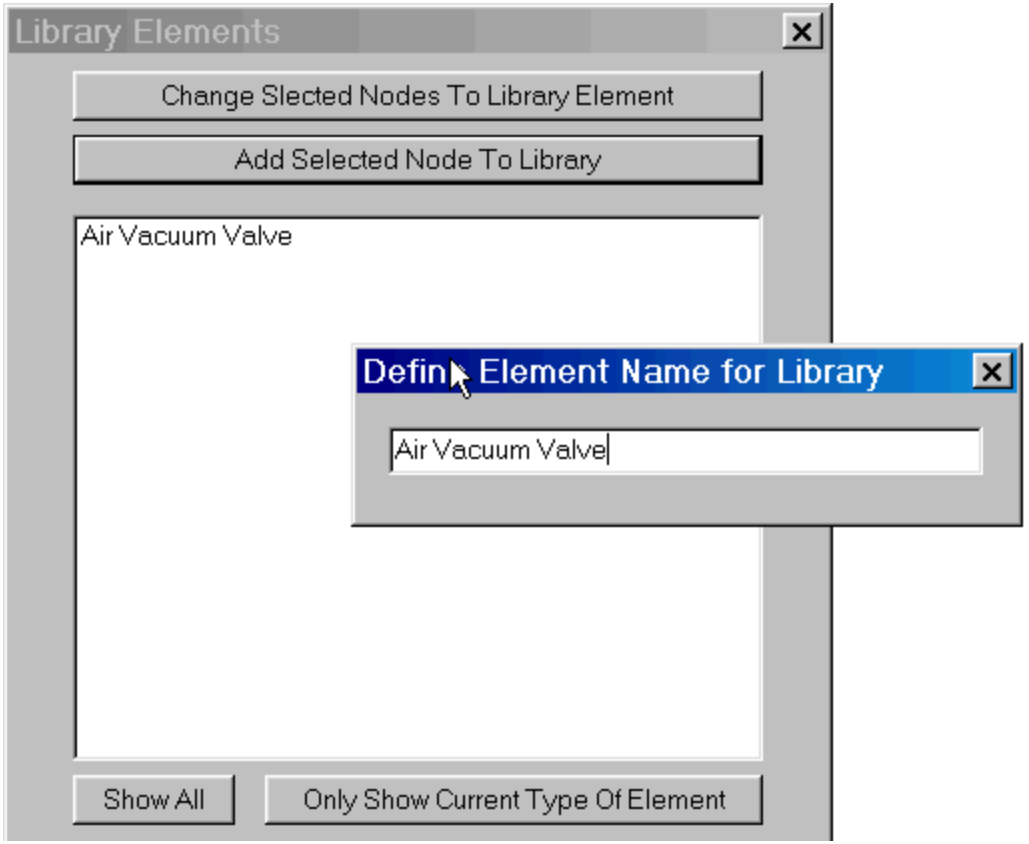
### How to Create a Pipe2018 Library

A library may be created for specific elements (such as air/vacuum valves).

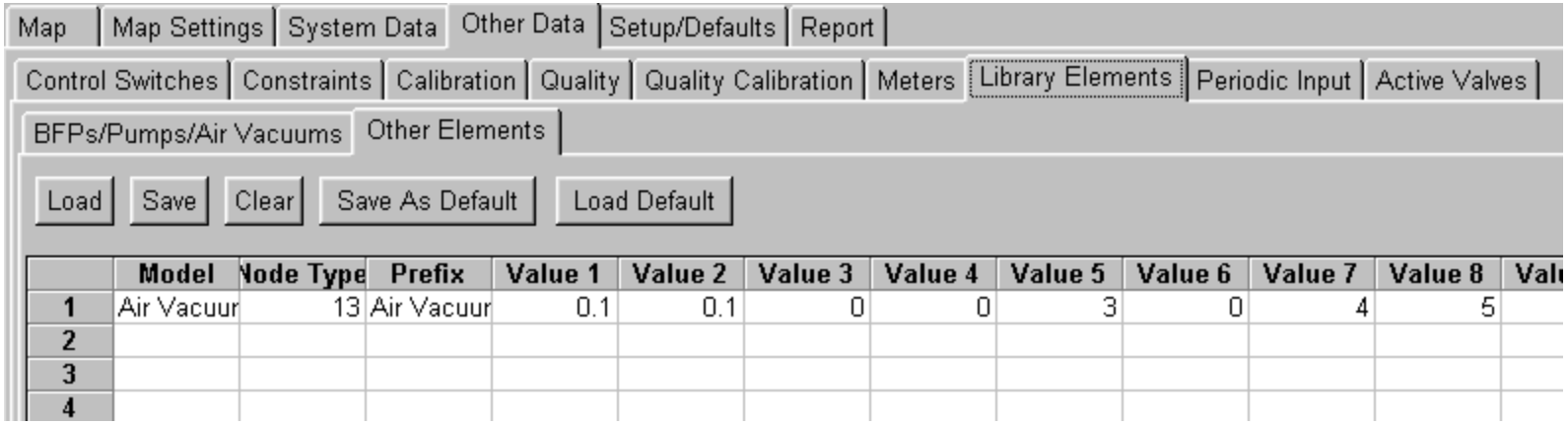
The following steps will allow you to create a library.

Before beginning, click on **or Other Data | Library Elements | Other Elements (Classic)**. Use the Save button to name a blank library.

1. In the map view, select the element for which the library will be created (such as a 3 stage air/vacuum valve)
2. Enter all the data for that element in the Node Information window..
3. Click on the Library symbol in the top right corner (Classic) or bottom (KYnetic) of the Node Info window. You will see the Library name. Click on it.
4. Click on **Add Selected Node to Library** and provide a descriptive title for that element such as a model name and number (Define Element Name for Library).



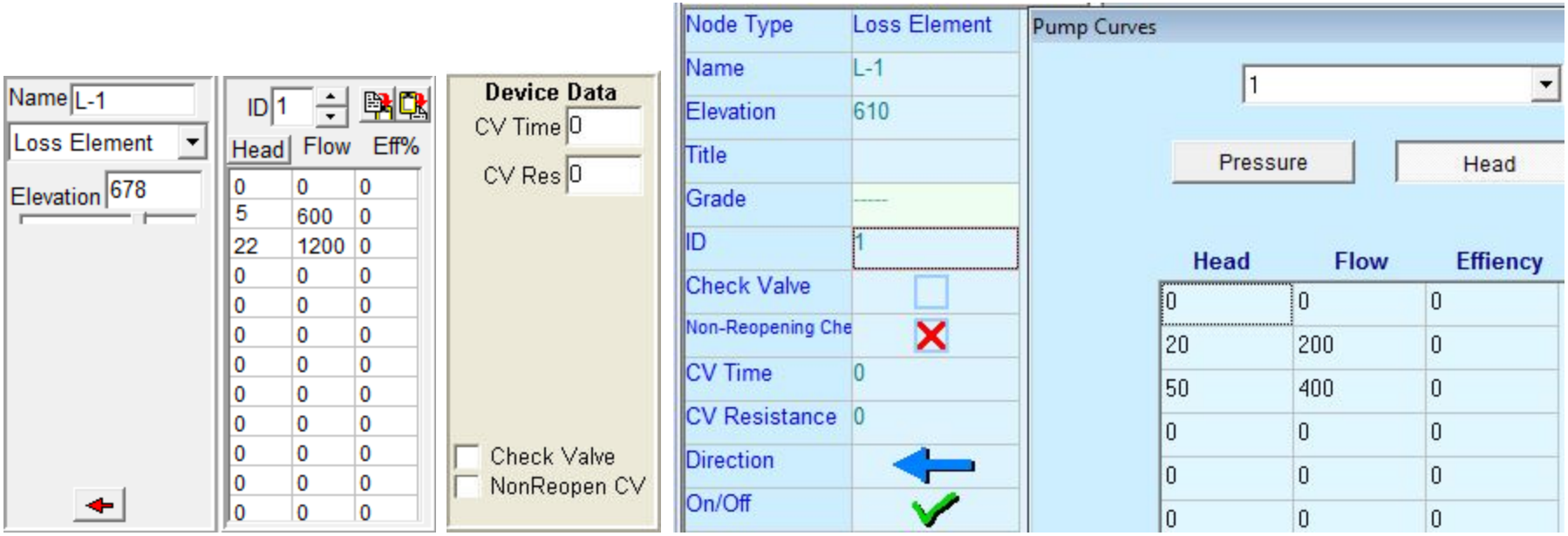

5. You have now created the Template for your element. You can repeat steps 2-4 until library is complete. This is the recommended approach. You may also directly enter all the desired data for this element into the library spreadsheet. Click on **or Other Data | Library Elements | Other Elements (Classic)**. Use the same columns as the data appears in the first row. If a number appears that you can't identify use the same number for all your elements (this is an internal key). Note that the information in the first column is what will appear in the library selection list so make this entry as descriptive as possible (you can use 20-30 characters if desired).

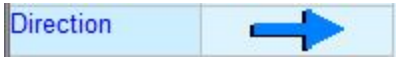



| | Model | Node Type | Prefix | Value 1 | Value 2 | Value 3 | Value 4 | Value 5 | Value 6 | Value 7 | Value 8 | Val |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | Air Vacuur | 13 | Air Vacuur | 0.1 | 0.1 | 0 | 0 | 3 | 0 | 4 | 5 | |
| 2 | | | | | | | | | | | | |
| 3 | | | | | | | | | | | | |
| 4 | | | | | | | | | | | | |

6. When you are done Save the file.

## 1.3.5.22 Loss Element Data




A loss element node is a directional end node where a head (pressure) loss occurs. The loss element operates on a head/flow curve based on data provided in a head/flow table. A loss element is a directional node and multiple pipe links may be connected to either side. The directional indicator and the connections must be consistent with correct operation. The direction can be switched using or 

and this can be done simply to improve the appearance of the model (so long as the operation is correct). **If no pipe links are connected to one side, this is assumed to be a reservoir connection and the reservoir HGL must be provided.**

**A note on calculations.** The headloss across elements changes with flowrate but typically the K value (minor loss) itself remain constant. This is generally true for losses over simple elements such as gate valves, elbows etc. However, loss elements such as heat exchangers do not follow this relationship (i.e., constant K value as in dH = K * (V^2/2g) ). The headloss vs. flow characteristics of loss elements are generally given by the manufacturer instead of single minor loss coefficient. Internally, Pipe2018 models a loss element as a pump which produce pressure drop across the element instead of pressure gain. Ideally, the user should provide multiple point data for a loss element, but Pipe2018 can build a curve internally if the first and last points are not specified in what is traditionally a three-point curve for a loss element. See below.

**Head/Flow Data Table**

A table of head (or pressure) - flow data which describes the operation of a pump, loss element or pressure supply. An entry for efficiency is also provided but this is used only for computing power costs for pumps and should be ignored for other applications. Note that there is a button to switch between head [ft. (m)] and pressure [psi (kpa)]. Each table has a unique integer identifier (ID).

**Loss Element ID**

An integer identifier for the head (pressure)/flow data table.

**Loss Element Data**

There are two options for entering head (pressure)/flow data for a loss element. Note the head (pressure) switch to select units for this entry. *Important*: Multiple data points should be entered in order of increasing flowrates. Flowrate is entered in your specified units (top bar). Also enter head (pressure) data in order of increasing head (pressure). Pipe2018 internally assigns a negative ( - ) to this data to indicate a loss and to satisfy the h1>h2>h3 data requirement. Efficiency data is not appropriate.

1. A single head (pressure) drop/flow point can be entered. This is used to generate a loss coefficient which results in a loss proportional to the square of the flow. If you look at the pump report setion of the output Report, you can see the other two points generated by the KYPIPE program for each of the loss elements modeled.
2. Three or more data points can be entered and a head loss/flow curve generated for the model. ****It is best to make the first point a cut-off point (0,0).**

Note that any entries for head (pressure) are assumed to be negative internally (represent a head loss).

**Device Data**

Check Valves- Pumps, active valves, and loss elements all can be equipped with a check (non-return) valve to prevent backflow. Either a normal check valve which will close or open depending on conditions or a non-reopening check valve which will close only once and remain closed can be designated. The closing time (CV Time) is the time it takes for the check valve to close once closing is initiated. The check valve resistance (CV Res) is the resistance (head/flow^2) when the valve is fully open.

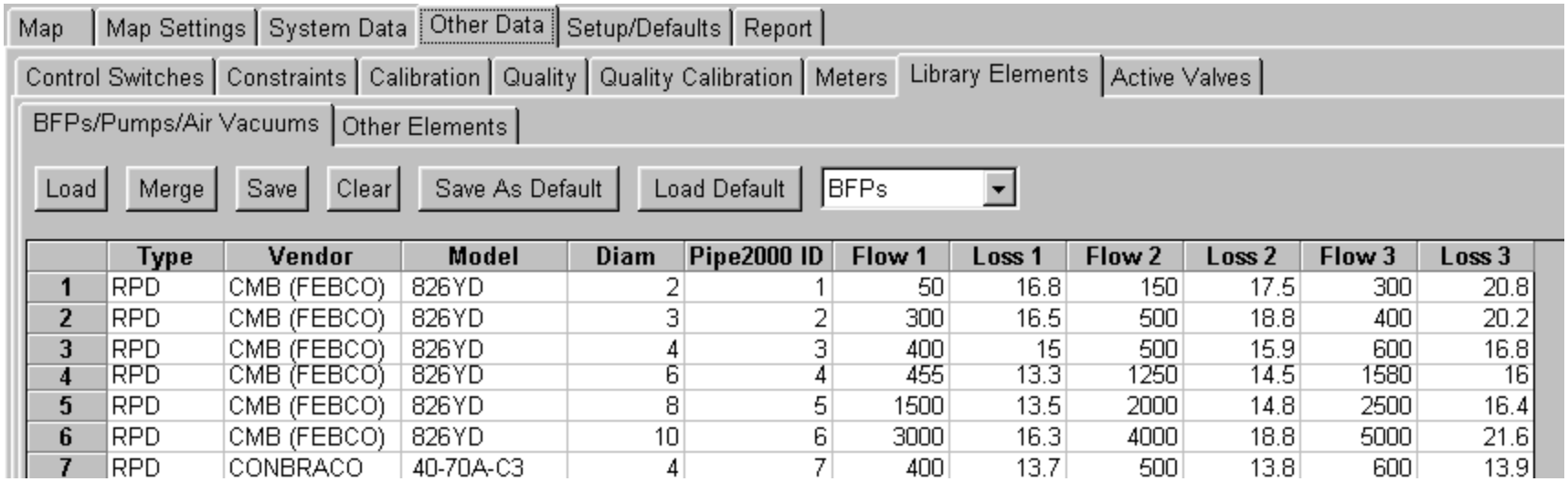
Minor Loss Components in Pipes versus Loss Elements

Both approaches allow for variation in flowrate (dictated by system hydraulics) across the element. In one case the program uses dH = K * (V^2/2g) relationship and the other case dH = A + c*Q^n where A, c, and n are constants obtained by fitting an exponential equation for the three point headloss vs. flowrate data. If A were zero and n is 2 in the power fit of headloss vs flowrate curve then both equations are basically identical and should produce identical results. If A is not zero and n is not equal to 2 then the results would be different.

## 1.3.5.23 Loss Element Library

Loss Element (LE) Library

In KYnetic, use the Components menu icon:




| | Type | Vendor | Model | Diam | Pipe2000 ID | Flow 1 | Loss 1 | Flow 2 | Loss 2 | Flow 3 | Loss 3 |
|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | RPD | CMB (FEBCO) | 826YD | 2 | 1 | 50 | 16.8 | 150 | 17.5 | 300 | 20.8 |
| 2 | RPD | CMB (FEBCO) | 826YD | 3 | 2 | 300 | 16.5 | 500 | 18.8 | 400 | 20.2 |
| 3 | RPD | CMB (FEBCO) | 826YD | 4 | 3 | 400 | 15 | 500 | 15.9 | 600 | 16.8 |
| 4 | RPD | CMB (FEBCO) | 826YD | 6 | 4 | 455 | 13.3 | 1250 | 14.5 | 1580 | 16 |
| 5 | RPD | CMB (FEBCO) | 826YD | 8 | 5 | 1500 | 13.5 | 2000 | 14.8 | 2500 | 16.4 |
| 6 | RPD | CMB (FEBCO) | 826YD | 10 | 6 | 3000 | 16.3 | 4000 | 18.8 | 5000 | 21.6 |
| 7 | RPD | CONBRACO | 40-70A-C3 | 4 | 7 | 400 | 13.7 | 500 | 13.8 | 600 | 13.9 |

---

**BFPs/Pumps/Air Vacuums**

The Library Elements editor is a comprehensive list of Back Flow Preventers which the user references when choosing a BFP for their system. The most important consideration when choosing a BFP within Pipe2018 is the units for the flow and loss data. In the BFP editor in the default data table, all values are in English units, ft for loss and gpm for flow. Therefore, to correctly use any of the BFP selections provided in Pipe2018, the Pipe2018 data file must be using English units. In order to use a BFP in a file using SI units, the user should enter the data in SI units (m for loss, l/s for flow) directly into the BFP editor spreadsheet. When entering data into the editor, the user may create and save a separate data spreadsheet.

**Other Elements**

This portion of the editor is a writable spreadsheet identical to the BFP spreadsheet. Values may be added, changed, and deleted. Separate data sets may be created and saved and even entered as the default.

See End Nodes.

## 1.3.5.24 LPS Tank

Low Pressure Sewer (LPS) Tank

For KYPipe EPS and Surge analysis.

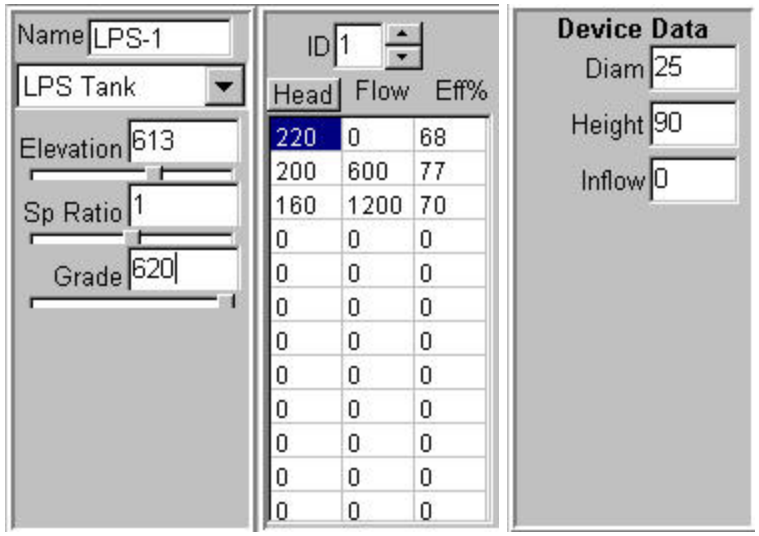
Low Pressure Sewer (LPS) tanks are for sewer (or water) applications where gravity flow is not possible and each service point requires storage (holding tank) and a pump to feed into the network. LPS tanks are constant diameter tanks with an ID Pump (variable speed) situated in the bottom of the tank. The pump is functionally located between the tank and the network.

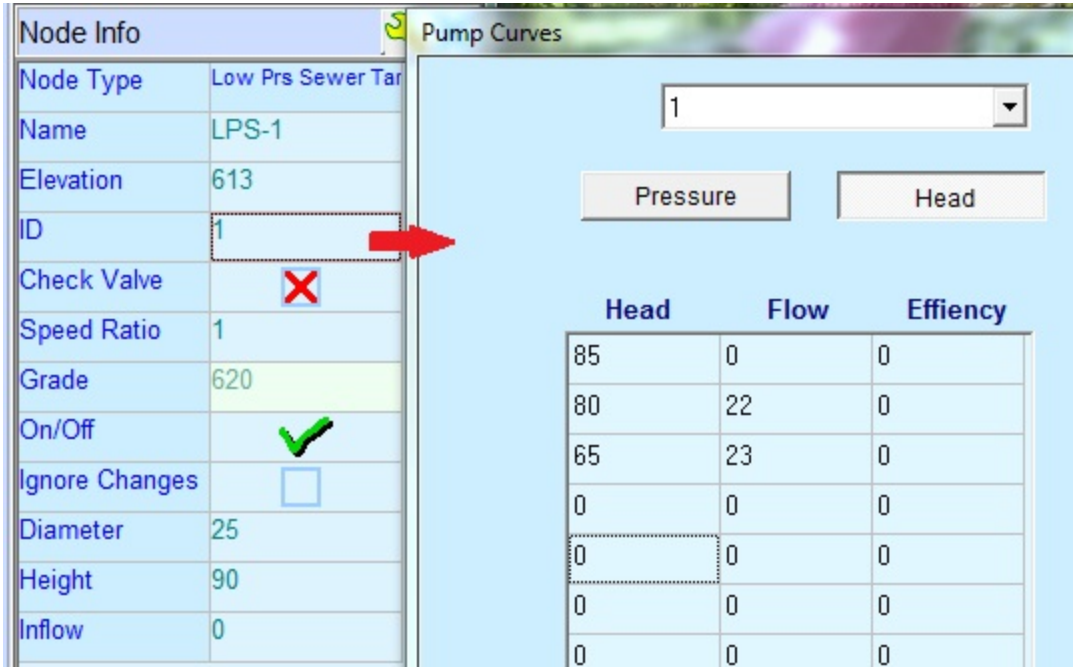
This is a discharge-only element. The pump empties the tank INTO the network. The results which are available include outlet pressure, head, and flow. The 'holding' tank is filled from an outside source - flow specified in the Inflow data field.

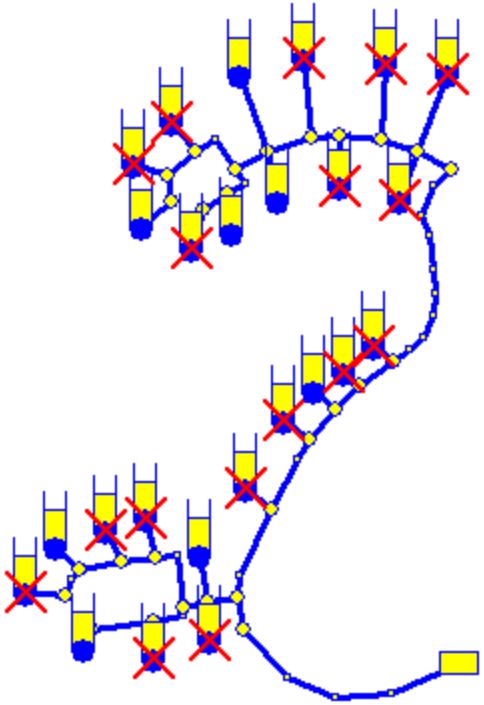
For the tank, instead of maximum and minimum elevations as with regular tanks, the Height (ft or m) of the tank above the applied elevation is defined, along with Diam(eter) (ft or m). The Grade (ft or m of elevation) of the tank is the initial water level elevation in the tank.

For the internal pump, the pump speed ratio and the pump ID are defined.

To view tank levels for LPS tanks, go to Report and view the Tank Report.



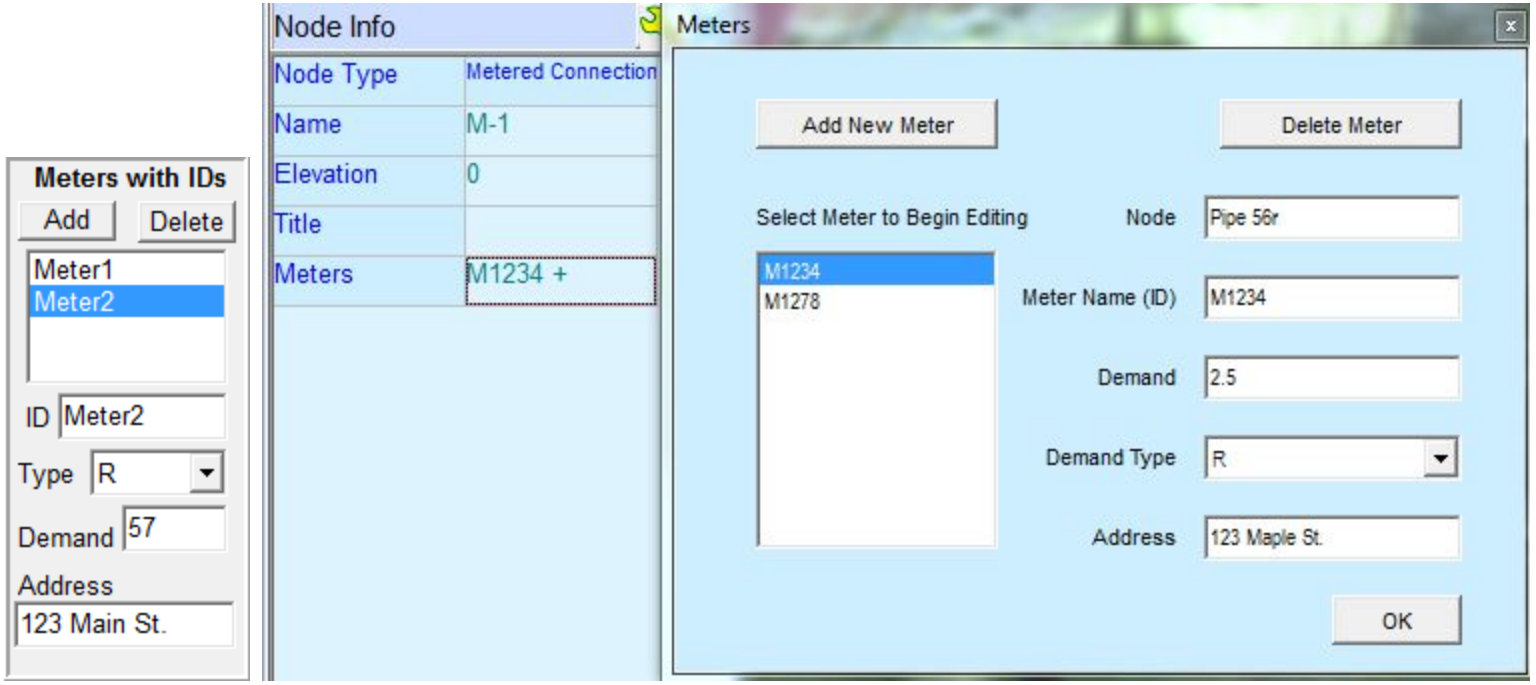

Node Info
Node Type
Low Prs Sewer Tar
Name
LPS-1
Elevation
613
ID
1
Check Valve
Speed Ratio
1
Grade
620
On/Off
Ignore Changes
Diameter
25
Height
90
Inflow
0
Pump Curves
1
Pressure
Head
Head
Flow
Effiency
85 0 0
80 22 0
65 23 0
0 0 0
0 0 0
0 0 0
0 0 0



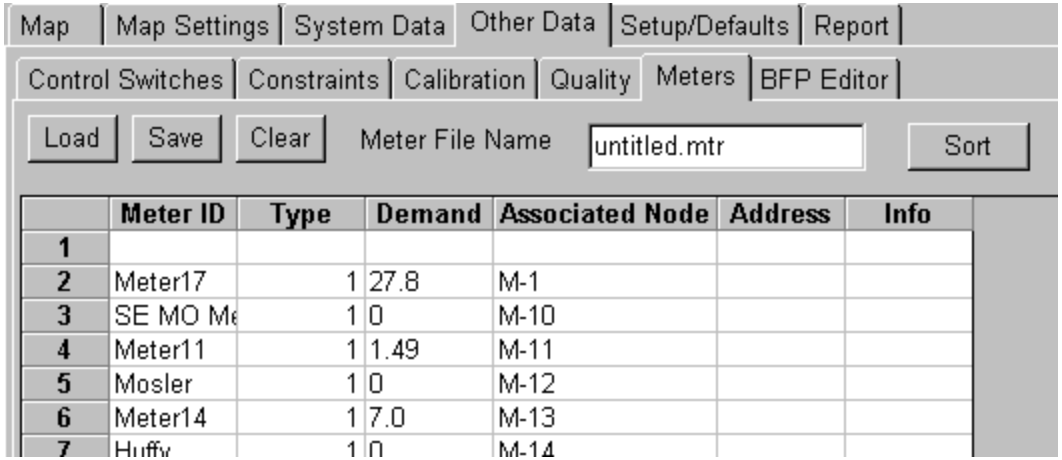
Typical Low Pressure Sewer System.

## 1.3.5.25 Metered Connection Data

Metered Connection Data




KYnetic Components menu icon:




See the Meter video on the KYPipe web site.

---

Metered Connection Node

This is an internal node where individual metered connections may be identified by ID. A demand and demand type is assigned to each meter. These demands are allocated to adjacent junction nodes. The allocation is based on the location within the pipeline. Thus demand is proportionally allocated, more demand going to the nearer junction, less to the further one. The Metered Connection Data is stored in a Meter Record File which can be updated externally. To view a **Meter Data Table**, go to Components menu - Meters (KYnetic) or Other Data | Meters (Classic).

Add - Click here to add a meter to list below.

Delete - Click here to delete

Selected Meter (list) - Select a specific meter from the list.

ID - Provide an ID for the meter. The ID is the name of the meter in the list (hit Refresh if it does not appear in the meter list). Note that the NODE NAME for the meter (e.g. 'M-1') is independent of the associated meters connected to that meter node.

Type - Select demand type (or key in type)

Demand - Enter demand for meter in specified flow units.

Address - For user reference, the address of the meter location may be entered.

**Metered Connection Table**

This table contains the information for metered connections. The data is stored in an Excel format file which can be saved and re-loaded, and also generated or updated externally. With this feature it is possible to use **meter record data** to generate the meter record file and update your model.

Each Metered Connection in a model can have several associated meters, each with it's own demand, demand type, name, and user data (e.g. address). Each

row of data in the Meters table refers to an individual meter associated with a Metered Connection in the model. The data columns are as follows:

**Meter ID** - This the meter name/title. This will appear as the meter name in the **Meters with IDs** box for the associated Metered Connection (see Associated Node below). Please note this in not the Node Title. The Node Title applies to the Metered Connection, not the individual meters associated with a Metered Connection.

**Type** - refers to the demand type specified by the user, 0-9.

**Demand** - this is the demand in the user-specified flow units.

**Associated Node** - This is the Metered Connection with which the individual meter is associated. This node must be an existing node in the Pipe2018 file. When the Metered Connection is selected in the map screen, the data for the individual node will appear when selected in the **Meters with IDs** box (scroll through the boxes with the pointing fingers if this box doesn't appear in the Node Information window).

**Address** - This is user information. It appears when the individual meter is selected in the **Meteres with IDs** box in the Node Information window.

**Info -** This is user information with only appears in the Meters table (Components menu or Other Data | Meters).

**Data Import/Entry.**
There are several ways to input data to the Meters table. Besides simply keying in data, the one of the simplest ways to import data is by copying and pasting blocks of data from an Excel file into the Meters table.

To import an existing Excel file (assuming the data is in the same format as the Meters table), rename the Excel file with an .mtr file extension, then load it (use the Load button and browse for it) into the Meters table (Other Data | Meters) in Pipe2018.

To create a new Excel file for the purpose of importing into Pipe2018, first go into Other Data | Meters and save a new file in the meter table (blank or with some data keyed in). Open the .mtr file in Excel and edit. You will likely be given text conversion information. Once done editing, save and exit Excel, then load the mtr. file in the Meters table.

## 1.3.5.26 Meters

There are several types of meters associated with Pipe2018

Residential Meters - a number of connections associated with a pipe, all with the same demand.

Metered Connection Data - may be used to incorporate meter records. It is a location within a pipe which may represent multiple meters.

In-line Meter - Inserted in a pipe to track total flow at that location.  Results appear in the Report.

For water utility modeling one of the most time consuming tasks is to allocate demands for junction nodes. Demands represent the metered water distribution throughout the system and these are typically combined and associated with junction nodes. Pipe2018 incorporates meters into the model and the logic to allocate the associated demands. The quickest and most simple approach is to provide as input data the number of residential meters connected to each pipe link and the average value for the Residential Meter Demand (System Data/Other) which represents an average value per resident. In addition, data for individual metered connections at any location can be provided. Pipe2018 automatically allocates the demands associated with meters to the adjacent junction nodes.

## 1.3.5.27 Minor Loss Components

Minor Loss Components

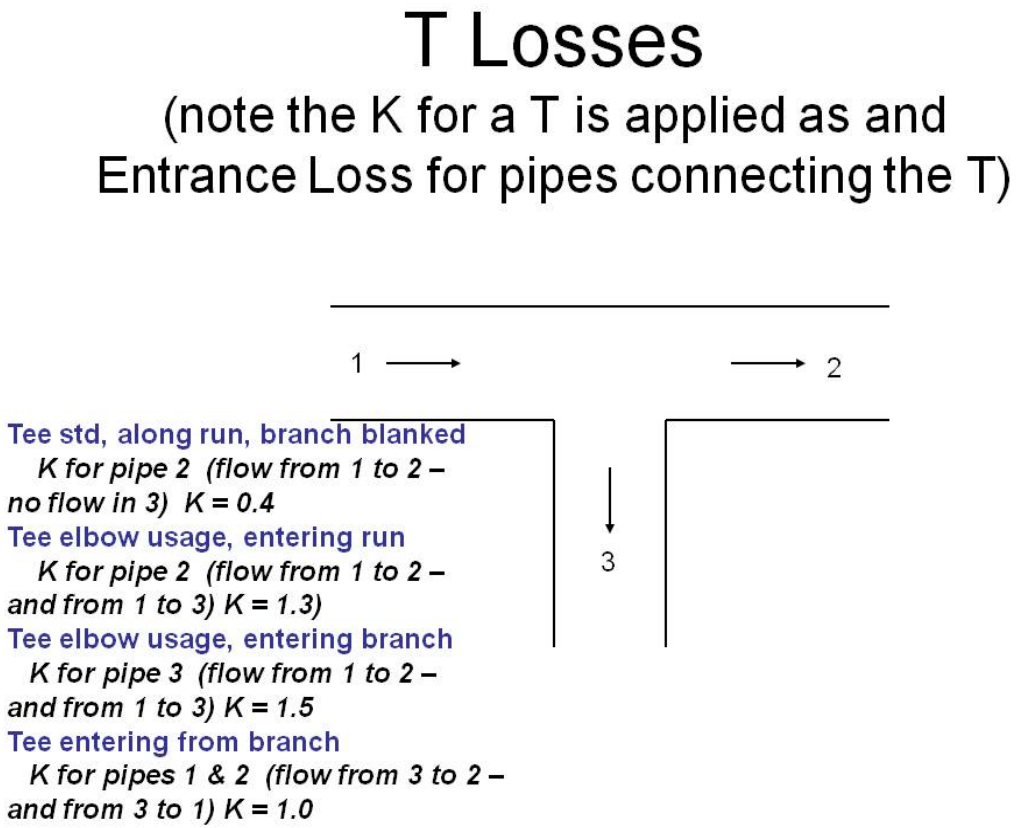
---

A number of components in a pipe system (such as valves, junctions, bends, meters, etc.) produce a head loss which may be substantial and should be included in an analysis of the flow distribution of that system. The need to include such losses depends on the relative importance of these losses compared to the line losses and this judgment must be made by the user. These losses are included by using the concept of a minor loss coefficient (K) which is a non-dimensional term which multiplies the velocity head to give the concentrated head loss at the component. Hence, the loss is given by:
hLM = Sum K V*V /2g

where hLM is the head loss in feet (meters) head, V is the line velocity in ft/s (m/s), Sum K represents the sum of all the minor loss coefficients for that pipe and g = 32.17 ft/s2 (9.807 m/s2 ). The minor loss coefficient may vary somewhat with flow conditions but it is usually sufficient to consider this to be a constant for a certain component. KYPIPE uses a single data entry for each pipe section for Sum K to incorporate minor losses and some representative values of K which may be used for common fittings are given in the Minor Loss Coefficients Table.

It is often necessary to compute a value for K from data (observed or furnished by the manufacturer) for a particular component. If the pressure drop across a component is known for a specific flow, the value of K is easily computed.

If a single value for K does not adequately represent the head loss-flow relationship for a component, it may be necessary to input several values of head loss-flow and utilize a curve fitted to this data. KYPIPE has a special component for this approach (Loss Element). For this application, the data consists of 3 pairs of head loss - flow points.




## 1.3.5.28 Network Elements

Network Elements

See the Elements video on the kypipe.com web site.

---

Pipe distribution systems are constructed using the following two elements:

1. Pipes - Links and Segments

   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.

   See the Layout and Model videos on the kypipe.com web site.

2. 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.

All development is carried out using only these two elements. Additional important definitions follow:

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.

## 1.3.5.29 Node Types and Properties

Node Types and Properties

---

## *End Nodes*

**End Node Properties** - 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 are accepted.

**End Node Types** - These include the following node types:

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 (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.

## *Internal Nodes*

**Internal Node Properties** - 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.

**Internal Node Types** - These include the following node types that are situated at a connection of two pipe segments with identical basic pipe properties:

1. **Intermediate Node** - No device at this location - usually represents a change of alignment.

2. **Valve** - Indicates location of cut-off valves.

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).

### *Additional Information*

**All Nodes** - All nodes, including text nodes, have a location ('**x**' and '**y**' coordinate) and an elevation ('z' coordinate). A text title can be defined and displayed. Also, a bitmap image can be imported and displayed. Except for text nodes and intermediate nodes, all nodes have additional attributes (data). They are defined in the data information boxes that are displayed when the node is selected and in the data tables that can be accessed for all nodes.

## 1.3.5.30 On/Off Valves

On/Off Valves

---

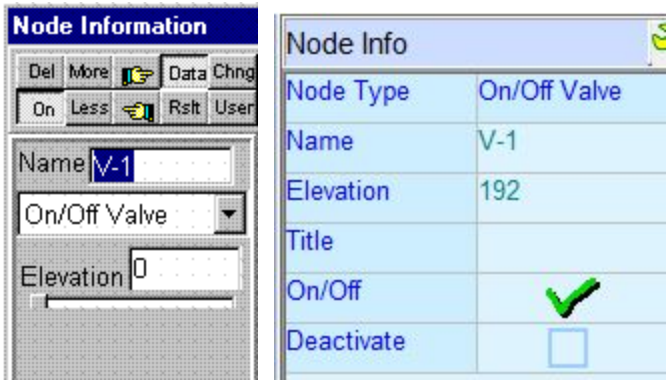
Valve on

Valve off

***What is an on/off valve?***

An on/off valve is an internal node in a pipeline which will control the open/closed status of the link. If you wish to account for the minor loss provided by that valve, it should be inlcuded in the fittings data for the pipe link.

To change the on/off status of the valve, select the valve and click on the On (or Off) button in the upper left corner of the Node Inofrmation Box.




The only data for the on/off valve is elevation.

On/off valves are used to do a pipe break simulation which locates and identifies the valves to be closed to isolate any location in your pipe system. See Pipe Break Simulation.

## 1.3.5.31 Positive Displacement Pump in Surge

A positive displacement pump may be modeled in Surge through the use of the periodic input function. See Surge Elements and Data Requirements / Creating a Transient - Periodic Variations for examples using this feature.

**Positive Displacement (reciprocating) Pump Example**

Figure 1 shows a Pipe2018 model to simulate the action of a Positive Displacement pump. This pump provides a periodically varying flow into the piping system. The elevation and initial pressure are shown at each node and the length and initial flowrate for each pipe. There is an initial demand of 50 gpm at the node noted. This demand is assumed to be pressure dependent and will increase and decrease with pressure changes.

All pipes have a diameter of 8 inches, a Hazen Williams roughness of 100 and a wave speed of 3600 ft/s as shown.

To simulate the action of the positive displacement pump a periodic input is simulated for the inflow at the pump node. For this example it is assumed that the pump provides a periodic inflow varying from 0 to 400 gpm at a rate of 1 cycle/second. Figure 2 shows the Pipe2018 input (Other Data/Periodic Input) to set this condition. A surge analysis is required to do carry out this simulation and a time of 10 seconds is chosen. Note the simulation must be sufficiently long to reach a repeatable solution. This is determined by inspecting the results.

Figures 3 and 4 show the pressure and flow variations at the pump. Note that a repeatable periodic response was attained after 3-4 cycles.

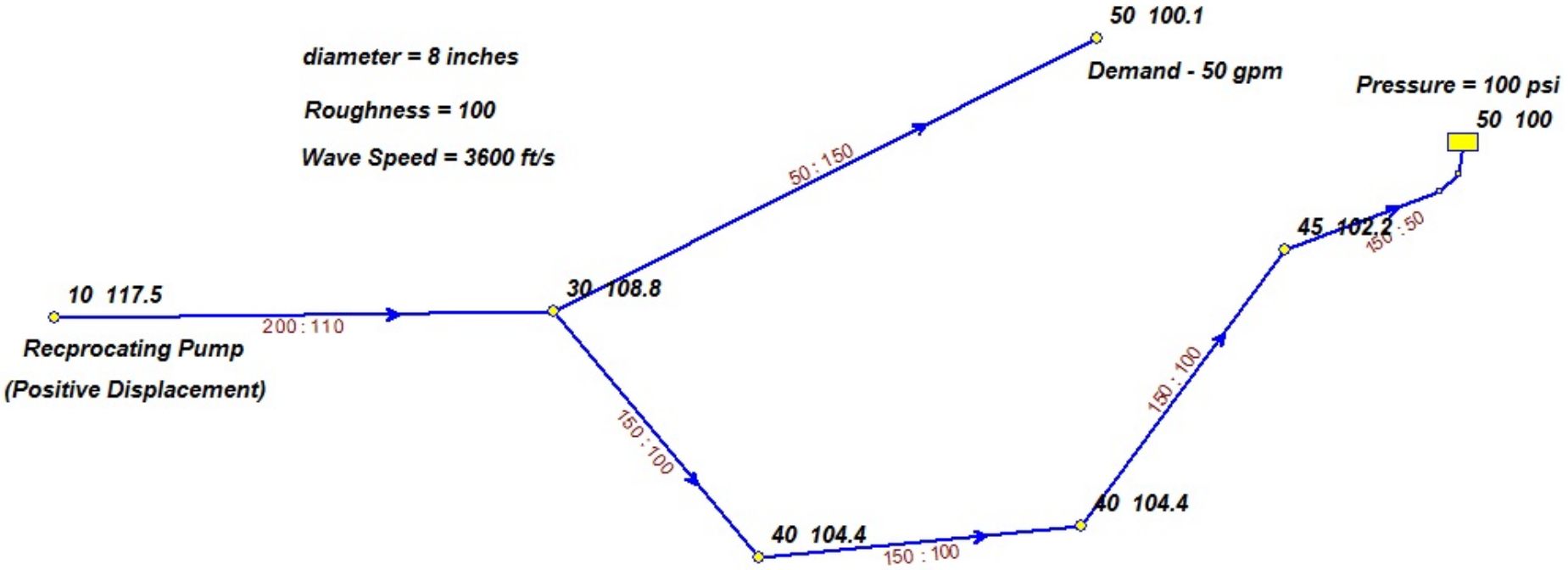



**Figure 1 Pipe2018 Model**

Meters | Library Elements | Periodic Input | Active Valves

Type: 0=Node 1=Connecting Rerservoir

| NODE | TYPE | PERIOD | PHASE SHIFT | AMPLITUDE | AVERAGE |
|---|---|---|---|---|---|
| J-4 | 0 | 1.0 | 0 | 200 | -200 |

**Figure 2 Setting up a Positive Displacement Pump**

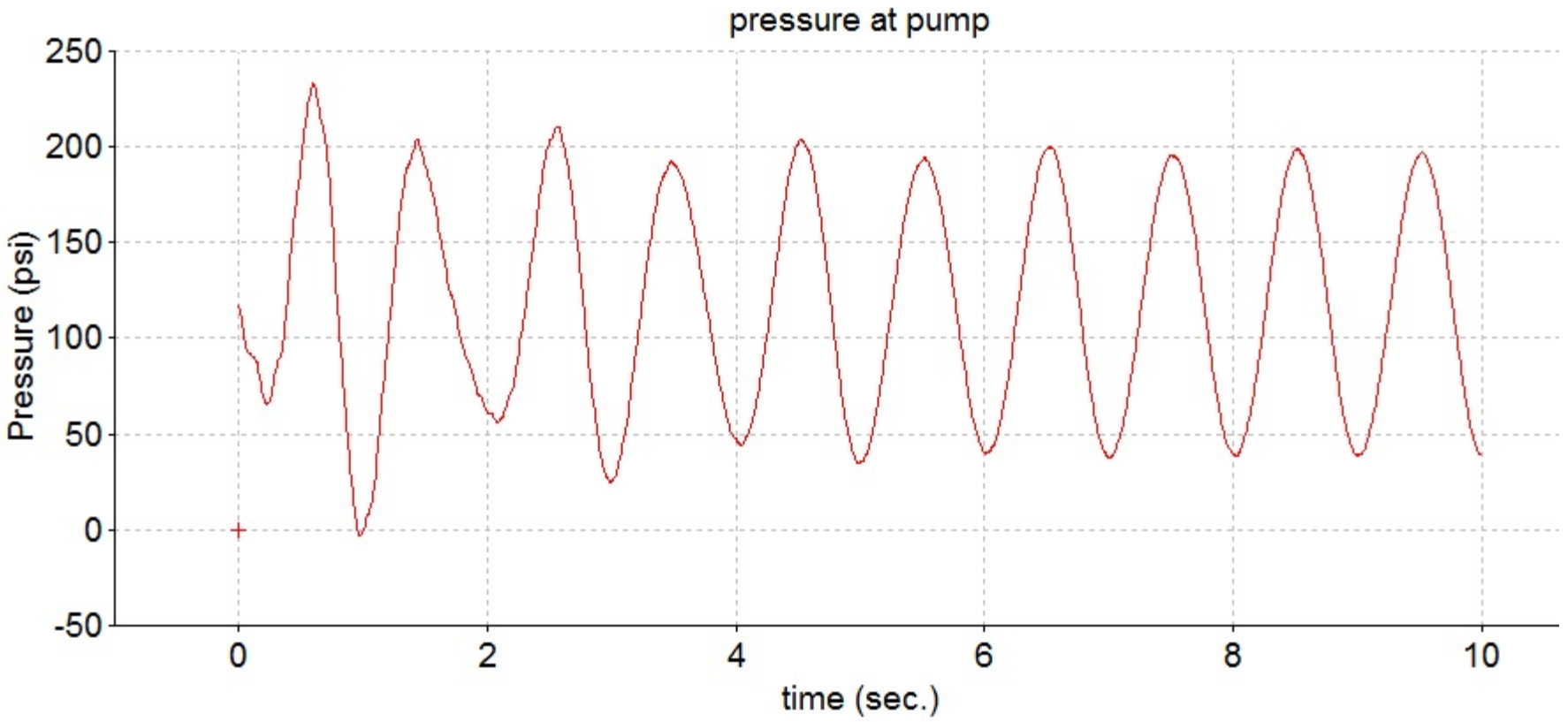



**Figure 3 Pressure Variation at Pump**

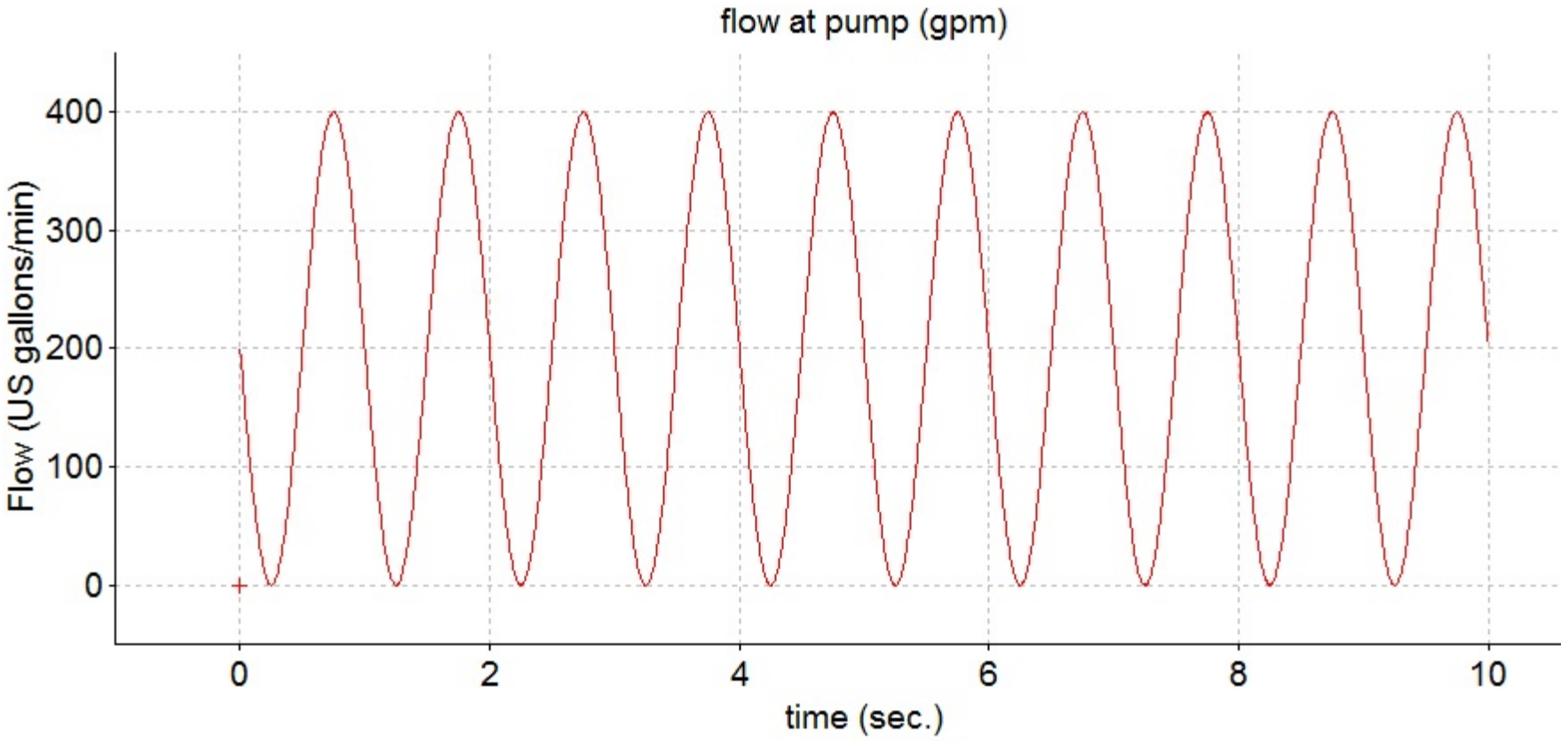
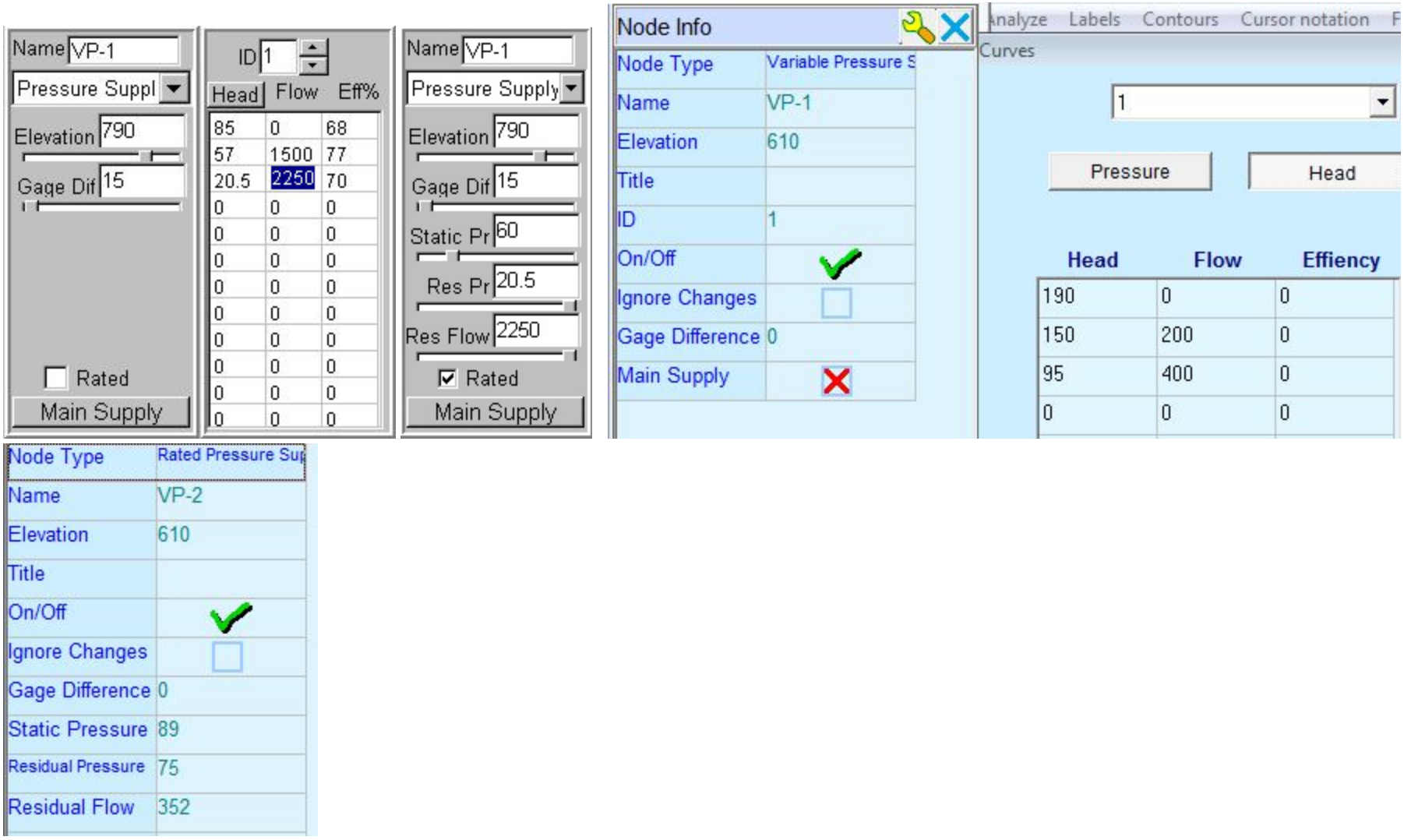



**Figure 4 Flow Variation at Pump**

## 1.3.5.32 Pressure Supply Data

Pressure Supply Data



---

*What is a Pressure Supply Node*?

A pressure supply node is an end node which represents a connection to a supply where the available pressure depends on the flow supplied. For example, a connection to a transmission main may represent a primary source for a system to be analyzed and the pressure available in the main may vary significantly with the amount of flow withdrawn at that point. Most connections to existing distribution systems should be modeled as variable pressure supplies. Head (pressure)/flow data must be provided for variable pressure supplies. Usually data to characterize the supply is obtained from a hydrant flow test. The head (pressure)/flow table must be created and the ID specified for each variable pressure supply.

Elevation

This is the elevation of the pipe connection.

Guage Dif

This is the elevation difference between the pipe connection and the pressure guage.

Rated

Check this box to define the pressure supply with hydrant flow test data. See Rated Pressure Supply below.

Main Supply

This is a GoFlow feature. See GoFlow Nodes.

* * * Additional Box * * *

*What is a head/flow data table*?

A table of head (or pressure) - flow data which describes the operation of a pump, loss element or pressure supply. An entry for efficiency is also provided but this is used only for computing power costs for pumps and should be ignored for other applications. Note that there is a button to switch between head [ft. (m)] and pressure [psi (kpa)]. Each table has a unique integer identifier (ID).

Pressure Supply ID

An integer identifier for the head (pressure)/flow data table.

Pressure Supply Data

There are two options for entering head (pressure)/flow data for a variable pressure supply. Note the head (pressure) switch to select units for this entry. Multiple

data points should be entered in order of increasing flowrates. Flowrate is entered in your specified units (shown on top bar of Map screen).

1. The first entry is tank pressure (head) and zero (0) flow and the second is residual pressure (head) and residual flow (specified units). this is normally obtained from a hydrant test on a hydrant close to the location of the pressure supply. A curve is generated from this data based on AWWA guidelines.

2. Three or more head (pressure)/flow data points are entered and an operating curve is generated from this data.

### Rated Pressure Supply

Static Pr

This is the measured static pressure at the pressure supply (psi or kPa).

Res Pr

This is the hydrant test pressure (psi or kPa).

Res Flow

This is the hydrant test flow (selected flow units).

The full, 3-point characteristic curve for a Rated Pressure Supply is obtained as follows:

$Q_0 = 0$
$P_0$, Static Pressure
$Q_1$, Residual flow
$P_1$, Residual pressure
$Q_2 = ?$
$P_2 = ?$

The program doubles the residual flow, making $Q_2 = 2Q_1$

The following equation is a form of Hazen-Williams.

$$Q_2 = Q_1\left(\frac{(P0-P2)}{(P0-P1}\right)^{0.54}$$

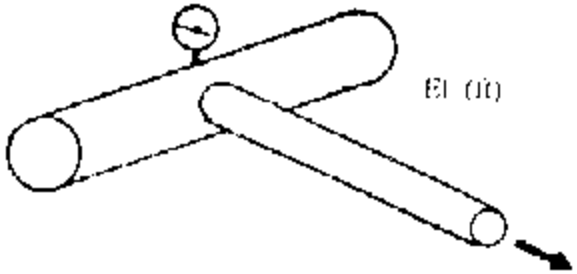
$$\frac{(Q2)}{(Q1)} = 2$$

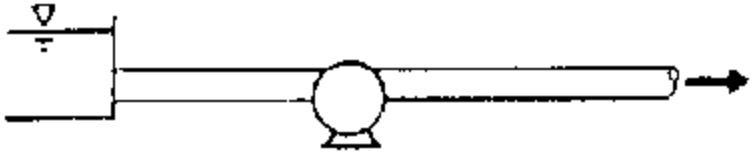
$$(2)^{\frac{1}{0.54}} = \frac{(P0-P2)}{(P0-P1)} = 3.61$$

simplify to

$P_2 = 3.61\ P_1 - 2.61P_0$

| flowrate (gpm) | available pressure (psi) | head increase (ft) |
|---|---|---|
| 0 | 70 | 161.5 |
| 1800 | 62 | 143.0 |
| 2500 | 53 | 122.3 |



pseudo pump

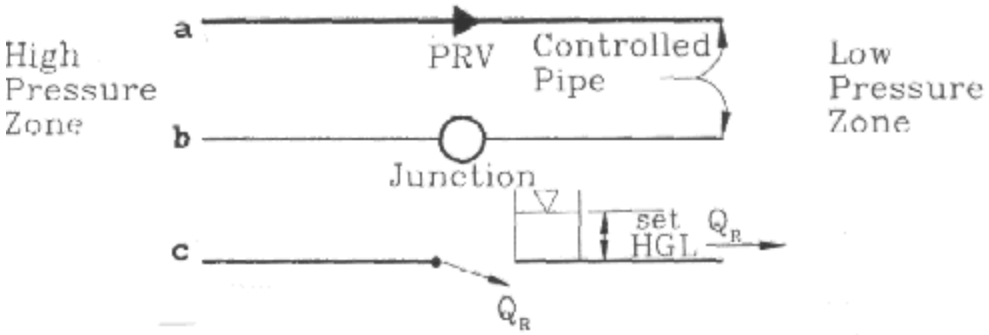
## 1.3.5.33 Pressure Regulating Valve (PRV)

Pressure Regulating Valve (PRV)

see also Regulator Data

---

These valves are designed to maintain a specified discharge pressure which is lower than the upstream pressure. A PRV location is modeled, as shown in line "b" below, as a junction node positioned at the upstream end of the controlled pipe. Within KYPIPE the PRV is modeled as shown in line "c" below as two nodes. The upstream node is a junction node with an added flow demand set (within the program) equal to the flow through the PRV. The downstream node becomes a FGN feeding the controlled pipe with the HGL head input equal to the elevation of the PRV plus the set pressure head.




Two situations can occur which can keep the PRV from operating in a normal fashion. l) Where the operating conditions and network configuration indicate flow reversal, the PRV cannot control the downstream pressure which exceeds the set pressure. The line downstream from the PRV will automatically close in this situation. 2) The upstream pressure drops below the set pressure. In this case maintaining the PRV setting would result in an increase in the pressure and act as a booster pump which is incorrect unless this specific application is desired. The analysis will automatically be carried out with the PRV removed (line wide open) if this situation occurs. A report concerning the valve operation is given in the results. You can choose to model abnormal situations where the set pressure is maintained under all conditions.

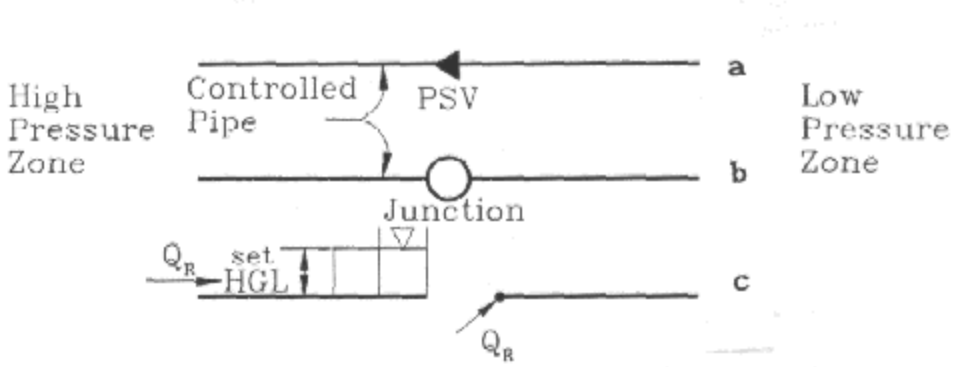
## 1.3.5.34 Pressure Sustaining Valve (PSV)

Pressure Sustaining Valve (PSV)

See also Regulator Data

---

This device is designed to sustain the upstream pressure at a designated location in the system (by limiting the flow if necessary) through the valve. This valve can be modeled as depicted below with the controlled pipe upstream from the junction node which represents the location of the PSV. Like a PRV the PSV can operate in three modes.




1. The valve is fully open and the upstream pressure is above the set value.

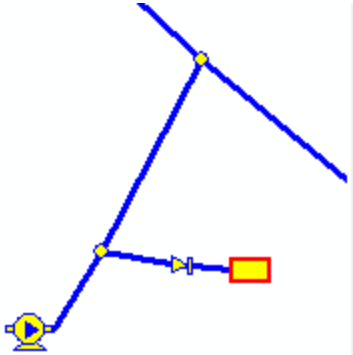
2. The valve is throttled and the upstream pressure is regulated at the valve setting.

3. The valve is closed and the upstream pressure drops below its set value but cannot be controlled by the valve.

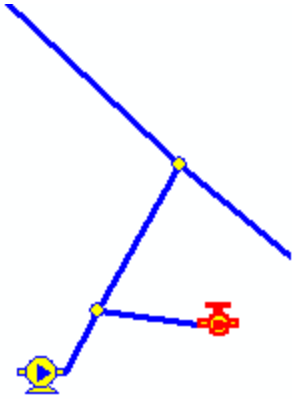
KYPIPE checks to determine which of these situations occurs and analyzes that situation. The regulating valve report clarifies the valve operation mode at all times.

## 1.3.5.35 Pressure Relief Valve in KYPipe

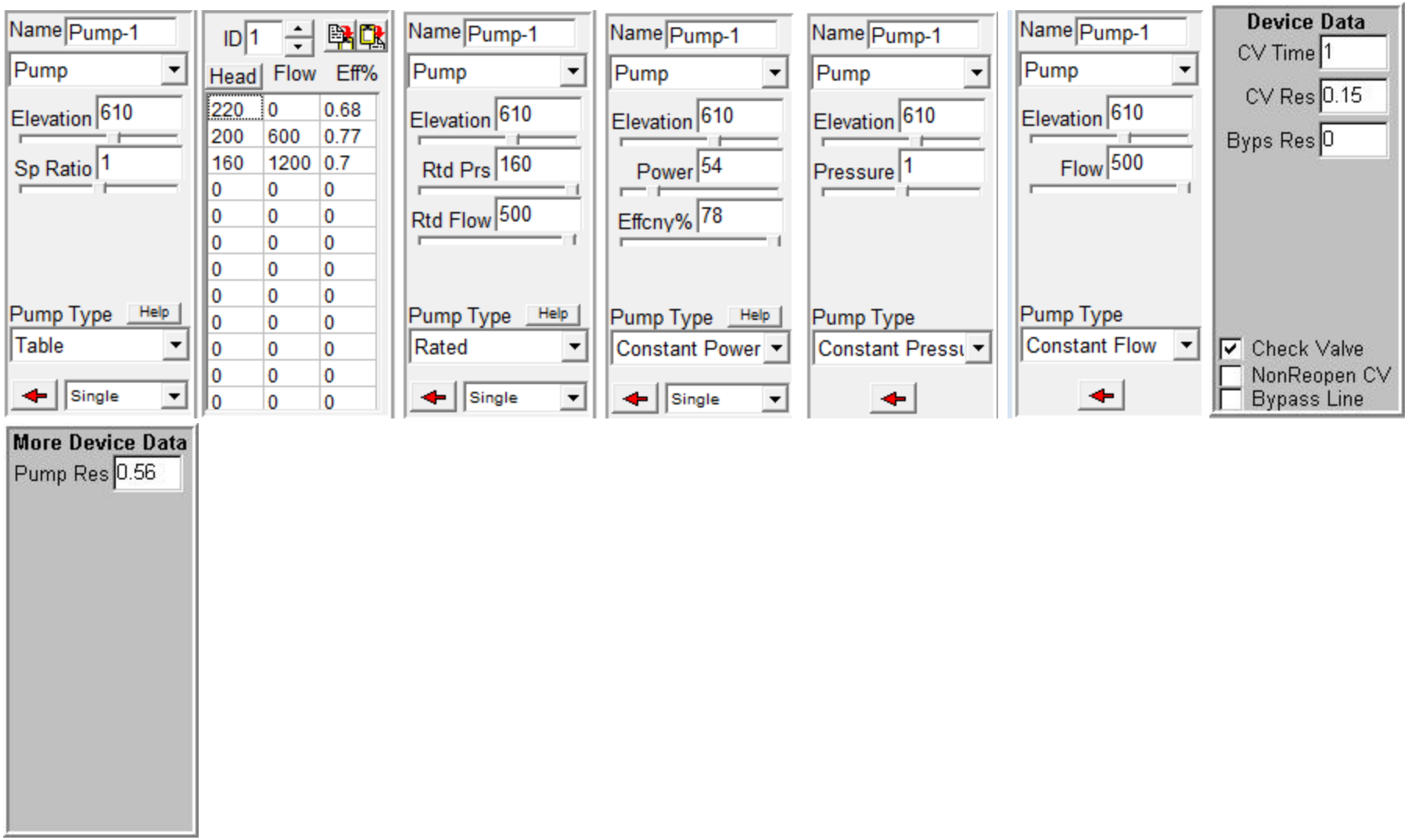
To model a pressure relief valve in KYPipe a pipe leading to a Reservoir (Pressure) may be used. The reservoir is set at the pressure of the relief valve. The pipe has a check valve allowing flow to the reservoir so when the pressure in the system is greater than the reservoir setting the the check valve opens and vents.

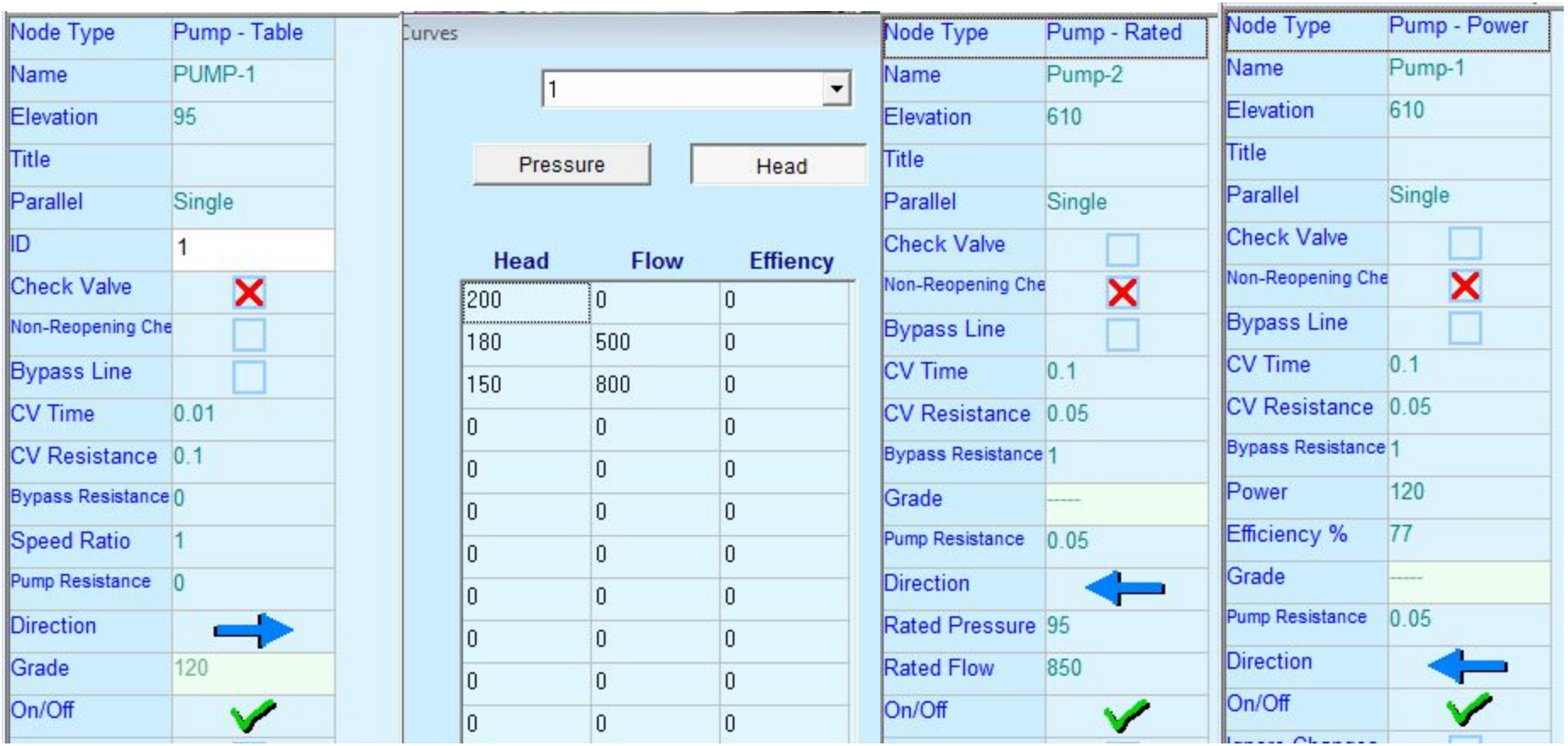


Another possibility is to use an active valve at the end of a short pipe with a reservoir on the exit side set to the relief valve pressure setting. The active valve has an internal check valve to allow only flow to exit. This would look more like a relief valve.



## 1.3.5.36 Pump Data


Name Pump-1
Pump
Elevation 610
Sp Ratio 1
Pump Type Help
Table
Single
ID 1
Head Flow Eff%
220 0 0.68
200 600 0.77
160 1200 0.7
Name Pump-1
Pump
Elevation 610
Rtd Prs 160
Rtd Flow 500
Pump Type Help
Rated
Single
Name Pump-1
Pump
Elevation 610
Power 54
Effcny% 78
Pump Type Help
Constant Power
Single
Name Pump-1
Pump
Elevation 610
Pressure 1
Pump Type
Constant Pressu
Name Pump-1
Pump
Elevation 610
Flow 500
Pump Type
Constant Flow
Device Data
CV Time 1
CV Res 0.15
Byps Res 0
Check Valve
NonReopen CV
Bypass Line
More Device Data
Pump Res 0.56



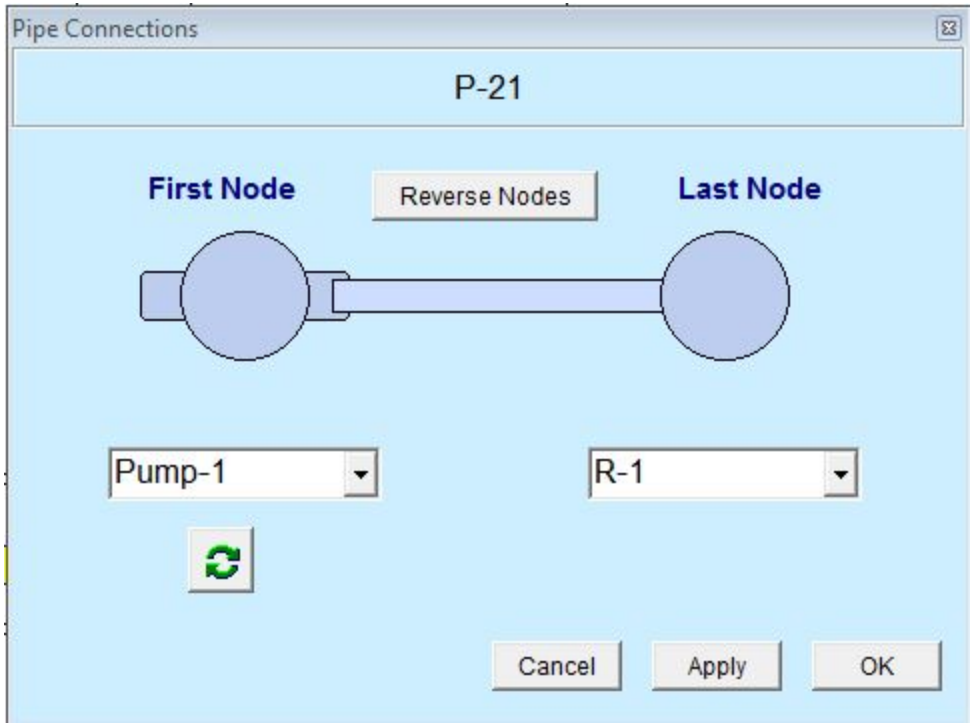

Pump and System Curves , Compressors , Modelling Wells , Pumps - About

---

## What is a Pump Node?

A pump node is an end node where a head (pressure) gain occurs. The pump may be a constant power device or operate on a head/flow curve based on data provided in a head/flow table. A pump is a directional node and multiple pipe links may be connected to either side.

The **pipe connections** are modified by selecting the connecting pipe(s) and clicking on the symbol in the pipe data box (Classic) or (KYnetic) clicking on the Node 1 cell and the following window appears:




The pump may be **closed** by selecting the pump node and clicking on

Classic - the on/off button at the top of the Node Information window


KYnetic -

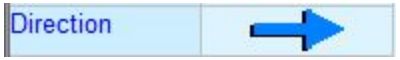

**NOTE**: If pipe links are connected only to **one side of the pump**, this side with no connections is assumed to be a reservoir connection. A data entry for the reservoir (Grade) will appear and the HGL must be provided.

### Pump Grade [ft (m)]

This appears only if no pipe links are attached to one side of pump and is the HGL [ft (m)] of the connecting reservoir.

### Pump Type

You can choose between a pump described by a data table (ID), one operating at constant power, one described by rated conditions or one described by a pump file (Surge application). For the data table option two boxes (above center) appear while for the other choices a single box (above on left or above on right) is displayed.

**Pump Direction** Classic: KYnetic: 


This button changes the pumping direction. Pump must be oriented in direction of flow.

**Pump Configuration**


This allows the user to specify several configurations of pumps; single, groups in parallel or groups in series. By specifying a group of pumps in parallel or series, all the pumps are considered to be of the type specified in this data box. If the user needs to represent different sized pumps together in series or parallel, then these must be defined separately in the model. With parallel or series pumps, it may be desired to define a pump resistance. See More Device Data.

### Device Data

CV Time (Check valve closure time) may be defined in seconds. CV Red (the check valve resistance) may be defined in units of headloss/(flow)^2. Bypass line applies to Pipe2018 : Surge applications.

(Classic) Click the 'More' button or pointing hands to view, if necessary.

Important note: When viewing the Pump Head in the Report, if a resistance value is entered here, the added head drop will be reflected in the total head drop in the Pump Report.

### (Classic only) More Device Data

Click the 'More' button or pointing hands to view, if necessary. A pump resistance for each pump may be defined in units of headloss/(flow)^2. A special tool is available to calculate the resistance, Resistance Calculations. In the Resistance Calculation Tool, specify 'Piping for Parallel and Series Pumps.'

Important note: When viewing the Pump Head in the Report, if a resistance value is entered here, the added head drop will be reflected in the total head drop in the Pump Report.

## Table Pump (VFD pump)

### Pump Speed [rpm]

The speed ratio (operating speed/rated speed) for a variable speed pump. For this application constant power pumps may not be used. Note that the speed may be changed using Change Pattern data and also may be used as part of a Constraint calculation

### Pump ID

An integer used to identify different sets of head (pressure)/flow data, entered by the user, for a particular element (pump, loss element or pressure supply). Different pumps can use the same pump ID.

In KYnetic, click on Pump ID number to bring up the ID input table.

- Copy and Paste buttons allow IDs to be copied entirely or in part. Also applies to Pressure Supplies, and Loss Elements..

### Pump Data

There are several options for entering head (pressure)/flow pump data into the head/flow data table. Note the head (pressure) switch to select units for these entries. Multiple data points should be entered in order of increasing flow rates. Flow rate is entered in your specified units (top bar).

1. A single (head/flow) point can be entered, which will be interpreted as rated conditions (HR, QR). Two additional data points will be automatically generated (1.4HR, 0) and (0.65HR, 1.5QR) and a power curve is generated based on these three points.

2. Three data points are entered and a power curve is generated.

3. Four or more points (up to 12) are entered and a quadratic fit is generated using three points in the vicinity of the operating point.

The efficiency data is optional (enter as a percent, 0 - 100) and used only for power costs calculations. You can enter three efficiency points (for corresponding head/flow data points) and an efficiency curve will be used. If you enter just one efficiency point, a constant efficiency based on that entry will be used.

## Constant Power Pump

### Power

The power (useful horse power or KW) for a constant power pump.

hp = (WRT/550)(k/(k-1))[(p2/p1)^((k-1)/k)-1] comes from the energy equation. It is isentropic. The standard units for pump power is ft-lbf/sec. 550 ft-lbf/sec = 1 Hp which is a non standard but popular unit. In SI units 1000 is used to convert standard power units (N-m/s which is same as Watt) into kW.

### Effcny (efficiency)

The efficiency (%) is for constant power pump only. This efficiency is for the user's reference and is not considered in the analysis. The efficiency of the pump should be factored in when the useful power is entered into the Power field.

## File Pump

This pump is used for pump trip simulation in Surge models. See Surge Elements

## Rated Pump

**Rtd Prs (Rated Pressure)** - the rated pressure for the pump (psi or kpa).

**Rtd Flow (Rated Flow)** - the rated flow for the pump (in specified units)

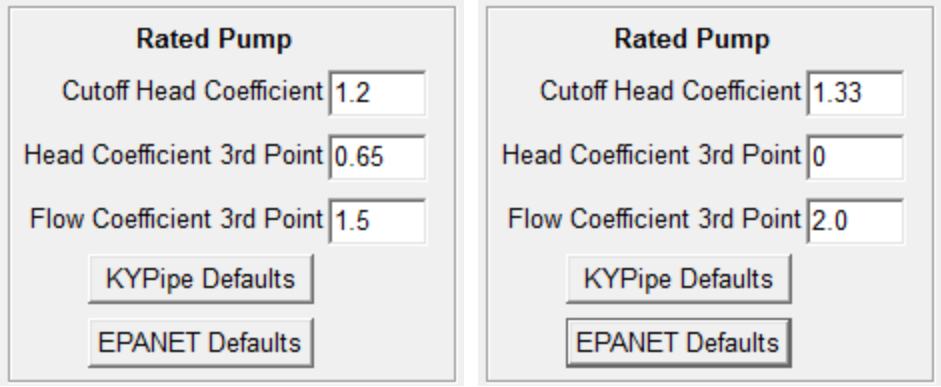
Note: a head-flow curve is generated for this pump using three data points:

There are three Options for the pump curve:

Option 1 (KYPIPE Default)
1.) cutoff pressure = 1.2*(rated pressure), flow (cutoff) = 0
2.) rated pressure, rated flow
3.) 0.65*(rated pressure), 1.5*(rated flow

Option 2 (EPANET Default)
1.) cutoff pressure = 1.33*(rated pressure), flow (cutoff) = 0
2.) rated pressure, rated flow
3.) 0.0*(rated pressure), 2.0*(rated flow

Option 3:
User enter data (System Data/Other (shown below))




### Constant Pressure Pump

Pressure - in psi or kPa

### Constant Flow Pump

Flow - in selected flow units

## 1.3.5.37 Pump Minimum Flow Valve

Pump protection valves are are recirculating valves which ensure a minimum flow rate through a pump at all times. If the flow through the pump falls below a certain level the bypass system opens and the fluid will be recirculated providing the required minimum flow for the pump.

Pump protection valves often work in a modulating way. Should the discharge increase through the following processes, then the amount of fluid recirculated by the bypass is automatically reduced- and vice versa. This increases the efficiency of the entire system and optimizes the economy in the low-load operation.

The recirculation could either be back to the pump suction or to a tank.

While there is no specific element in Pipe2018 for a Pump Minimum Flow Valve, a pressure relief valve set at the pressure where the corresponding flow reaches its minimum may be used. As the pump flow decreases the pressure increases. Pressures associated with a particular flow are known. Important note: Relief Valves tend to create transients and open/shut frequently so long times may be required for opening/closing to prevent this.

## 1.3.5.38 Pumps

Pump Data

---

A pump can be included at any location in the pipe system. The effects of pumps can be described in a variety of ways.

Constant Power
Head-Flow Data (single data point)
Head-Flow Data (multiple data points)
Variable Speed Pumps
Constant Head Pumps
Constant Flow Pumps

### Constant Power Pumps

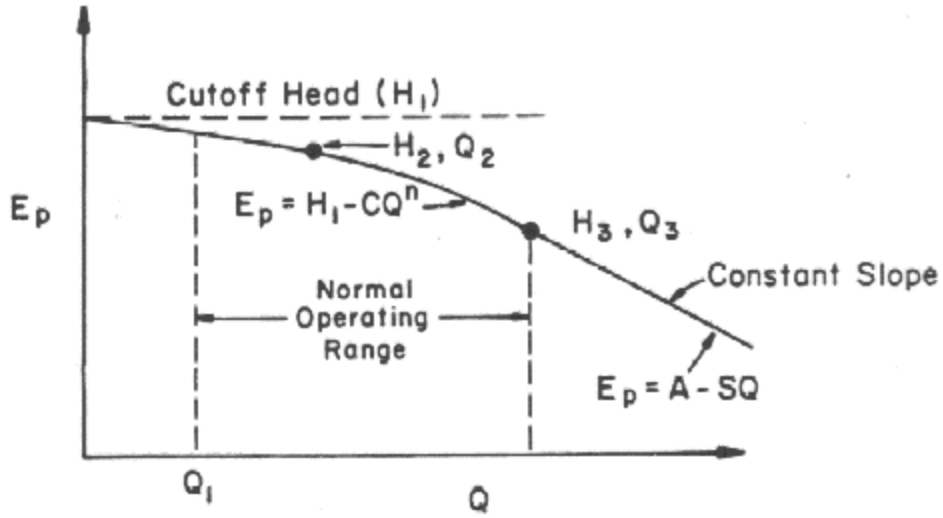
The useful power (horsepower or KW) of the pump can be specified as input data. The useful power refers to the actual power which is transformed into an increase in pressure head of the liquid as it passes through the pump. This method of describing a pump is particularly useful for a preliminary analysis or design when the specific operating characteristics of the pump are not known. In terms of the fluid density, $\gamma$, the flowrate, Q, and the pump head, Ep, the useful power, Pu, is given by:

$$Pu = Ep\ Q\gamma/Const$$

In SI units the constant is 1 (one) for kilowatts, and 550 for horsepower in English units.

### Head-Flow Data (3 data points)

A pump can be described by a curve fit to points of operating data. If the pump head-flow curve is smooth and, generally, concave downward as shown in the following picture




The preferred technique is to use three head-flow data points including cutoff head. KYPIPE fits an exponential curve to this data to obtain a pump characteristic curve describing the pump operation in the form:

Ep = H1 - CQ**m)

KYPIPE determines the coefficient C and exponent m for this curve. The pump cutoff head H1 , and two additional points of operating data (head - discharge) input are used to define the curve. The picture above depicts this representation.

The data points are shown along with the curve of the form of the above equation which passed through these data points.

The exponential relation given in Equation (2b) closely represents the pump between zero flow and the third data point (H3 and Q3) but is not suitable for flow reversal and may not be suitable for flows in excess of Q3. It is possible, however, that a solution of the hydraulic equations requires pump flowrate outside the range of pump representation by this equation (the pump may not be suitable for the conditions specified). In order to satisfactorily handle this, KYPIPE is designed to do the following:

a. A pump described by operating data will shut off if flow reversal occurs. This indicates that the pump cutoff head is not adequate to overcome the system grade caused by other factors, and the pump as described is unable to operate in a normal manner. **KYPIPE automatically places a check valve in a pump line to prevent flow reversal.**

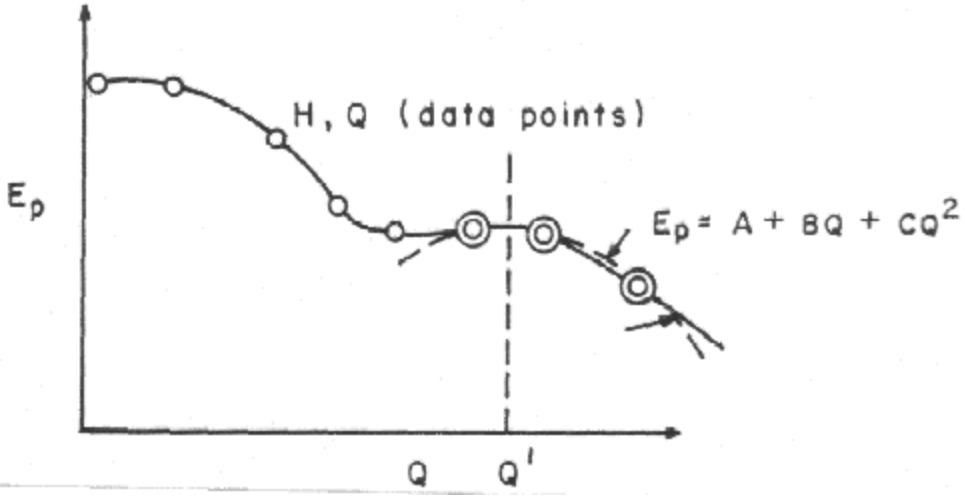
b. If the solution indicates that the pump is operating at a flowrate above the third data point (Q3) then it operates on a straight line

described by Ep = A - SQ where the slope is the same as given by Equation (2) at Q = Q3. KYPIPE determines the values of A and S. This gives a characteristic which is more realistic than Equation (2b) for Q > Q3 and will better stimulate a typical pump curve in this region.

c. If the pump operates out of the flow range 0 < Q < Q3 then the pump is considered to be out of its normal range and a message to this effect is generated. This alerts you to potential problems associated with the description of the pump. The use of pump data which are not compatible with the system requirements may lead to poor results.

### Head-Flow Data (multiple data points)

If the pump curve is not smooth but includes inflection or near inflection points such as the one depicted below, then additional data points will be required.




When more than three points of pump head-flow data are provided, KYPIPE fits an quadratic equation of the form

Ep = A + BQ + CQ**2

to the three data points closest to the operating point. As the solution is obtained this data used may vary. This will require additional trials and may cause convergence problems if several pumps are described in this manner. Therefore it is suggested that this approach only be applied if the pump curve is not smooth.

### Variable Speed Pumps

You can specify a dimensionless pump speed ratio, n, when providing pump data. The speed ratio is defined as the ratio of the actual pump speed, N, to the speed for which the data is applicable, NR. This ratio is used to modify the pump data by applying homologous considerations

H' = Hn*n , Q' = Qn

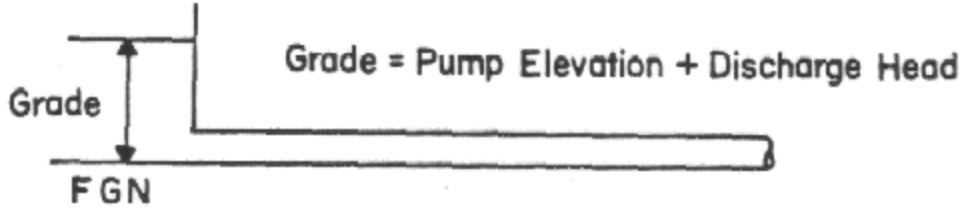
where H' and Q' are modified head-flow data and H and Q are the original data points. In this manner, solutions can be obtained for a pump described by a series of head-flow data points operating at a different speed. The modified exponential head flow curve for variable speed pump operation is

$$Ep = H_1 n^2 - \frac{Cn^2}{n^m} Q^m$$

In many instances variable speed pumps are operated such that a constant discharge head or a constant flowrate is maintained. For those applications the approaches described below provide accurate and simple modeling.

### Constant Head Pumps

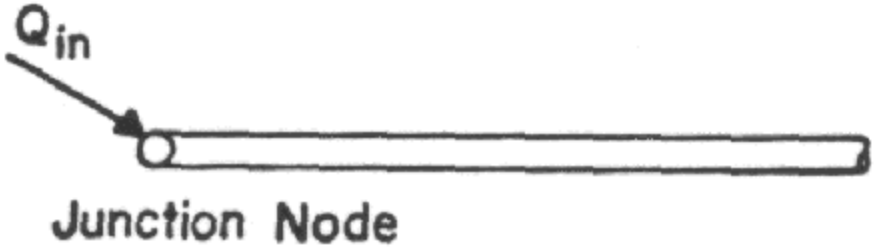
An alternate method of indirectly incorporating the effects of a pump into the system may be desirable if the analysis is to be made for a situation where the pump is operated such that the discharge pressure may be specified or is fairly closely known. For this application the pump discharge location is modeled as a fixed grade node (FGN) with the HGL computed using the elevation of the pump and specified pump discharge pressure. If the pump is on an external line this fixed grade node is simply a supply reservoir. If the pump is on an internal line (a booster pump) then this application is identical to that for a boosting pressure regulating valve with the regulated (boosted) pressure being the discharge pressure of the pump. For both applications the analysis determines the pump flowrate associated with the specified discharge pressure. This approach to pump modeling is illustrated below.




### Constant Flow Pumps

If a pump is operated such that it delivers a specified discharge into (or out of) the distribution system the pump can be represented as a junction node. The desired inflow can be specified at this node and the analysis will determine the pump discharge pressure which will be required to produce the

specified inflow. This approach is also illustrated below.




## 1.3.5.39 Regulating Valves

Regulating Valve Data

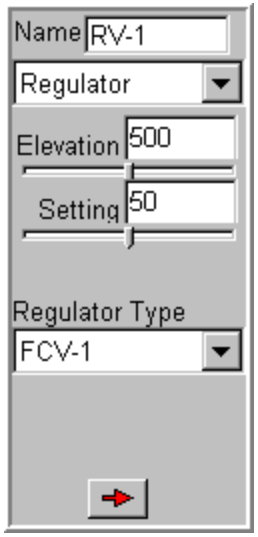
---

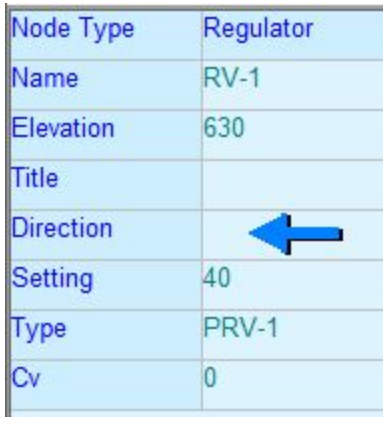
There are three types of regulating valves which can be automatically incorporated into your model. Pressure regulating valves regulate the pressure downstream from the valve. Pressure sustaining valves regulate the upstream pressure. Flow control valves regulate the flow. All of these valves are designed to operate in a throttled state and maintain a set condition. However, the valves may operate fully open or fully closed and be unable to maintain the set conditions. KYPIPE is designed to accommodate these three valves operating in both a normal (throttled) and abnormal (wide open or closed) mode. Data required for regulating valves includes the type of valve, the valve location (a junction node), the connecting controlled pipe and valve setting. A description of the techniques used to model these valves is described below.

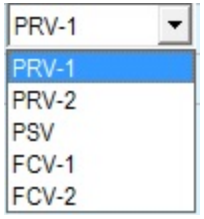
See Also: Pressure Regulating Valve (PRV)
Pressure Sustaining Valve (PSV)
Flow Control Valve (FCV)
Regulator Data

## 1.3.5.40 Regulator Data

Regulator Data




About Regulators

---

What is a regulator?

This directional node provides pressure or flow regulation and must have pipe links connected to both sides of the regulator. Five types of regulators can be modeled.

Setting

provide the pressure setting (psi or kpa) for pressure regulator or flow setting (specified flow units) for flow regulator.

Regulator Type

select a regulator type from the dropdown list. The choices are:

PRV-1 pressure regulating valve (normal operation) - may close or operate wide open if inlet < outlet

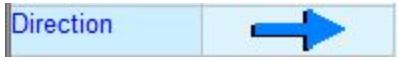
PRV-2 pressure regulating valve (always provides set value) - boosts pressure when inlet < outlet

PSV-1 pressure sustaining valve (normal operation)

FCV-1 flow control valve (normal operation) - may close or operate wide open if inlet < outlet

FCV-2 flow control valve (always provides set value) - boosts flow when inlet < outlet

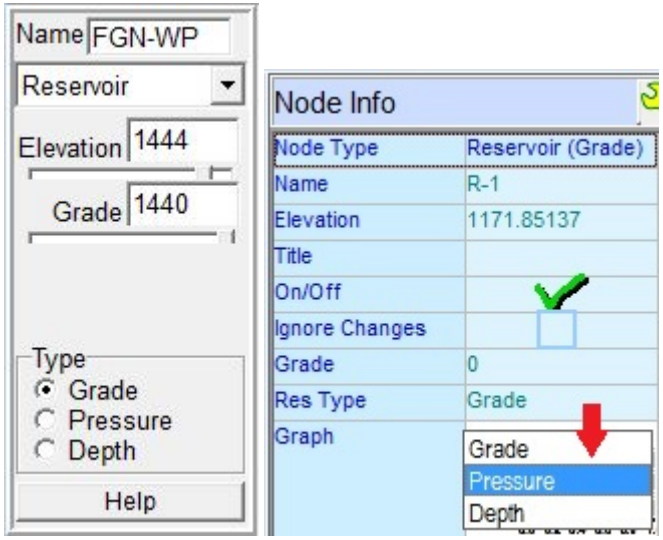
A PRV requires a downstream pressure setting (psi or kpa), a PSV an upstream pressure setting (psi or kpa) and a FCV a flow setting (in specified flow units).

Direction KYnetic: 


This button changes the direction of the regulator. Important note: Make sure that the regulator is set for the correct direction and that the pipe links are connected to the correct side of the regulator. To change the connection side of a regulator to a pipelink, select the pipe. In the Other Data box, click on the green directional arrows next to the regulator node.

## 1.3.5.41 Reservoir Data

Reservoir Data



*What is a Reservoir Node*?

A reservoir is a fixed head supply node such as a lake or fixed level storage basin. Data required may be entered as:

Grade - HGL elevation and pressure head for the reservoir level, ft or m.
Pressure - psi, kPa, or Bar
Depth - height of water level above the Elevation, ft or m

**Note that if a pump or loss element has no connections on one side this is assumed to connect to a reservoir and an entry for the reservoir HGL is required.**

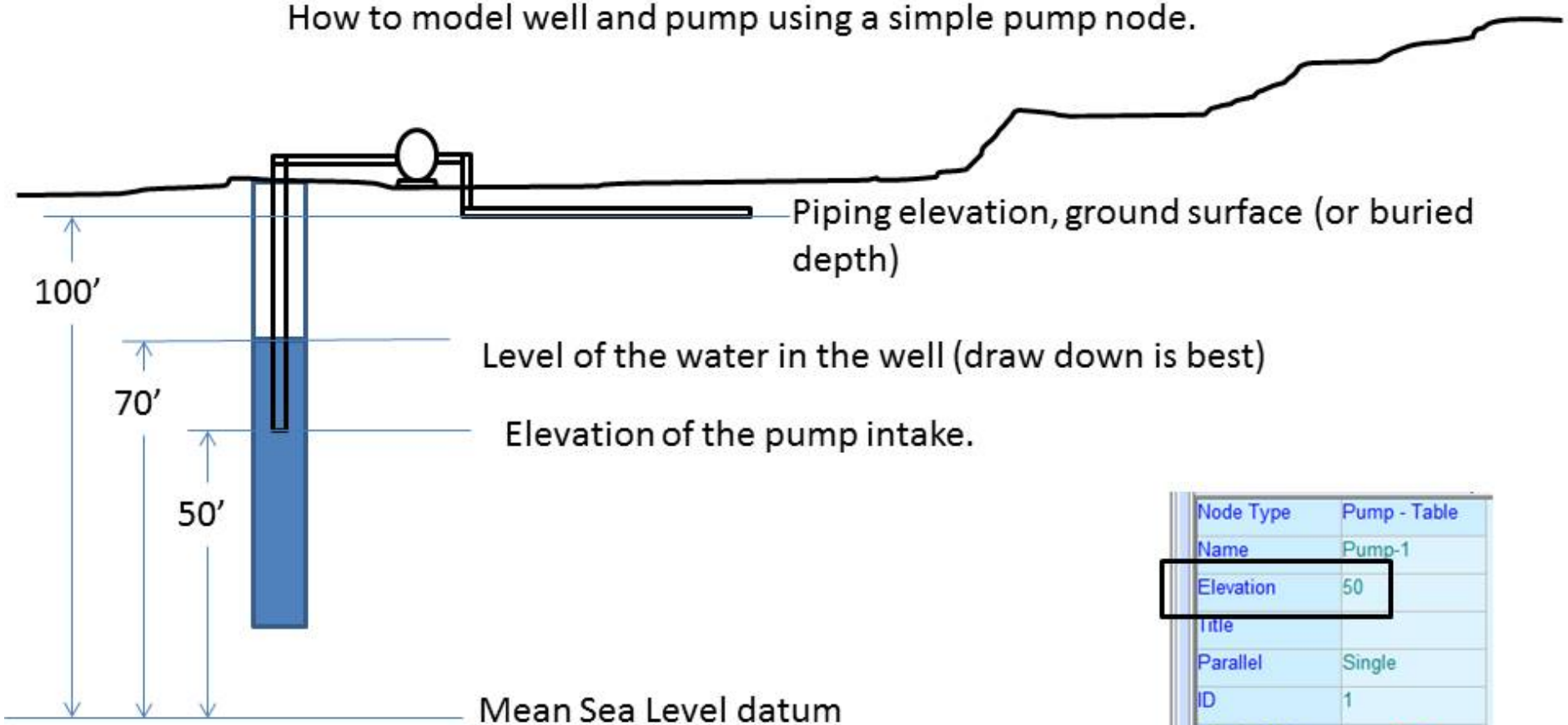
grade [ft (m)]

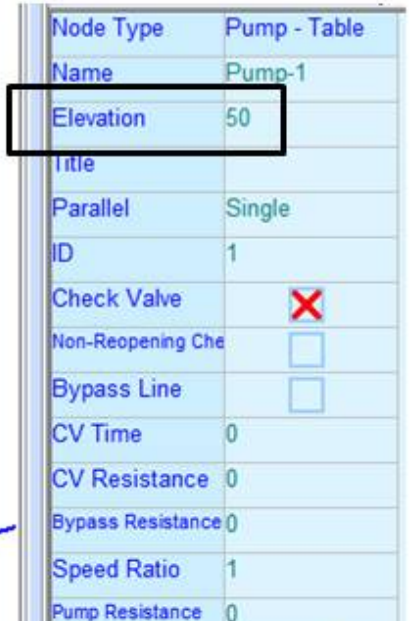
The hydraulic grade (elevation + pressure head = HGL) for the reservoir based on the designated datum.

Modelling Wells

When modelling a well with a pump, either a reservoir element plus a pump element may be used or a just pump element by itself as an end node (at the end of a line with grade data entered). The elevation of the water in the well is the grade, either for the reservoir or for a pump as an end node. The elevation of the pump is the elevation of the pump impeller (usually down in the well). See also Pump Data.

How to model well and pump using a simple pump node.



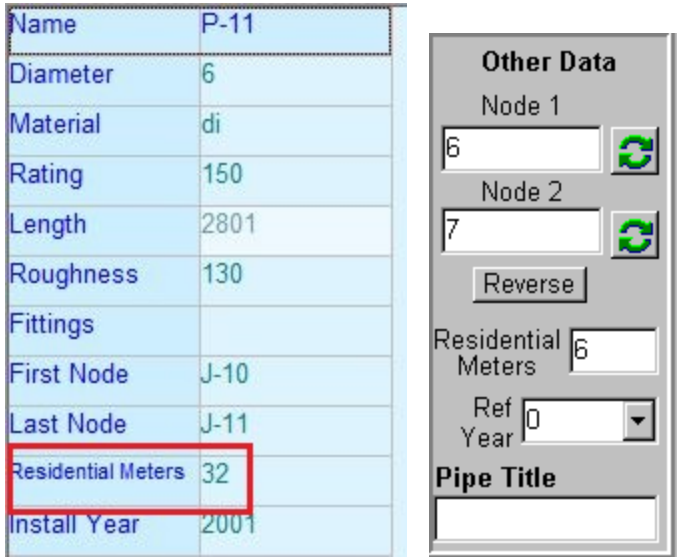

Elevation data

## 1.3.5.42 Residential Meters

Residential Meters

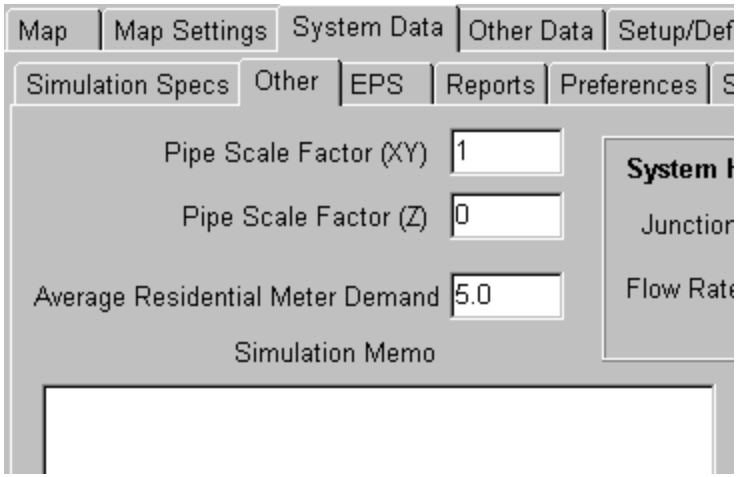
---

Residential Meters may be placed on a pipe to represent the number of domestic connections in a line. The data is entered in the Pipe Information window in the Other Data box as shown.



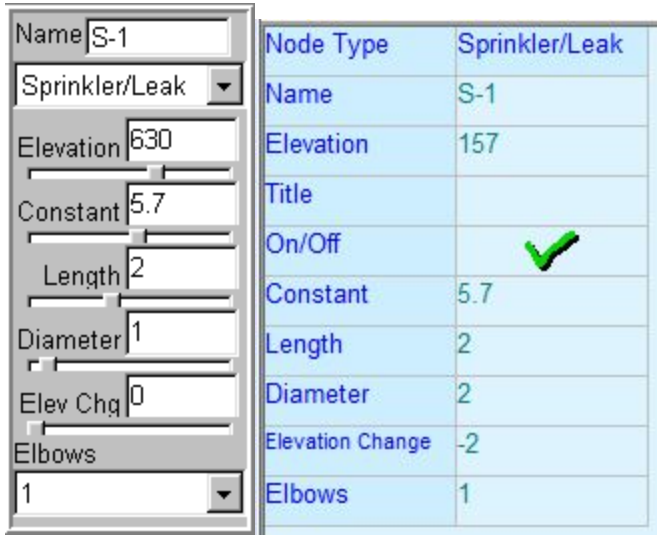

An average domestic connection demand is then assigned to each residential meter in the system. This data is entered under Other menu (KYnetic) or System Data | Other (Classic). The total demand in the line due to residential connections is distributed evenly to the nodes at either end of the pipe.

KYnetic Other Menu icon:




## 1.3.5.43 Sprinkler Data

Sprinkler Data




---

*What is a Sprinkler Node*?

A sprinkler node is an end node where flow discharges to the atmosphere through a sprinkler orifice. A sprinkler may also be used to simulate a leak.

Elevation - this is the elevation of the discharge orifice.

Sprinkler Data

The sprinkler constant (Ks) must be provided. This is based on the relation Q = Ks(Δp)^0.5 where Q is the flow rate (gpm or l/s) and Δp is the pressure drop (psi or kPa). These units apply no matter which flow units are specified (see Sprinkler Constant). Some standard values for Ks are:

| Orifice Size | Ks |
|---|---|
| 1/4" | 1.4 |
| 3/8" | 2.8 |
| 1/2" | 5.6 |
| 5/8" | 11.2 |

| | |
|---|---|
| 3/4" | 14.0 |

Defining a Sprinkler Connection is done by defining the three data items below. For no connecting pipe, ignore these data items (entries = 0).

Length (ft. or m) - length of the sprinkler connection

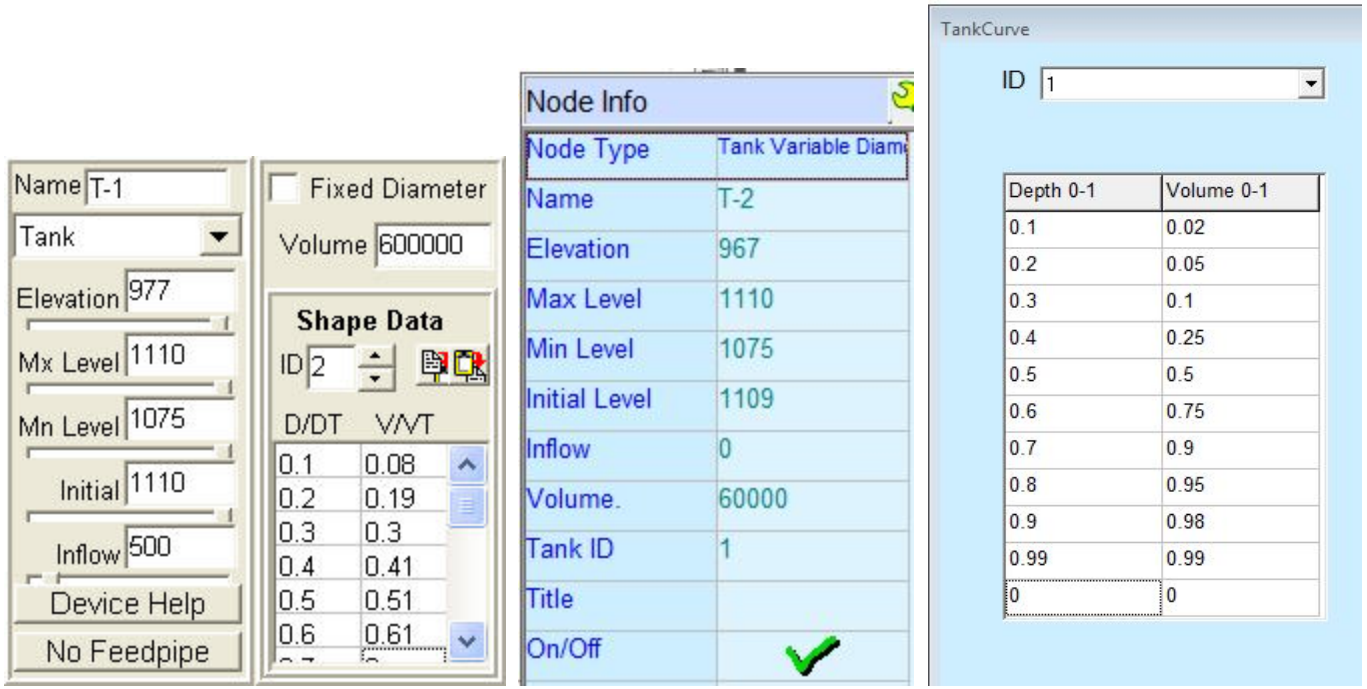
Diameter (in. or mm) - diameter of the sprinkler connection

**Elevation Change - this is the sprinkler head elevation minus the elevation of the connection to the branch pipe. A negative entry means the sprinkler orifice is below the branch pipe.**

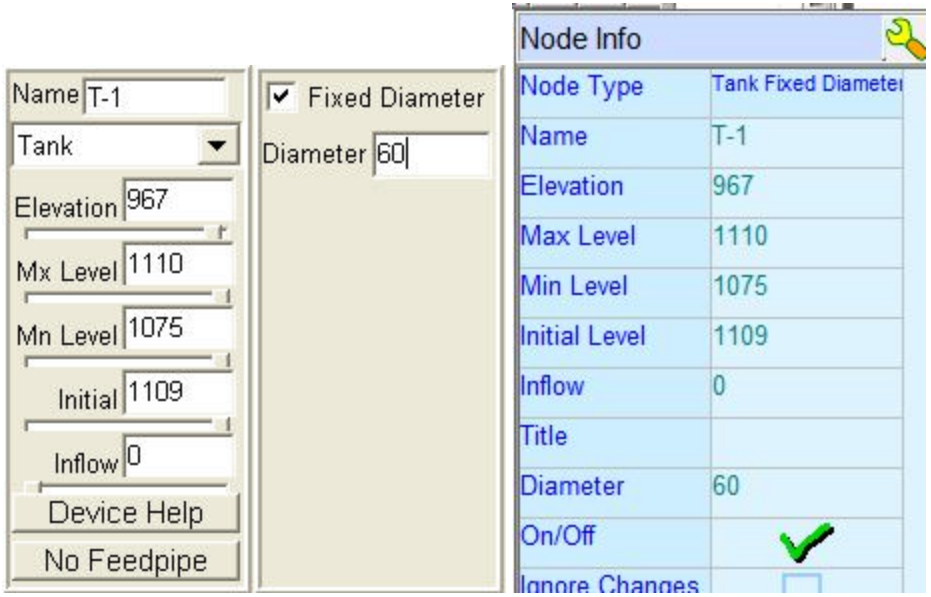
Elbows - user may define the number of elbows in the sprinkler connection.

## 1.3.5.44 Tank Data

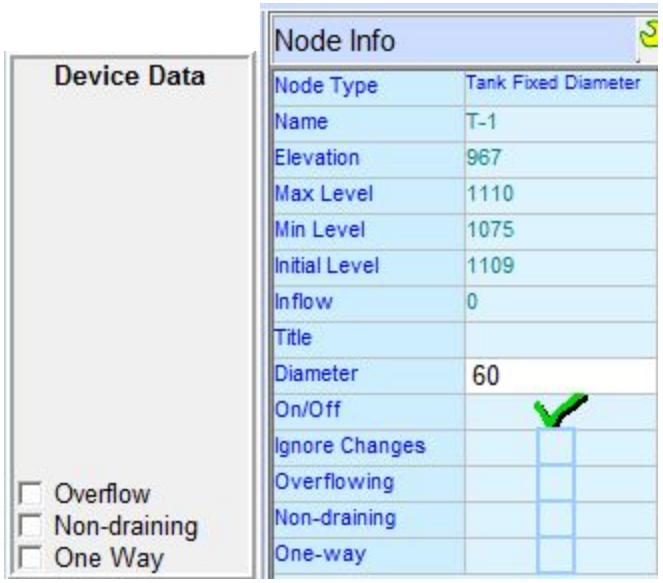
Tank Data




Variable Area




fixed diameter




Options for Overflowing, Non Draining, One-Way (Pipe2014 and later versions)

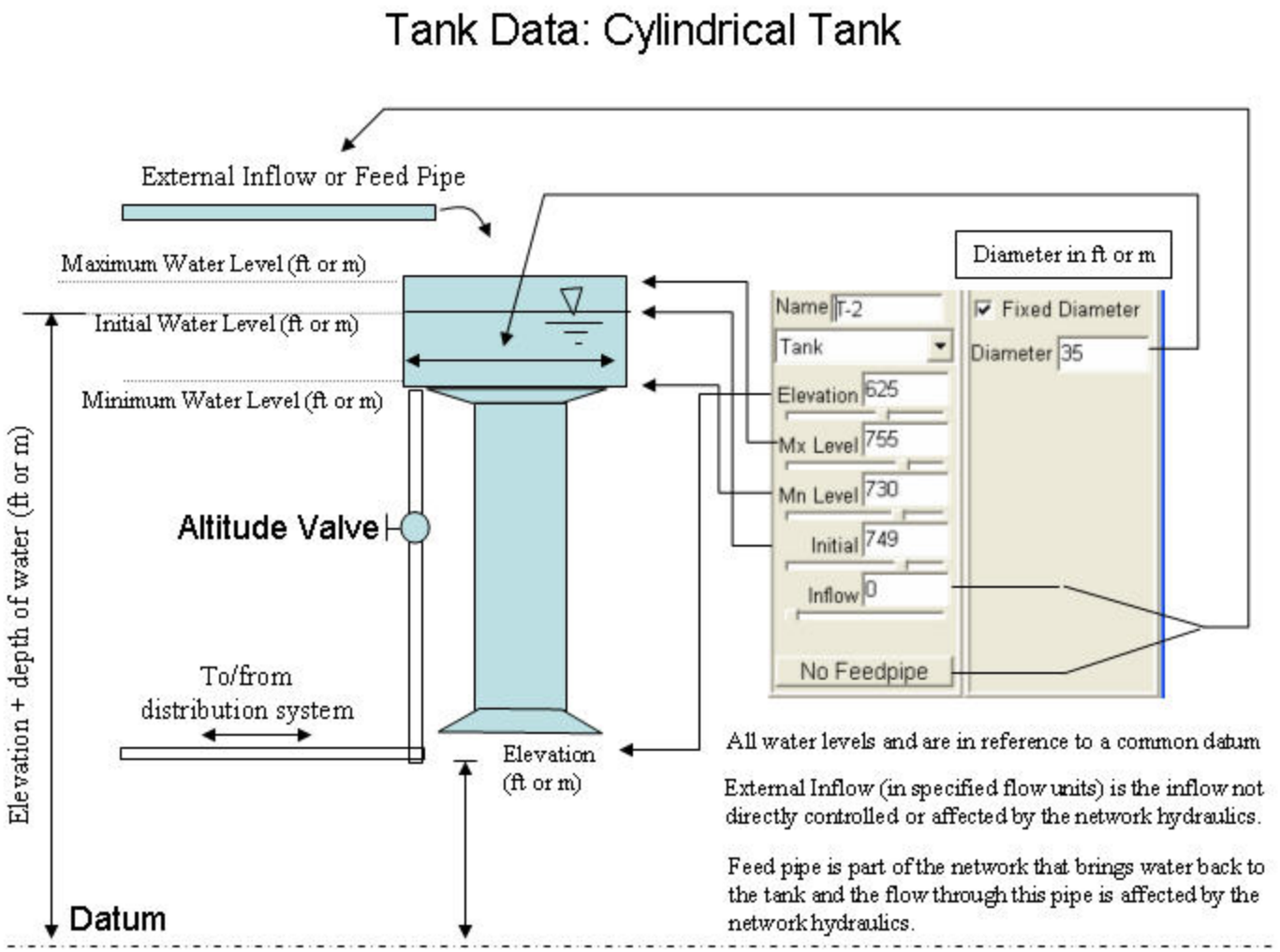
LPS Tank Hydrotank

---

What is a Tank Node?

A tank node is an end node which represents a connection to a storage tank. The tank level varies during an Extended Period Simulation (EPS). For a regular simulation the tank is modeled as a constant level reservoir operating at the initial level specified in the tank data.

If the tank is full no additional flow from the pipe system can enter the tank, and if it is empty no additional flow can leave the tank. However, flow can go in the opposite direction in either case. This models the action of an altitude valve. Note: For EPS, check valves are not permitted in lines connecting to variable level storage tanks.




Tank Data - The following additional data entries are required:

Maximum Level

The overflow level for the tank. No inflow to the tank at this level.

Minimum Level

The low level for the tank. No outflow from the tank at this level.

Initial Level

The starting level for the tank (time = 0 for EPS ). For regular simulation this is the grade for this FGN.

Inflow

The flow rate into the tank from **external** source at (time = 0), in specified flow units (note top bar). This does not represent flow from the network. A demand pattern may be applied to tank inflow for an EPS or Surge simulation. Under System Data | Other, a Demand Type number is specified. This Demand Type is then used in the Demand Pattern Table to define

No Feedpipe / Feedpipe

A button to specify a feedpipe discharging into this tank. If a feedpipe is specified enter the name of a pipe whose discharge feeds this tank in the space provided (Node Image box). The feedpipe should be modeled as discharging into a reservoir.

* * * Additional Box * * *

Fixed Diameter (Tanks)

Check this box for tanks with a fixed diameter. The units for the diameter will be in feet (or meters for SI units).

Shape ID (Tanks)

An identifier for the tank shape table. The same ID can be used for any number of tanks.

- Copy and Paste buttons allow IDs to be copied entirely or in part.

Tank Shape Data

Variable level tanks can be fixed (constant) or variable diameter vessels. For fixed diameter tanks (Left box) check Fixed Diameter and enter Diameter [ft. (m)]. For variable diameter tanks (Right box) enter total Volume referenced to flow units as follows:

| Flow Units | Volume Units |
|---|---|
| CFS | cubic feet |
| GPM | gallons |
| MGD | gallons |
| l/s | liters |
| CMS | cubic meters |
| l/min | liters |
| lb/s | cubic feet |
| BPH | cubic feet |
| kg/s | cubic meters |
| USER English | cubic feet |
| USER SI | cubic meters |

For variable diameter tanks a shape ID is specified. This ID is associated with the table displayed which contains pairs of depth/total depth and volume/total volume ratios. It is recommended that you enter 9 pairs of data using depth/total depth ratios of 0.1, 0.2, 0.3, - 0.9 and the default for the tables uses these values. * Note: If another number of pairs of data is used for a Shape ID, for example twelve data spaces, then space 1 must equal 0 and space 12 must equal 1 for the interpolation to be properly calculated.

**Options for Overflowing, Non Draining, One-Way (Pipe2014 and later versions)**

Overflowing - When tank reaches Max Level, HGL will remain the same, but tank will overflow until HGL drops below Max Level and tank begins to drain.

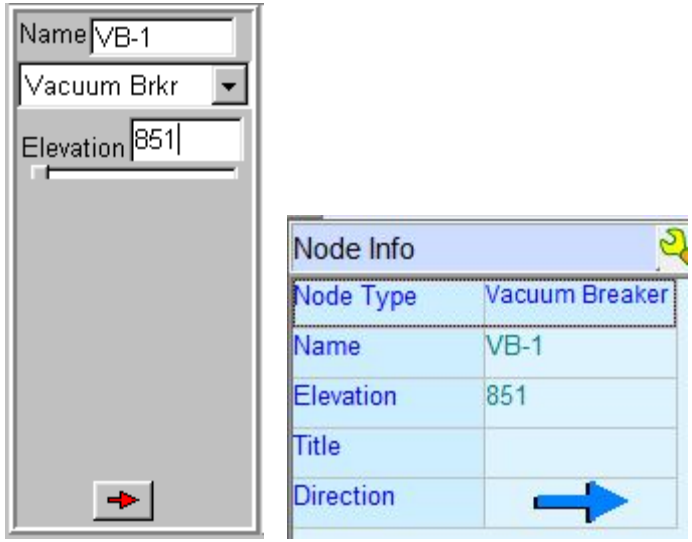
Non-Draining - An internal check valve prevents flow from leaving the tank, but will accept flow in.

One-Way - An internal check valve prevents flow from entering the tank, but flow may drain out.

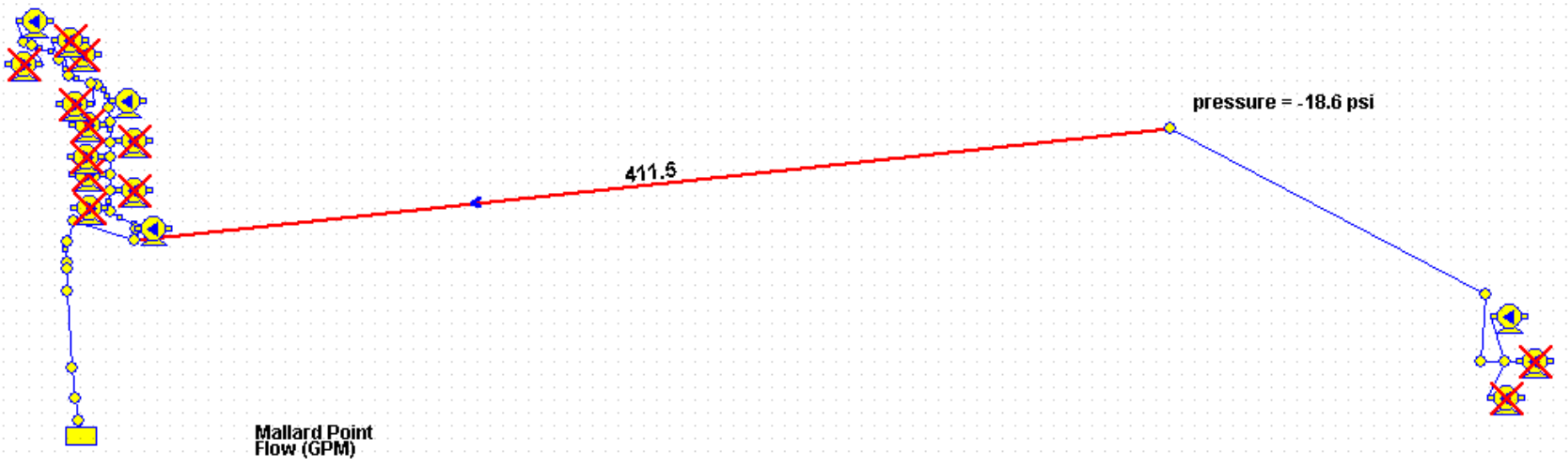
## 1.3.5.45 Vacuum Breaker element

Vacuum Breaker element

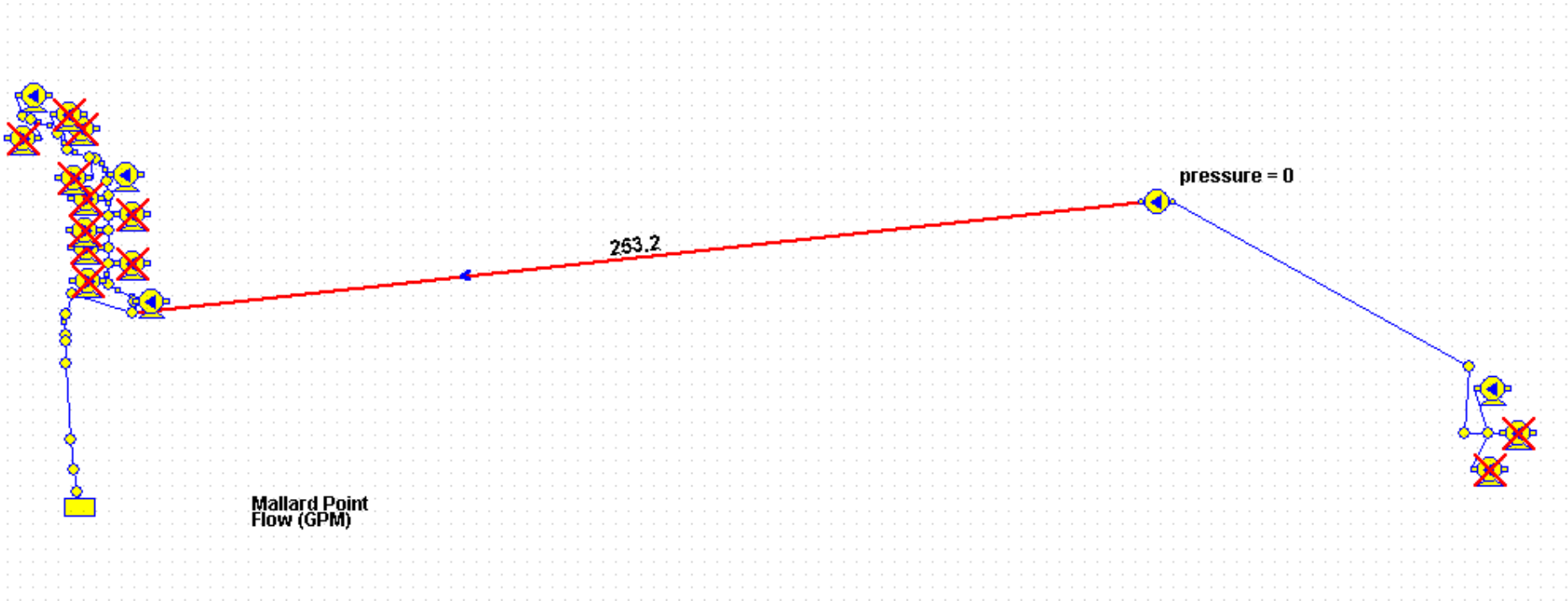
## Vacuum breaker



An element for KYPipe systems. This element is used to prevent a vacuum at high points in a system. The pipe is vented to the atmosphere, at atmospheric pressure. Elevation is the only required data. If the vacuum breaker is activated, the flow will be decreased and the pipe may flow partially full in regions beyond the breaker. Flow data is valid beyond the vacuum breaker, but pressure data may not be valid.




**Flow Without a Vacuum Breaker**




**Flow With a Vacuum Breaker**

**This is illustrated above. The upper schematic shows the flowrate calculated without a vacuum breaker. The calculated pressure at the high point is -18.6 psi which is not feasible. The second schematic shows the same calculations with a vacuum breaker. The pressure at the highest point is atmospheric (p=0) and the flow is considerably reduced.**

**Note - to the left (downstream) of the vacuum breaker the pipe will flow partially full. Element is functionally like a PSV set to 0 psi.**

## 1.3.5.46 Valves

Valves

---

**Valves**

Pipe2018 models a variety of valves which provide a wide range of features. Among them are:

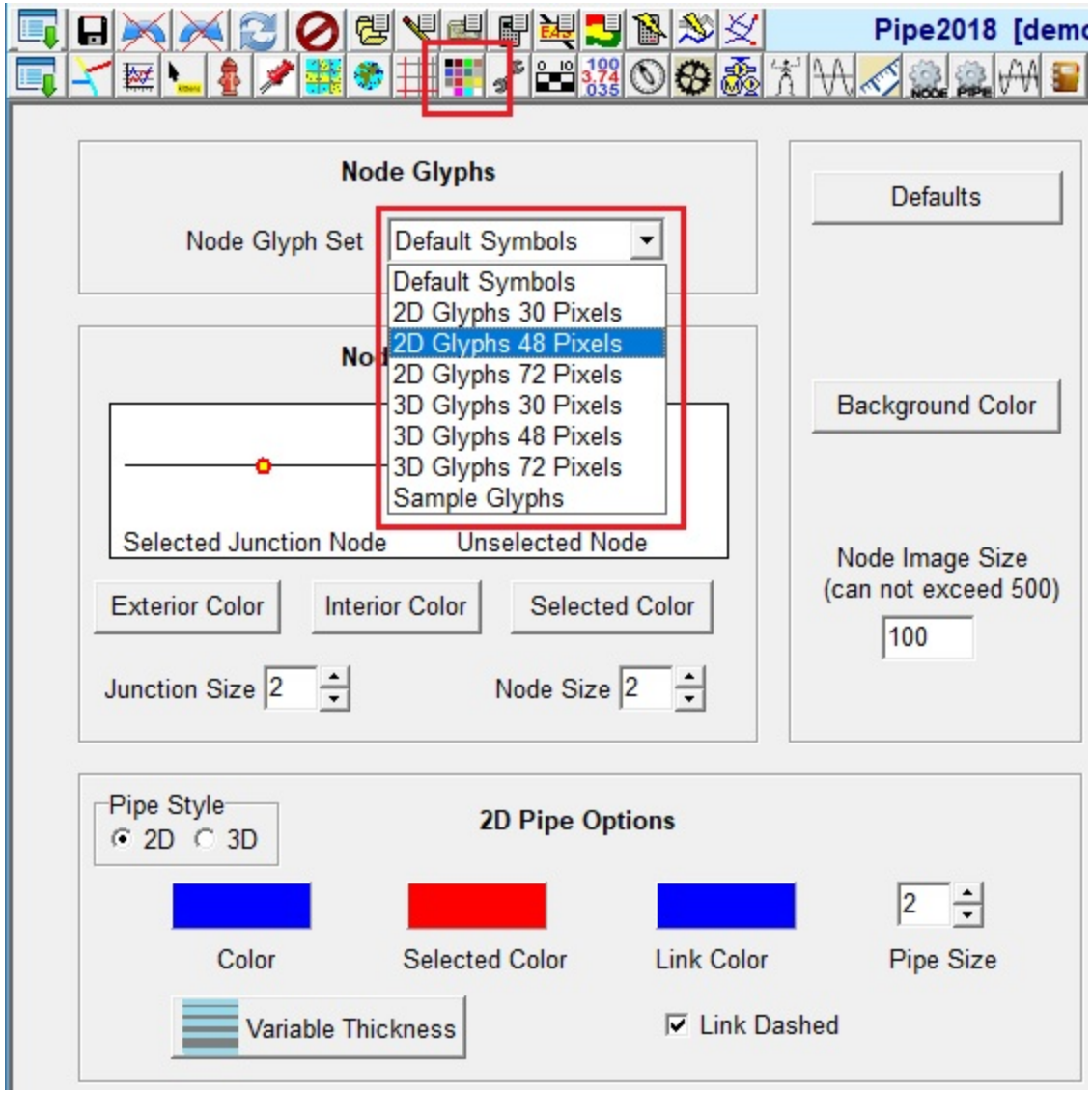
- on/off valves
  - pipe break simulations
- active valves
- regulating valves
- check valves

## 1.3.6 Backgrounds and Map Tools

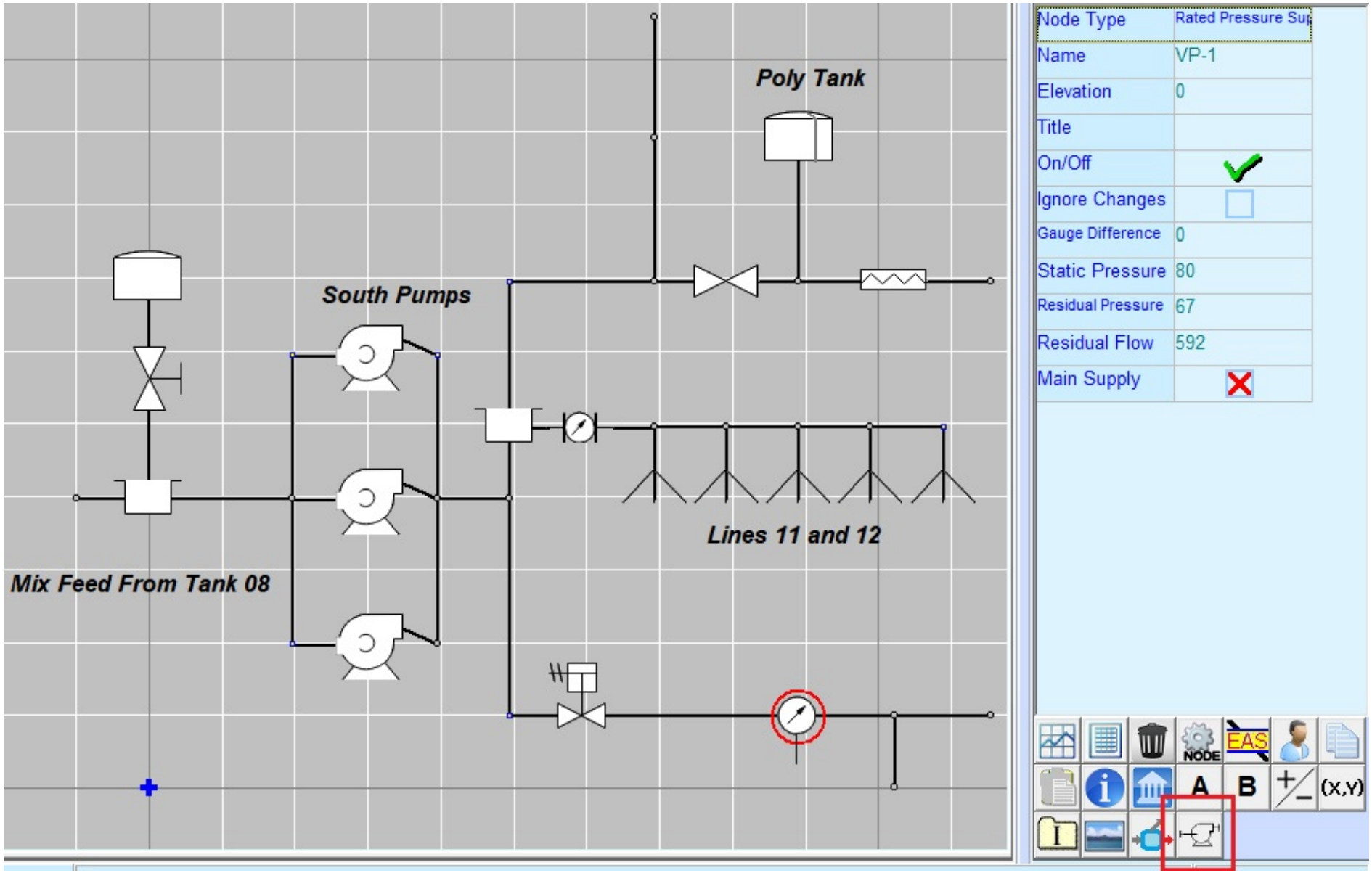
### 1.3.6.1 Node and Pipe Glyphs

**Piping and Instrumentation Diagram Symbols (PID) and Node Glyphs**

Colors menu, use drop-down to select the glyph set.



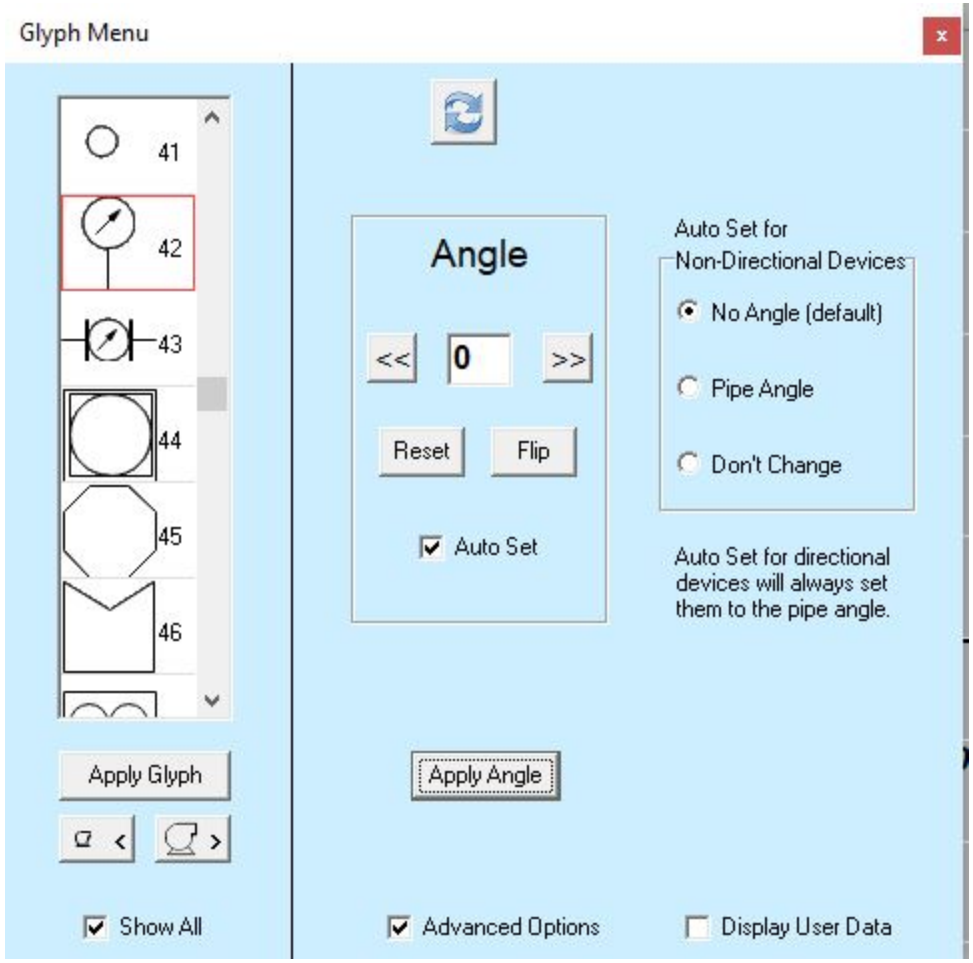

PID (2D) Glyph Example.


Poly Tank
South Pumps
Lines 11 and 12
Mix Feed From Tank 08
Node Type
Rated Pressure Sup
Name
VP-1
Elevation
0
Title
On/Off
Ignore Changes
Gauge Difference
0
Static Pressure
80
Residual Pressure
67
Residual Flow
592
Main Supply
NODE
EAS
A
B
(X,Y)

Selected glyphs are circled in red.

To change an individual glyph, select the Glyph menu icon in the Node Information window as shown above.

This will open the Glyph Menu, which include the **glyph library** as shown:




Show All - will show the full list of available glyphs and symbols.

To apply a new glyph, select it, then click Apply Glyph.

To angle the glyph, set the angle value, then click Apply Angle.

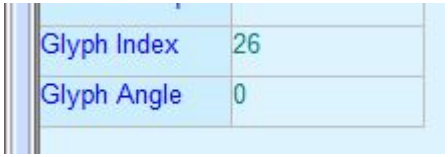
Auto Set (angle) - uses the orientation of the connecting pipes to set the angle for directional nodes. Uncheck to make all glyphs default to 0 angle.

Autoset does not apply to non-directional nodes like tanks, hydrants, and on/off valves

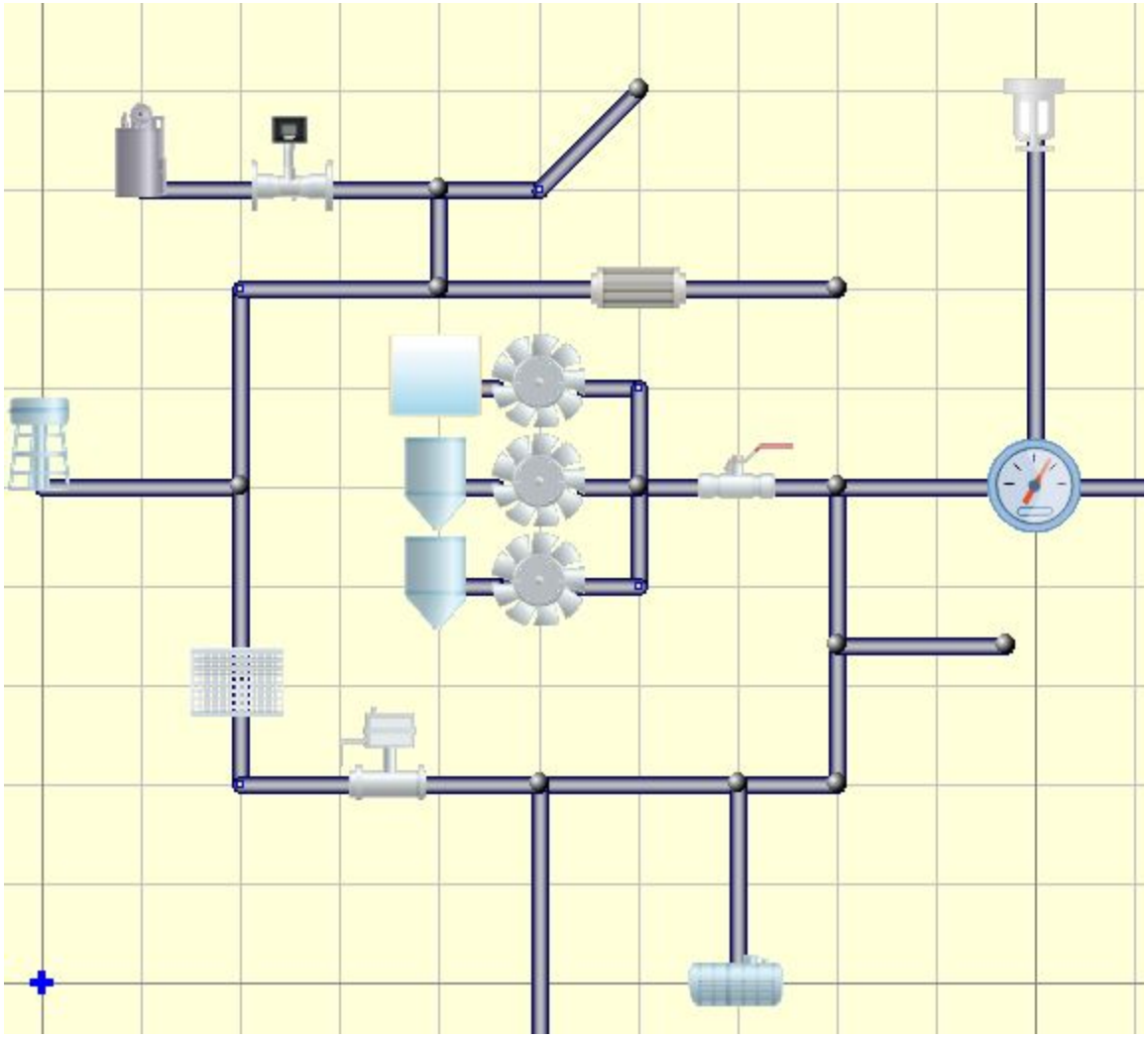
Advanced options:

Angles may be set for non-directional nodes using advanced options.

User data may be displayed. This adds glyph attibutes to the Node Info window, which allows the information to be used/queried to create node groups, edit the glyph symbol for a group, and change the angle of a group.




3D (shaded, realistic) Glyph Example.

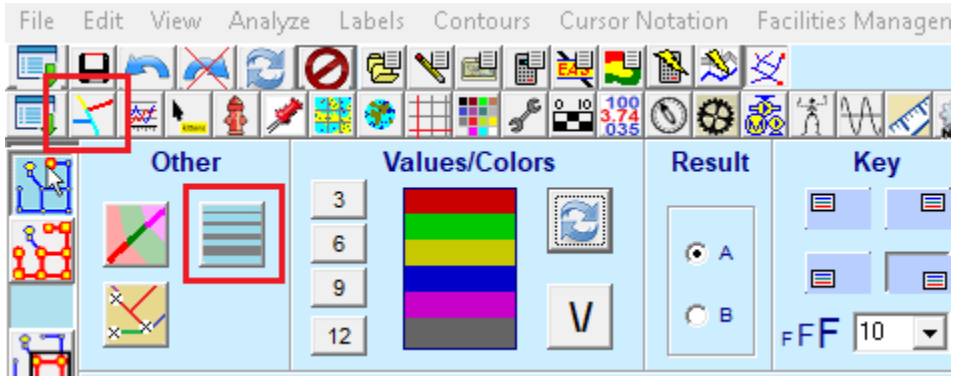


**Variable Pipe Thickness and 3D (shaded) pipes**

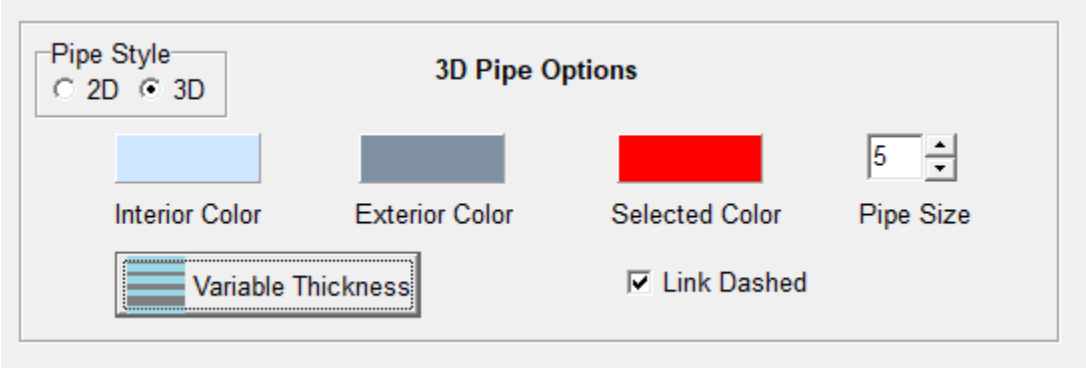
Pipes may have variable thickness based on diameter or other parameters.

Pipes may also be displayed as 3D, which shades the pipe to have depth as shown in the image above for 3D glyphs.

Two ways to launch:

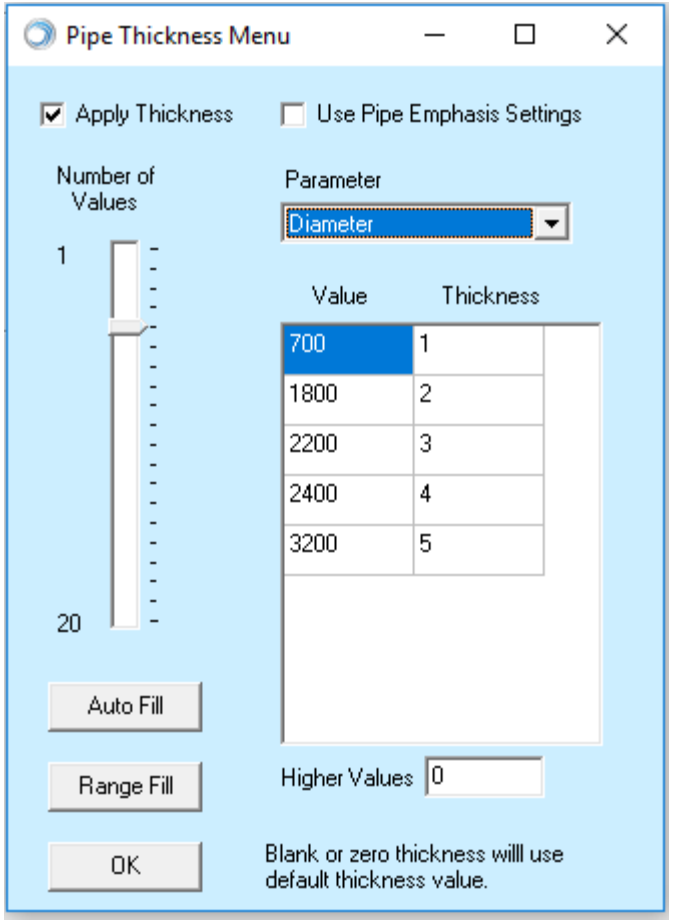



Or from the Colors Menu:




Use the Interior and Exterior colors to alter the quality and texture of the 3D shading

Pipe Thickness menu:




Apply Thickness - this turns on or hides the pipe thickness settings.

Use Emphasis - if pipe emphasis is already set up, these settings can be applied to pipe thickness

Use the slider to set the number of values.

Parameter - although pipe thickness will usually be based on Diameter, any attribute with a numerical value can be used to define varied thicknesses.

Auto Fill - will make equal increments from the range of existing pipe values

Range Fill - enter a low and high value and this will fill in the values between in equal increments.

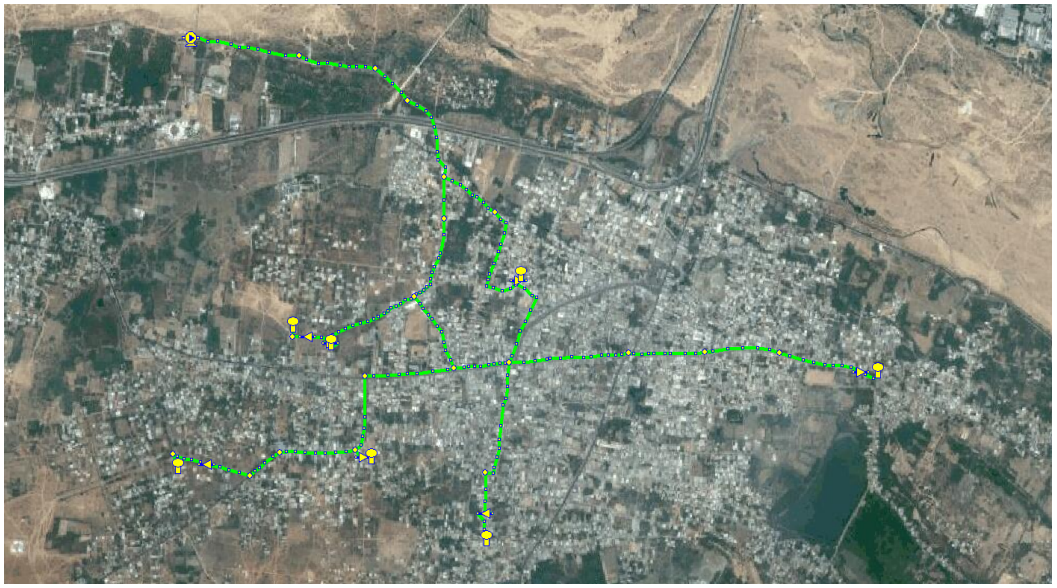
Once the Values are applied to the thickness table, enter numerical values greater than zero.  Demical values are ok, range from >0 through 10 is typical.

## 1.3.6.2 Plan View-Profile View

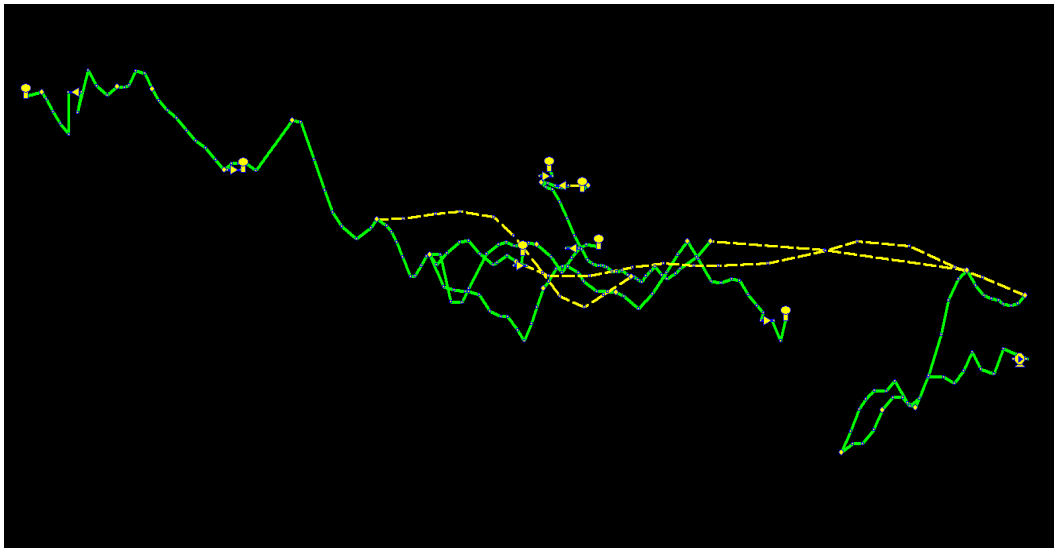
**Important:** Must make a profile and save the .p2k file first. This creates some internal data that is used in the profile view generation.

Go to View menu.

Plan View of a piping system

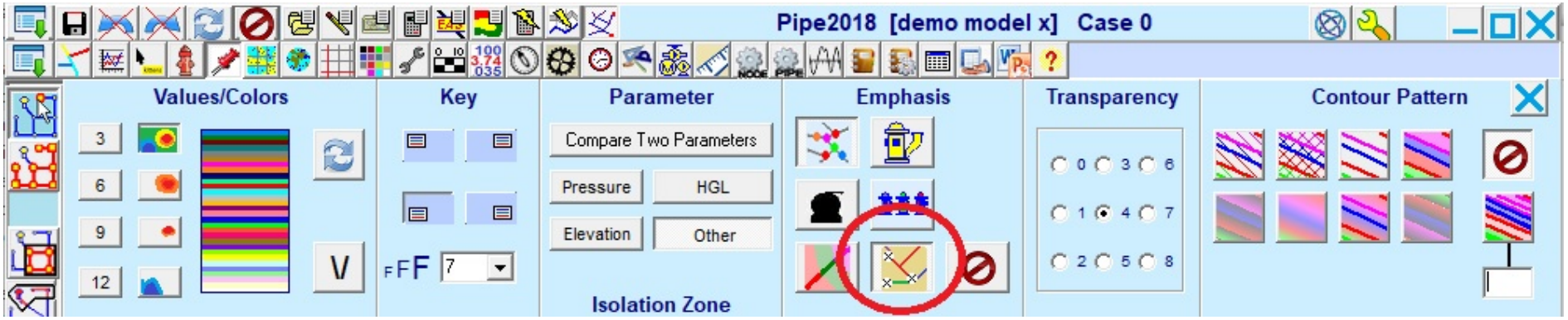


Profile View

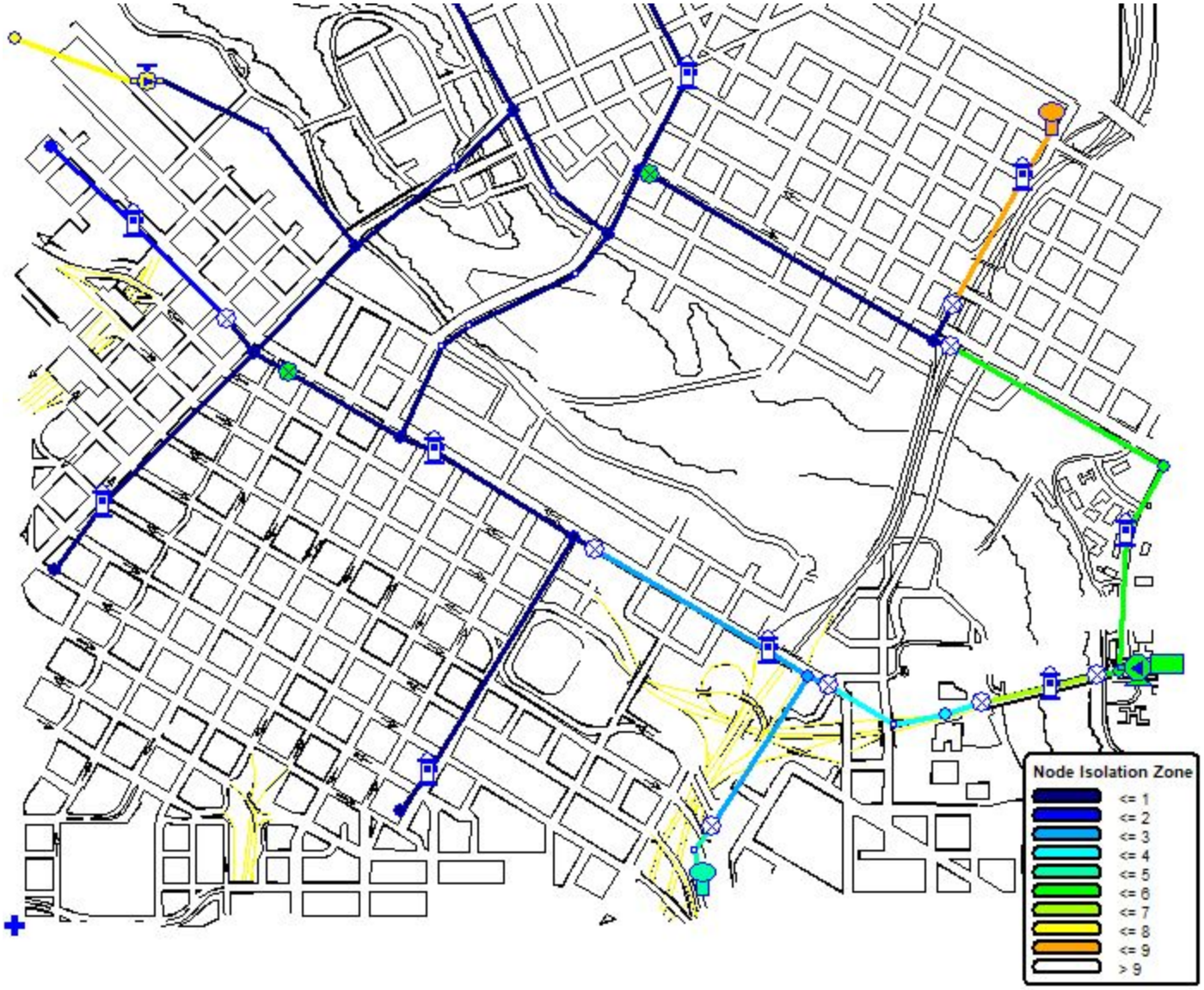


## 1.3.6.3 Isolation Zones

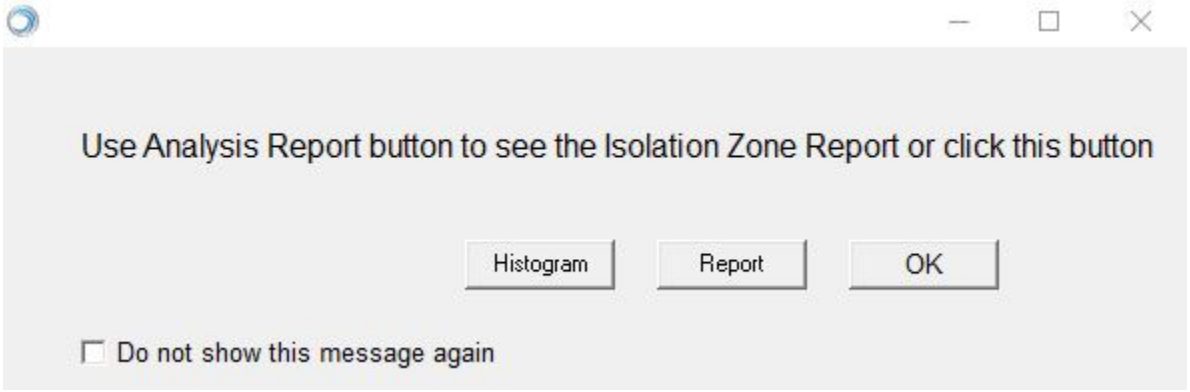
Go to Node Contours menu and select Isolation Zones.




All of the potential isolation zones are emphasized in the system as shown in the example below.




A histogram of the effected demands may be created either by selecting the option in the prompt as shown.




Or by going to the Report as shown.

Histogram:

Here is an example Isolation Zone Report.

```
+-------------------------------------+
|   Pipe2000 Isolation Zone Report    |
+-------------------------------------+

System Name: demo model x.p2k
Date: 01/11/18
Number of Zones: 9
Demand Units: US gallons/min

Zone      Demand       Extra       Total   Isolated Zones
  1        273.0        45.0       318.0   2 8
  2         45.0         0.0        45.0
  3        120.0         0.0       120.0
  4          0.0         0.0         0.0
  5          0.0         0.0         0.0
  6         12.5         0.0        12.5
  7          0.0         0.0         0.0
  8          0.0         0.0         0.0
  9          0.0         0.0         0.0

Zone  Closed Valves
  1   AV-1          V-10          V-3           V-5           V-7
  2   V-3
  3   V-5           V-8           V-9
  4   V-2           V-9
  5   V-8
  6   V-1           V-7
  7   V-1           V-2
  8   AV-1
  9   V-10

Zone  Pipes (ft)
  1   P-1 (2855)    P-10 (1649)   P-11 (422.9)  P-14 (172.2)  P-16 (1546)
      P-17 (2950)   P-18 (1942)   P-19 (2018)   P-20 (1238)   P-22 (1429)
```

```
     P-23 (2663)    P-4 (392.5)    P-5 (3296)     P-6 (1983)     P-7 (678)
     P-8 (231.3)    P-9 (3138)
2    P-11 (2378.1)
3    P-12 (215.5)   P-13 (1726.6) P-8 (2401.7)
4    P-12 (1248.5) P-15 (374.6)
5    P-13 (672.4)
6    P-14 (2368.8) P-15 (211.6)   P-2 (2075)     P-21 (292)     P-3 (190)
7    P-15 (1148.8)
8    P-24 (1339)
9    P-4 (1944.5)
```

## 1.3.6.4 Internet Background Maps and Automatic Elevation Retrieval

---

Internet Background Maps - Pipe2012 and later versions.

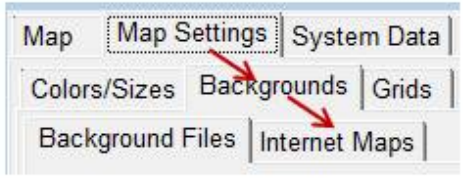
Google map and Google elevation source - Pipe2014 and later versions.

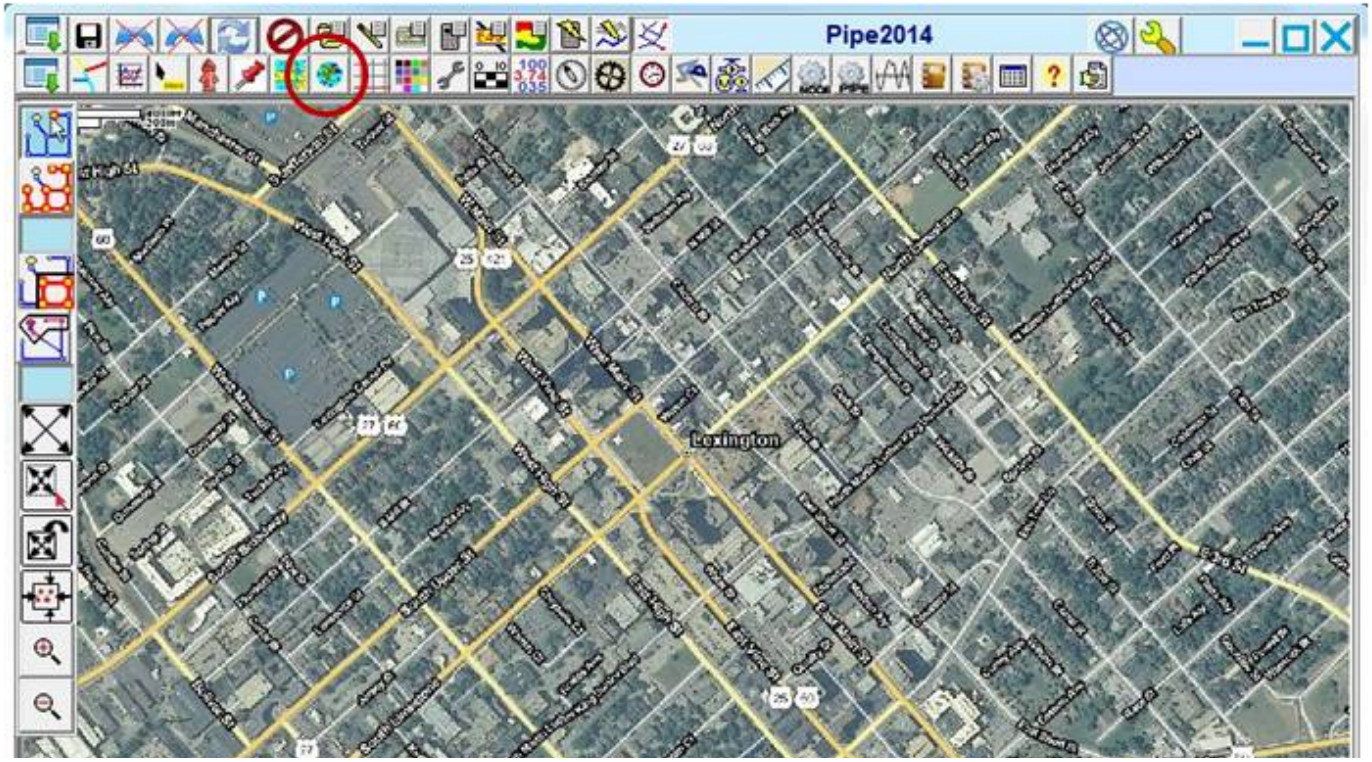
Built-in UTM and State Plane Coordinate systems - Pipe2014 and later versions

.

.

# Coordinates and Internet Maps



Layout Mode
(6678396, 4978606)
Pressure - Flow

.

_____

.

# Coordinates Options

The following coordinates may be applied:

- Built-in coordinates.
  Pipe2014 and later versions. Many coordinate systems have been built in such as State Plane and UTM coordinates.

- WGS84 and Lat/Long
  These are default systems used by Google (Pipe2014 and later) and Mapquest.

- Default coordinates.
  Sometimes no specific coordinate system is associated with a model, in which case a default coordinate set is used. Advanced Scaling is used to align the model with the Internet map. Coordinates for the model will be

changed to WGS84 (or Lat/Long) by the scaling process.

10

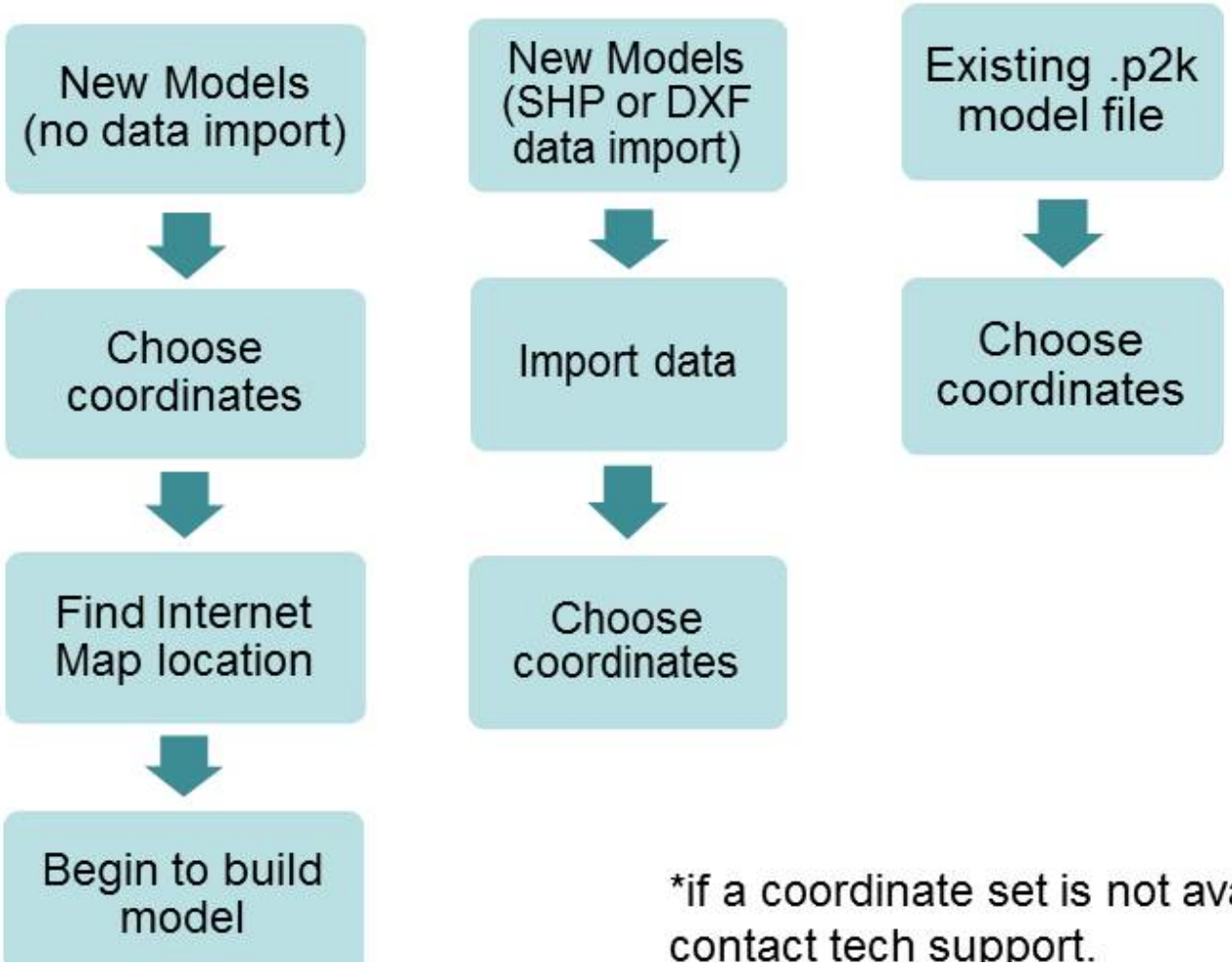
.

_____

.

## Using **Built-In Coordinates** *

State Plane and UTM (Pipe2014 and later)




*if a coordinate set is not available, contact tech support.

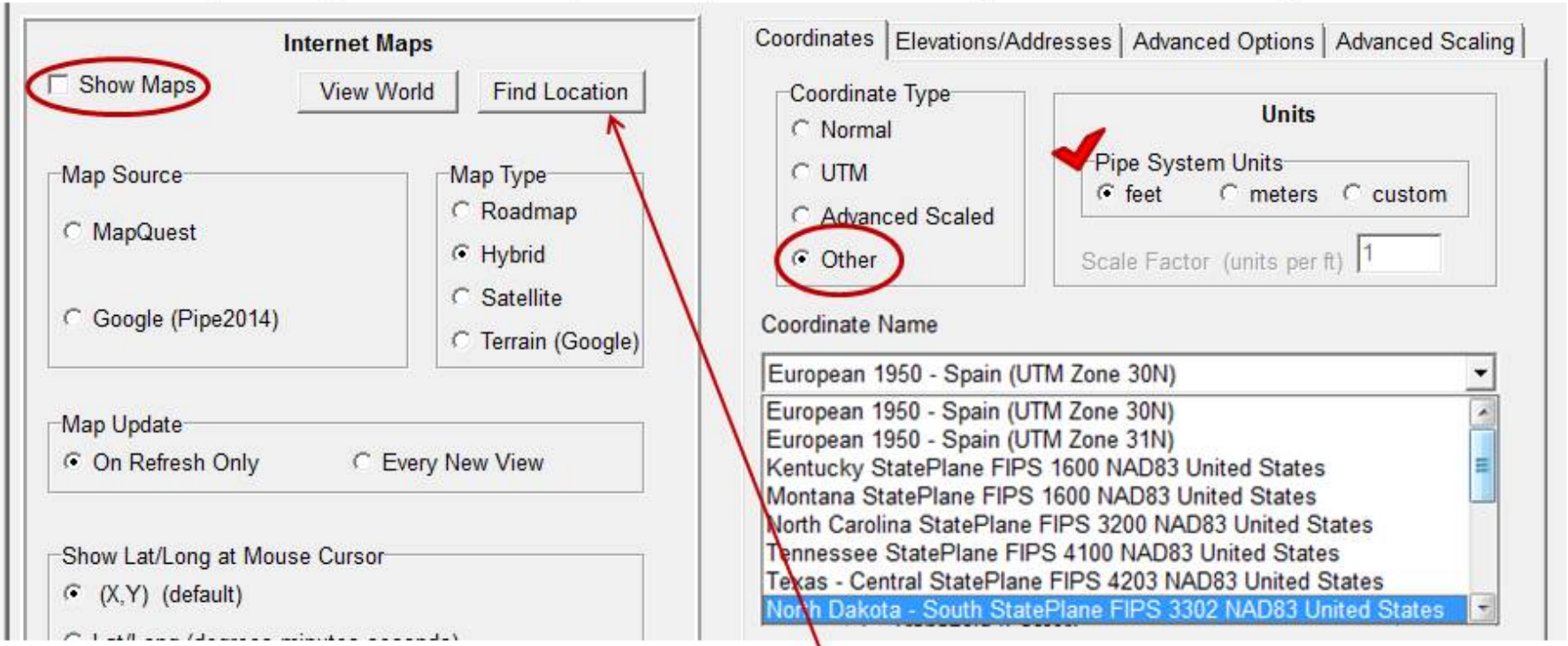
11

.

.

# Built-In State Plane (and old UTM)

After opening new, existing, or imported model, go to Internet Maps menu.




1. Check Show Maps.
2. Select “Other” for Coordinate Type.
3. Choose Coordinate Name.
4. Set the Units to ft or m.

5. (New models with no imported data only) Use Find Location button and enter city or landmark. 12

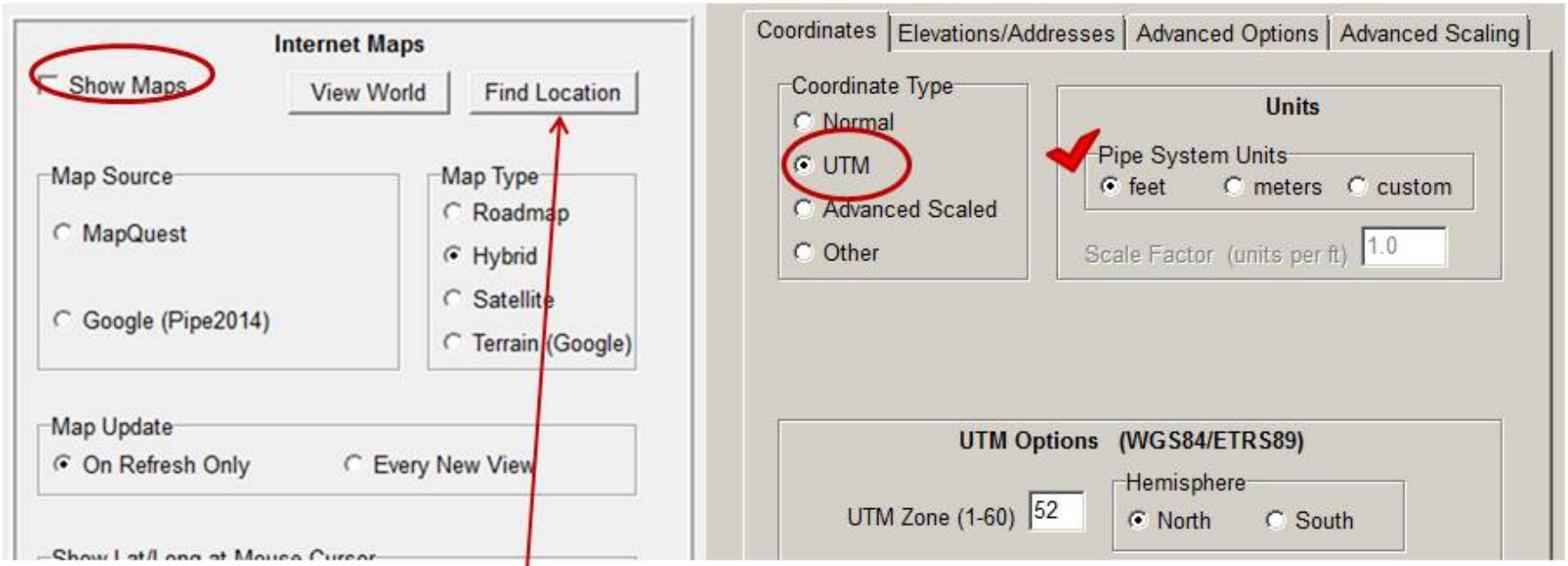
.

---

.

# Built-In UTM Coordinates

After opening new, existing, or imported model, go to Internet Maps menu.




1. Check Show Maps.
2. Select "UTM" for Coordinate Type.
3. Set the Units to ft or m.
4. Enter UTM Zone.
5. Select Hemisphere

OR

1. Select UTM
2. Click Find Location button and enter city or landmark.
3. Correct UTM zone will be found. Be sure Units are correct

6. (New models with no imported data only) Use Find Location button and enter city or landmark.

correct.

13

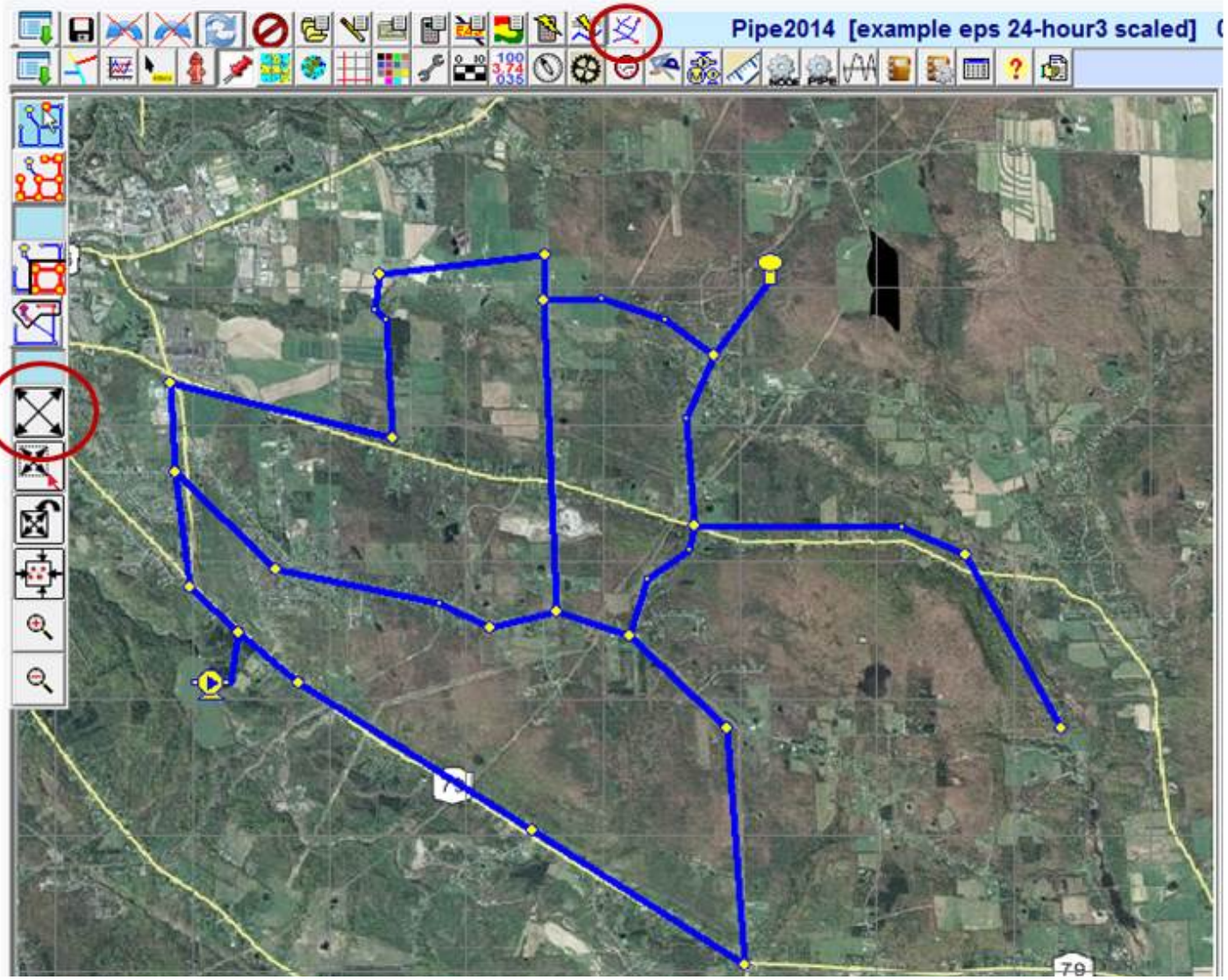
.

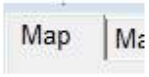
---

.

Once coordinate selection is complete, map view is shown with Internet Background

Use Zoom All to position pipes in map window (if applicable)



14

.

_____

.

# WGS84 and Lat/Long

New Models (no data import)

↓

Find Internet map location

↓

Begin to build model

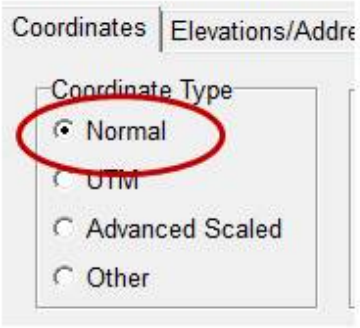
New Models (SHP or DXF data import)

↓

Import data*

↓

Find Internet map location




Select “Normal”

LAT/LONG - contact tech support for import and/or conversion of lat/long.

15

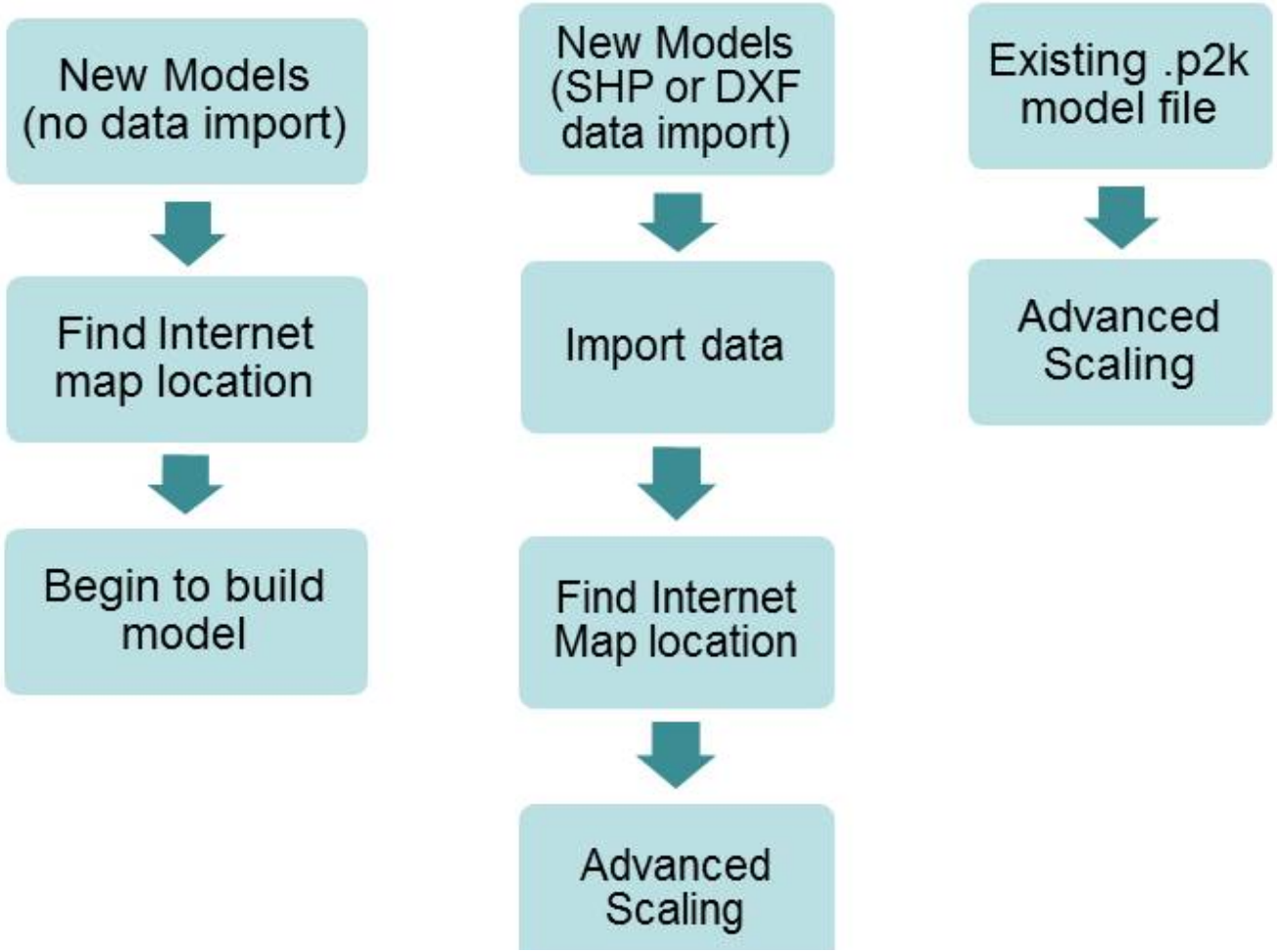
.

_____

.

# Default Coordinates

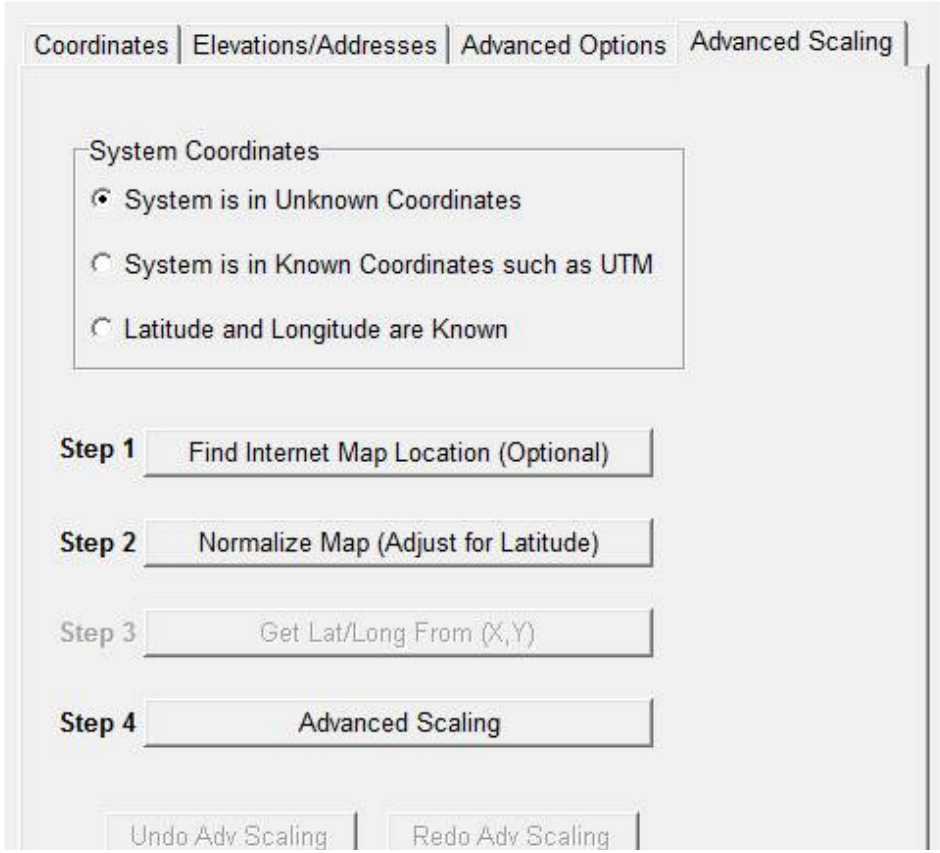
Model uses no specific coordinate set



16

# Advanced Scaling

For existing and imported models with no specified coordinates




1. Check Show Maps box
2. Go to Advanced Scaling tab.
3. Select System is in Unknown Coordinates
4. Find Internet Map location (**Step 1**)
5. Normalize Map (**Step 2**). Answer "Yes" to "Move and scale pipes to this location?".
6. Create Text nodes (see next slide).
7. Once text nodes are created, click Advanced Scaling (**Step 4**).

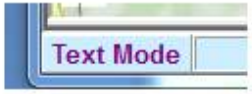
If the system does not align properly, the process may be repeated with new text

nodes (reload unsaved model or use Undo Adv Scaling button).

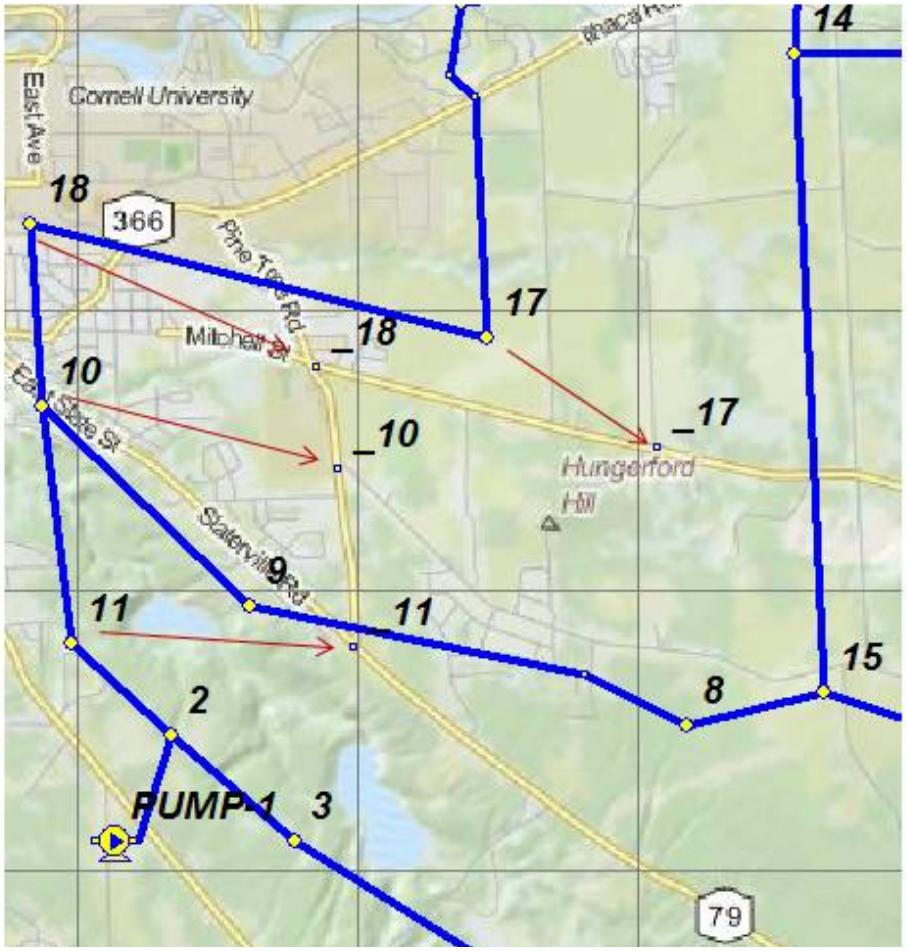
.

---

# Laying Out Text Nodes for Advanced Scaling



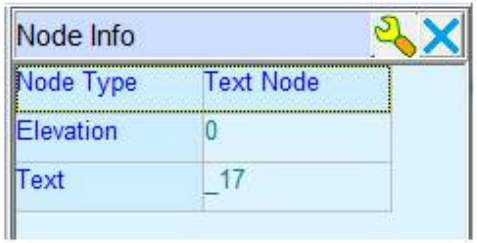
Classic




1. Go into Text Mode, click bottom left corner (KYnetic).

2. Click to place text nodes in the location to where corresponding junction node will be placed as shown.

3. Enter the name of the corresponding node in the Text field prefixed with an underscore "_"




4. Do this for several nodes spread throughout the system, at least 3.

Junction names and text nodes

.

---

.

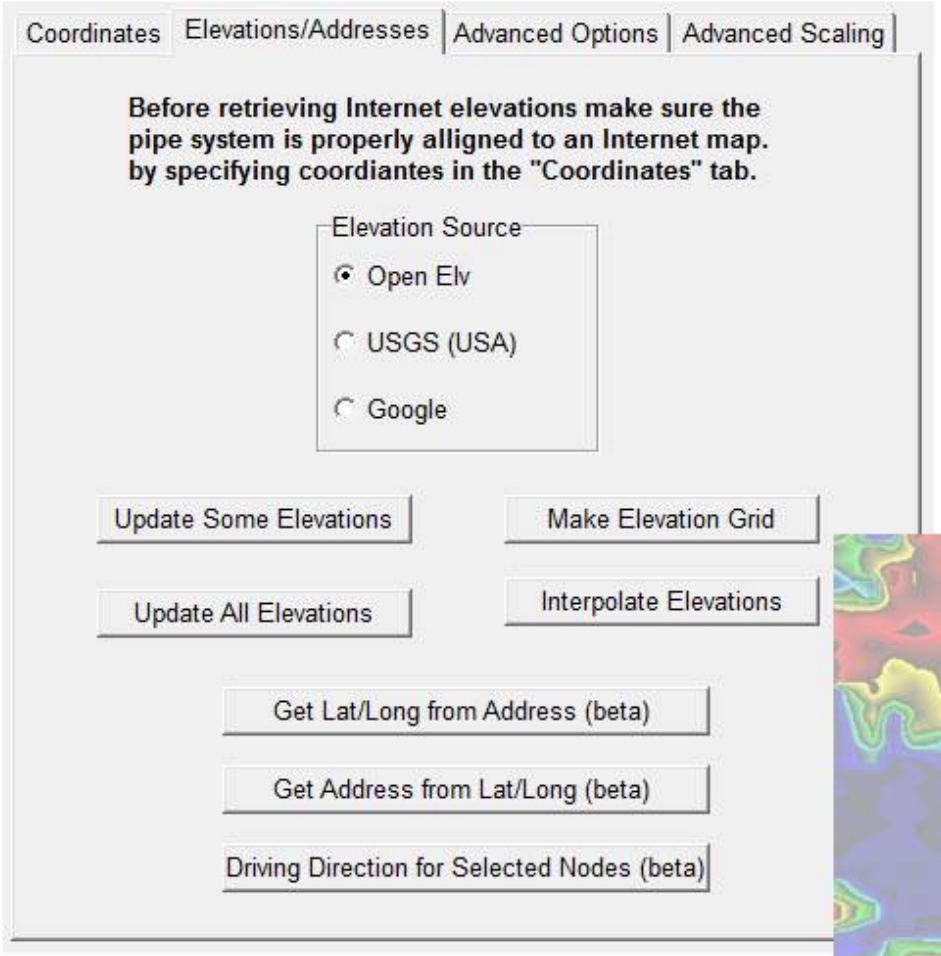
# Options for Aligned Internet Maps

- Import Elevations
- Import Elevation Grid
- Map Appearance and Source
- Mouse Cursor Coordinate Values
- Lat/Long Pipe Lengths

- Other Lat/Long Data Uses

.

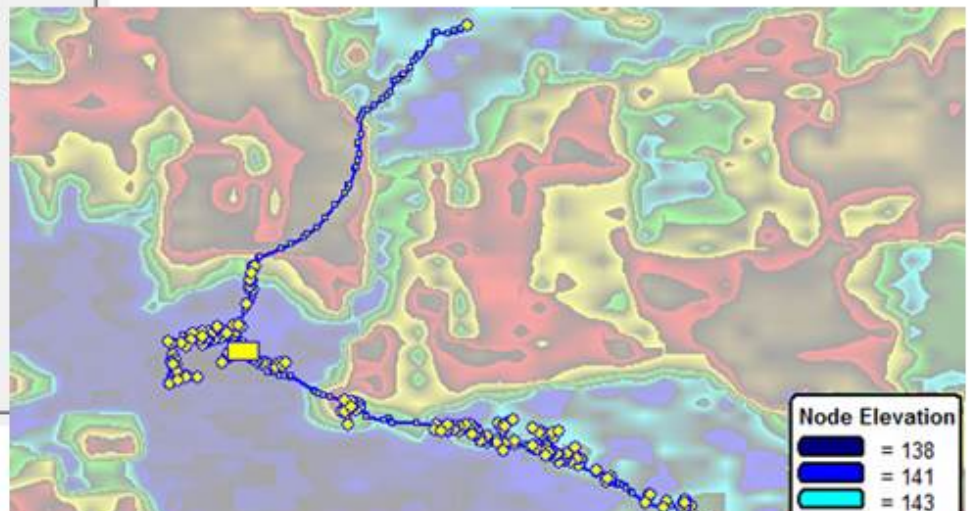
# Import Elevations




Select Elevation Source.

Click “Update All (or Some) Elevations” or “Make Elevation Grid”.

Elevation grid adds reference points outside pipe model for contouring elevations.



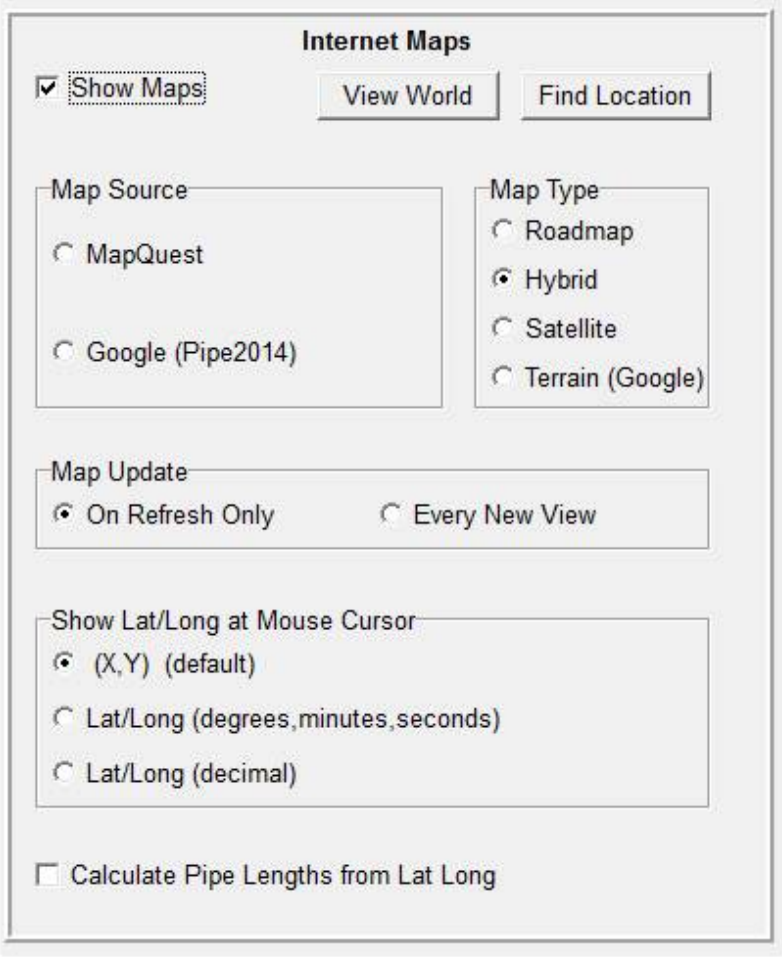

Interpolate Elevations - assigns

elevation values to zero elevation nodes interpolated from adjacent node elevations.

.

---

# Internet Maps - Settings




Map Source
- •Mapquest
- •OpenStreetMaps – Mapquest
- •OpenStreetMaps – OSMA
- •Google (for Pipe2014 version)

Map Type

Map Update

To save loading time, default is "On Refresh Only". Use the Refresh button as you pan and zoom.

Show Lat/Long at Mouse Cursor (see next slide)

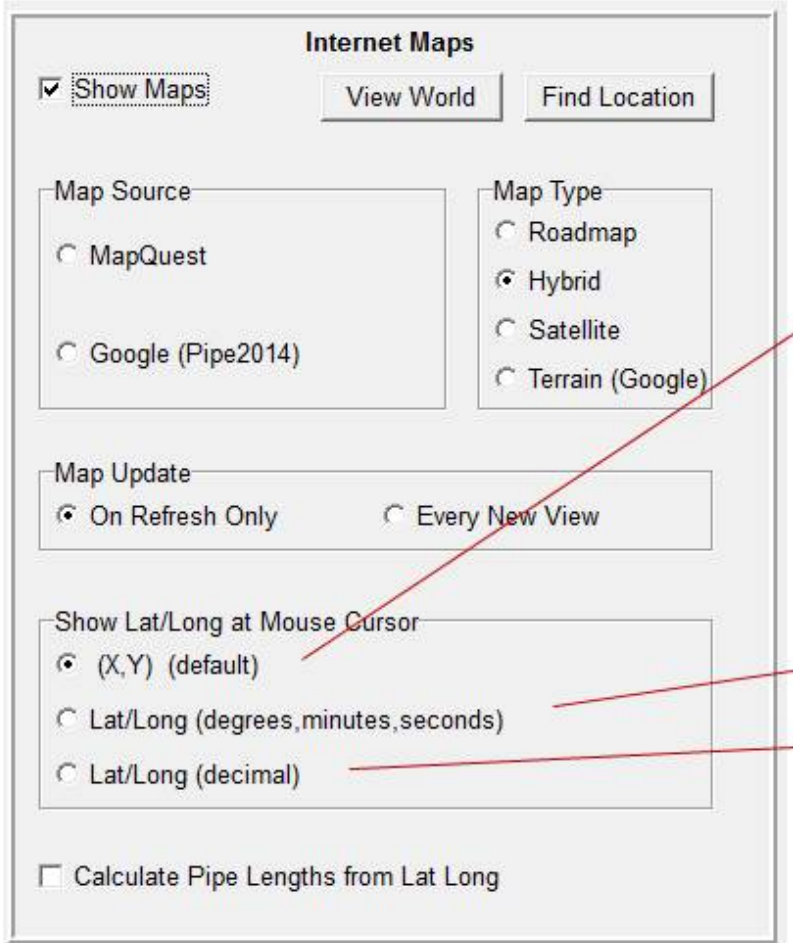
Calculate Pipe Lengths form Lat Long – when checked, pipe length from Advanced Scaling option may be ignored in favor of Lat/Long reference. Do not use with Built-in coordinate systems.

.

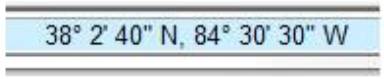
_____

# Mouse Pointer Values

Options for showing coordinate value at mouse pointer.



The values shown are the coordinates in which the model was created. Note: the Advanced Scaling option can changes these values.

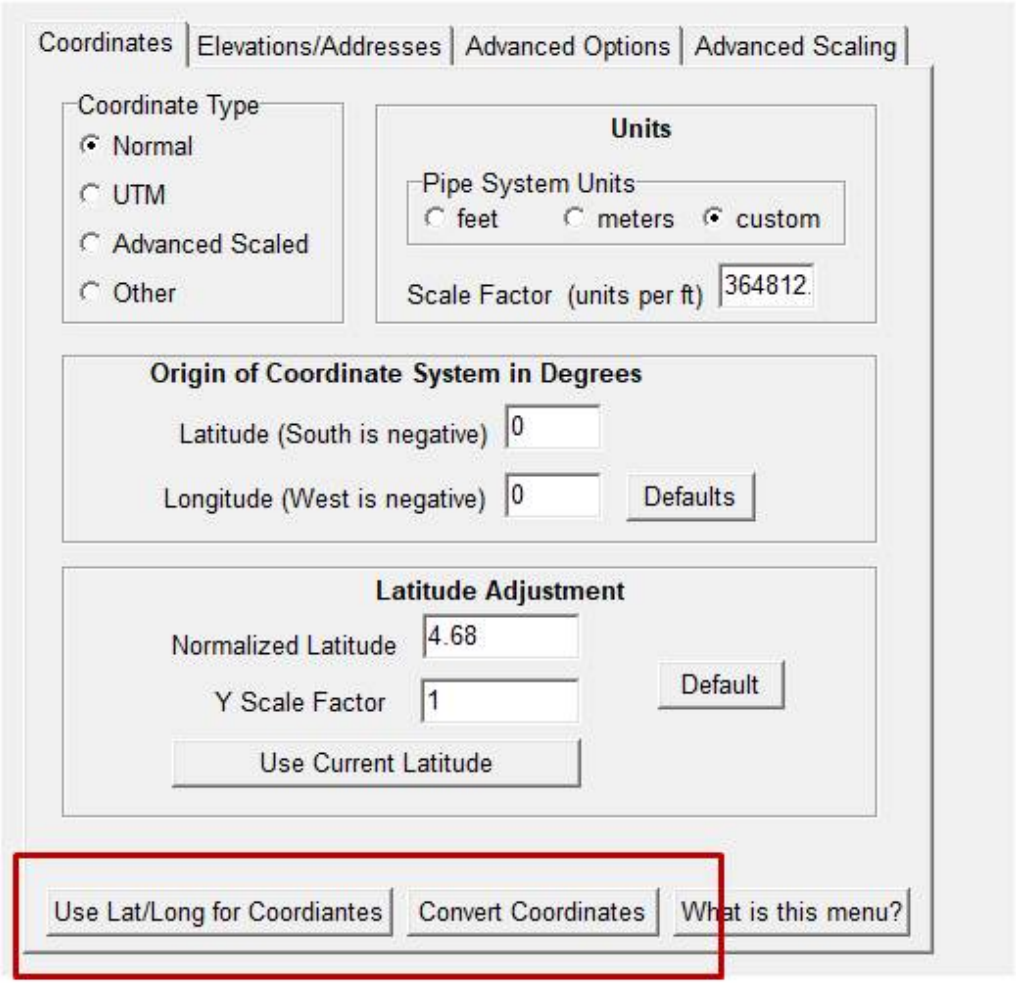


Lat/Long are WGS84, same as Google and Mapquest maps.

.

Lat/Long and Convert Coordinates

# Lat/Long and Convert Coordinates

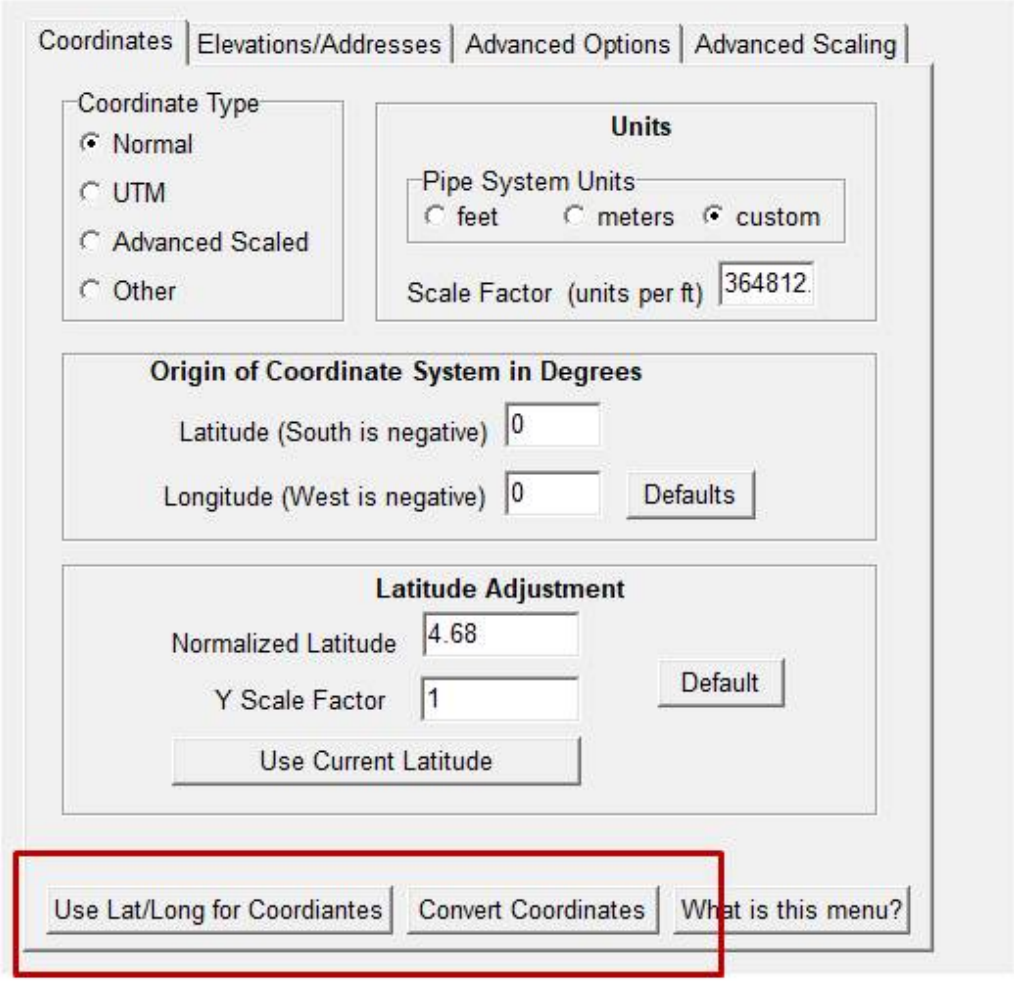


# Use Lat/Long

If a system's (X, Y) coordinate values are in Lat/Long format then this feature will configure the background maps to display properly for this situation. For example, a node at (X, Y) location (39,-96.67) will be displayed on the background map at latitude 39 degrees north and longitude 95.67 degree west which is near Topeka, Kansas.

Sample location for original coordinates

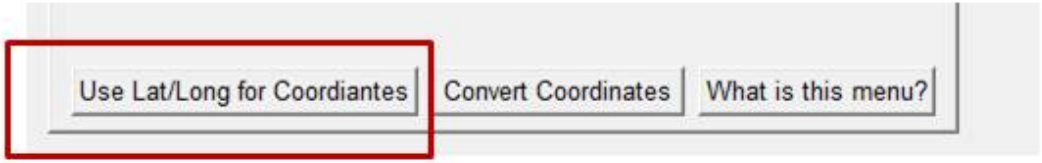
# Lat/Long and Convert Coordinates



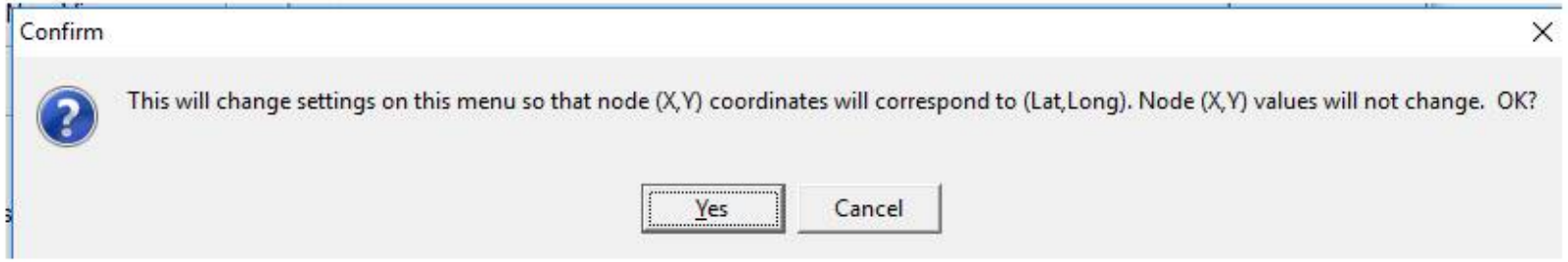
# Use Lat/Long

If a system's (X, Y) coordinate values are in Lat/Long format then this feature will configure the background maps to display properly for this situation. For example, a node at (X, Y) location (39,-96.67) will be displayed on the background map at latitude 39 degrees north and longitude 95.67 degree west which is near Topeka, Kansas.

Sample location for original coordinates

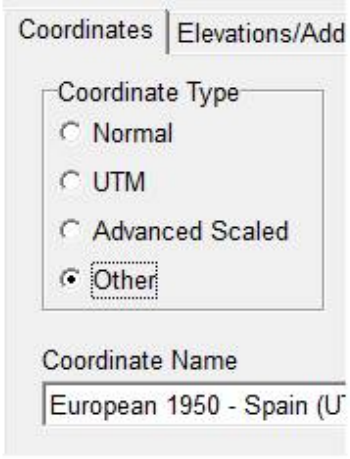



You will be prompted as follows:



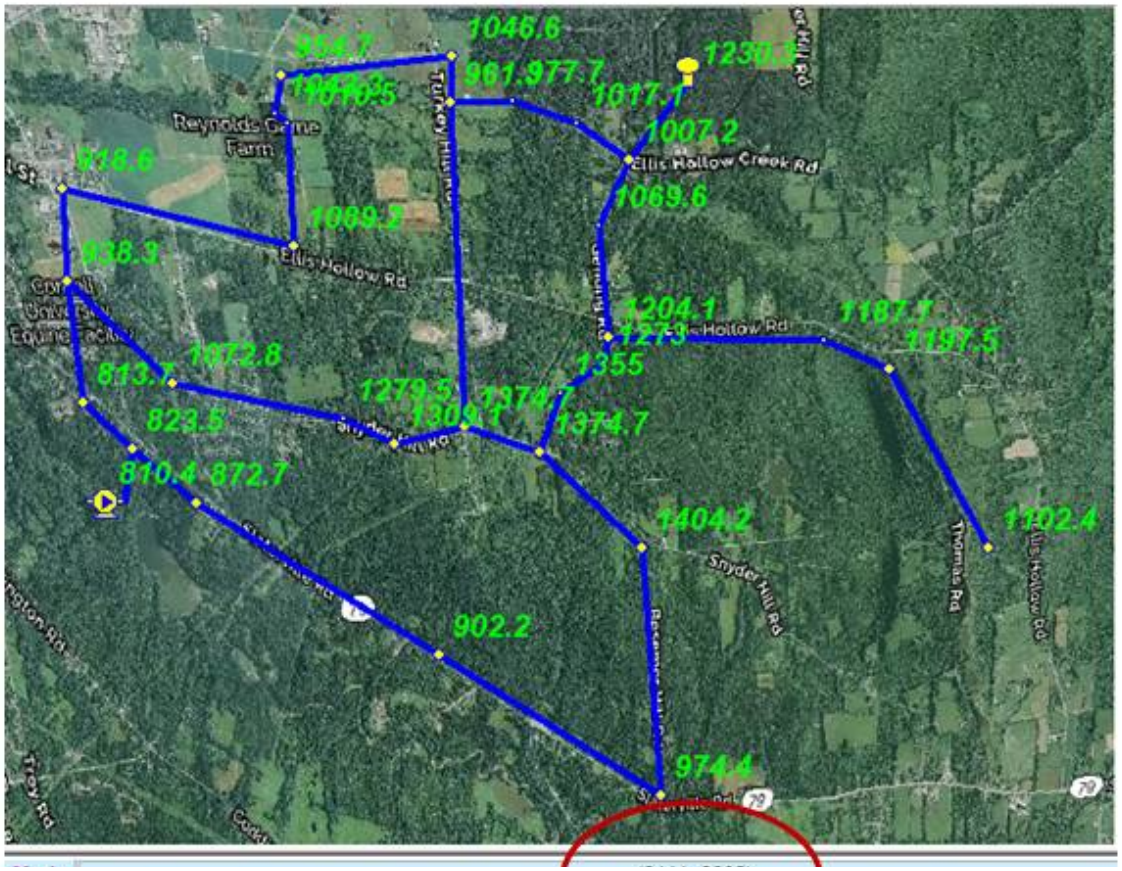
# Convert Coordinates

- Coordinates can be converted to Lat/Long or one of the coordinates from the list of available coordinates. Select Other under Coordinates tab to choose from the built-in coordinates.

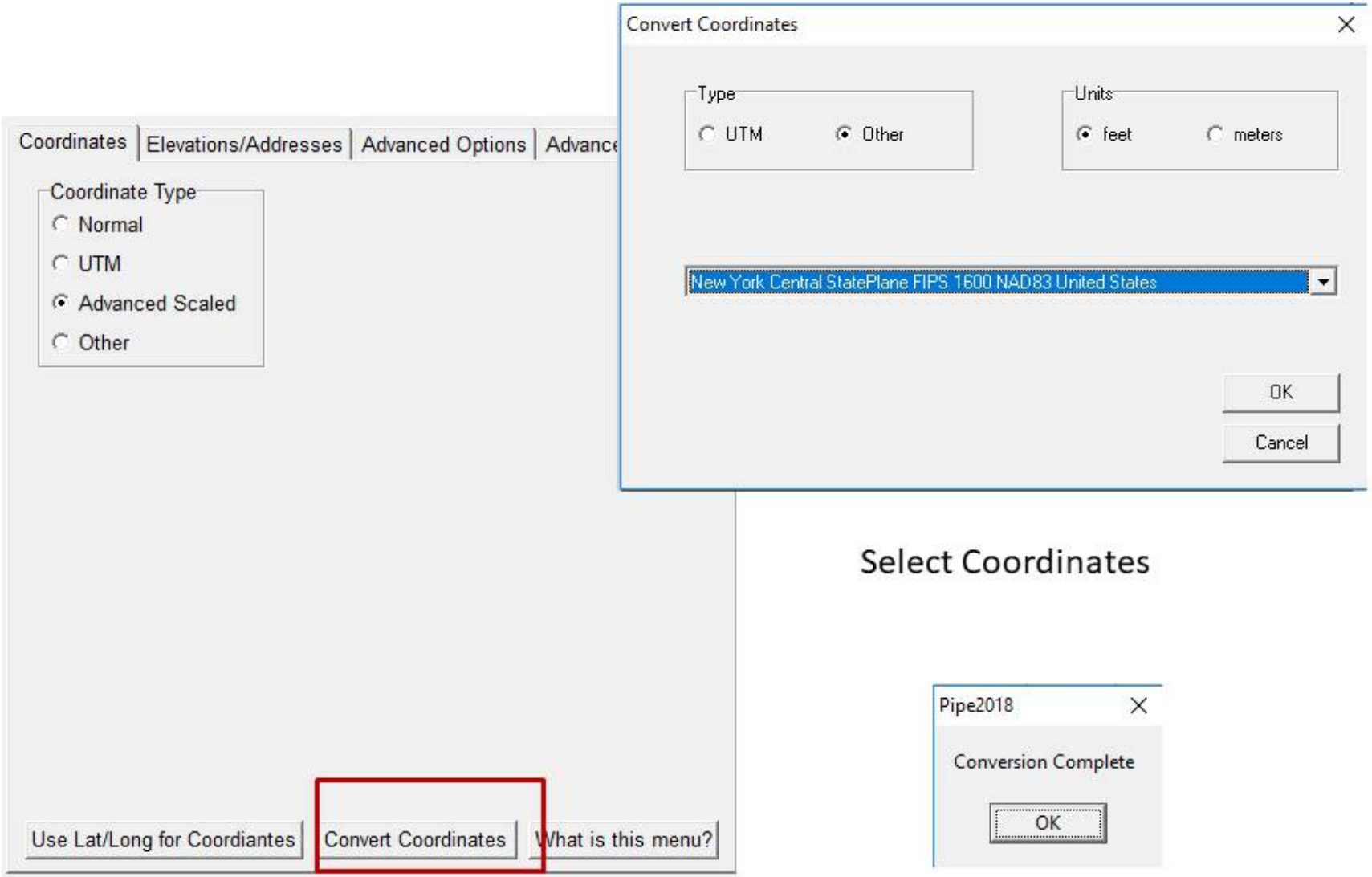


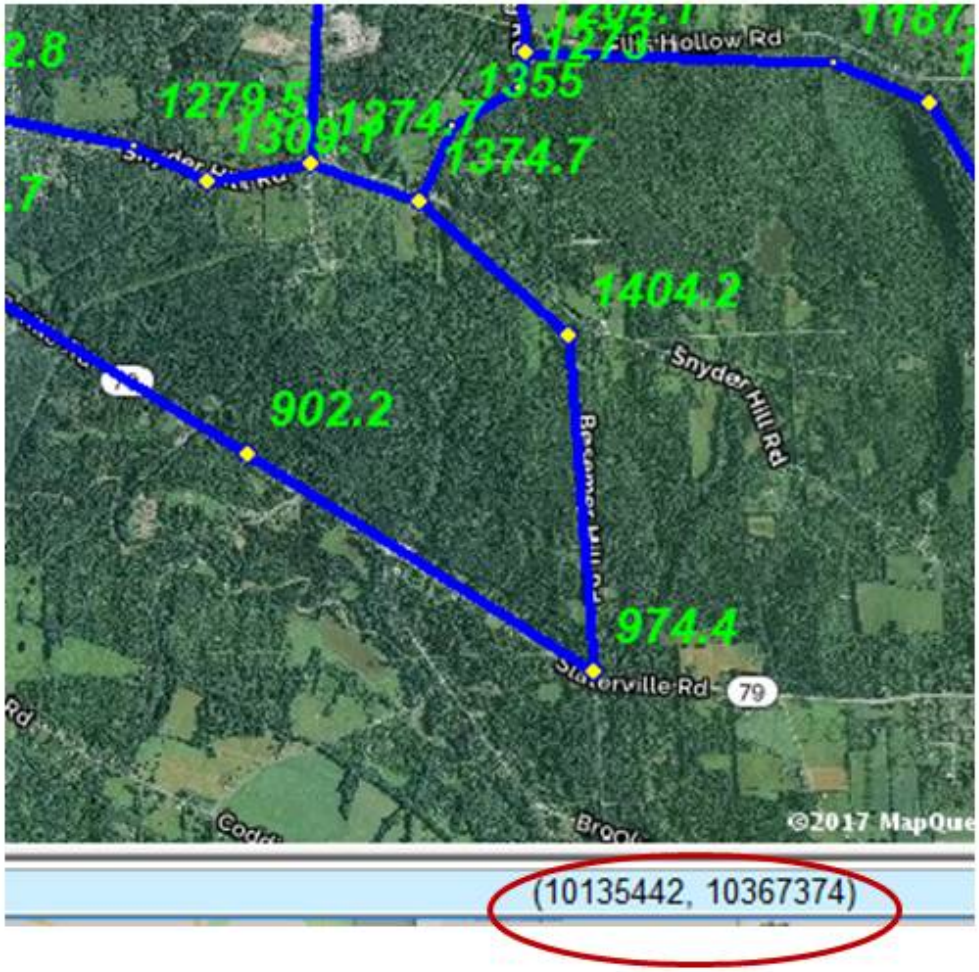
# Convert Coordinates

An advanced scaling process on an pre-existing model has resulted in a nice alignment and good imported elevation data. However, the coordinates are "meaningless"



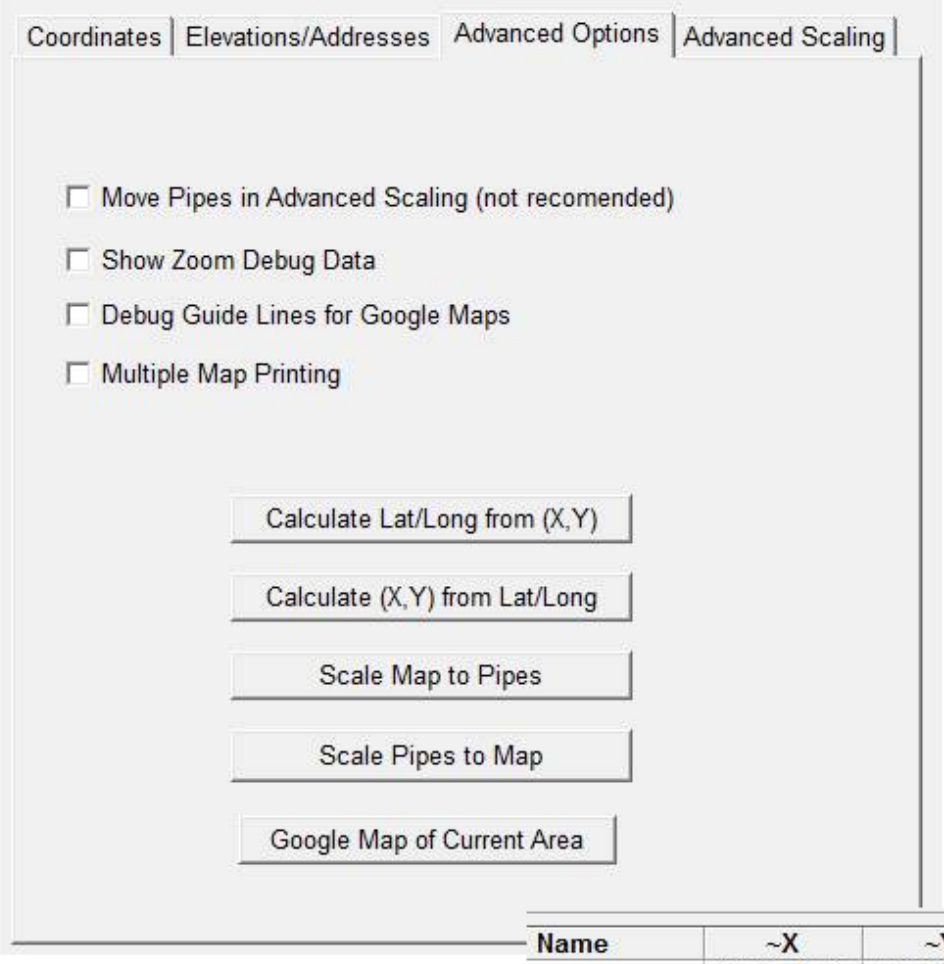
Sample coordinate value


Coordinates
Elevations/Addresses
Advanced Options
Coordinate Type
Normal
UTM
Advanced Scaled
Other
Use Lat/Long for Coordiantes
Convert Coordinates
What is this menu?
Convert Coordinates
Type
UTM
Other
Units
feet
meters
New York Central StatePlane FIPS 1600 NAD83 United States
OK
Cancel
Select Coordinates
Pipe2018
Conversion Complete
OK




Location is preserved, coordinates are converted.

# Advanced Options




Once scaling is complete, "Calculate Lat/Long from (X,Y)" may be used to add Lat/Long values to the data table.

Go to Data Table (Hidden) to view Lat Long values.

"Calculate (X, Y) from Lat/Long" can be used if the original model data was imported with lat/long (WGS84). First select the coordinate system from the Coordinates tab, then use this button to obtain the (X, Y) values for each node in the selected coordinate system.

Google Map of Current Area - will open Google Maps in the default Internet browser.

| Name | ~X | ~Y |
|---|---|---|
| | 1848558.497 | 456513.6741 |
| | 1849344.672 | 456082.7979 |
| | 1849476.214 | 455997.5919 |

| ılts | Latitude | Longitude |
|---|---|---|
| | 38.0394296 | -84.4969634 |
| | 38.0611842 | -84.485132 |
| | 38.0630013 | -84.4870308 |

.

---

.

## 1.3.6.4.1 Driving Directions - Internet Maps

Pipe2014 and later versions. Only available if an Internet Map is applied to the model.

To get Driving Directions to node locations in a model, go into Group Mode and select the nodes.

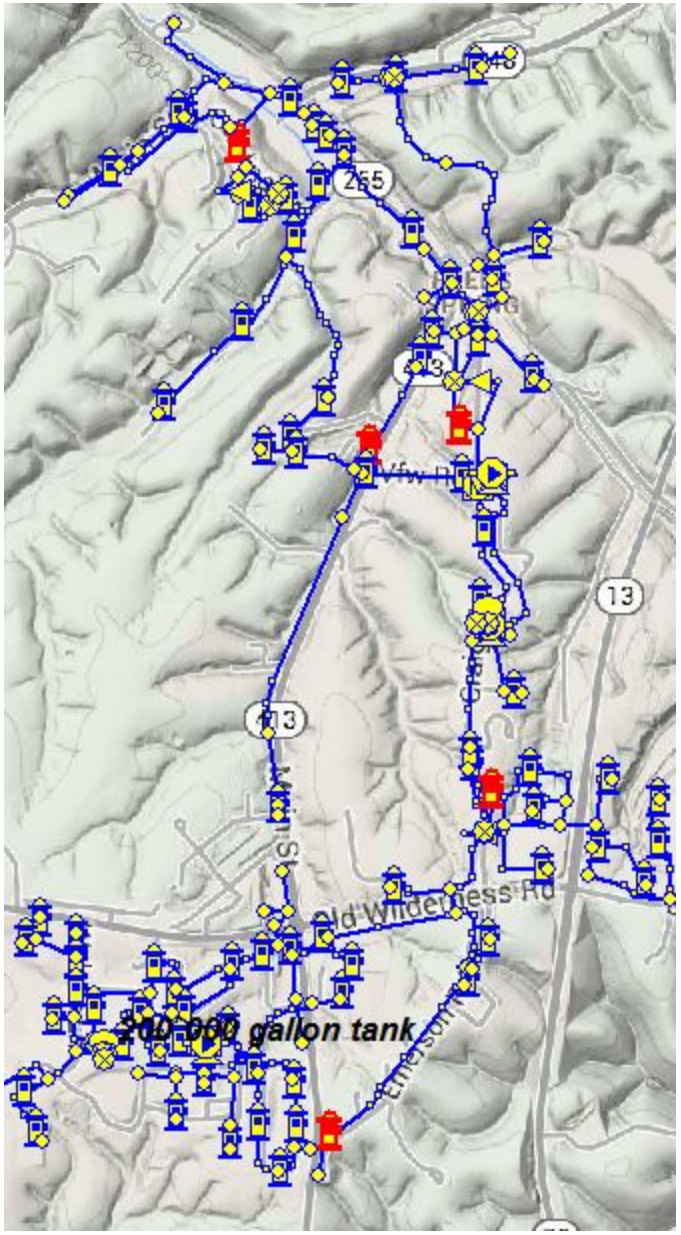
Go to Internet Maps menu and click the Elevations/Addresses tab.

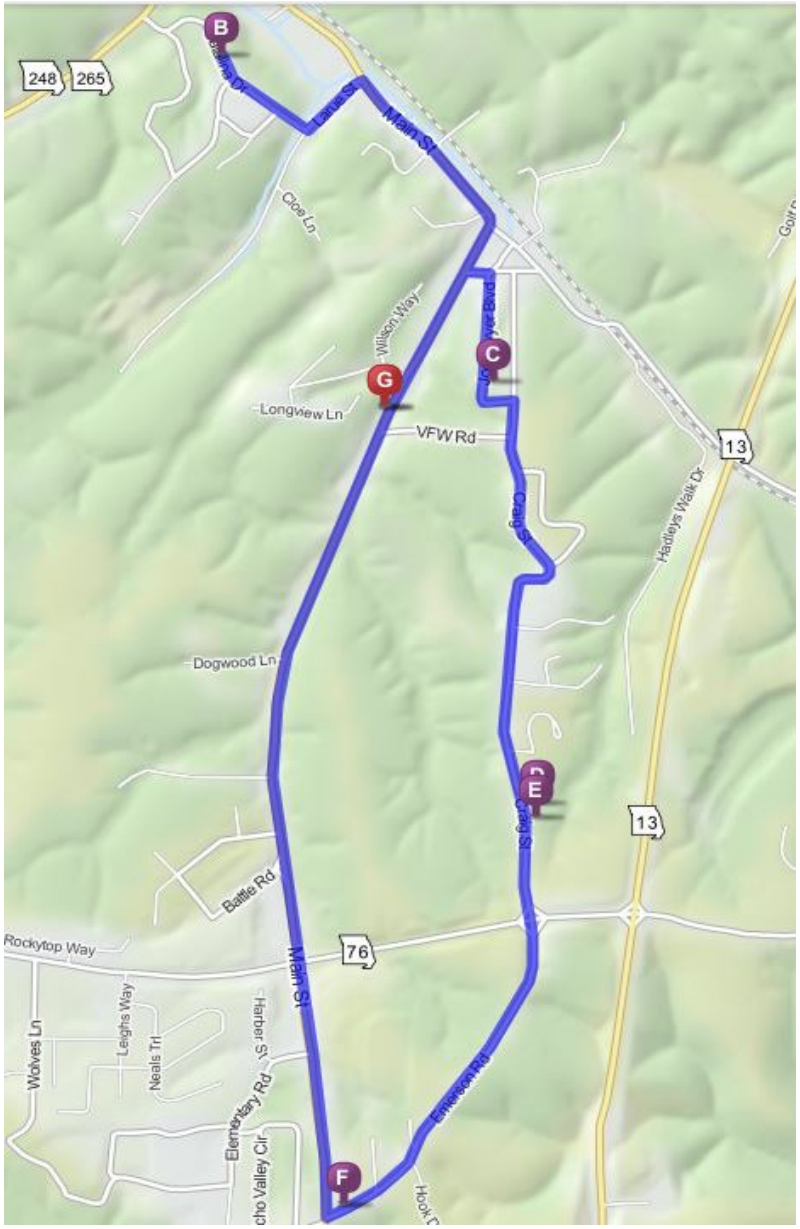
Click the button Driving Directions for Selected Nodes.

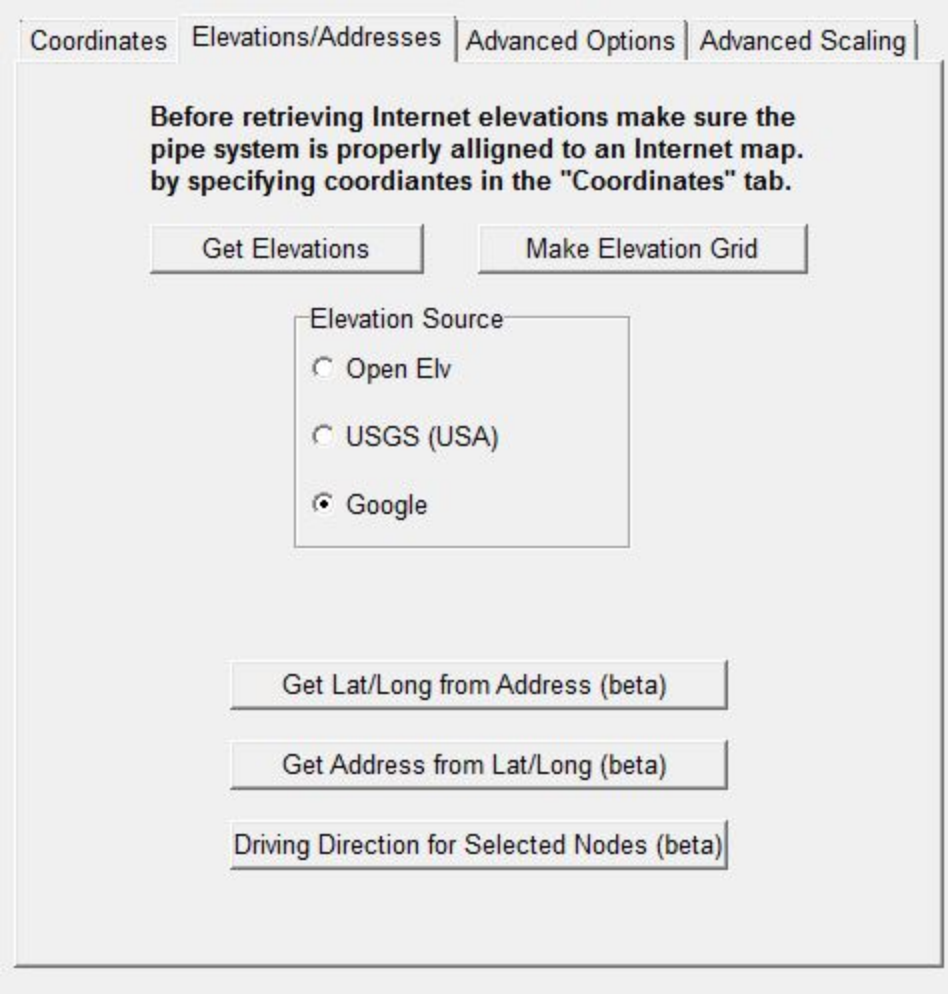
An MapQuest Internet map window will appear in your internet explorer application with a driving route to all points.
Tips:

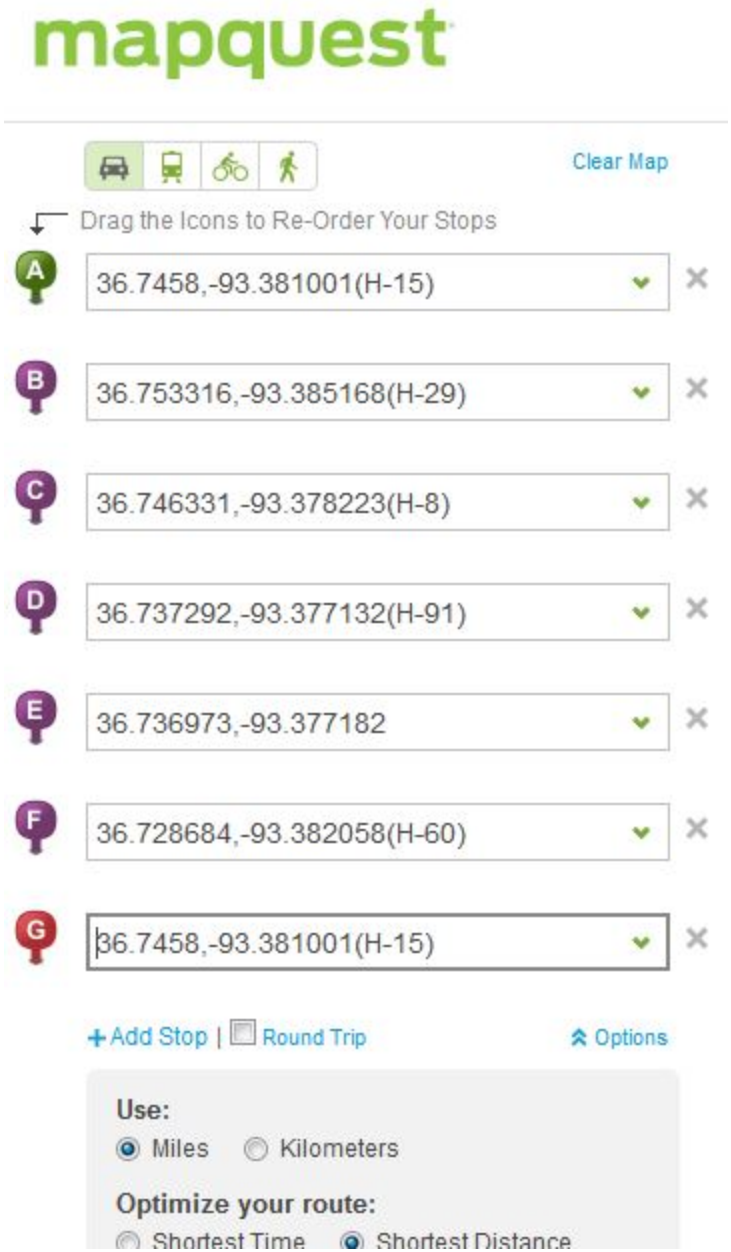
The last node selected in Group mode will be the starting and ending location for the route.

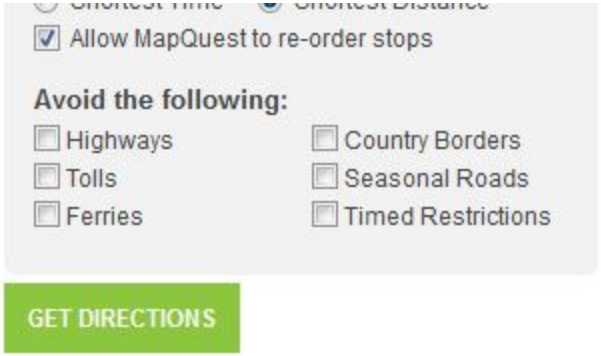
On the left side of the MapQuest window, under the list of Stops is an Options drop-down.  Scroll down to see image.  Check the box "Allow MapQuest to Reorder Stops" and click "Get Directions" to recalculate the most efficient route.


265
13
13
13
Old Wilderness Rd
Main St
200,000 gallon tank

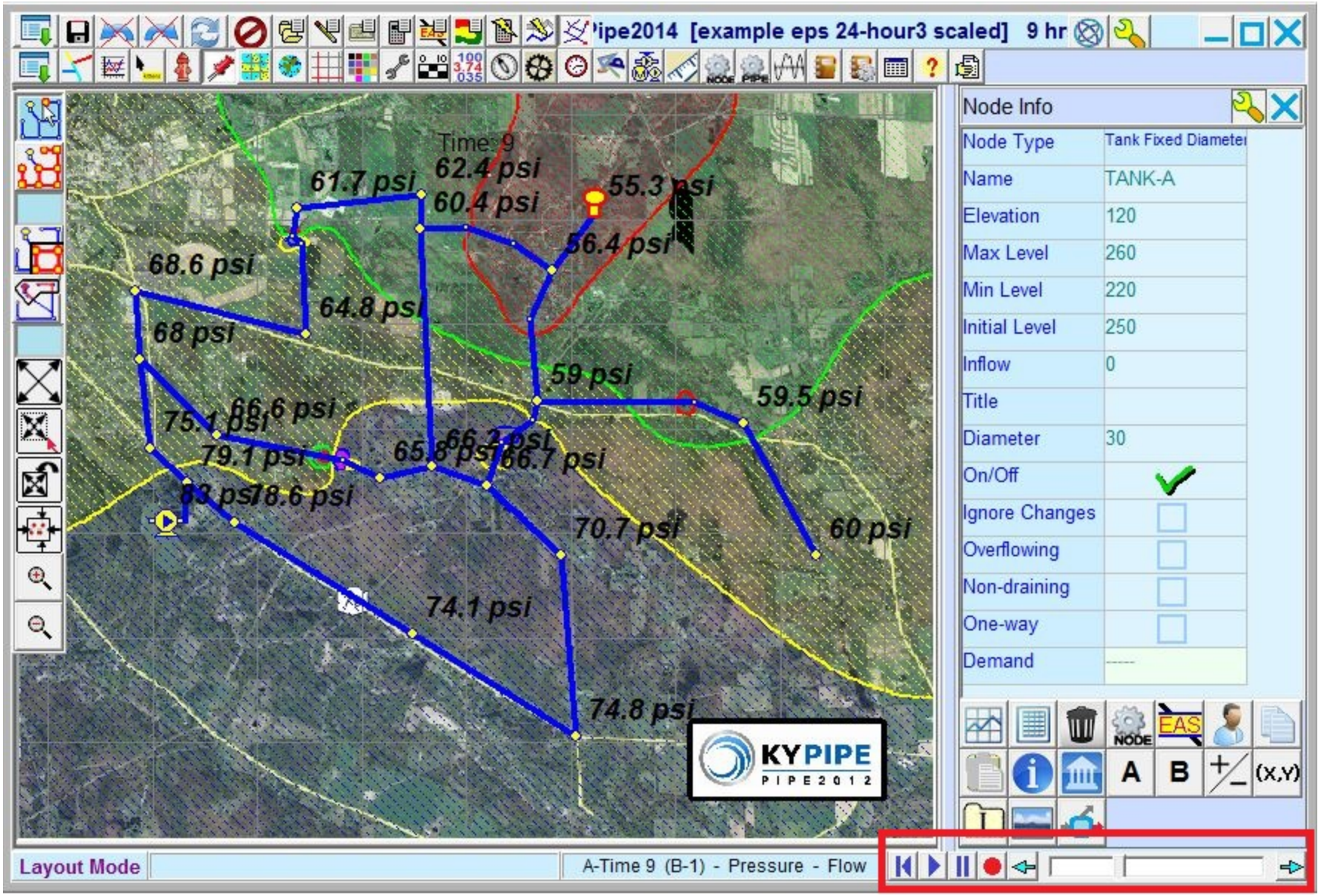

B
248
265
Main St
Close Ln
Wilson Way
C
G
Longview Ln
VFW Rd
13
Craig St
Hadleys Walk Dr
Dogwood Ln
E
13
Craig St
Battle Rd
Rockytop Way
76
Main St
Wolves Ln
Leighs Way
Neals Trl
Elementary Rd
Emerson Rd
F


Coordinates
Elevations/Addresses
Advanced Options
Advanced Scaling
Before retrieving Internet elevations make sure the pipe system is properly alligned to an Internet map. by specifying coordiantes in the "Coordinates" tab.
Get Elevations
Make Elevation Grid
Elevation Source
Open Elv
USGS (USA)
Google
Get Lat/Long from Address (beta)
Get Address from Lat/Long (beta)
Driving Direction for Selected Nodes (beta)


mapquest
Clear Map
Drag the Icons to Re-Order Your Stops
A
36.7458,-93.381001(H-15)
B
36.753316,-93.385168(H-29)
C
36.746331,-93.378223(H-8)
D
36.737292,-93.377132(H-91)
E
36.736973,-93.377182
F
36.728684,-93.382058(H-60)
G
36.7458,-93.381001(H-15)
+ Add Stop | Round Trip
Options
Use:
Miles
Kilometers
Optimize your route:
Shortest Time
Shortest Distance




### 1.3.6.5 Animate Map




Use the controls to step through results of an EPS or Surge analysis, or multiple steady state scenarios. Use the red Record button to create an AVI.

In Classic interface this is found under View in the main menu.

## 1.3.6.6 Background Images

Background Images

---

Pipe2018 allows you to use several types of background images at the same time, which can guide your pipe system layout and provide a scale to determine pipe length.

Internet Background map (Google, MapQuest)

Grids

Raster (CAD) Files

Vector (Image) Files

Node Images

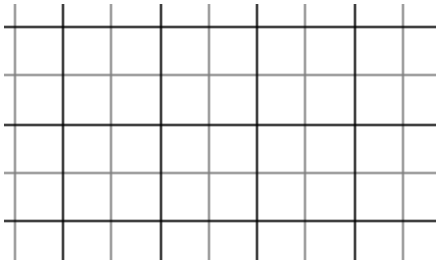
See also Backgrounds

Scaling Background Maps

Note regarding **shapefiles** as background images: While some shapefiles will successfully load in the background using the Add Map Button within the Backgrounds Menu, shapefiles are not an officially-supported background format. The appearance of shapefiles imported as background images is inconsistent and cannot be controlled. A better option may be to export the shapefile to a CAD format and use the resulting CAD file as the background image. The export to CAD function is available in many GIS mapping programs. Some of the more advanced CAD mapping programs allow the import of shapefiles into CAD format.

GRIDS

You can turn on grid lines of any spacing. The spacing can be changed at any time. This feature is accessed using the Grids menu or Map Settings / Grid tab (Classic).



RASTER FILES

Raster files are picture files in which every pixel has a specified color. A photographic image is typically stored in a raster file. Raster file background layers can be loaded and turned on or off as desired. This feature is accessed using the Background menu or Map Settings / Backgrounds (Classic) tab. These drawings can occupy all or a portion of your drawing area. A number of file formats are supported (.pdf (Pipe2014) .bmp, .tif, .bml, .shp, .jpg, .mif). Other formats can be converted to the (.tif) format using the To TIFF module which is included in the Pipe2018 package.

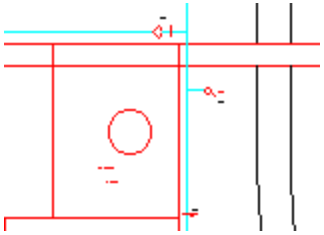
Raster files require a coordinate reference file which may be modified using the Map Link module which is included.

See Scaling Background Maps for detailed instructions.

VECTOR FILES

Vector files are files that describe the size, length, color, and position of lines (vectors). Vector files are typically used to represent things like plat maps and CAD drawings. Vecor file background layers can be loaded and turned on or off as desired. This feature is accessed using the Backgrounds menu or Map Settings / Backgrounds tab (Classic). These drawings can occupy all or a portion of your drawing area. A number of 2D file formats are currently supported and include AutoCad DXF and DWG and MicroStation DGN (may require saving in earlier version formats). The actual drawing coordinates are used to position the image on your pipe system coordinates. However, drawings can be shifted and scaled.

See Scaling Background Maps for detailed instructions.



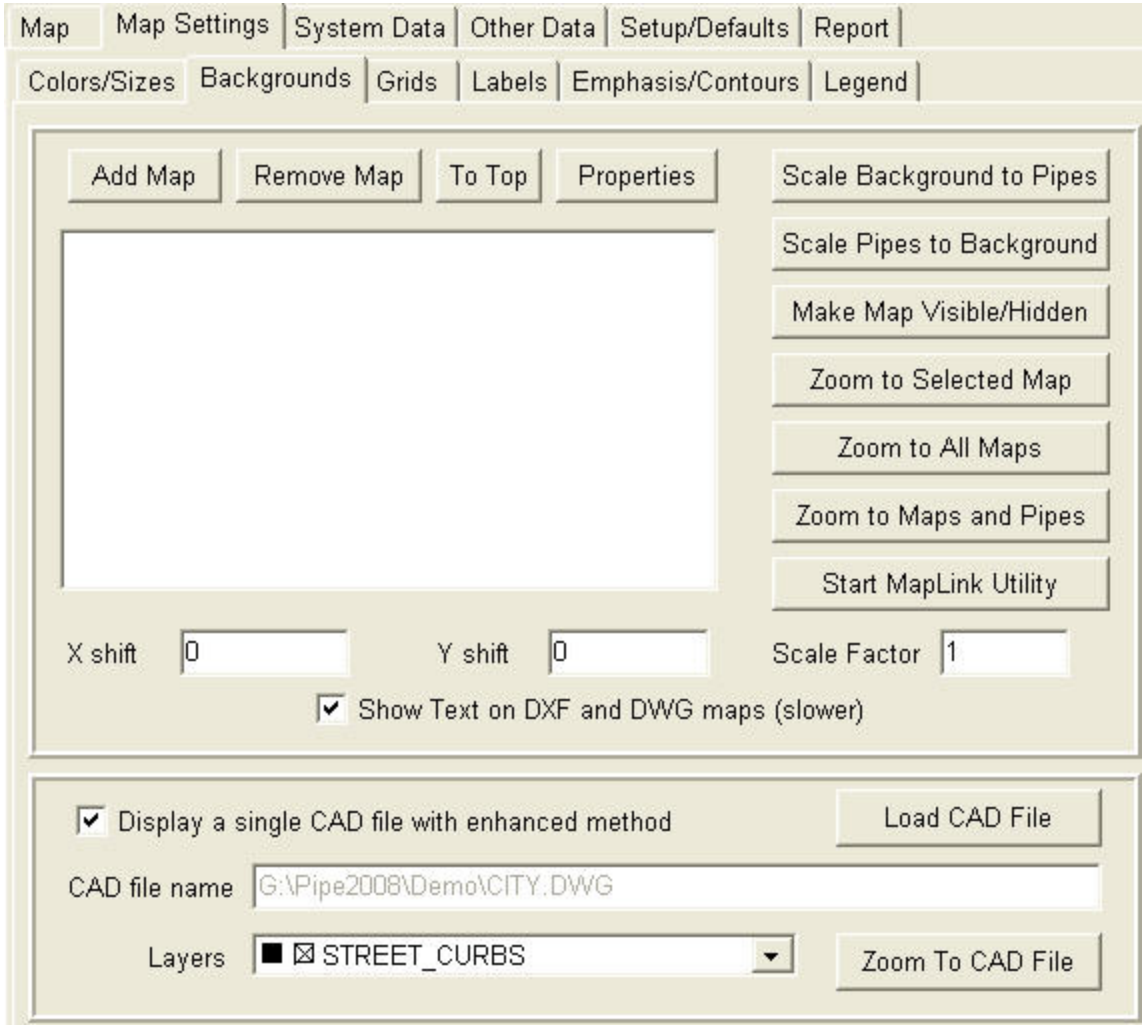
NODE IMAGES

See Node Images and Text Nodes.

### 1.3.6.7 Backgrounds

## Backgrounds

## KYnetic icon:




---

The Backgrounds tab is used to control the loading and display of background images. Pipe2018 can input a background maps or drawings in a variety of vector and raster formats. Using a scaled background map or grid lines will allow pipe links to be precisely scaled (length calculated).

Supported formats:

**PDF (Pipe2014 and later versions)**
SHP and MRSID (some source formatting limitations)
BMP
JPG
TIF (some geo-referenced TIFs also)
GIF
for other formats see MapLink Utility

CAD DWG and DXF files may be added using section at the bottom of the menu (more information below)

Internet Maps supported via different menu.

Add Map

This button brings up a file selector that allows you to choose a picture file as a background for your piping system. You can have more than one picture loaded at a time. For example, you may want to have your pipe system overlaying a plat map overlaying an aerial photograph. See Scaling Background Maps

Remove Maps

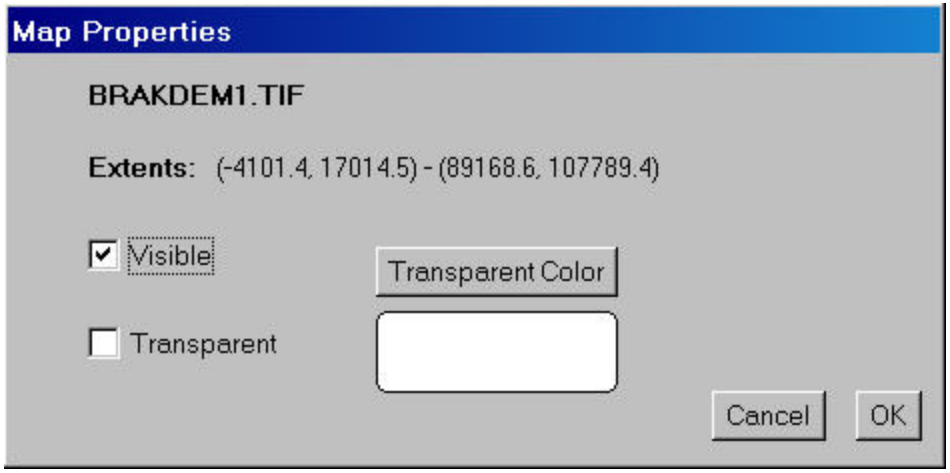
This button will remove a background picture from your data file. You must first click on the name of the picture that you want to remove and then click this button.

To Top

This button will change the order that the pictures are drawn on the screen. The order that the filenames appear in the list is the order in which they will be drawn. This is significant in the case where you are overlaying one picture on top of the other (incorrect ordering can cause one of the pictures to be obscured). To move one of the pictures to the top of the list first click on the picture name and then click on this button.

Properties

Under Map Settings | Backgrounds, the Properties button accesses the window below. The Map Properties window displays the title, extent and visibility status of the selected map. For raster-type maps, transparency may be specified along with the transparent color. This is useful for multiple map layers.




Scale Background to Pipes

This button allows you to alter the scale of the background pictures to fit the scale of your piping system. This will change the X shift, Y shift and Scale Factor of your background. These settings will be saved when you save your Pipe2018 file. Keep in mind, however, that these settings will change if the system is rescaled for any reason. See Scaling Background Maps.

Scale Pipes to Background

This button allows you to alter the scale of the piping system to fit the scale of your background pictures. Note that this will change the length any pipes that are not fixed (see Pipe Data Boxes - length). Coordinates will also be changed. This option will also orient the system to the map by rotating the network model. See Scaling Background Maps.

Make Maps Visible / Hidden

By first clicking on the picture name in the list and then clicking this button you can make the picture appear (Visible) or not appear (Hidden) in the drawing area.

Zoom to Selected Map

By first clicking on the picture name in the list and then clicking this button you can cause the current map viewport to be set as large as possible while still containing the selected picture in its entirety.

Zoom to All Maps

By clicking this button you can cause the current map viewport to be set as large as possible while still containing all the background pictures in their entirety.

Zoom to Maps and Pipes

By clicking this button you can cause the current map viewport to be set as large as possible while still containing the piping system and all the background pictures in their entirety.

Start MapLink Utility

The MapLink Utility allows the user to create a reference file for a specific background picture, recording the scale and location for that background. See MapLink.

X-shift

This entry causes all the background pictures to be shifted along the X axis the distance (in coordinate units) specified (positive is to the right).

Y-shift

This entry causes all the background pictures to be shifted along the Y axis the distance (in coordinates units) specified (positive is up).

Scale Factor

This entry causes the size of all the background pictures to be scaled by this amount (numbers <1 decrease the size).

See Scaling Background Maps.

### Show Text on DXF and DWG maps (slower)

When dxf or dwg files are used as a background the user has the option of displaying or hiding the text within that file. Keep in mind that this display option is slower.

### Display a single CAD file with enhanced method

A CAD file may be loaded in this section instead of the main map list. When loaded here, individual CAD layers may be turned on and off. The entire CAD map may be turned on/off without losing the layer settings by checking/unchecking the box "Display a single CAD file with enhanced method"

**Layers** - To view layers, simply click the drop-down selector arrow. The layers toggle between off and on by clicking on the row.

## 1.3.6.8 Complex CAD Formats Background Method

# Complex CAD Formats Background Method

**Using CAD files as background images.** CAD files from AutoCAD 2010 and later, plus some CAD files from applications that are not AutoDesk often need to be exported to earlier AutoCAD formats such as AutoCAD2004 to be successfully loaded as a background image. In addition, if XReferences are associated with the CAD file, these need to be removed.

With the Complex CAD Formats option, most of these CAD drawings may be successfully loaded without export or other modifications. To use the Complex CAD Formats feature

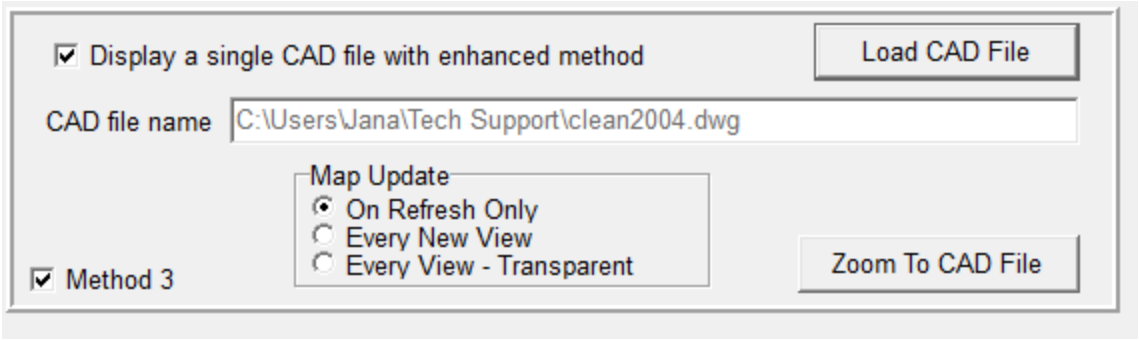
check the Complex CAD Formats box (Method 3)

Click Load CAD File button and browse to load the CAD file.

Map Update setting. The default setting is “On Refresh Only” so that the background will re-load when the user **manually refreshes the map** view using the refresh button:

KYnetic Classic

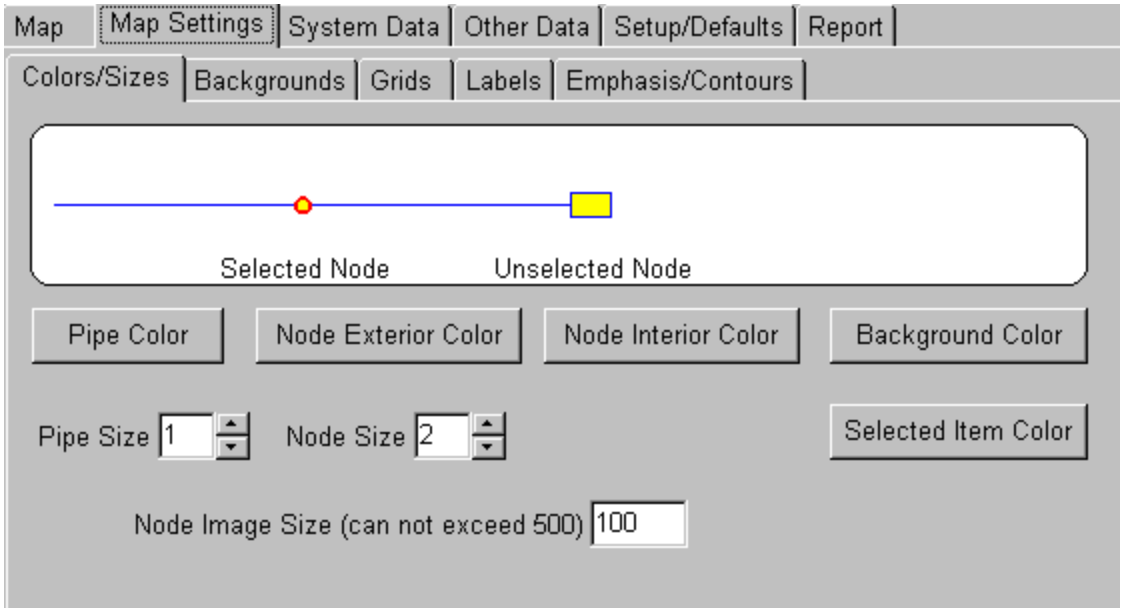
In some instances the user may wish to automatically refresh the map view every time the map window changes. This option can be slow. The “Every New View “ option may be useful in addressing the Map Object LT ESRI conflict which occurs with some installed applications.



## 1.3.6.9 Colors

Colors

KYnetic Colors menu icon




---

The Colors / Sizes tab is used to set the display properties of nodes and pipes on the map. Changing these settings can make a system much easier to see if you are running at high resolutions (> 1280 x 1024) or if you have a complicated background picture.

Pipe Color

Sets the color that pipes will be drawn

Node Exterior Color

Sets the color that will be used to outline node pictures

Node Interior Color
Sets the color used to fill node pictures

Background Color
Sets the base color for the map. This essentially sets the color of the "paper" that the system is drawn upon.

Pipe Size
Sets the thickness (in pixels) of the lines used to represent pipes.

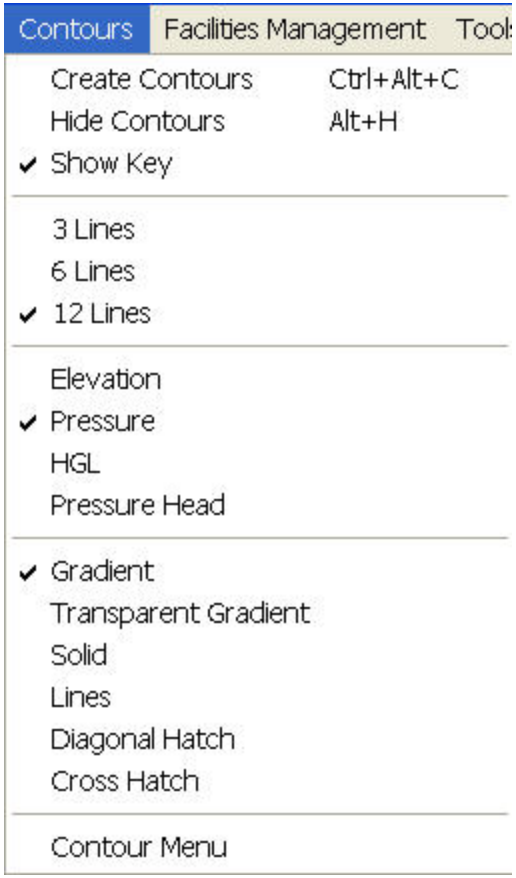
Node Size
Sets the size of the pictures used to represent nodes.

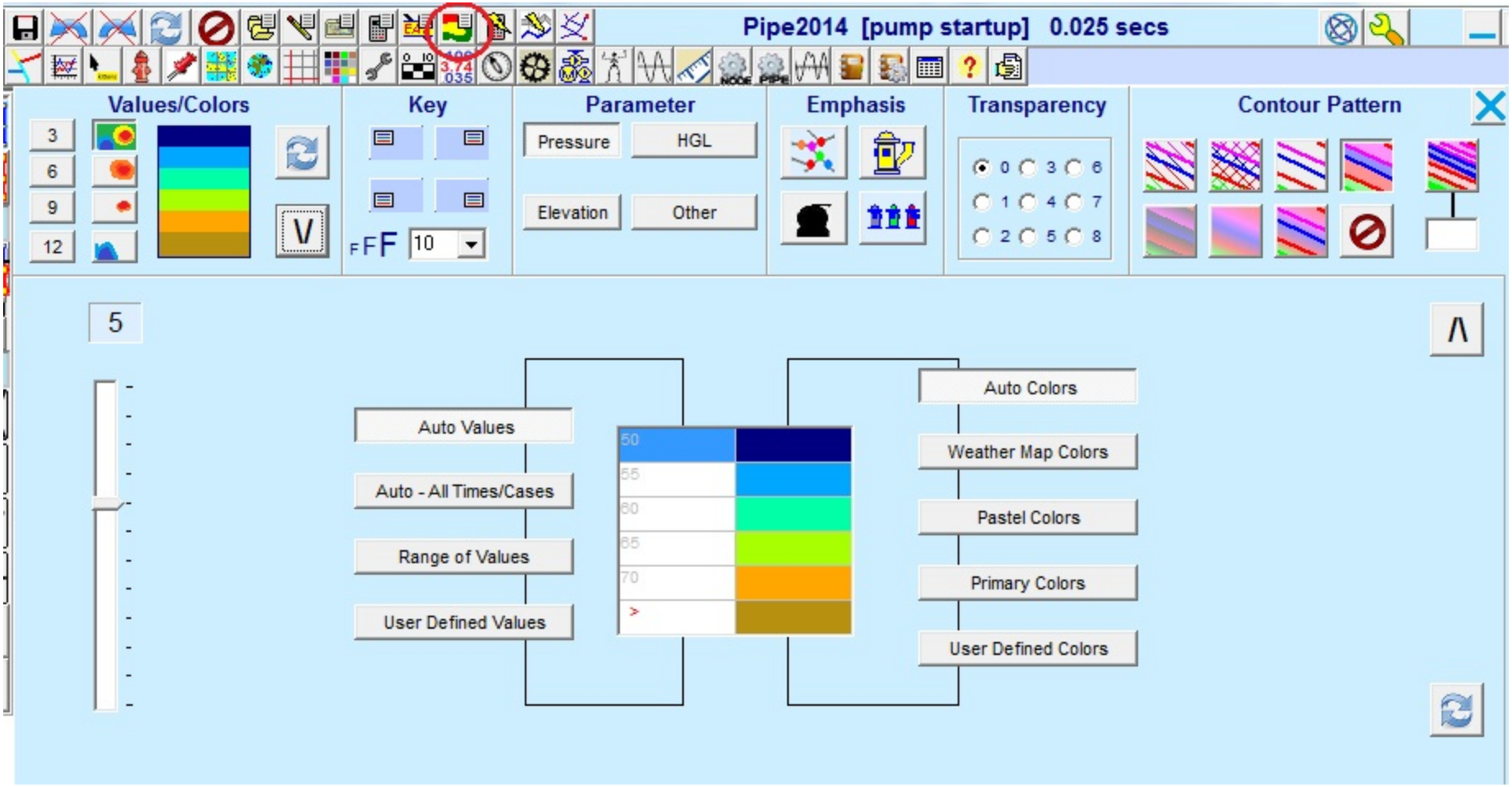
Selected Item Color
Allows the user to choose the color of the items currently selected under Group Mode.

Node Image Size
Sets the size of the bitmap pictures (images) that can be stored for each node. Often these images are digital photos of the site of the node.

## 1.3.6.10 Contours - Main Menu

Contours - Main Menu

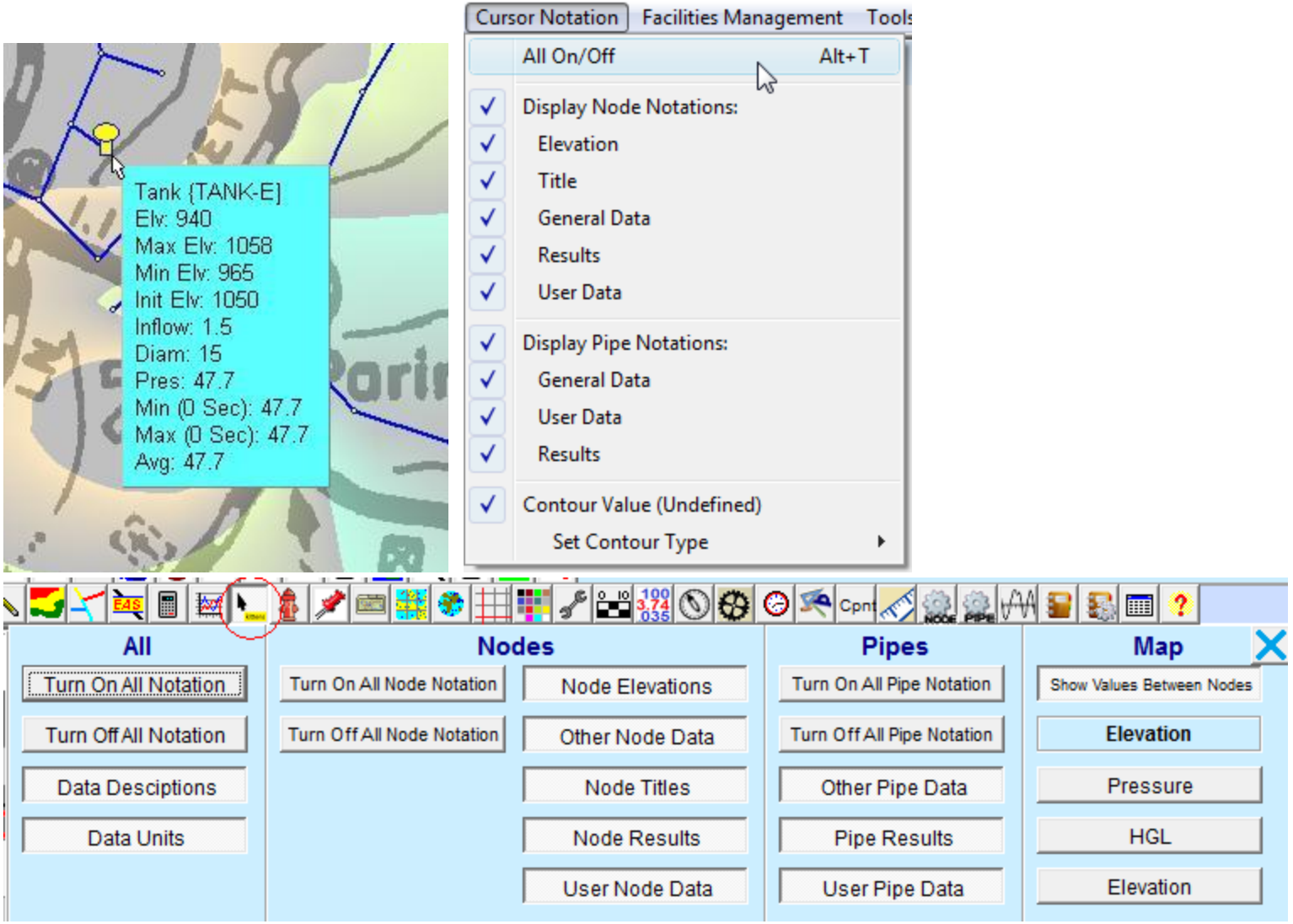

Contours
Facilities Management
Tool
Create Contours Ctrl+Alt+C
Hide Contours Alt+H
Show Key
3 Lines
6 Lines
12 Lines
Elevation
Pressure
HGL
Pressure Head
Gradient
Transparent Gradient
Solid
Lines
Diagonal Hatch
Cross Hatch
Contour Menu




This shortcut menu is available to quickly build contours and pipe emphasis for the map screen.

See also Map Settings | Emphasis/Contours and Pipe Emphasis.

## 1.3.6.11 Cursor Notation




The cursor may be configured to display any combination of the options listed in the drop-down menu shown above.

Set Contour Type - the mouse, when hovering over any point on the map, will display the interpolated value of the parameter selected in this drop-down sub

menu.

## 1.3.6.12 Emphasis / Contours - Nodes

Emphasis / Contours - Nodes

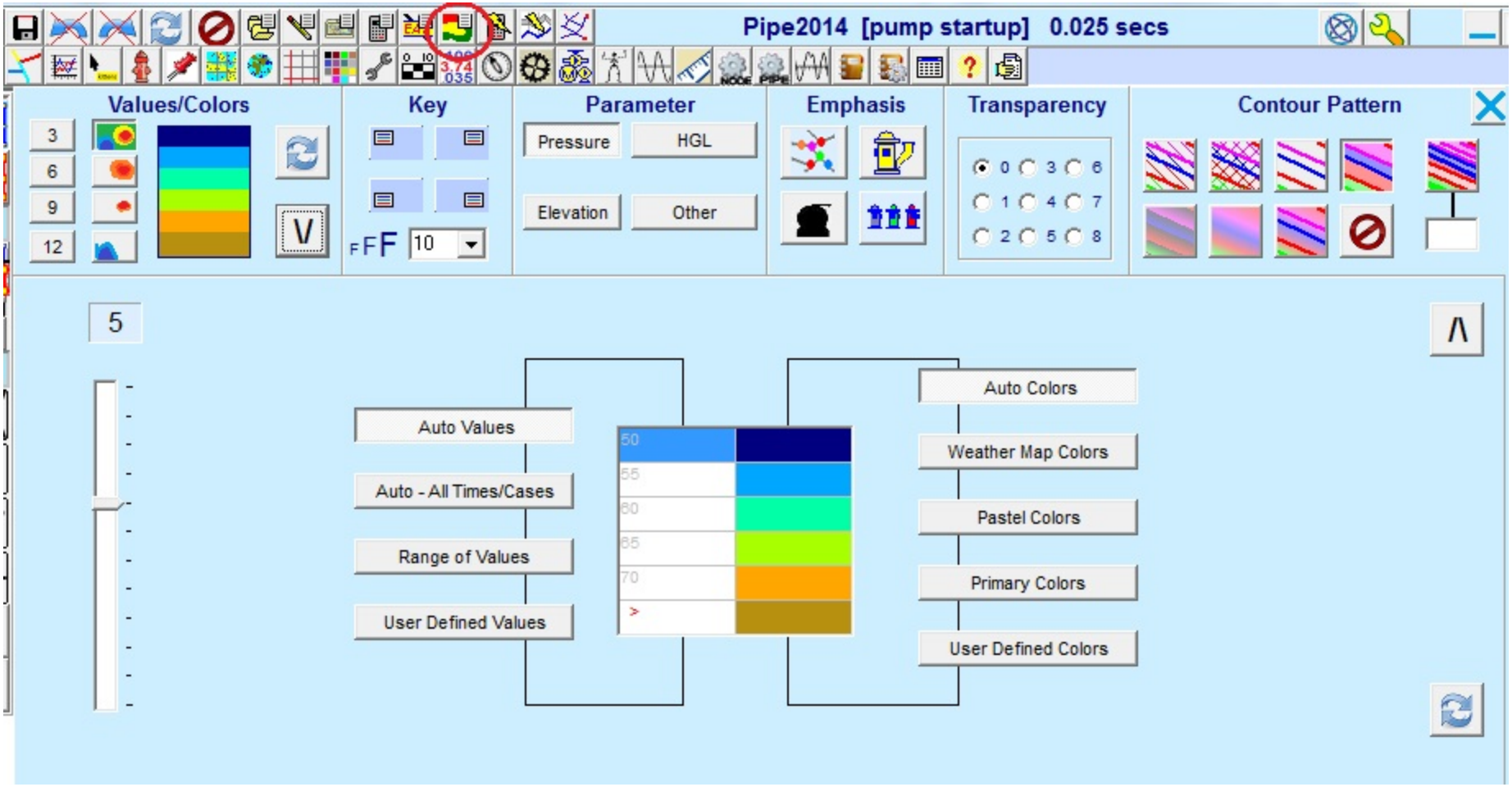
This menu is used to set up Contours or color emphasis for nodes. Contours show colored areas on the map that contain nodes within a specified data range. Color Emphasis sets the color of the node pictures based upon the value of their data.

Please note - the Refresh button KYnetic: Classic: will regenerate the contours. This is useful, for example, when zooming in to redraw the contours at the maximum resolution.

See the Contours video on the kypipe.com web site.

See also Pipe Emphasis. or Contours Main Menu shortcut

KYnetic:




Value/Colors - These are the values used to determine the range used for each color. Colors are applied to values up to an including each range increment (values that are less than or equal to the target value are set to the target color). So if Pressure values of 50 are blue, then values above the previous increment and up to and equal to 50 are blue.

Select the number of increments, 3, 6, 9, 12

Select the range for contour values - full range, high values, very high values, and low values.

- Drop down additional menu options. For all of the above settings, the user can override and specify specific colors, values and ranges.

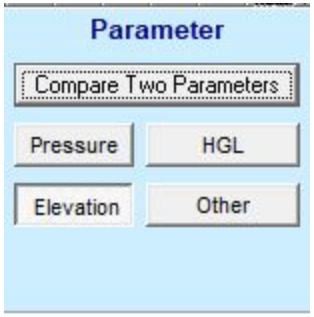
Set a specific number of values

Use one of the default ranges or click User Defined Values to enter your own

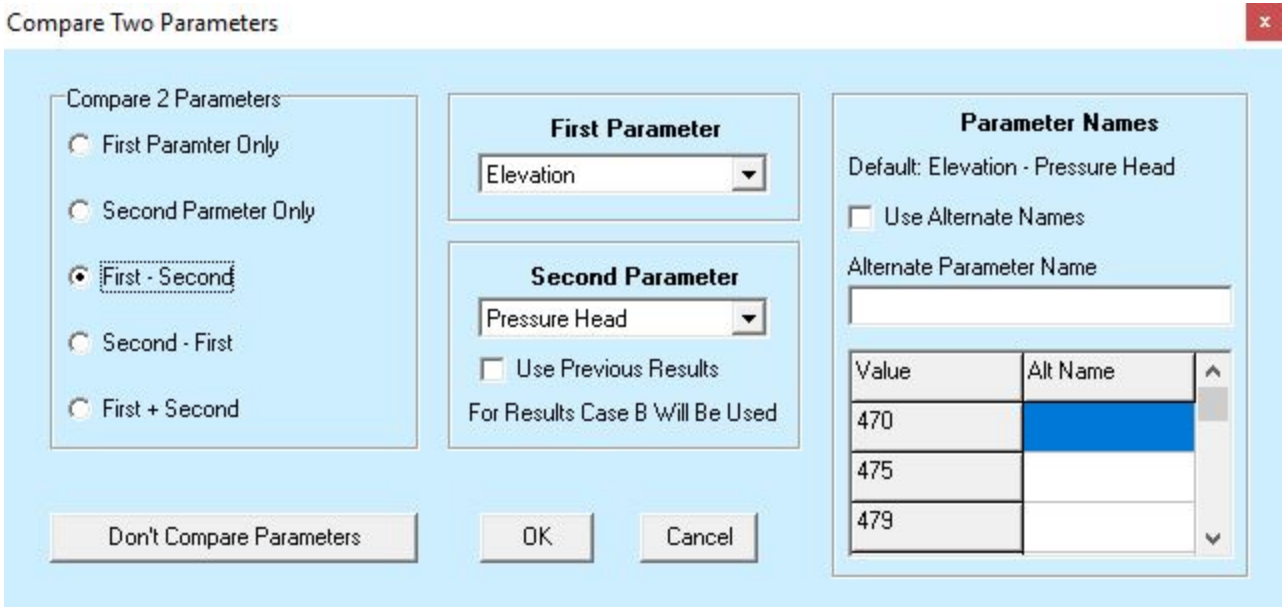
Use one of the default color paletts or selecte User Defined Colors to select your own

Key - set key location and font size

**Compare Two Parameters**




The button in the Contours menu opens the following menu:

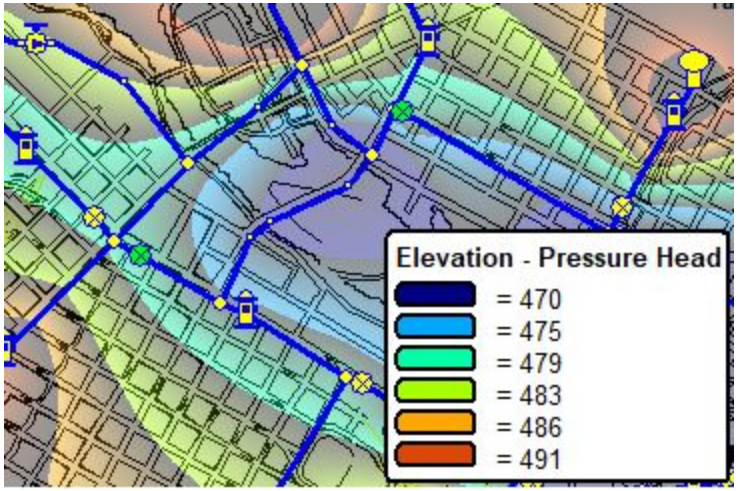



Here you can select the two parameters to subtract. Results from a previous analysis may also be selected.

Customized Parameter names may be specified.

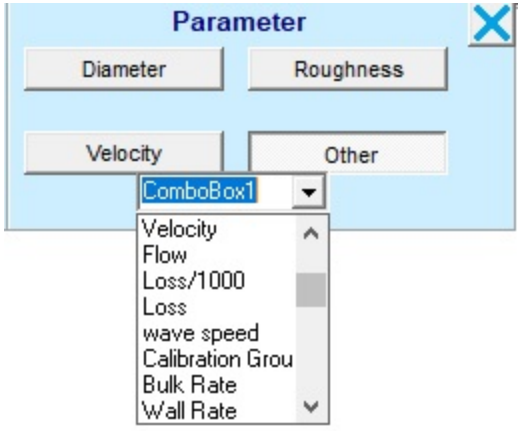
Once entry is complete, close and **it is important to refresh the values list** using the Auto option as shown:

This produces a contour of the difference between the two values, as shown:




**Parameter** - Setting the parameter also operates as the contour on/off function. Select Other for more contour parameter options.

**Added in Pipe2018 - Loss and Loss/1000**




Emphasis - colors the interior of each node icon

The first button turns the emphasis off and on, using the contours settings

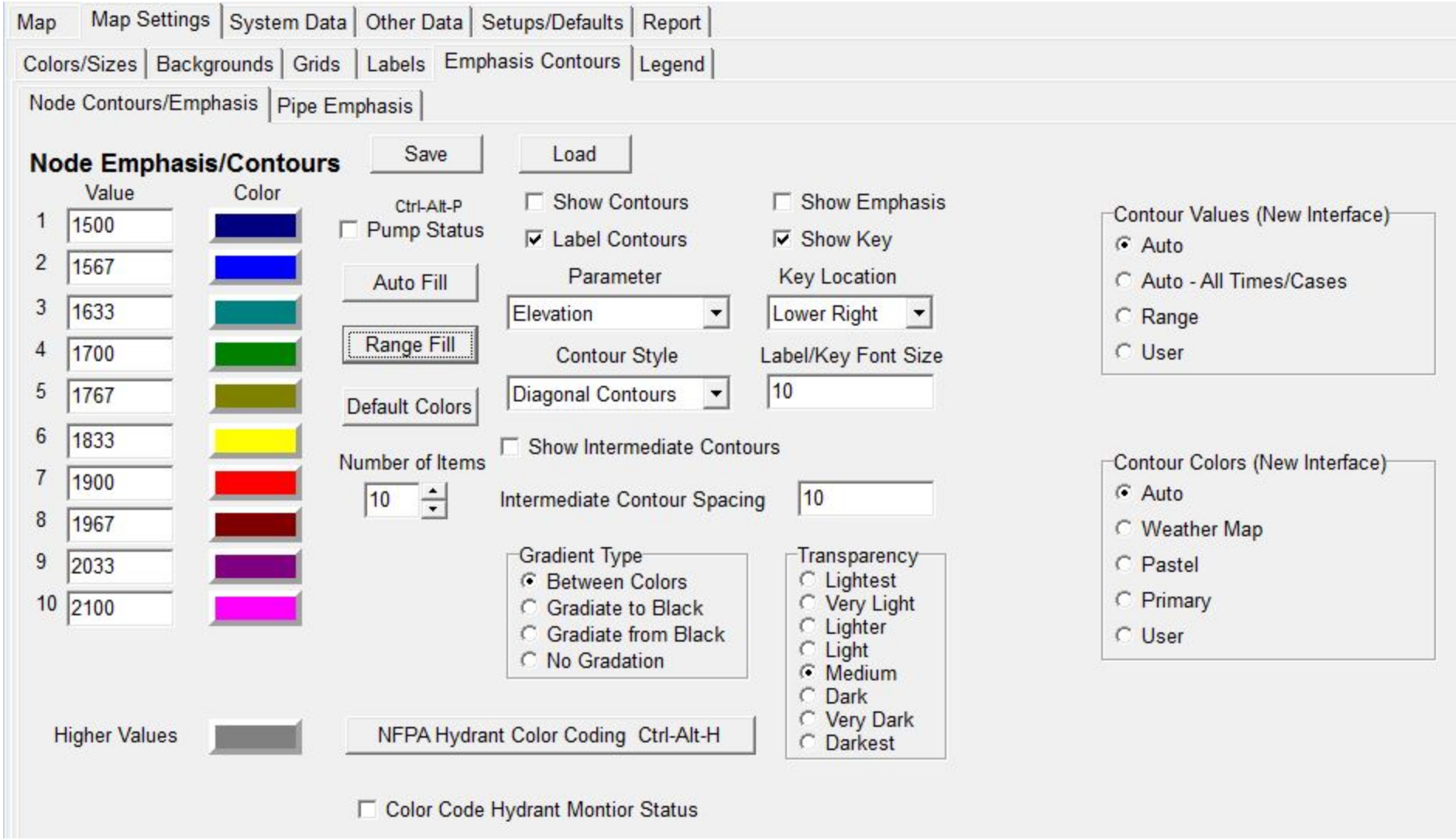
Three options for built-in emphasis are Hydrant Monitor status, Pump status, and NFPA Hydrant Rating.

Transparency settings for solid or gradient contours, allows background images to show through.

Contour Pattern - icons indicate the appearance. The top right button allows for added intermediate contours to be added. Note does NOT apply to solid contours.

---

Classic Interface:




Value

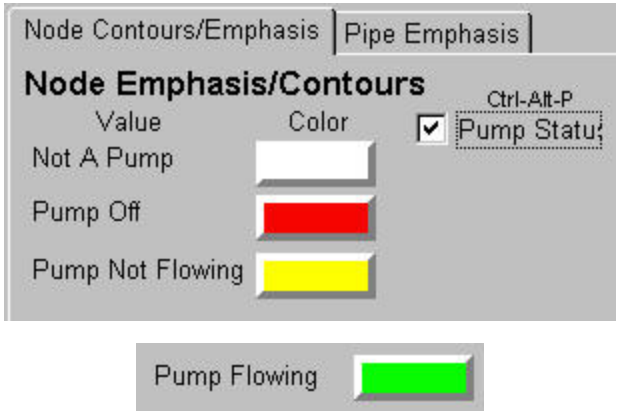
These are the values used to determine the range used for each color. Ranges are filled from the highest range down, and values that are less than or equal to the target value are set to the target color. In the example above Nodes with Elevations <=20 will be yellow, nodes with elevation >20 and <=40 will be purple, etc.

Color

These are the colors used to fill the nodes that fall within the specified range of values

Pump Status

**When this box is checked, the menu pictured below will appear. While checked, Node Contours will be replaced by Pump Status Emphasis. All nodes which are not pumps will be emphasized with the first color. A pump that is turned off in the in the baseline data (set to 'off' through the Node Information window) will be emphasized with the second color. A pump that has been turned off during a simulation or is experiencing flow reversal will be emphasized with the third color. A flowing pump will be empasized with the specified Pump Flowing color.**




Selector Boxes

Clicking on these boxes allows you to set the target color to the left.

Auto Fill

This box automatically sets the values in the value boxes to establish equal sized ranges to span the data type selected in the Parameter box.

Range Fill

This button causes the Value boxes to be filled in to establish equal sized ranges between the bottom and top value boxes. Note that you should fill in the first value box with the lowest value and the top box with the highest value BEFORE pressing the Range Fill button.

Default Colors

**Reset colors to default setting.**

Number of Items

This sets the number of user specified ranges that will be present.

Show Contours check box

Selecting this box causes the defined contours to be shown on the map.

Show emphasis check box

Selecting this box causes the nodes on the map to be colored according to the ranges.

Label Contours check box

Selecting this box causes the contours generated to be labeled with the parameter value.

Show Key check box

Check this box to have key or legend appear on the map showing contour values

Parameter

This selects the data item used to set the ranges. Note that if a result is selected the "A" box on the main menu may be used to identify which specific result you are referring to (in the case of Extended Period Simulations or Changes).

### Contour Style

This popup box allows the user to select the style that the contours will be filled with. The choices are:Diagonal, Crosshatch, Solid, and Lines.

### Key Location

This popup box allows the user to select the corner of the map in which the key will appear.

### Label/Key Font Size

**This box allows the user to choose the font size for the Key and for contour labels.**

### Show Intermediate Contours

This check box will add additional contours based on the Intermediate Contour Spacing specified below. Will be in the same color as the range within which the intermediate contours fall. Not applicable for solid contours.

### Intermediate Contour Spacing

**In the same units as the Parameter specified, the intermediate contour spacing is specified. For example, if the contours are elevations and range 1 is 556 to 671 and the contour spacing is 20, then there will be 6 intermediate contours added to range 1.**

### NFPA Hydrant Color Coding

After running a hydrant analysis, go to node emphasis click button that says “NFPA Hydrant color coding” Or you may hit ctrl – Alt – H, or to go to Labels |

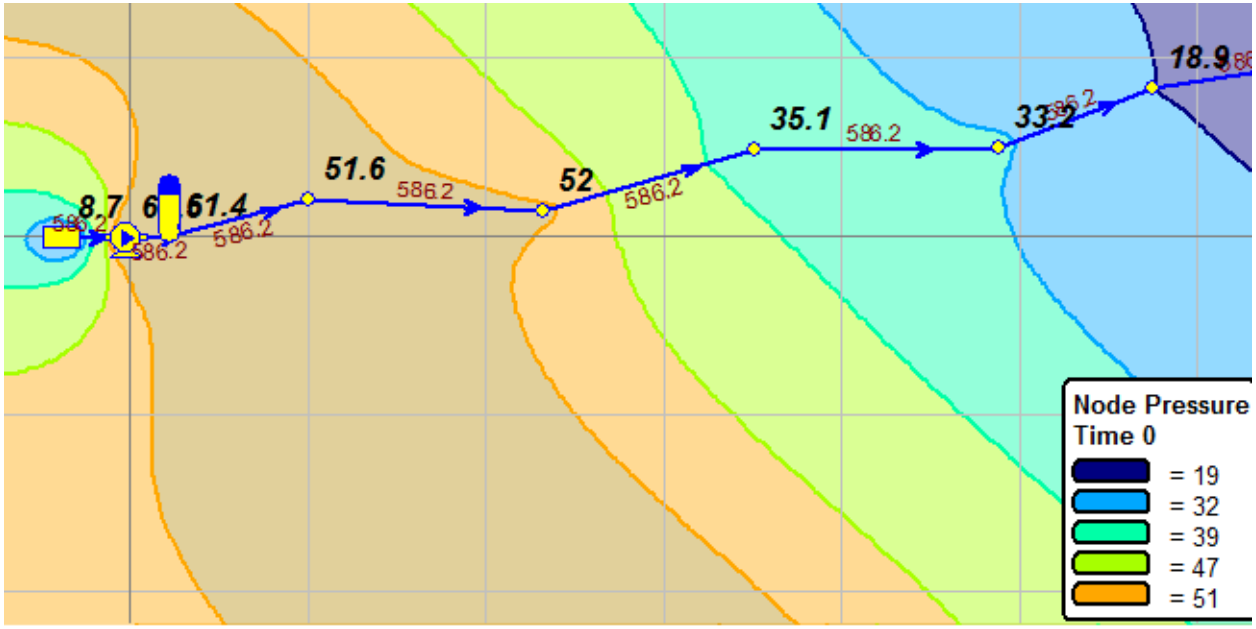
Results A or B | Fireflow/Static/color code NFPA. Emphasizes the hydrants.

## 1.3.6.13 Emphasize Pipes as Nodes

# Emphasize Pipes as Nodes

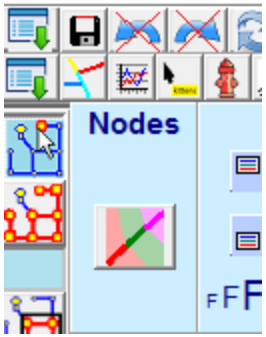
Emphasizes pipe with colors and values from Node Contours and Node Emphasis.

Pressure contours from the Node Emphasis Menu:

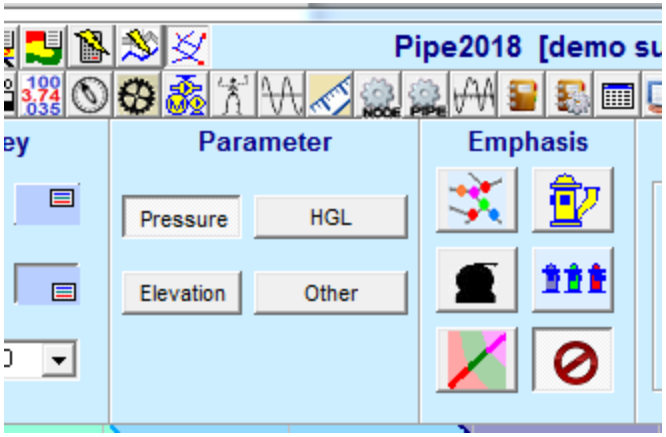



Click the Emphasize Pipes as Nodes box.

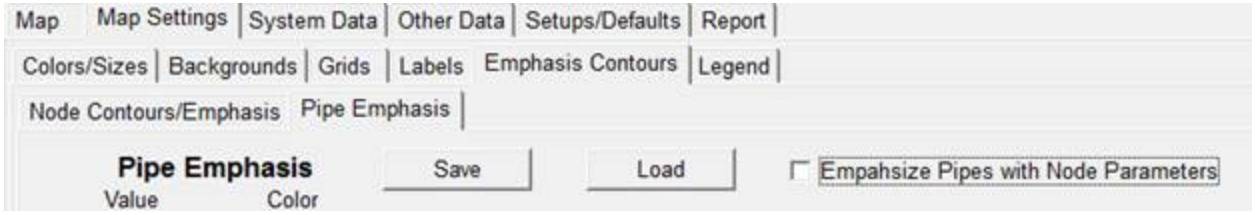
It can be found in the Pipe Emphasis menu in KYnetic:



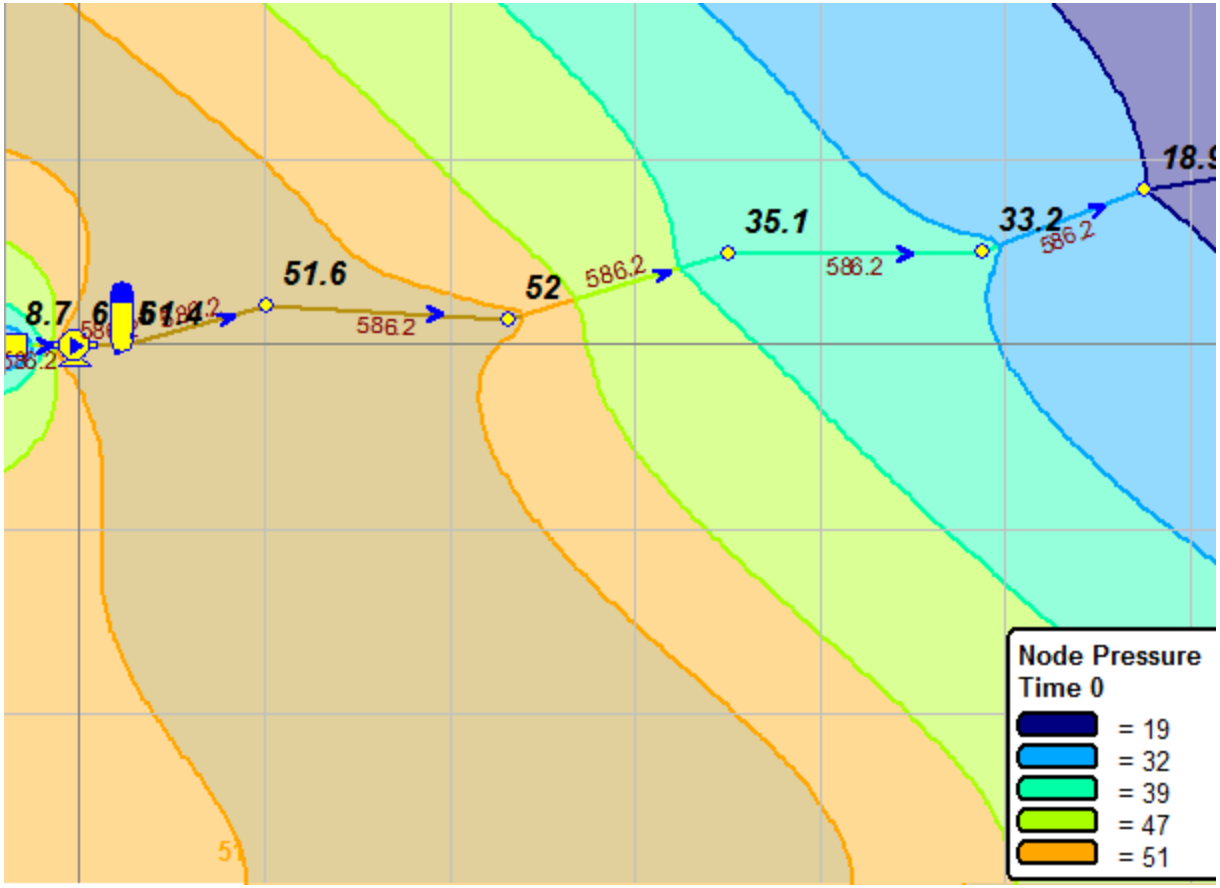

In the Node Contours menu in KYnetic:



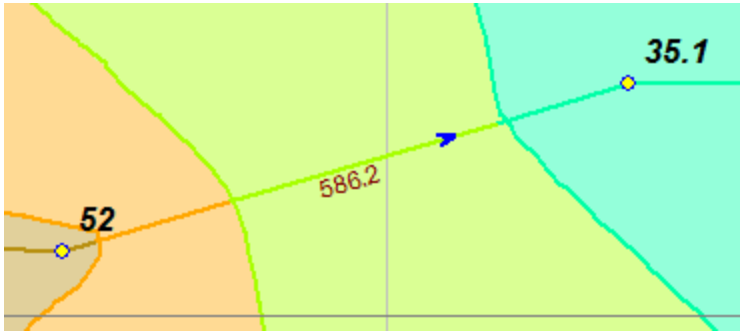

And as a check box in the Node Contours or Pipe Emphasis menus in Classic:




Once it is checked, the pipes are colored the same as the node contours:



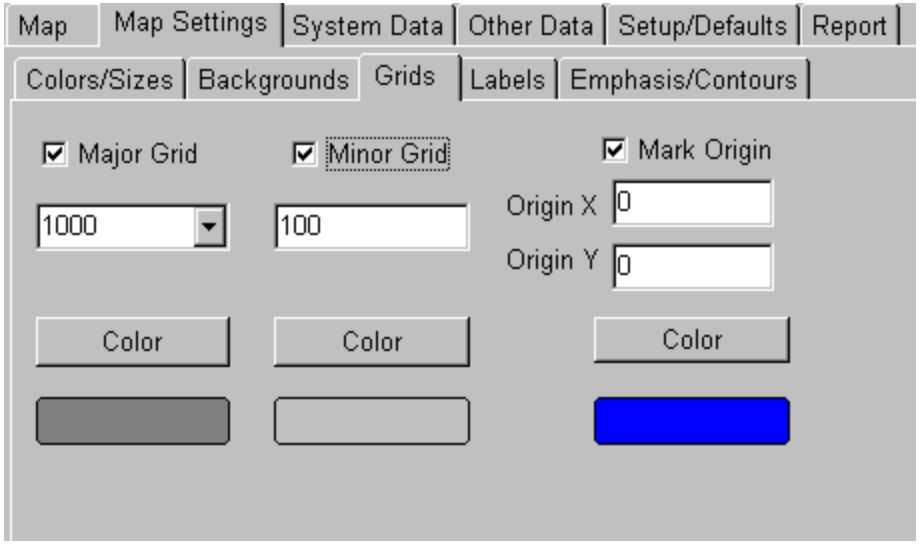

Close-up view:




## 1.3.6.14 Grids

Grids

KYnetic Grid menu icon:




See the Model1 video on the KYPipe web site.

---

**This window controls the color and size of the grids on the map. The map allows for major and minor grid lines. The major lines are typically an order of magnitude further apart than the minor grid lines. By choosing different colors for the two types of grid lines one can establish a good visual indicator or approximate position of features on the map.**

What are Grids?

Grid lines are scaled horizontal and vertical lines which may be displayed on the map and printouts. You can set the grid scale and use the resulting grid to produce a scaled layout of your system.

Major Grid

This checkbox determines if the major grid lines are shown.

Major Grid Value

This drop-down selector box allows you to set the spacing of the major grid lines.

Major Grid Color

This button brings up a window that allows you to choose the color of the major grid lines.

Minor Grid

This checkbox determines if the minor grid lines are shown.

Minor Grid Value

This box allows you to set the spacing of the minor grid lines.

Minor Grid Color

This button brings up a window that allows you to choose the color of the minor grid lines.

Mark Origin

This checkbox determines if the origin is marked on the map.

Origin X, Origin Y

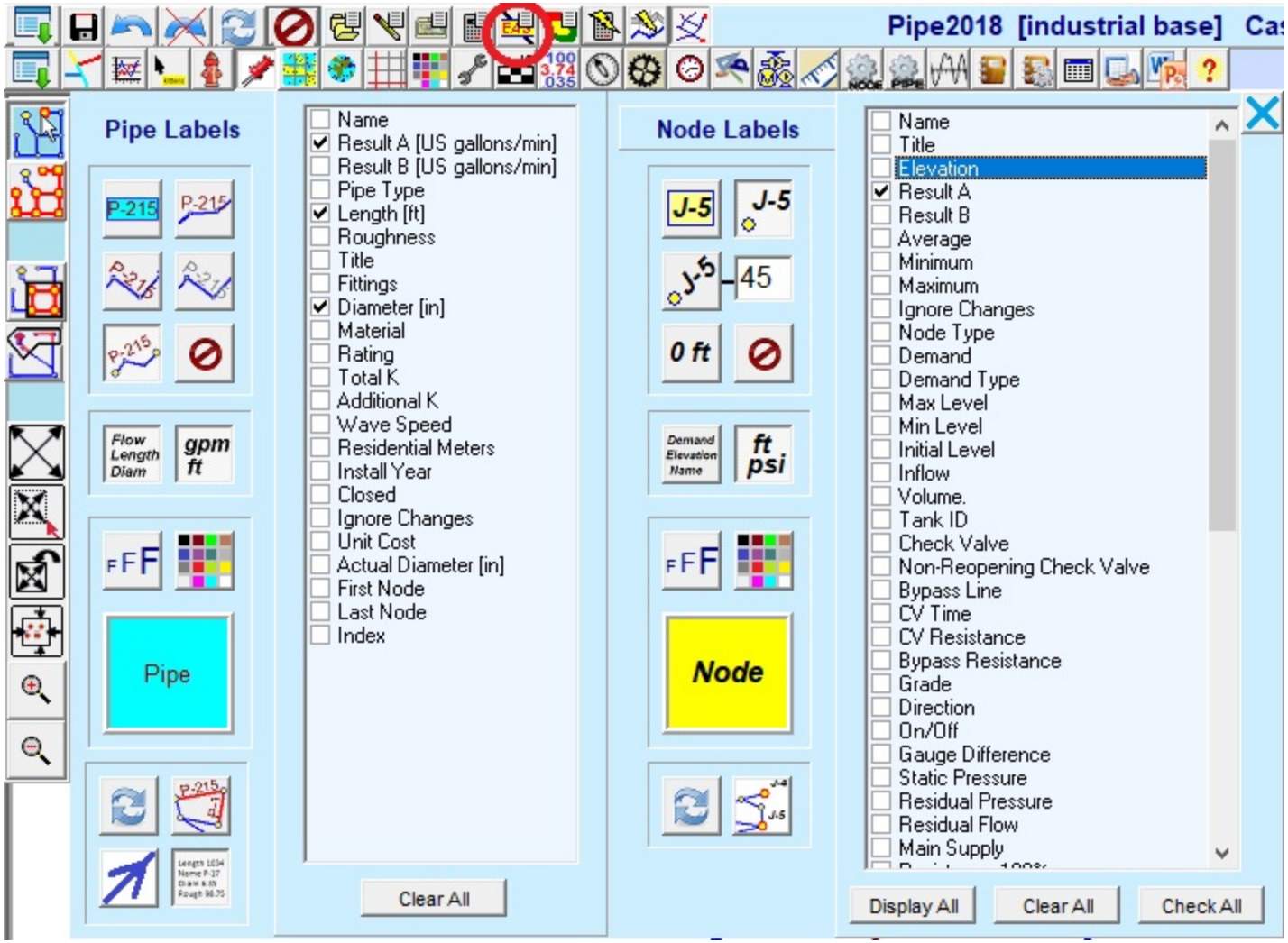
These boxes allow you to set an X and Y value of a coordinate to be marked as the origin of your map

Origin Color

This button brings up a window that allows you to choose the color of the origin symbol.

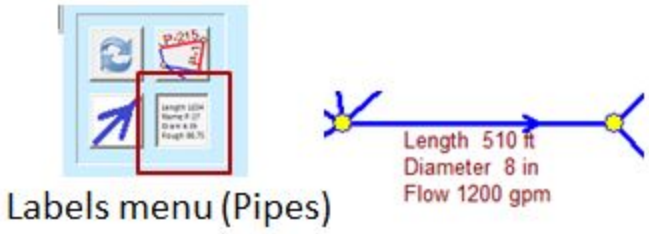
## 1.3.6.15 Labels

Selected Labels Only - or troubleshoot: why won't my labels show up?



KYnetic interface Labels menu

**Multi-line Pipe Labels (Pipe2018)**




**SHOW "ZERO" LABELS**

If this box is checked, node labels which are equal to zero (e.g. nodes with no elevation data) will be displayed on the map.

**LABEL TYPE**

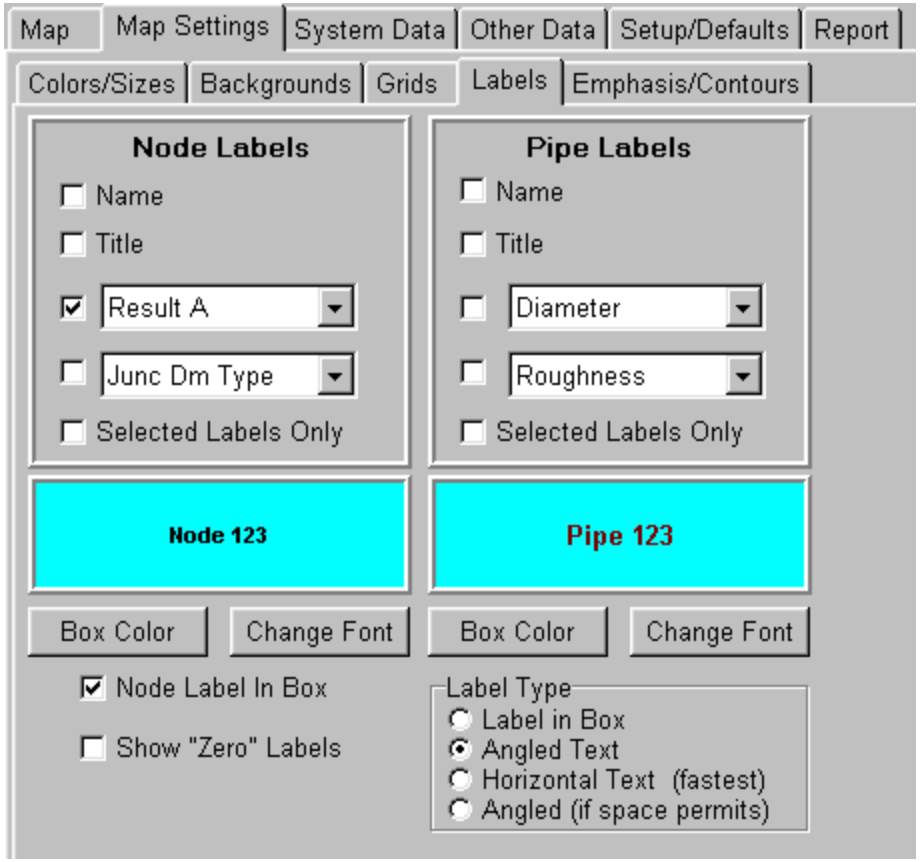
Label in Box - The label will be displayed horizontally within a box.

Angled Text - Aligns the labels along the pipe. For a big system, generating this display will be slower than other alignment options.

Horizontal Text (fastest) - Makes the label text horizontal.

Angled (if space permits) - Shows labels aligned with the pipe. Only those labels, based on font size and zoom level, which fit within the length of the pipe will be displayed.

**Classic Interface Labels Menu**




Check the boxes to select labels to display.

Use the two selector boxes to add specific labels

Selected Labels Only - See below

Box Color - changes the background color when "Label in Box" option is checked/selected.

Change Font - allows font size, color, and style to be changed.

(Pipes) Angled Text - label is parallel to a line connecting the two pipe end nodes.

(Pipes) Horizontal Text - label is horizontal. This is a good option is the map loading time is slow

(Pipe) Angled (if space permits) - only label which fit between the pipe end nodes will be displayed.

**Selected Labels Only (KYnetic and Classic)**

Click the Selected Labels Only button (or check the box) for Pipes or Nodes.

Use Group Mode to create a group of pipes or nodes. Once the group is created, go back to Layout mode.

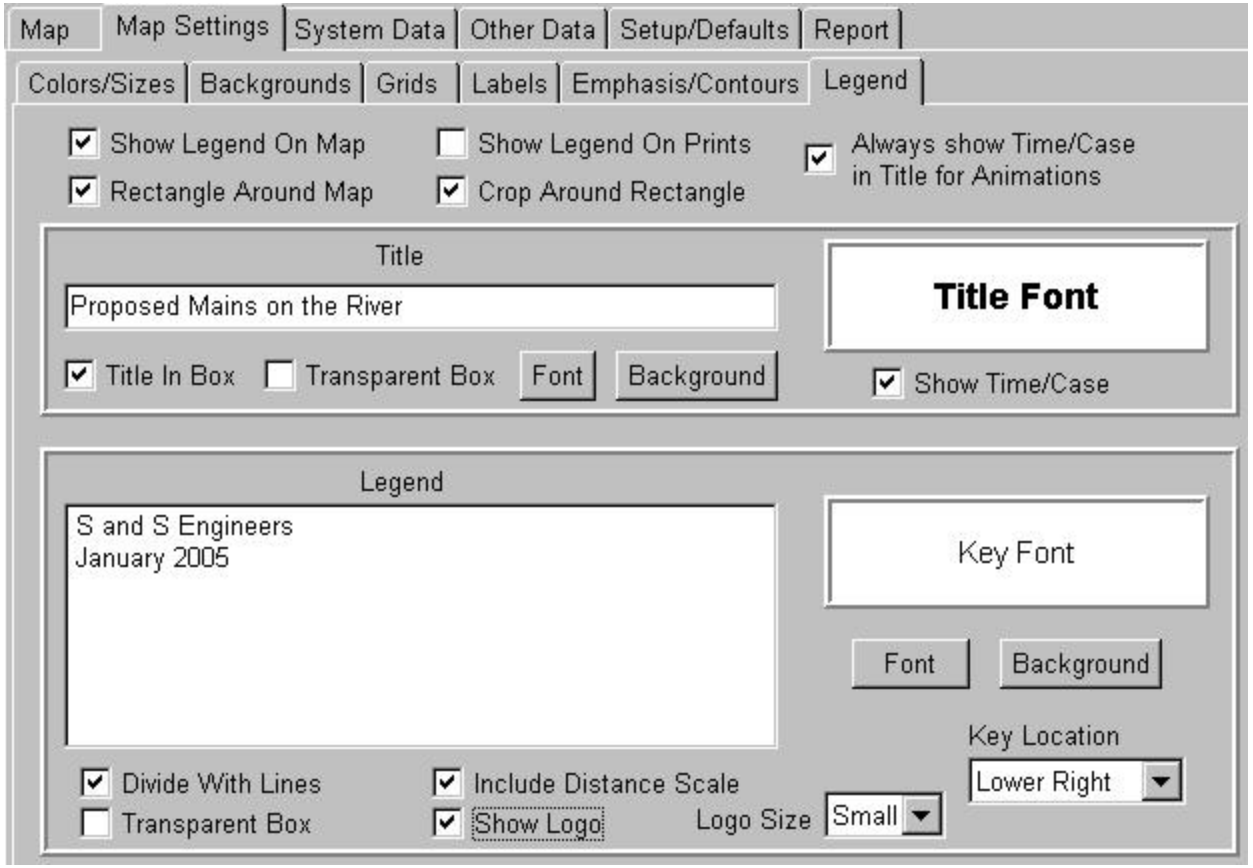
To be able to create a group, clear the group, then easily select the group again, create a User Data attribute

## 1.3.6.16 Legend

Legend

---

KYnetic Legend icon:




Show Legend On Map - causes the legend to be visible while viewing the map

Rectangle Around Map - adds a frame around the map as shown below.

Show Legend On Prints - causes the legend to be included when printing the map

Crop Around Rectangle - the optional frame is inset from the edges of the map, this visually crops any portions of the background or pipes which appear outside the frame.

Always show Time/Case in Title for Animations - for use with the View | Animate feature.

Title - a title may be added to the map

Title In Box - adds a frame around the title

Transparent Box - allows the background and pipes to show within the title frame.

Font - set the font of the title

Background - allows the user to set the color of the background of the title frame.

Show Time/Case - time/case is displayed in the title bar

Legend - User may enter the desired text.

Divide With Lines - draws a line between each line of text (separated by hitting Enter).

Transparent Box - allows the background and pipes to show through the Legend frame

Include Distance Scale - add a scale to the map.

Show Logo - User may create a bmp called Logo.bmp and save in the Pipe2018 folder. This may be checked to display this logo in the Legend.

Logo.bmp may be placed in one of three locations (path names relevant for Win7). In the order shown, the logo.bmp which is found first is the one applied to the map.

C:\Users\Public\Public Documents\KYPipe Docs\Images
C:\ProgramData\KYPipe
C:\Program Files(x86)\KYPipe\Pipe2018 (or other version)

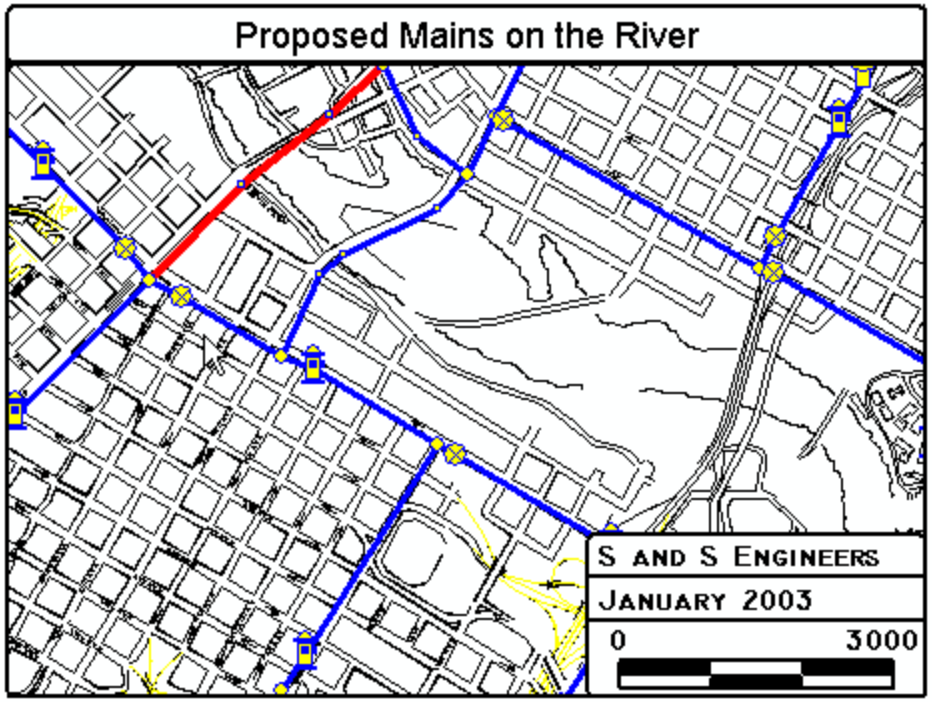
If the logo.bmp is not being recognized, make sure there is no other logo.bmp in any of the locations that take precedence.

Logo Size - sets the size of the logo from a choice of five settings.

Font - set the font of the Legend.

Background - allows the user to set the color of the background of the Legend frame.

Key Locations - the Legend may be placed in any one of the four corners of the map.




## 1.3.6.17 Map Link

Map Link

See also Scaling Background Maps

Utility Programs

---

Adding Maps – General Information

Both raster (jpg, gif, bmp, etc.) and vector (dwg, dxf) files may be used as background maps in Pipe2018. With vector files, the coordinates and scale are built in and may not need to be set by the user, but can be if desired. The scale or location of a vector map may be modified using the X Shift, Y Shift and Scale in the Map Setting | Background screen. For raster files, the user will almost certainly need to set a scale. The location will need to be set if a raster map is added to an existing model or if the user wishes to reference coordinates. These may both be set using the Map Link utility. See Scaling Background Maps.

## Creating a reference file with Map Link

Map Link is a Utility program used to create a reference file for a Raster file to be used as a Background Map. A reference file specifies location and scale for a Raster Background Map. Reference files have different file extensions depending on the file type. Those extensions are as follows:

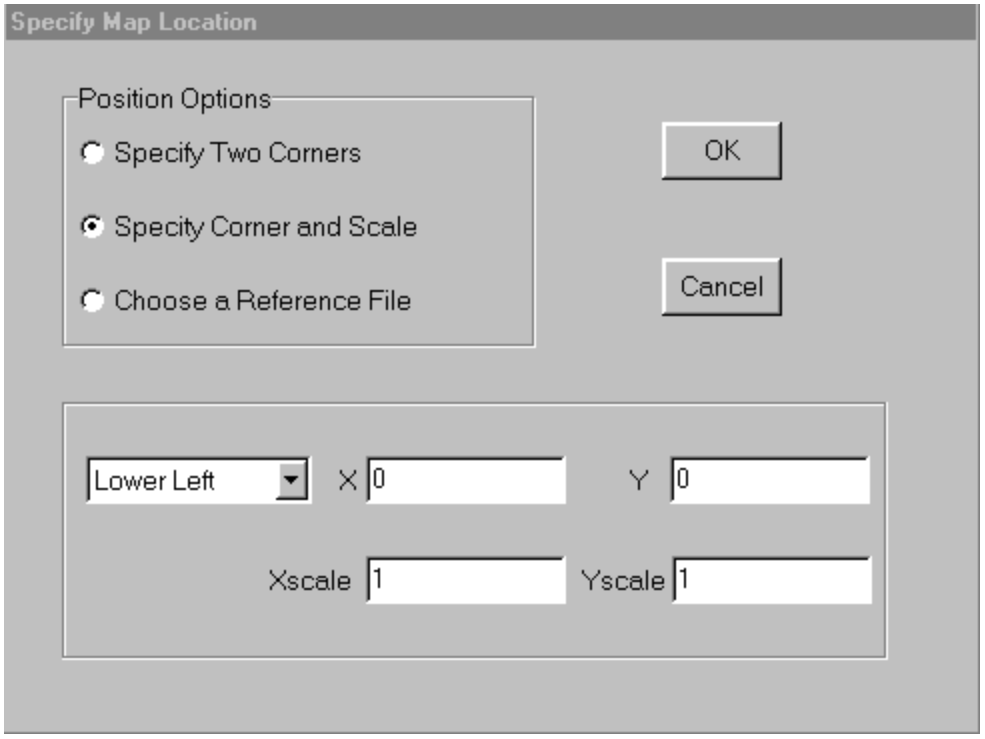
| file type | reference file |
|---|---|
| bmp | bmpw or bpw |
| jpg, jpeg | jpgw or jgw |
| tif, tff, tiff | tfw |
| gis | gsw |
| lan | lnw |
| bil | blw |
| bip | bpw |
| bsq | bqw |
| sun | snw |
| rs, ras | rsw |
| rlc | rcw |

The Map Link program is accessed by clicking on the Map Link icon in the Pipe2018 directory.

A new Raster Background Map may not be added without a reference file. An old reference file from another Raster Background may be used, but it is generally preferable to create a new reference file so that the position of the old Raster Background is not changed for future use or if it is currently in use.

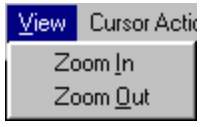
To create a new reference file enter the Map Link Utility and, under File, click Load Map. After choosing the map file, you will be prompted to specify your Map location. If you are adding the map to an existing system and you know the coordinates of that system, you may try to approximate where the corners of your Map

may fall (keep in mind, coordinates correlate with feet), or where one corner falls and assume a scale. This is not a necessary step however, and serves only to speed up the Map Scaling process. In general, it is recommended to choose Specify Corner and Scale under Position Options and for the Lower Left Corner (the default selection) to enter 0 and 0 for the X and Y coordinates and 1 and 1 for the Xscale and Yscale. Specifying a position (any position) creates a reference file. You may now exit Maplink and the Background Map may now be added then viewed on the Pipe2018 Map screen. See Scaling Background Maps to reset a position and scale for the map if necessary. If you don't see the map, use the Zoom to Selected Maps feature under Map Settings / Background tab in Pipe2018, or scroll your Map screen to the coordinates you specified in Map Link.




## Other Map Link functions

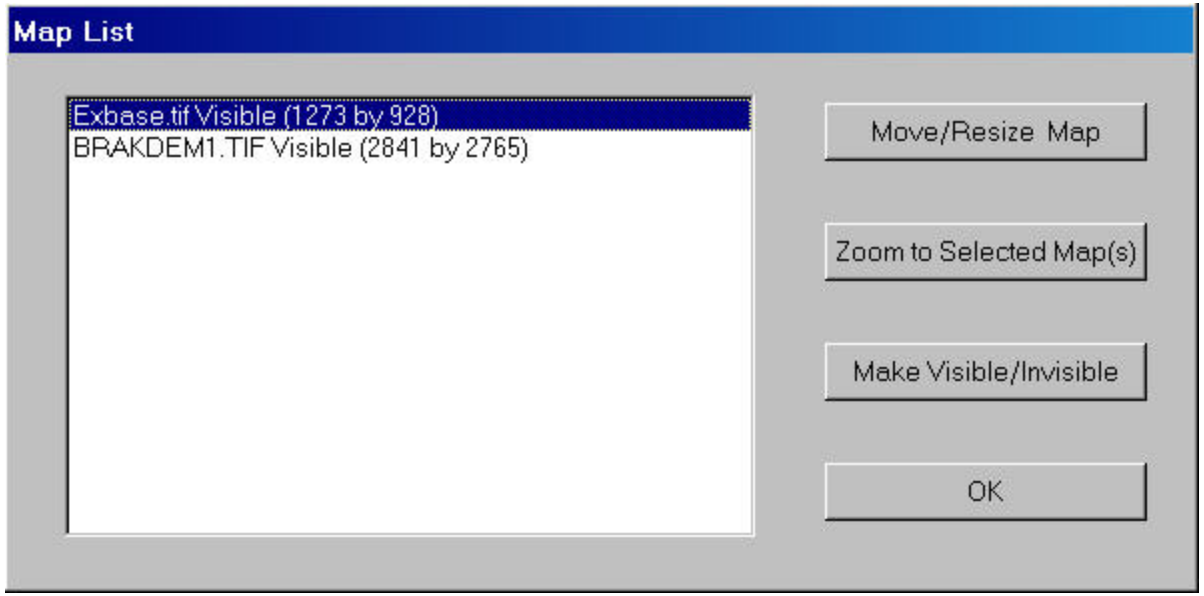
### Viewing options




Under View, there are standard Zoom In and Zoom Out options.

### Editing options

Clicking here will bring up the following screen:




**Map List**

This is a list of all of the maps currently loaded into the Map Link utility.

**Move/Resize Maps**

This will bring up the Specify Map Location Box shown above, allowing the user to specify the map location using coordinates and scale or with a reference file.

**Zoom to Selected Map(s)**

This will cause the viewport in Map Link to zoom to the map or maps which are selected (highlighted) in the Map List.

**Make Visible/Invisible**

This toggles the map selected in the Map List between visible and invisible in the Map Link viewport without removing from the Map List.

**Aligning new raster maps using Map Link** - When creating a new model, a user may have several background maps he or she would like to overlay in the Pipe2018 Map screen and use to lay out the graphical piping system. Maplink may be used to align several raster maps to each other at once. (To align background maps to existing pipes see Scaling Background Maps.) To do this, load the desired maps, specifying a

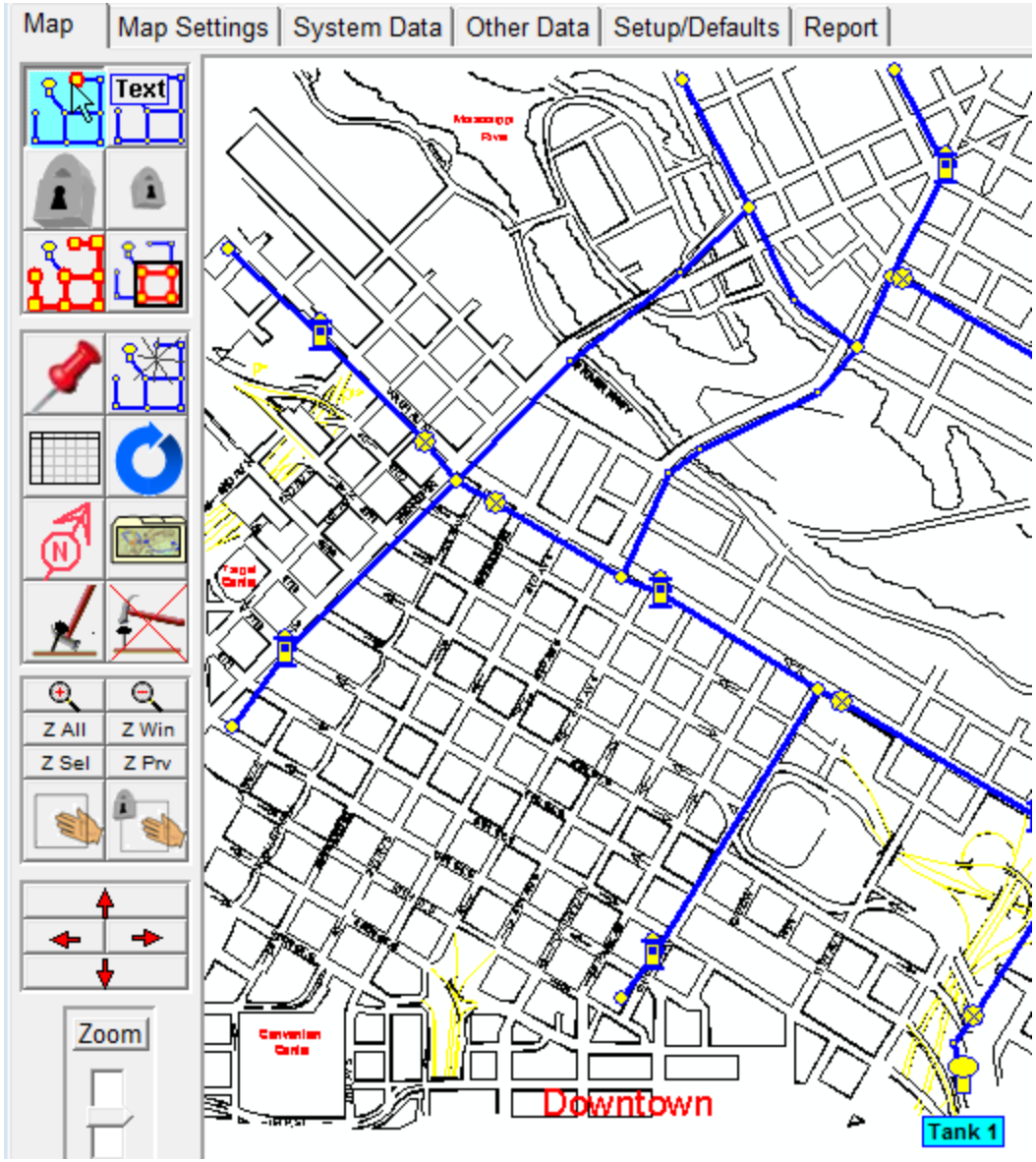
temporary position such as described above. Then under Edit Maps, select Move/Resize/Edit Maps. Then select Move/Resize Map with the target map highlighted in the Map List. Coordinates and scale may be changed as needed to achieve alignment of the maps.

## 1.3.6.18 Map Screen

This is the main drawing area. You system will be shown graphically here. This is the main space in which you will work to enter or edit your piping system.

KYnetic Return to Map icon:

Classic:


Map
Map Settings
System Data
Other Data
Setup/Defaults
Report
Text
Z All
Z Win
Z Sel
Z Prv
Zoom
Downtown
Tank 1

Operating Modes

Panning Controls

Pipe System Layout and Modification

## 1.3.6.19 Minimum View Scale

Minimum View Scale

---

If, upon zooming in with a vector background map, Pipe2018 crashes, follow these simple steps. This will allow the user to set the minimum view scale and zoom further in, past the map's viewable point .

1. In Pipe2018, open the file with the vector background map. Click Zoom All. Then begin Zooming In using the button. Count the number of zoom levels used until the program crashes. For example, lets say you zoom in 11 times and on the 11th time it crashes.

2. Reboot Pipe2018. Open the file again and click Zoom All again. Now zoom in one fewer times than it took to cause the program crash. Using our example you would then zoom in 10 times.

3. Under Map Settings | Backgrounds click on the Properties button. Read and make note of the number in the Scale field. Let's assume it says 500.

4. Click on the Properties button (within the current Properties window). In the field called Minimum View Scale, enter the value you read in step 3 minus 1. In our example, this would be 499.

5. Now the user will be able to zoom in past the minimum viewable size of the vector map without problems.

## 1.3.6.20 Node Images and Text Nodes

Node Images and Text Nodes

See the Images video on the KYPipe web site.

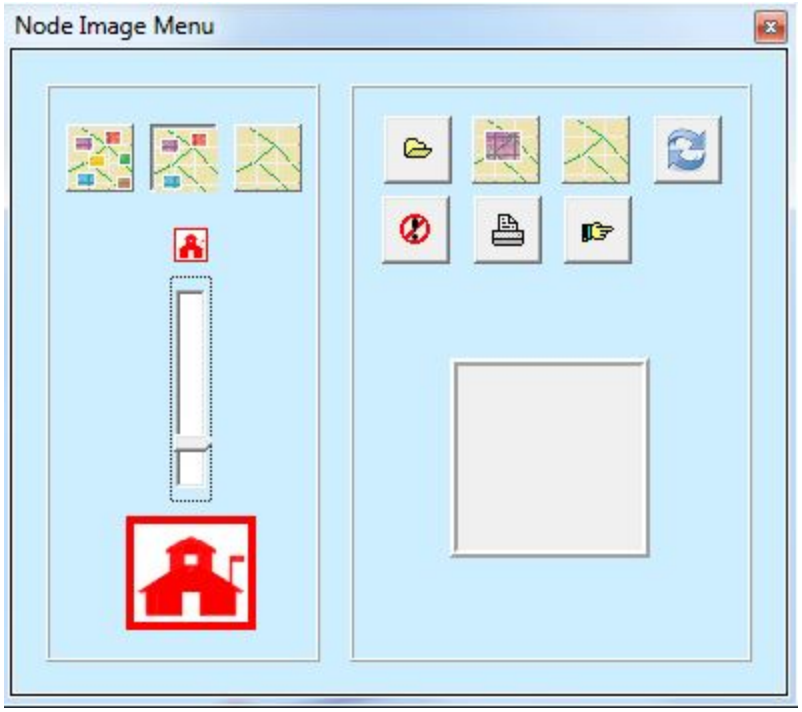
---

See also Node Data Boxes

Node Images
An image file (bitmap) can be loaded and displayed for any node (including Text Nodes) in the system. This feature can greatly enhance your model and can be used for a variety of purposes. This includes primarily cosmetic information such as a photograph of a tank or functional information such as a detailed map of a shutoff valve location.

**KYnetic Interface:**

- Image icon, brings up the Image menu shown below:




The three buttons on the top left are used to 1) Display all node images, 2) Display only the node images which are turned on for that node or 3) hide all images.

The slider changes the size of the currently selected image as it appears on the map.

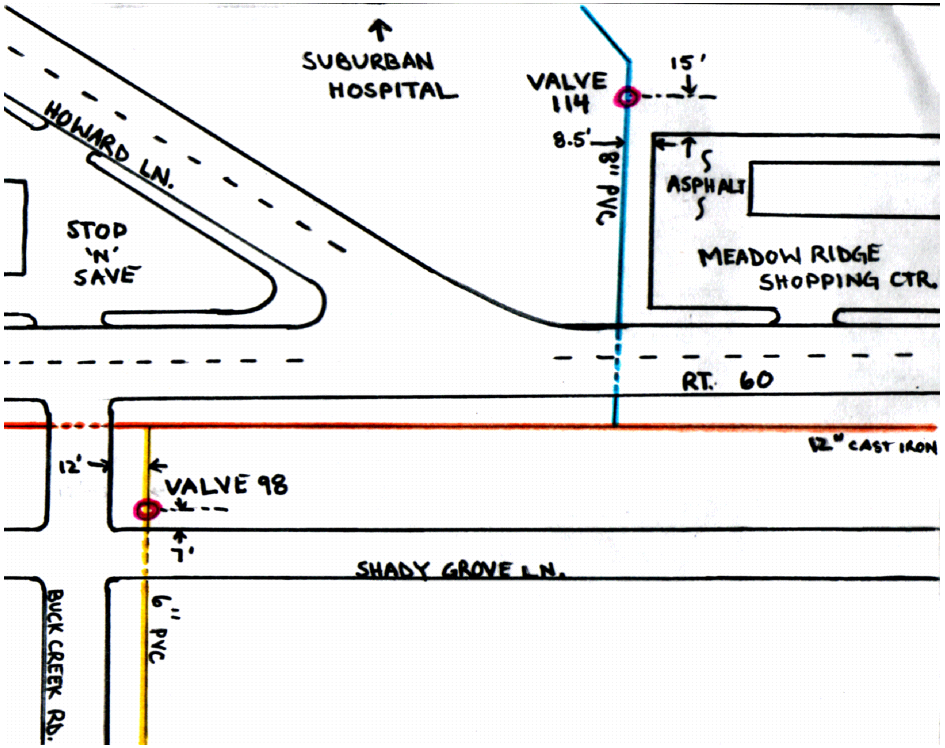
On the top left, the buttons will load an image, show, hide or refresh the image.

In the next row the options are to delete the image from the node, print, or change the location (quadrant) of the image on the map.

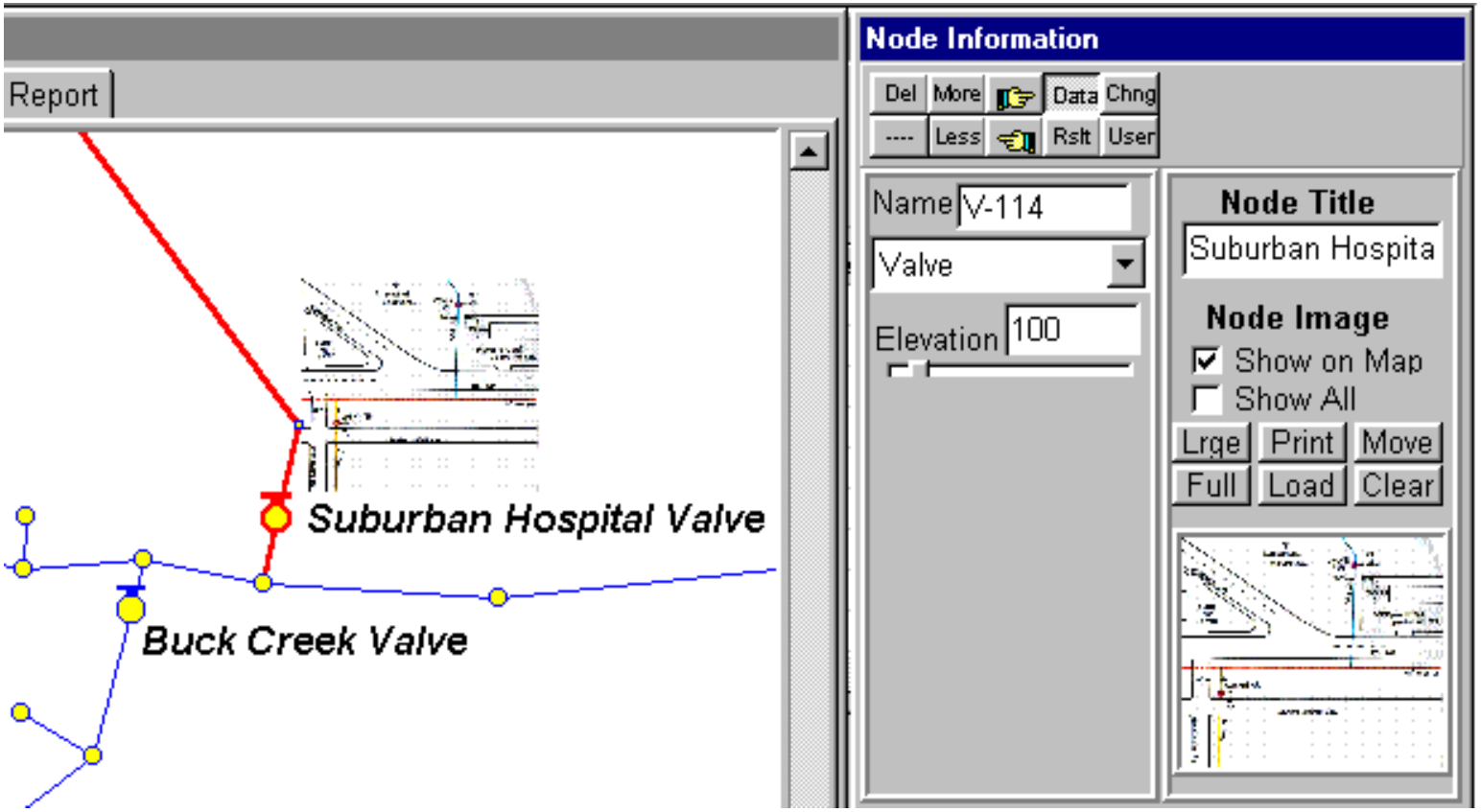
**Classic Interface:**

Here a photo of a pump which has been attached and can be displayed when the pump is selected on the graphical system.

Here is a map which field personnel may use to quickly find the location of a particular valve:




Once the map is scanned to create a bitmap file, it may be attached to the selected node or element by using the Load button. Viewing options within the Node Information Window toggle between Full, Large and Small (shown here). The Print button will print the image. The Clear button removes the image from the node. The Move button will change the quadrant of the image relative to the node on the map.




Text Nodes

These may be placed anywhere on your map to provide information. They can be easily added, moved or deleted.

See Text Nodes and Text Node Data

Back to Internal Node Data

## 1.3.6.21 Panning Controls

Panning Controls

---

**KYnetic Interface:**

Zoom with mouse wheel or with these buttons:

Pan - simply click and drag the map window in Layout mode.

Important note: In group mode, click and drag will move select nodes! Go back to Layout mode to pan.

Zoom All - This button causes the view in the Drawing Area to be zoomed to the point that all the features of the drawing are visible. You may wish to turn off node and or pipe labels if this view appears cluttered.

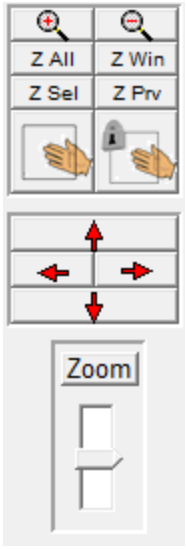
Zoom Window (**Zoom Lock**) - Allows the user to draw a box around the area to be magnified. **Right click this button for Zoom Lock, or CTRL+SHIFT+W**

Zoom Previous - Allows user to zoom to the previous zoom setting.

Zoom Selected - To be used with Group Mode. Zooms in to fit all elements highlighted in Group Mode within the viewport.

**Classic Inteface:**

These buttons allow you to quickly pan across the picture my moving in the direction of each arrow a large piece at a time.




The first two buttons zoom directly in and out.

Z All - This button causes the view in the Drawing Area to be zoomed to the point that all the features of the drawing are visible. You may wish to turn off node and or pipe labels if this view appears cluttered.

Z Win - Allows the user to draw a box around the area to be magnified.

Z Sel - To be used with Group Mode. Zooms in to fit all elements highlighted in Group Mode within the viewport.

Z Prv - Allows user to zoom to the previous zoom setting.

Pan (hand) - Allows the user to use the cursor to click and drag the map within the viewport to the desired location.

Pan Lock (hand + lock) - Lock into pan mode.  To cancel pan mode, simply click again.

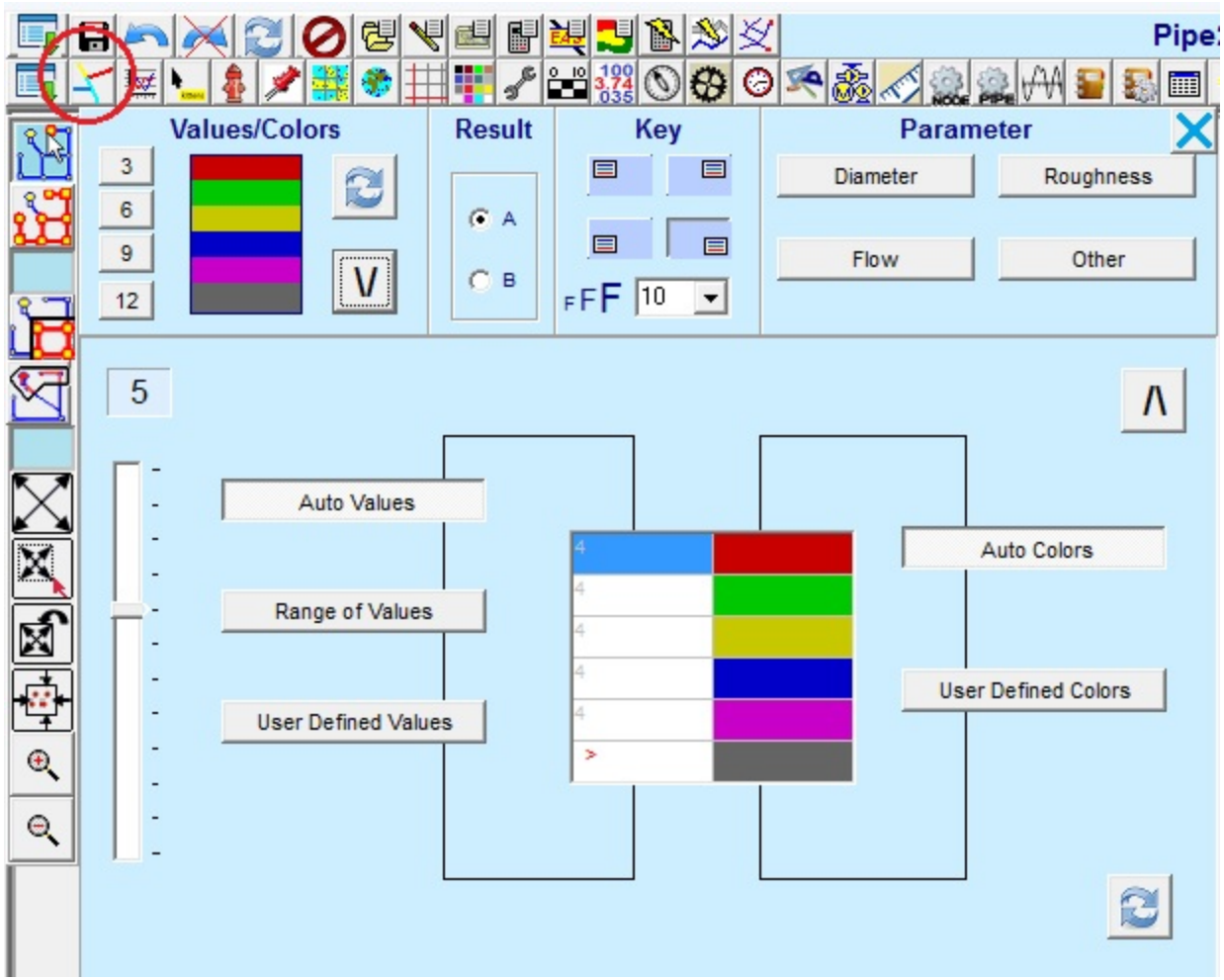
Arrows - The arrows pan the viewport toward the direction of the arrow (thus exposing the portion of the map opposite the arrow direction) a distance of half the

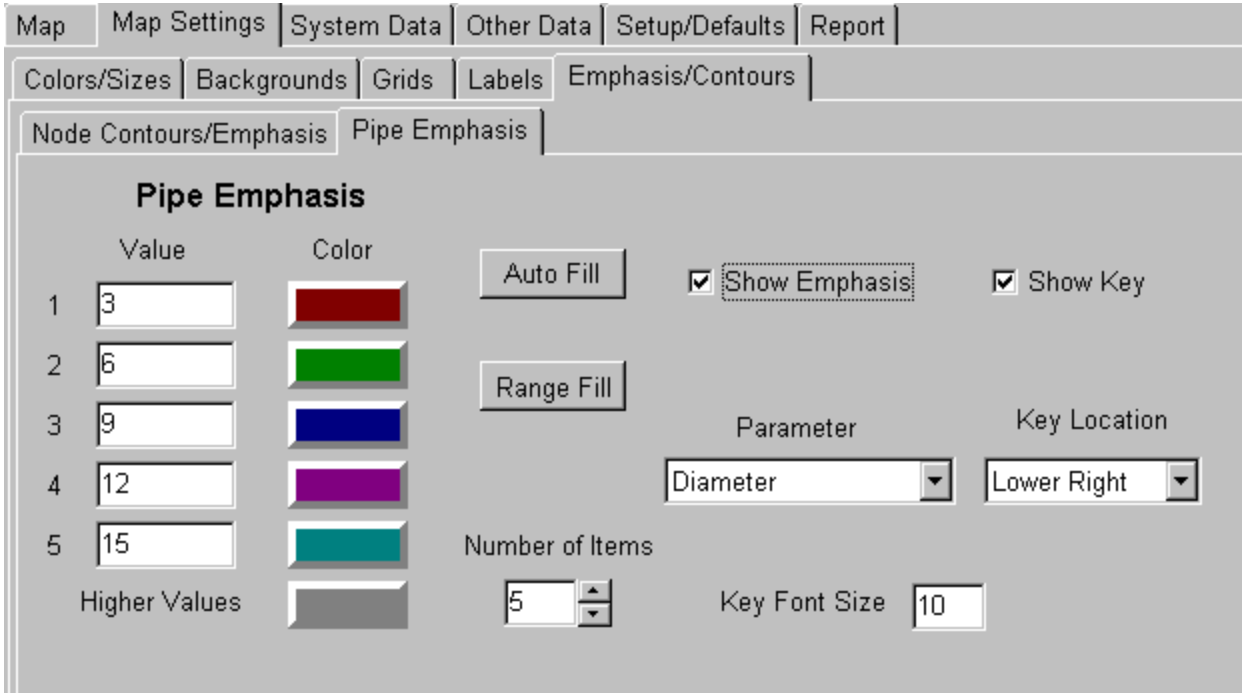
viewport screen.  Direction of pan may be switched in System Data | Preferences.

Zoom Slider - Zooms to custom level by click and drag slider bar.

### 1.3.6.22 Pipe Emphasis

Pipe Emphasis


Values/Colors
Result
Key
Parameter
3
6
9
12
A
B
10
Diameter
Roughness
Flow
Other
5
Auto Values
Range of Values
User Defined Values
Auto Colors
User Defined Colors




---

This tab is used to set up color emphasis for pipes. Color Emphasis sets the color of the pipes based upon the value of their data.

Value - These are the values used to determine the range used for each color. Values that are less than or equal to the target value are set to the target color.

Auto Fill - This box automatically sets the values in the value boxes to establish equal sized ranges to that span the data type selected in the Parameter box.

Range Fill - This button causes the Value boxes to be filled in to establish equal sized ranges between the bottom and top value boxes. Note that you should fill in the first value box with the lowest value and the top box with the highest value BEFORE pressing the Range Fill button.

Number of Items - This sets the number of user specified ranges that will be present.

Color - These are the colors used to draw the pipes that fall within the specified range of values

Classic - click on the color to bring up a color palette.

Auto Colors - uses default color set

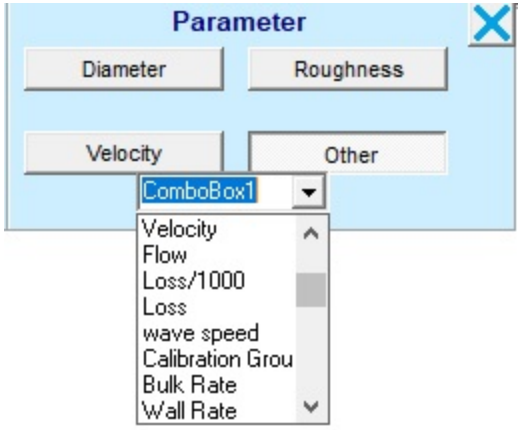
User-Defined Colors - user may set individual color values, click on color to bring up a palette.

Turn on/off emphasis - Classic - use "Show emphasis" check box, KYnetic - select the parameter button (Diameter, Flow, etc)

Parameter - In Classic this selects the data item used to set the ranges.

In KYnetic, this doubles as the on/off function for emphasis display.

**Added in Pipe2018 - Loss and Loss/1000**

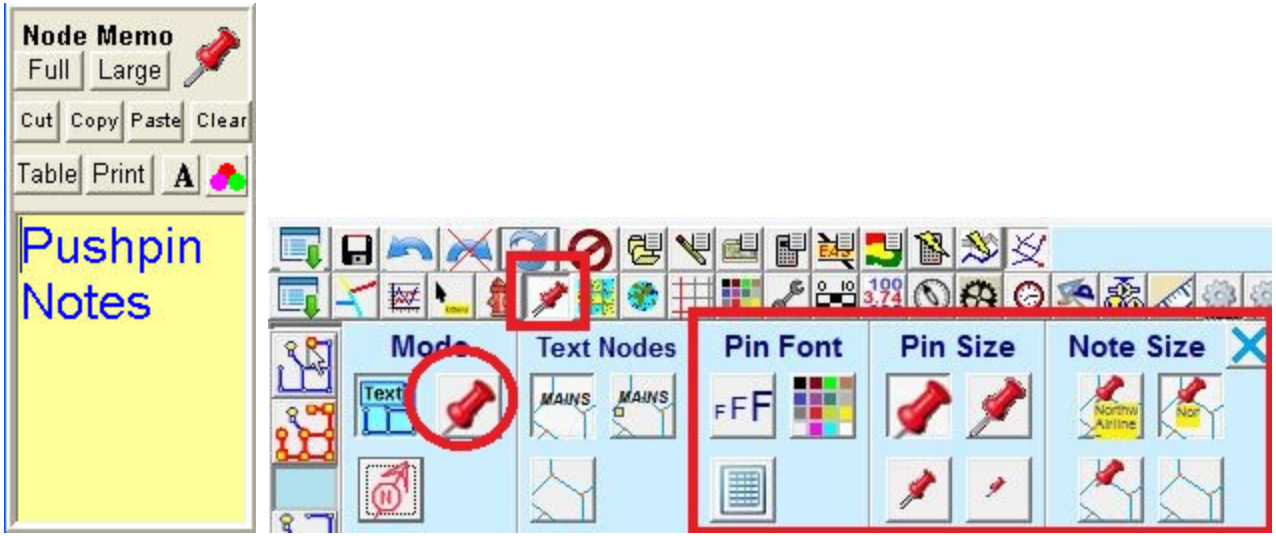



**Key / Key Location -** This allows the user to select the corner of the map in which the key will appear. In KYnetic, the location button functions as on/off button. In Classic, Key display is turned on/off with the **Show Key checkbox**

**Key Font Size -** This box allows the user to choose the font size for the Key.

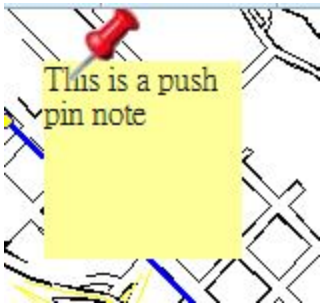
## 1.3.6.23 Pushpin

Pushpin




---

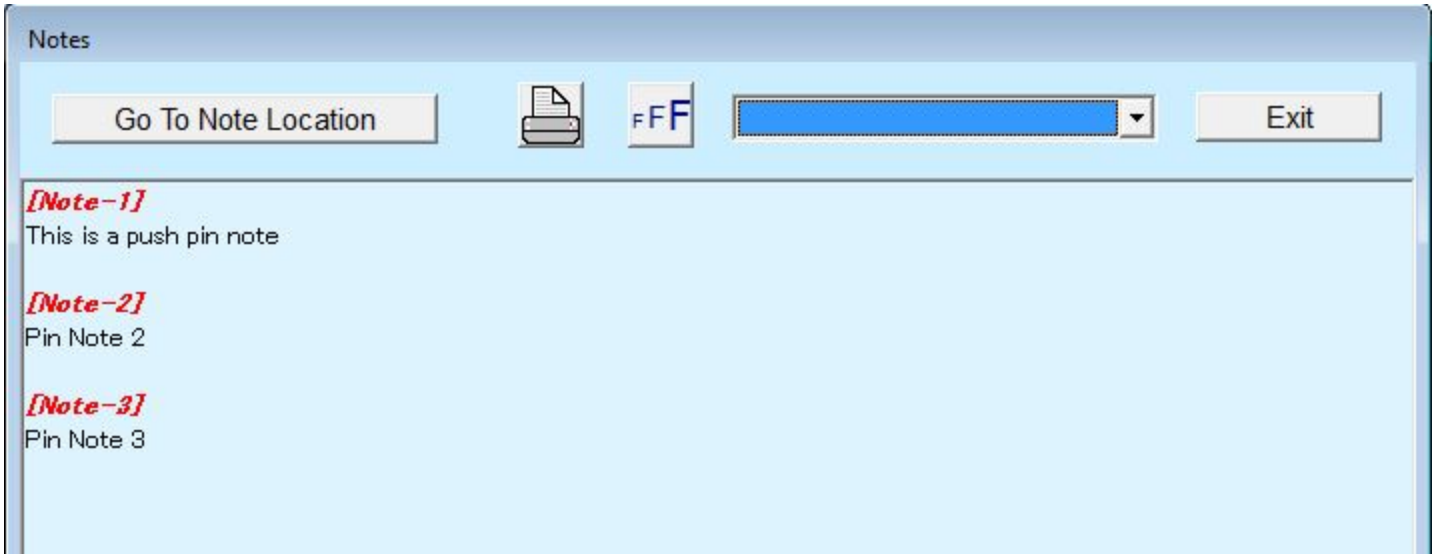
When you click the pushpin symbol on the left-hand side of the map screen, a node or area of the map screen may be selected. A blank 'note' will appear to which text may be added.




The font and color may be set for the pushpin note. Also, all notes are stored in a table. Notes may be printed singly or all together in the table format.

To make notes invisible click the Clear button. To make them reappear, click the pin again (then OK and the pushpin button again if you do not want to add a new pushpin note).

A table summary of all Notes may be created (KYnetic only) by clicking the Table icon under Pin Font.




## 1.3.6.24 Scaling Background Maps

Scaling Background Maps

Adding and Scaling Vector Files: to new Pipe2018 files

to existing Pipe2018 files

Adding and scaling raster files: to new Pipe2018 files

to existing Pipe2018 files

see also Backgrounds

Background Images

---

### Vector Files (dwg, dxf, etc.)

Vector files are those created in AutoCad or similar environment where coordinates and a scale are inherent. Pipe2018 recognizes these parameters and will use

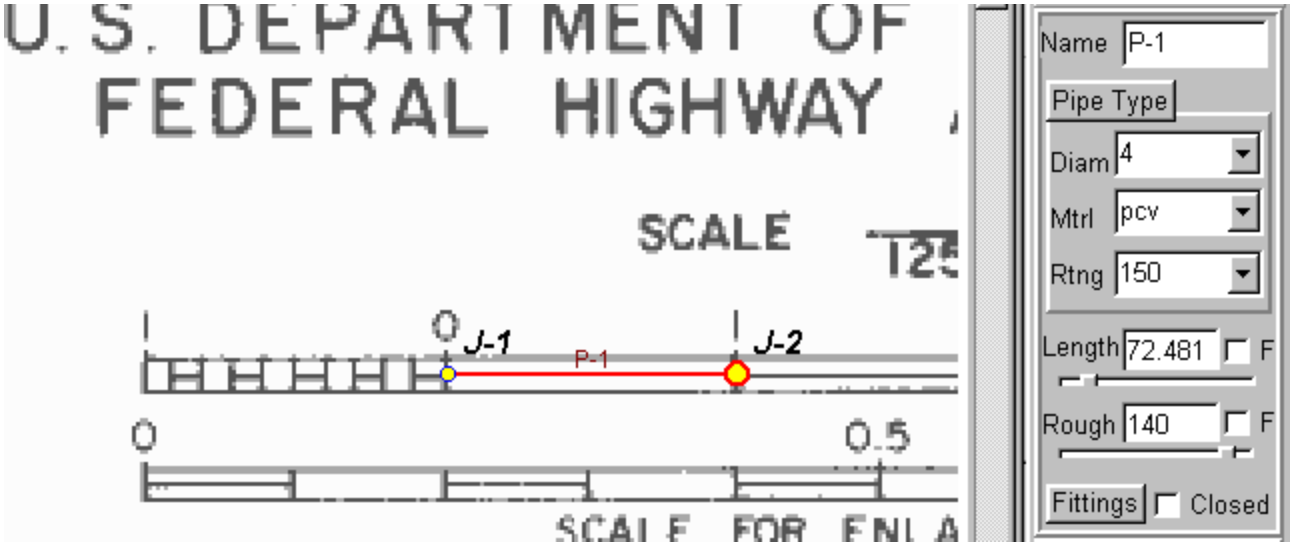
them as the default. In order to use different settings, Pipe2018 must be instructed to do so. For vector files, this is done using the X Shift, Y Shift and Scale under Map Settings | Backgrounds. For problems zooming in with a vector file see Minimum View Scale.

Kynetic Background menu icon:

## Vector files - New Pipe2018 file (no pipes entered):

1. When adding a vector file, under Background menu or Map Settings / Background (Classic), simply click Add Map and select the desired file. If your scale and coordinates are acceptable, begin laying out your system.

2. If the scale of your vector file does not match the scale you would like to use in laying out your Pipe2018 model (e.g. meters vs. feet), then use the Pipe Scale Factor under Other menu or System Data / Other (Classic) to set the desired scale. If the scale factor is not known, then lay out a pipe of known length on the background. Use the map legend scale (as in the example below) if one is included on your background. Click on the pipe. Compare the scaled length in the Pipe Information box to the length you want your pipe to be. Use this ratio (desired length/scaled length) in the Pipe Scale Factor setting in Other menu or System Data / Other (Classic). In the example below, the length we want the pipe to be is 1 mile or 5280 feet, which corresponds to the scale shown on the legend. The scaled (Pipe2018 assigned) pipe length is 72.481 feet. The Pipe Scale Factor would be 5280 / 72.481 which is equal to 72.85. Enter this into the Scale Factor box.




3. If the coordinates of your vector file do not match those you would like to use in the Pipe2018 model, first make sure scale factor is set as above, then place a node on the background map for which the coordinates are known. Set the X Shift and Y Shift in Map Settings / Background accordingly (to change a location from an x coordinate of 9 to 5 the X Shift would be -4). Note that this will affect any other maps already loaded onto your system, and it must be taken into consideration when adding new maps.

### Vector files - Existing Pipe2018 file:

1. If you are adding a vector file as a background to an existing system model, click Add Map under Background menu or Map Setting / Background (Classic)and view the background in the Map screen (you may need to use the Zoom to Selected Map feature). If the coordinates and scale used in generating the vector file are identical to the ones you want your Pipe2018 model to be in, then the map should be aligned with the piping system. If there is a discrepancy, use the above instructions in items 2 and 3 to set the coordinates and scale.

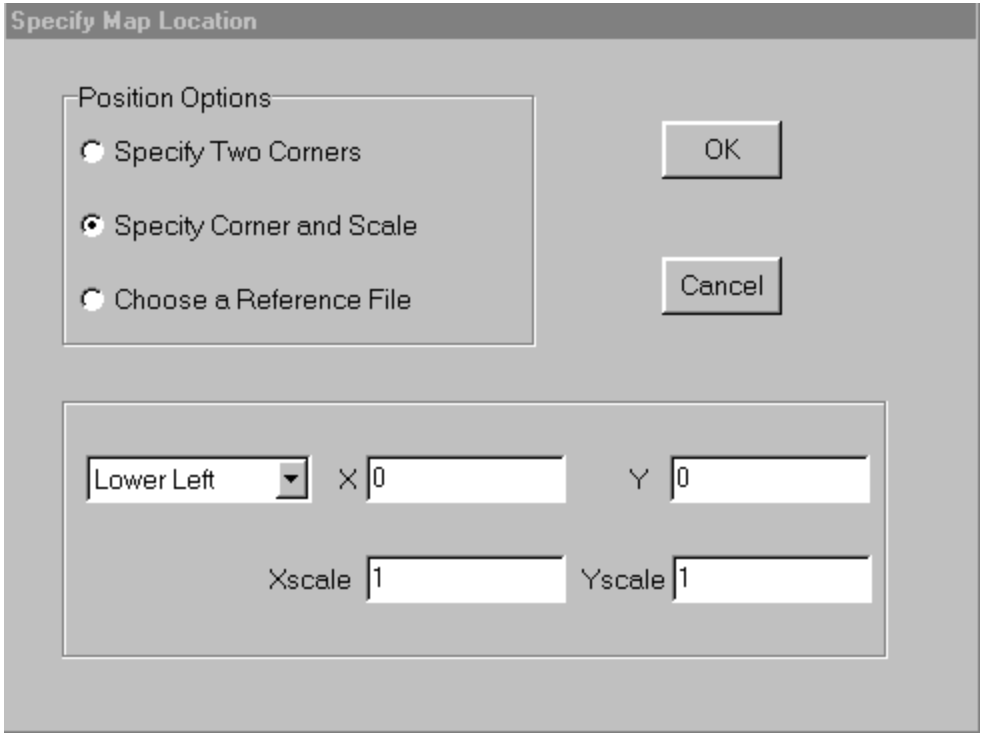
## Raster files (bmp, gif, tif, etc)

Pipe2018 uses a reference file to specify the permanent location and scale of a background map. The scaling options under Background menu or Map Settings/ Backgrounds (Classic)are view-related settings when used with Raster files. Scale and position set this way may be saved with your Pipe2018 .P2K file, but will be need to be manipulated with the addition of other backgrounds. Hence the need for a permanent, separate reference file. The scaling options are invaluable in obtaining the correct scale and coordinates to be recorded in the permanent reference file. The reference file is created using the utility called Map Link.

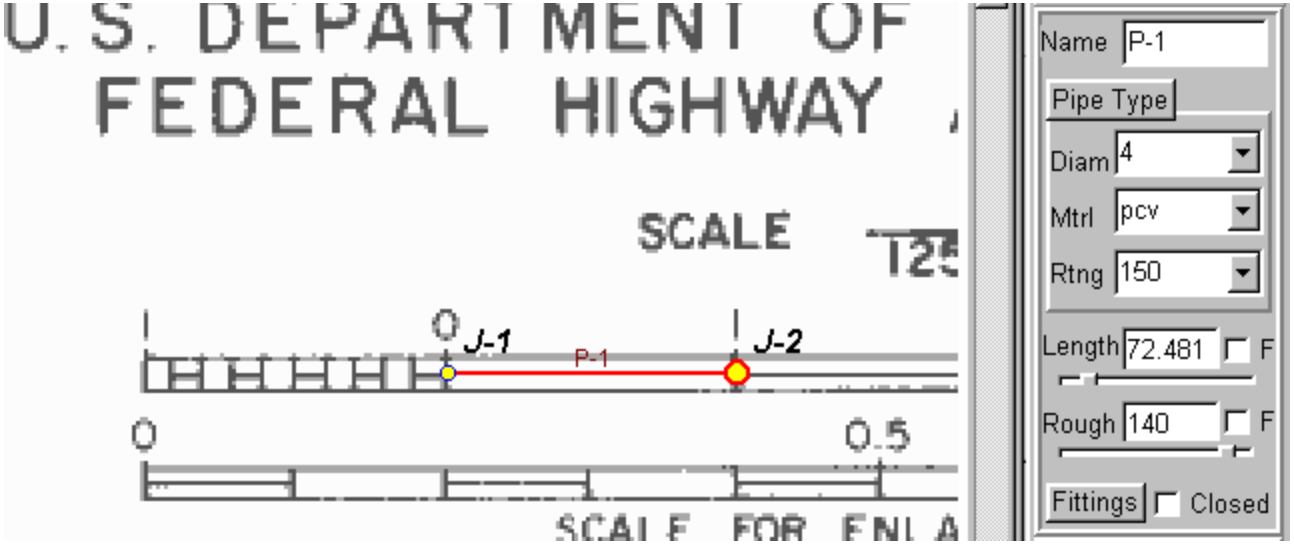
Note that there are a few Raster formats not supported within Pipe2018. Most of these may be converted to a .tif format usable in Pipe2018 with the To TIFF utility.

### New Pipe2018 file (no pipes entered):

1. When a background file has not been used before it will have no reference file. Therefore, when it is added in Pipe2018 (Add Map button) it will be placed by default at the origin (0,0). This default postition is fine when only one background map is used and there is no need for specific coordinate data. To define a different coordinate location for the background map, create a reference file by clicking on Map Link. In Map Link, under File, click on Load Maps. Select the desired file. When the Specify Map Location Box appears, three options are given. Use one of the first two to set new coordinates. Next the user will need to scale the map.




2. Scaling the background. There are two easy ways to determine the scale for a raster background map. If the map has a legend, you can zoom in on the map scale, place a pipe along the scale, compare the length of the pipe with the scale and calculate the ratio of the lengths (desired length/scaled length). Otherwise, place a pipe of known length on the map (e.g. from the intersection of Main Street and 1st Street to the intersection of Main Street and 2nd Street, which is known to be 350 feet.). In the example below, the length we want the pipe to be 1 mile or 5280 feet, which corresponds to the scale shown on the legend. The scaled (Pipe2018 assigned) pipe length is 72.481 feet. The Pipe Scale Factor would be 5280 / 72.481 which is equal to 72.85.



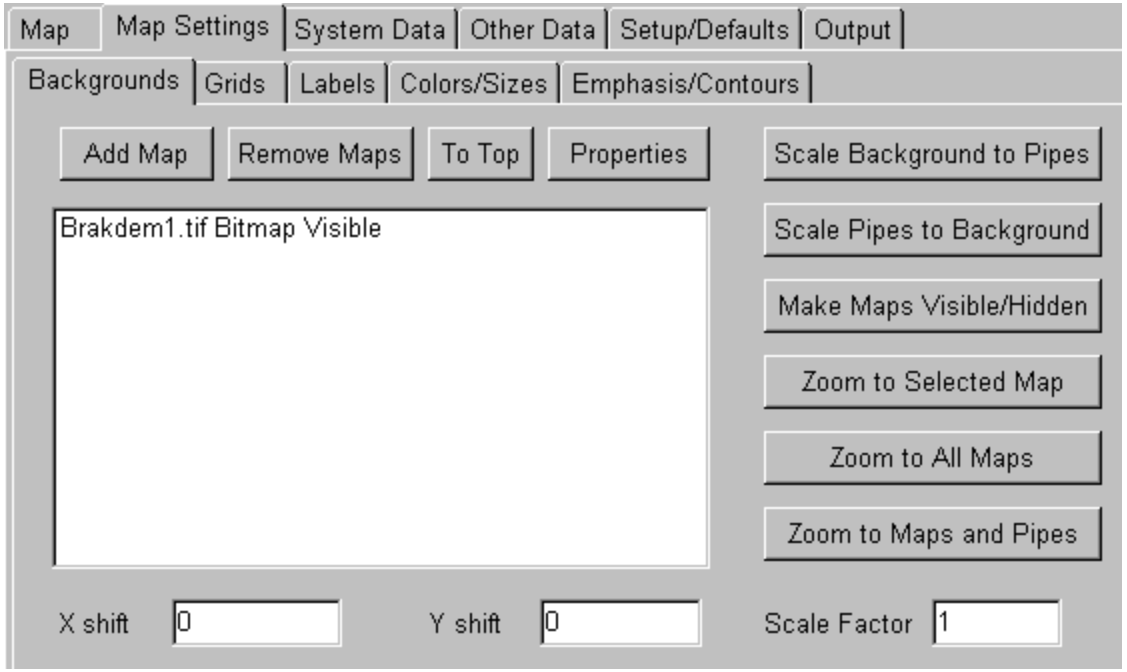

3. If you are only using one map, you may enter the scale (for our exampple 72.85) into the Scale Factor box. Otherwise to create a permanent reference file, go back into the MapLink utility. Load the map. Under Position Options, choose Specify Corner and Scale. Make sure the Lower Left corner is set to 0, 0 (unless you want to specify different coordinates). Enter the scale you calculated in step 2 in the Xscale and Yscale boxes.

4. Your map should now be scaled. You may begin to lay out your piping system.

## Existing Pipe2018 model:

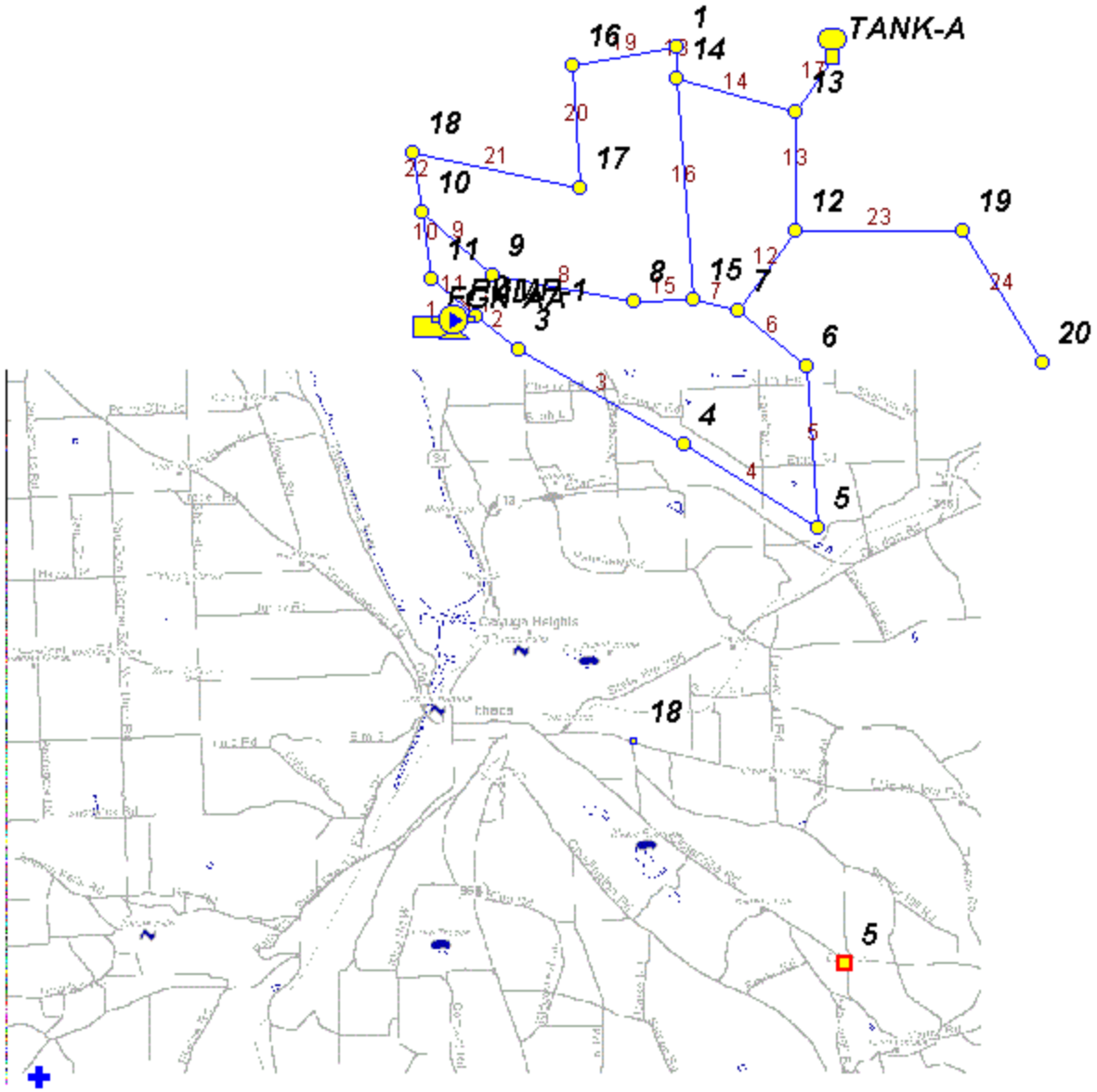
1. When a background file has not been used before it will have no reference file. Therefore, when it is added in Pipe2018 (Add Map) it will be placed by default at the origin (0,0).

KYnetic Backgrounds menu icon:




2. With the newly added background selected (highlighted), click Zoom to Selected Map. Go back to your model by clicking the Map tab.

3. If you have a model already set up, your background and your network will not be aligned. The best way to establish the correct position for your background is with the Scale Background to Pipes option. In order to do this, establish a correlation between two nodes at opposite corners of your piping system and their corresponding position on the background. In the Text Mode (vertical task bar on the left of your Map screen) add two text nodes onto the map in the location you would like the system nodes to be and give them the same names as their corresponding system nodes.




4. Next, in Map Settings / Backgrounds, select Scale Background to Pipes. View the result in the map screen. If the background and pipes are not lined up to your

satisfaction, repeat step 3, zooming in on the target text node locations for accuracy.

5. Now, the map is aligned, but the coordinate and scale data must be recorded in the reference file so that Pipe2018 will always recognize the location of this map, even if subsequent maps are added.

a. Under Map Settings / Background, view and make note of the X-Shift, Y-shift and Scale.

b. Then, set these parameters back to 0, 0, and 1, respectively.

c. Next, remove the map.

d. Go back into the Map Link utility.

6. In Map Link, load your background again. Repeat step one, entering the x, y, and scale data collected from step 5. This will reset your reference file to the location established in step 3.

7. In Pipe2018 add your background as in step 1. Maps and network model should now be aligned.

## 1.3.6.25 Scroll Bar

Scroll Bar

**The scroll bars show your position of the Drawing Area relative to the extents of the picture. By clicking on the end arrows or by dragging the indicator box in the middle you can shift the area that the Drawing Area contains.**

See Input and Editing Shortcuts

## 1.3.6.26 Text Nodes

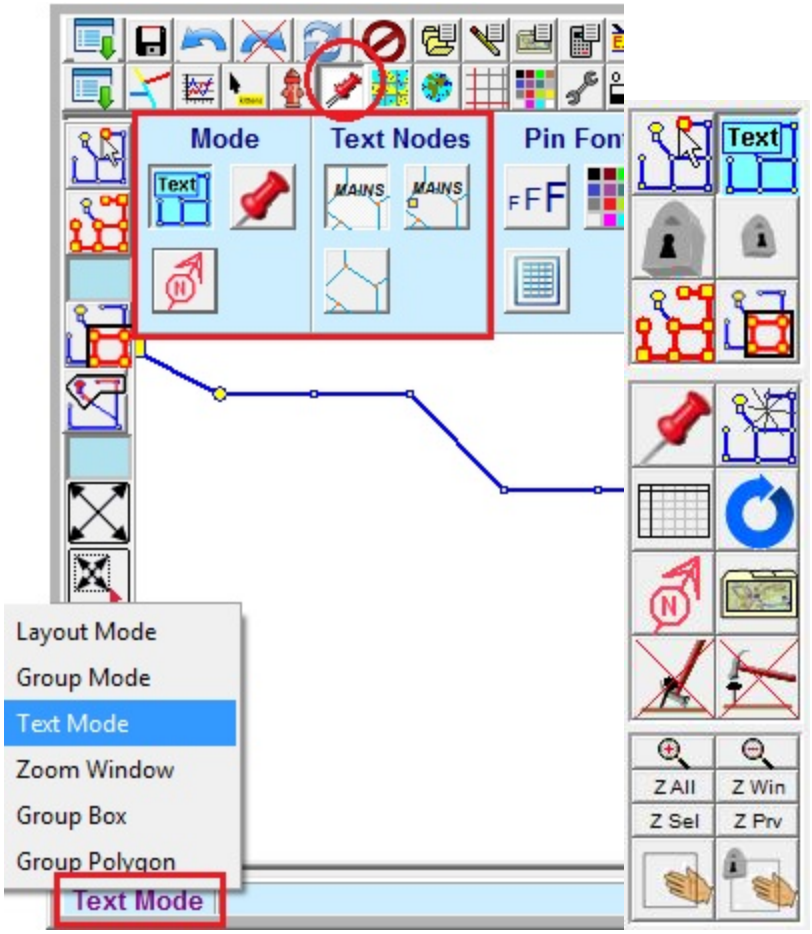
Text Nodes

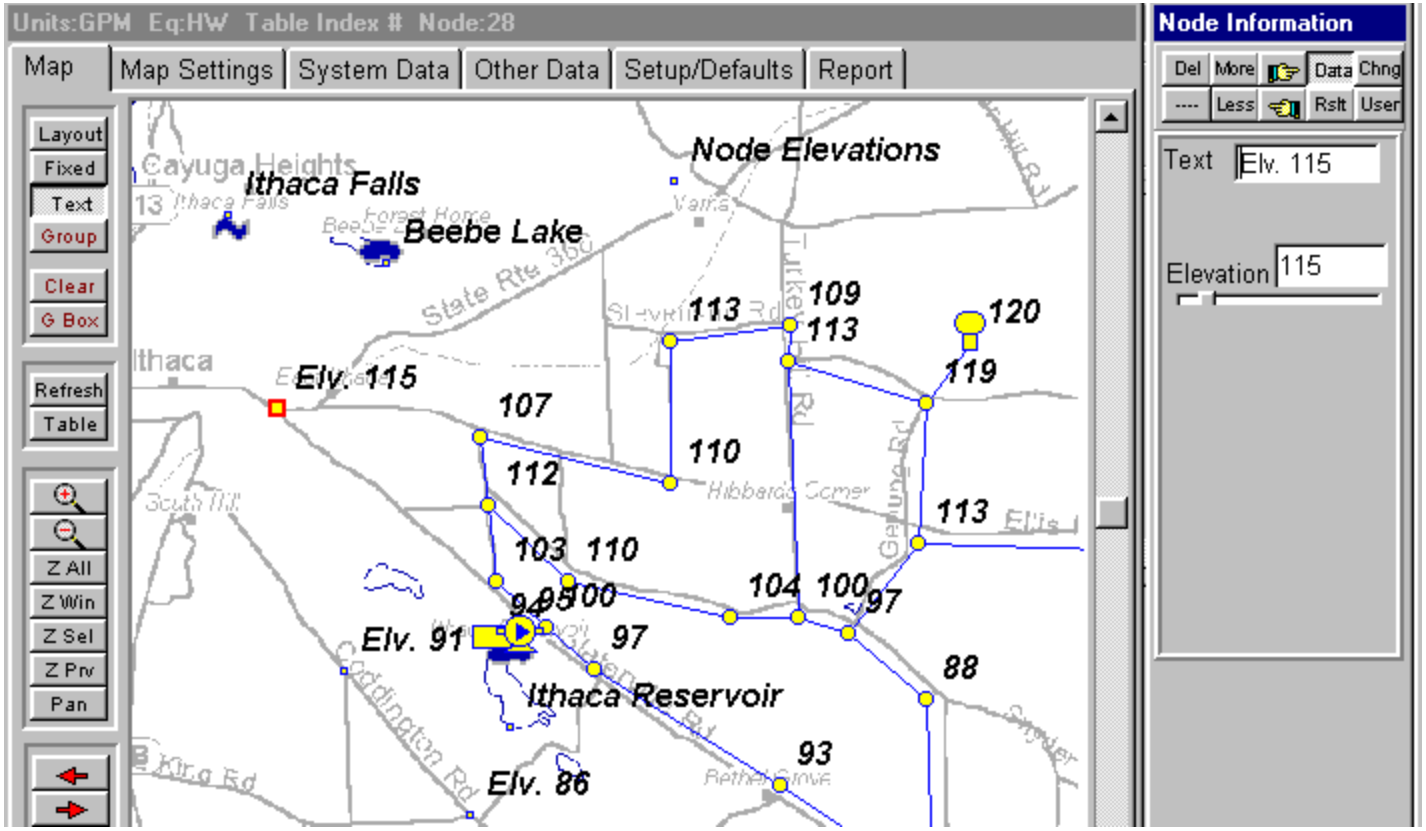
See the Images video on the KYPipe web site.

See also Map Legend.

---

**Text Nodes** - Text nodes are used to provide text information and can be located anywhere on your map. They are added and deleted in the '**TEXT**' mode and several options are available for displaying this information. See Text Node Data and Node Images and Text Nodes


Mode
Text Nodes
Pin Fon
Text
MAINS
MAINS
Text
Layout Mode
Group Mode
Text Mode
Zoom Window
Group Box
Group Polygon
Text Mode
Z All
Z Win
Z Sel
Z Prv




## 1.3.6.27 Text Node Data

Text Node Data

See the Images video on the KYPipe web site.

See also Map Legend.

---

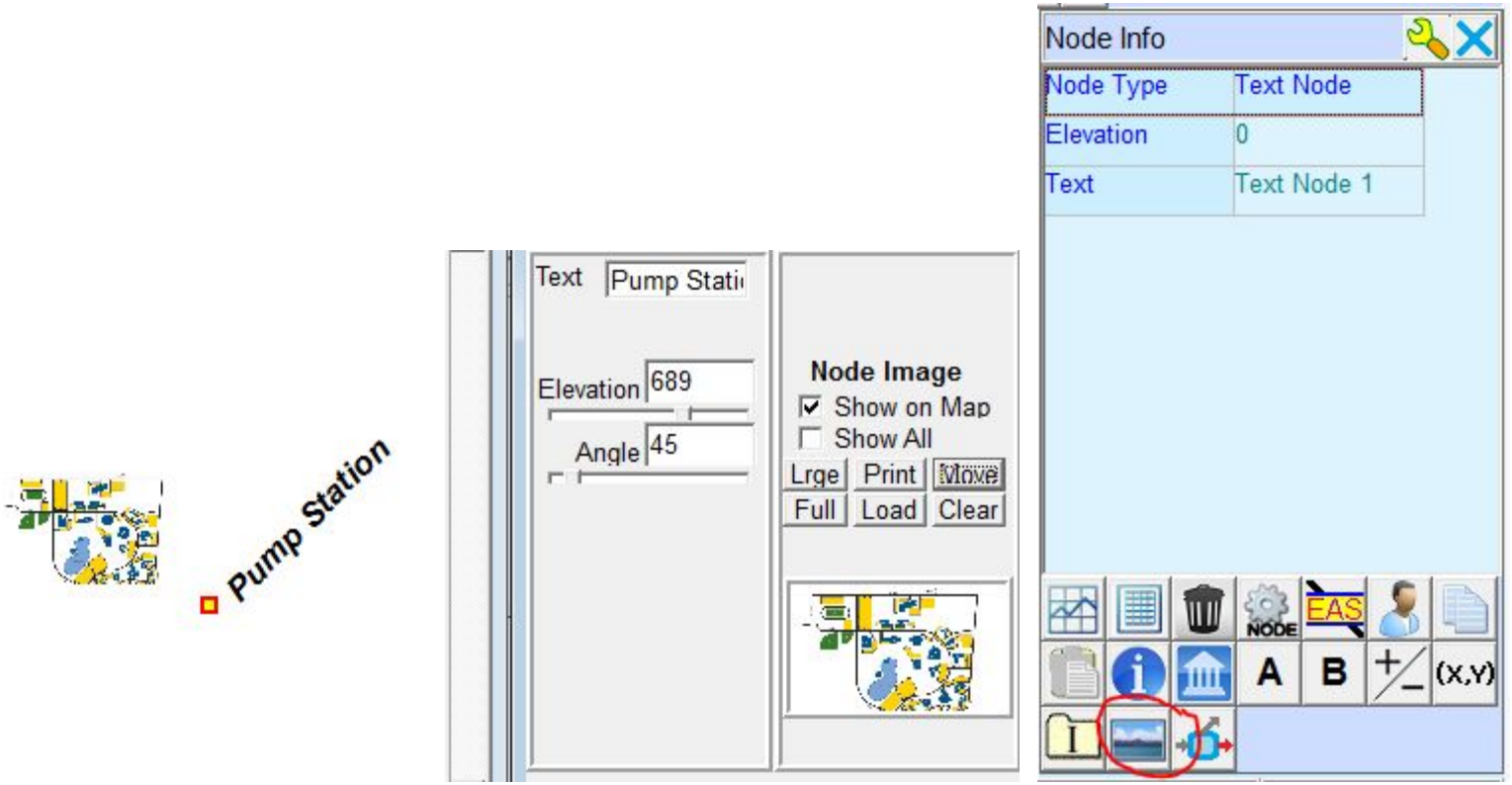
## Creating and Locating Text Nodes

To create a Text Node, select the Text operating mode (button located in the left-hand, vertical toolbar in the Map screen). Then move the mouse to the desired location and left click. This creates a small square symbol (locator box) to mark the text location. This box is also used to click and drag the location of the text. In the associated Node Data Box, the title is entered, which is the text which will be displayed at the text node location (this may be left blank).

Angle - define an angle for the text label.

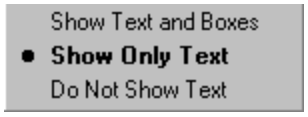
To add or delete a text node, you must be in Text Mode by clicking on the "Text" button.

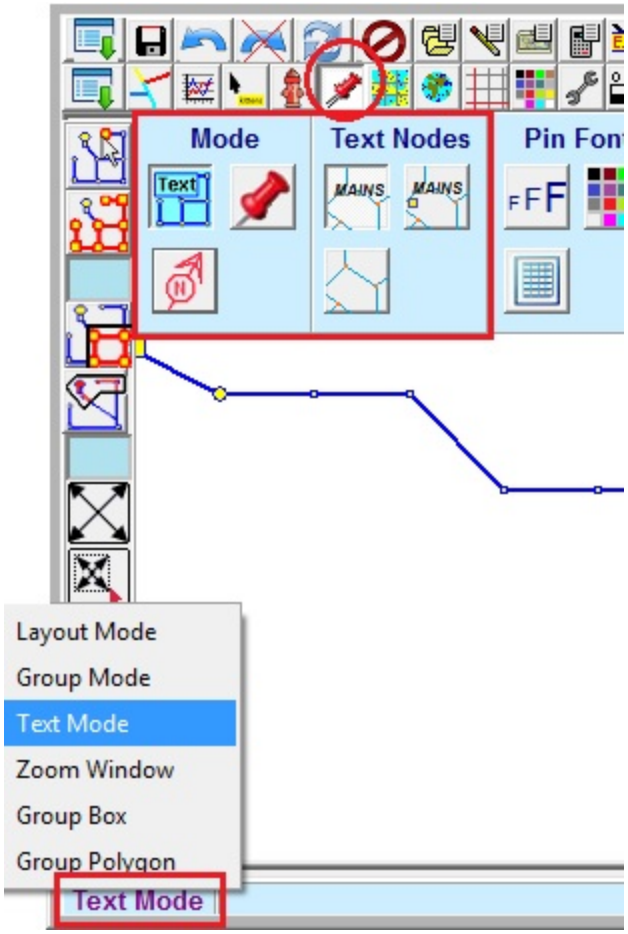
A bitmap image may also be loaded in the Node Image Box and displayed on the screen (see Node Images and Text Nodes). Lastly, an elevation may be assigned to the text node location. This is useful in creating elevation contours.



## Viewing Text Nodes

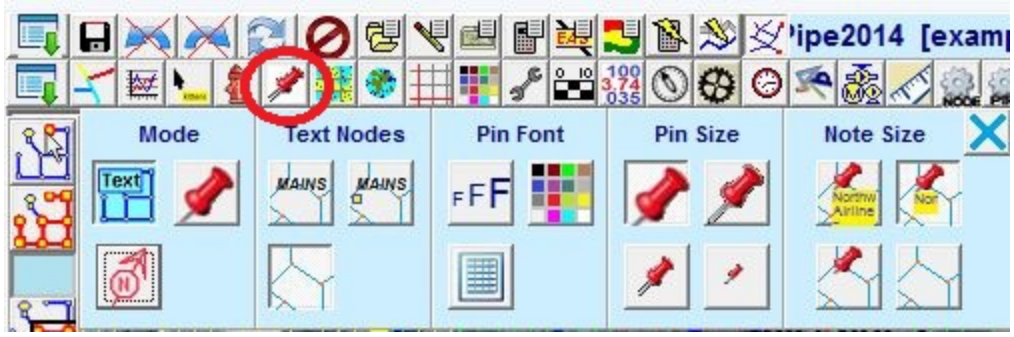
Under View (Main Menu) or Push Pin and Notes icon (KYnetic), select Show Text. The following options are displayed:






To select a Text Node (to move, edit text, or delete), the first selection must be activated and the locator box displayed. Then simply point to the locator box and left click to select.

## 1.3.6.28 Text and Notes menu




Enter Text Mode

Add North Arrow

Add Push Pin Note

Show Only Text

Show Text and Boxes

Do Not Show Text

Change Push Pin Font Size and Appearance

View all Push Pin Nodes in a Table

Set Push Pin Size

Set Note Size

## 1.3.6.29 URL and SCADA for nodes

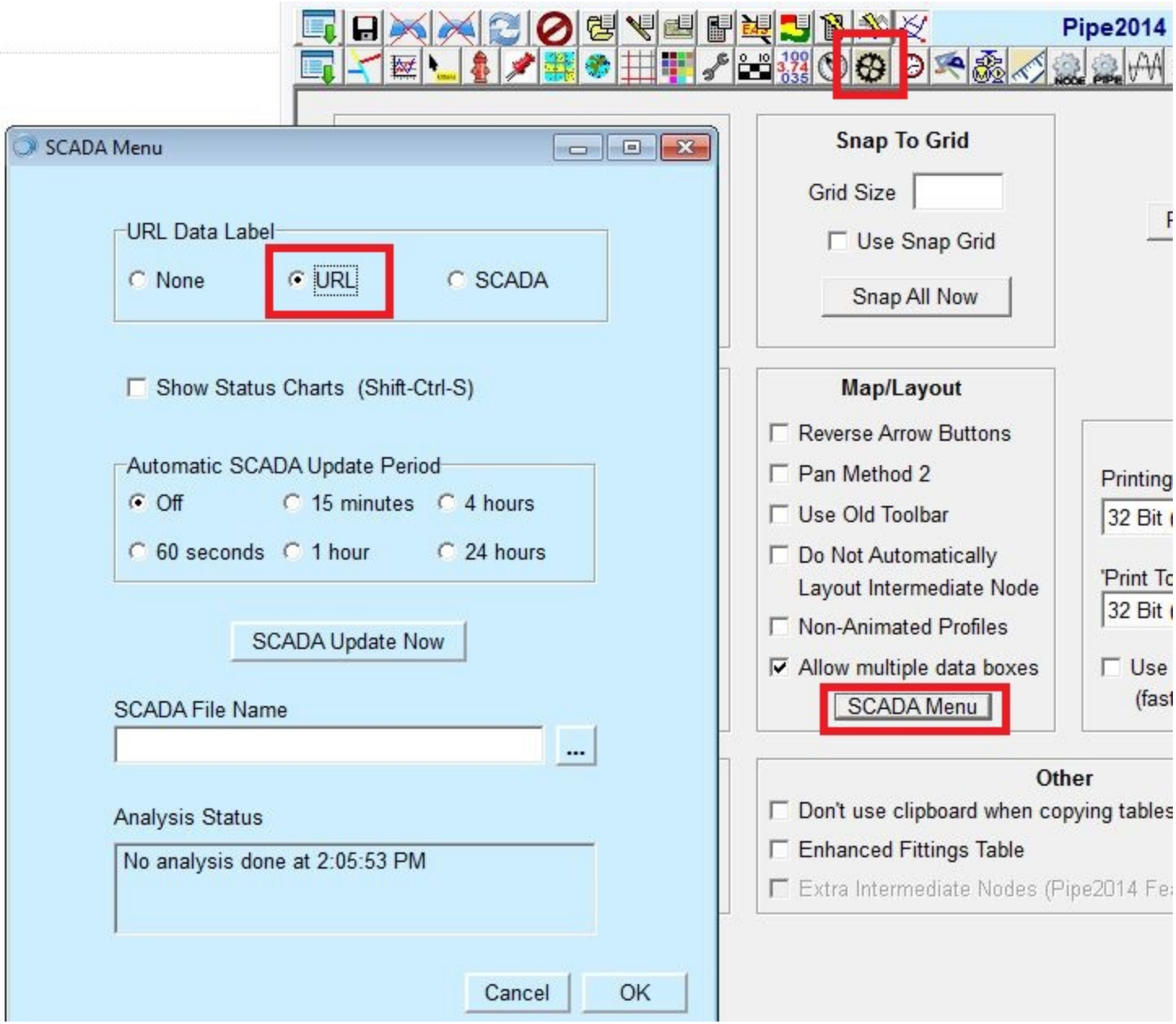
### Adding an URL link to a node

Pipe2014, KYnetic interface only.

Go to Preferences menu.

Under Map/Layout, click Scada Menu button.

Select URL


Pipe2014
SCADA Menu
URL Data Label
None
URL
SCADA
Show Status Charts (Shift-Ctrl-S)
Automatic SCADA Update Period
Off
15 minutes
4 hours
60 seconds
1 hour
24 hours
SCADA Update Now
SCADA File Name
...
Analysis Status
No analysis done at 2:05:53 PM
Cancel
OK
Snap To Grid
Grid Size
Use Snap Grid
Snap All Now
Map/Layout
Reverse Arrow Buttons
Pan Method 2
Use Old Toolbar
Do Not Automatically Layout Intermediate Node
Non-Animated Profiles
Allow multiple data boxes
SCADA Menu
Printing
32 Bit
Other
Don't use clipboard when copying tables
Enhanced Fittings Table
Extra Intermediate Nodes (Pipe2014 Fe

Back in Map view, select node.

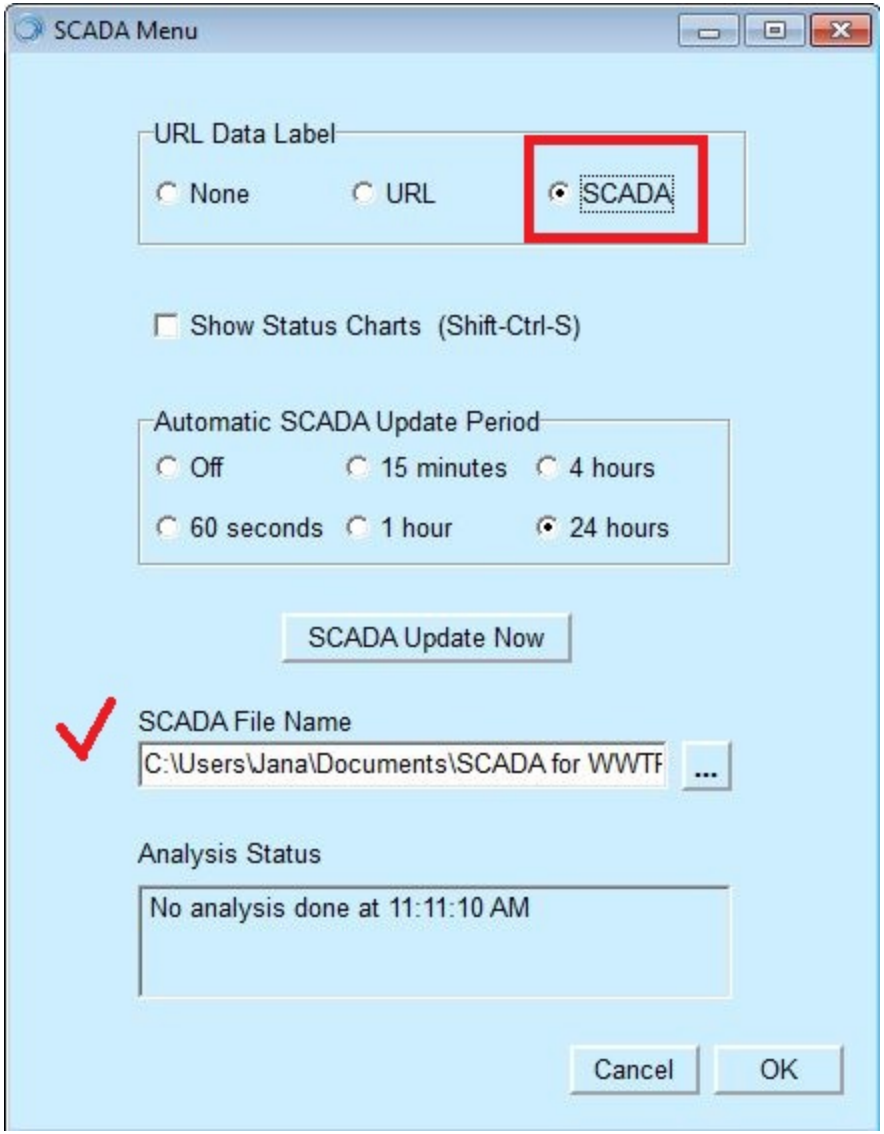
Copy in URL.

You may click on orange URL icon at the bottom of the Node Info window to go to the location.

## SCADA for Nodes

Select SCADA for URL Data Label.

Locate SCADA-format text file, which is used to update boundary conditions (tank level, pump status, etc) in the model.  Optionally specify how frequently the the SCADA file is re-loaded, or use "Update Now" button.




## 1.3.7 Analysis and Design Tools

## 1.3.7.1 Calibration

### 1.3.7.1.1 Calibration of Hydraulic Networks

Calibration of Hydraulic Networks

See also Calibration Examples

Optimized Calibration Data

Calibration Wizard

Optimized Calibration

---

CALIBRATION OF HYDRAULIC NETWORK MODELS

By
Lindell E. Ormsbee and Srinivasa Lingireddy
Department of Civil Engineering
161 Raymond Building
University of Kentucky
Lexington, KY 40506-0281
Ph: (859) 257 5243
Fax: (859) 257 4404
Email: Lindell@kypipe.com

1 Introduction

1.1 Network Characterization

1.2 Network Data Requirements

1.3 Model Parameters

2 Identify the Intended Use of the Model

3 Determining Model Parameter Estimates

3.1 Pipe Roughness Values
3.1.1 Pipe Roughness Chart
3.1.2 Pipe Roughness Field Estimation
3.1.2.1 The Parallel-Pipe Method
3.1.2.2 The Two-hydrant Method
3.1.2.3 General Suggestions
3.2 Nodal Demand Distribution
3.2.1 Spatial Distribution of Demands
3.2.2 Temporal Distribution of Demands
4 Collect Calibration Data
4.1 Fire Flow Tests
4.2 Telemetry Data
4.3 Water Quality Data
5 Evaluate Model Results
6 Perform Macro-level Model Calibration
7 Perform Sensitivity Analysis
8 Perform Micro-level Model Calibration
8.1 Analytical Approaches
8.2 Simulation Approaches
8.3 Optimization Approaches
9 Future Trends
10 Summary and Conclusion
11 References

## 1 Introduction

Computer models for analyzing and designing water distribution systems have been available since the mid 1960's. Since then, however, many advances have been made with regard to the sophistication and application of this technology. A primary reason for the growth and use of computer models has been the availability and widespread use of the microcomputer. With the advent of this technology it has been possible for water utilities and engineers to analyze the

status and operations of the existing system as well as to investigate the impacts of proposed changes (Ormsbee and Chase, 1988). The validity of these models, however, depends largely on the accuracy of the input data.

### 1.1 Network Characterization

Before an actual water distribution system may be modeled or simulated with a computer program, the physical system must be represented in a form that can be analyzed by a computer. This will normally require that the water distribution system first be represented by using node-link characterization (see Figure 1). In this case the links represent individual pipe sections and the nodes represent points in the system where two or more pipes (links) join together or where water is being input or withdrawn from the system.

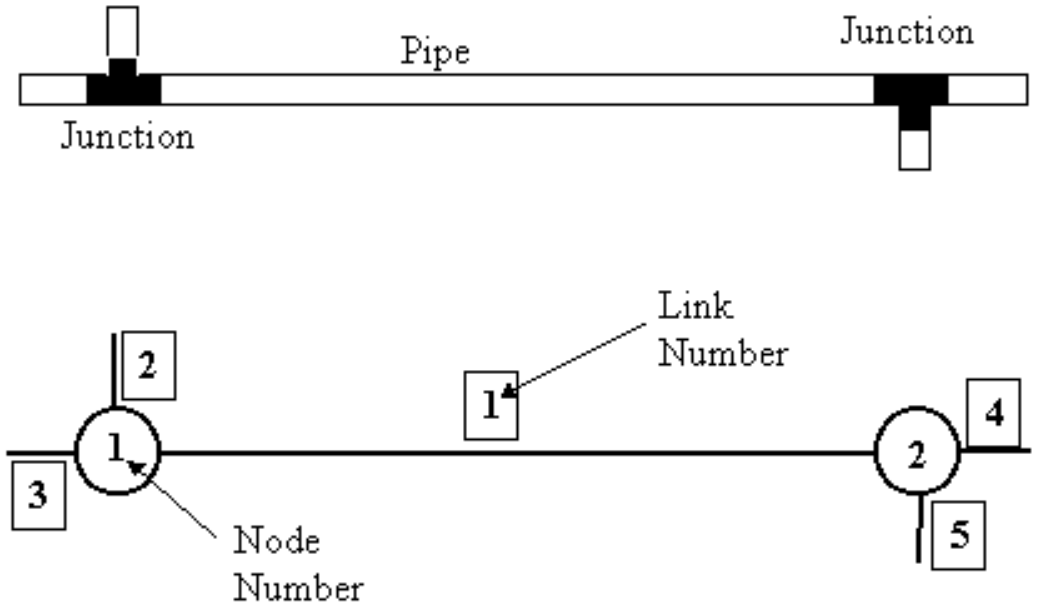



Figure 1: Node-Link Characterization

1.2 Network Data Requirements

Data associated with each link will include a pipe identification number, pipe length, pipe diameter, and pipe roughness. Data associated with each junction node will include a junction identification number, junction elevation, and junction demand. Although it is recognized that water leaves the system in a time varying fashion through various service connections along the length of a pipe segment, it is generally acceptable in modeling to lump half of the demands along a line to the upstream node and the other half of the demands to the downstream node as shown in Figure 2.

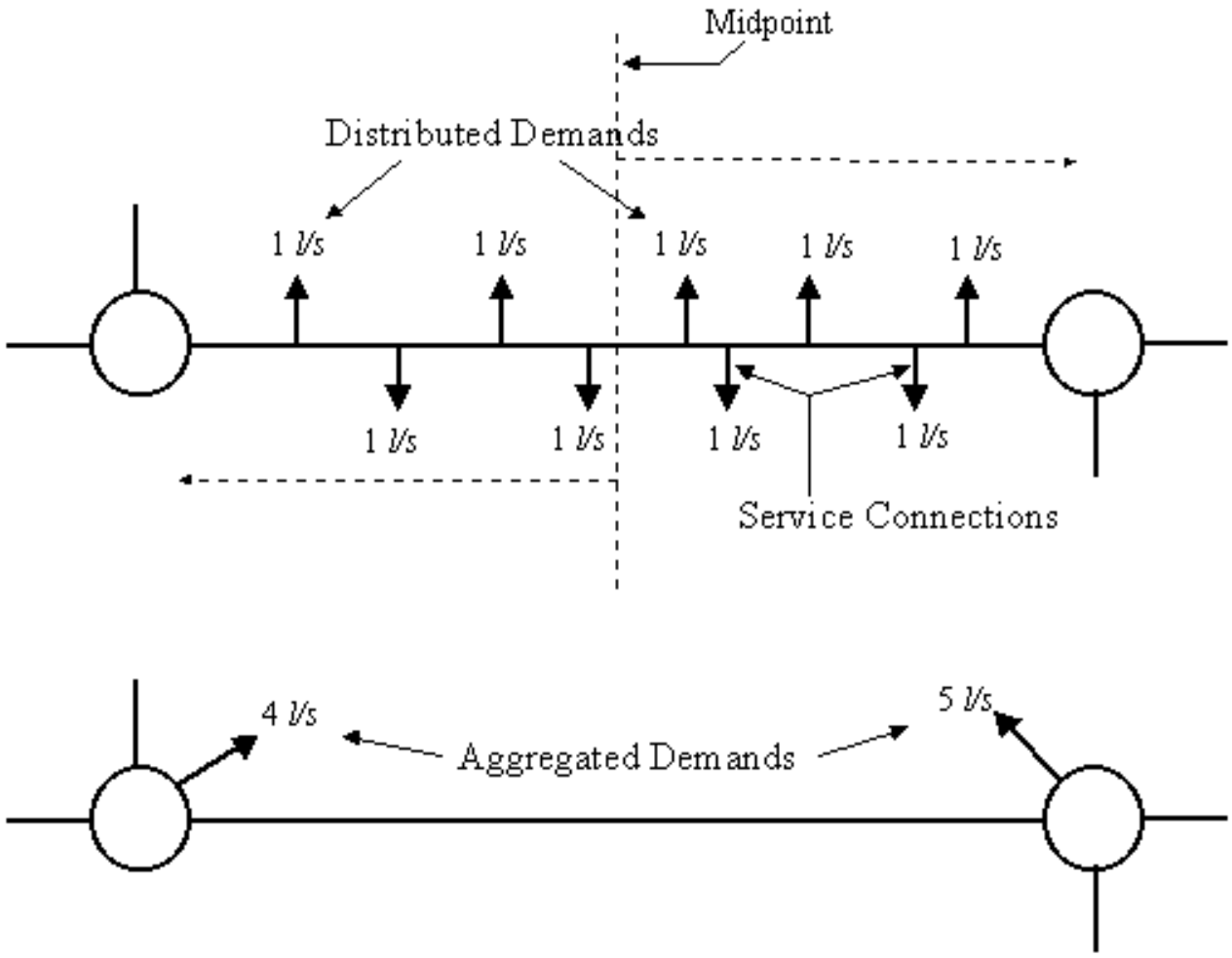



Figure 2: Demand Load Simplification

In addition to the network pipe and node data, physical data for use in describing all tanks, reservoirs, pumps, and valves must also be obtained. Physical data for all tanks and reservoirs will normally include information on tank geometry as well as the initial water levels. Physical data for all pumps will normally include

either the value of the average useful horsepower, or data for use in describing the pump flow/head characteristics curve. Once this necessary data for the network model has been obtained, the data should be entered into the computer in a format compatible with the selected computer model.

1.3 Model Parameters

Once the data for the computer network model has been assembled and encoded, the associated model parameters should then be determined prior to actual model application. In general, the primary parameters associated with a hydraulic network model will include pipe roughness and nodal demands. Due to the difficulty of obtaining economic and reliable measurements of both parameters, final model values are normally determined through the process of model calibration. Model calibration involves the adjustment of the primary network model parameters (i.e. pipe roughness coefficients and nodal demands) until the model results closely approximate actual observed conditions as measured from field data. In general, a network model calibration effort should encompass seven basic steps (see Figure 3). Each of these steps is discussed in detail in the following sections.

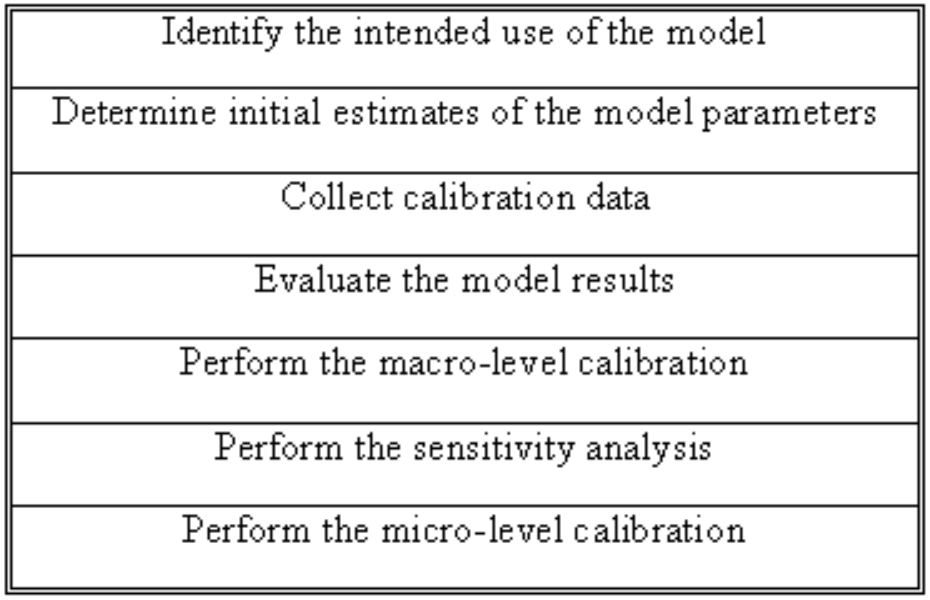



Figure 3: Seven Basic Steps for Network Model Calibration

## 2 Identify the Intended Use of the Model

Before calibrating a hydraulic network model, it is important to first identify its intended use (e.g., pipe sizing for master planning, operational studies, design projects, rehabilitation studies, water quality studies) and the associated type of hydraulic analysis (steady-state versus extended-period). Usually the type of analysis is directly related to the intended use. For example, water quality and operational studies require an extended-period analysis, whereas some planning or design studies may be performed using a study state analysis (Walski, 1995). In the latter, the model predicts system pressures and flows at an instant in time under a specific set of operating conditions and demands (e.g., average or maximum daily demands). This is analogous to photographing the system at a specific point in time. In extended-period analysis, the model predicts system pressures and flows over an extended period (typically 24 hours). This is analogous to developing a movie of the system performance.

Both the intended use of the model and the associated type of analysis provide some guidance about the type and quality of collected field data and the desired level of agreement between observed and predicted flows and pressures (Walski, 1995). Models for steady-state applications can be calibrated using multiple static flow and pressure observations collected at different times of day under varying operating conditions. On the other hand, models for extended-period applications require field data collected over an extended period (e.g., one to seven days).

In general, a higher level of model calibration is required for water quality analysis or an operational study than for a general planning study. For example, determining ground evaluations using a topographic map may be adequate for one type of study, whereas another type of study may require an actual field survey. This of course may depend on the contour interval of the map used. Such considerations obviously influence the methods used to collect the necessary model data and the subsequent calibration steps. For example, if one is working in a fairly steep terrain (e.g. greater than 20 foot contour intervals), one may decided to use a GPS unit for determining key elevations other than simply interpolating between contours.

## 3 Determining Model Parameter Estimates

The second step in calibrating a hydraulic network model is to determine initial estimates of the primary model parameters. Although most models will have some degree of uncertainty associated with several model parameters, the two model parameters that normally have the greatest degree of uncertainty are the pipe roughness coefficients and the demands to be assigned to each junction node.

### 3.1 Pipe Roughness Values

Initial estimates of pipe roughness values may be obtained using average literature values or directly from field measurements. Various researchers and pipe manufacturers have developed tables that provide estimates of pipe roughness as a function of various pipe characteristics such as pipe material, pipe diameter, and pipe age (Lamont, 1981). One such typical table is shown in Table 1 (Wood, 1991). Although such tables may be useful for new pipes, their specific applicability to older pipes decreases significantly as the pipes age. This may result due to the affects of such things as tuberculation, water chemistry, etc. As a result, initial estimates of pipe roughness for all pipes other than relatively new pipes should normally come directly from field testing. Even when new pipes are being used it is helpful to verify the roughness values in the field since the roughness coefficient used in the model may actually represent a composite of several

secondary factors such as fitting losses and system skeletonization.

Table 1: Typical Hazen-William Pipe Roughness Factors

| Pipe Material | Age (years) | Diameter | C Factor |
|---|---|---|---|
| Cast Iron | New | All Sizes | 130 |
| | 5 | >380mm (15in) | 120 |
| | | >100mm (4in) | 118 |
| | 10 | >600mm (24in) | 113 |
| | | >300mm (12in) | 111 |
| | | >100mm (4in) | 107 |
| | 20 | >600mm (24in) | 100 |
| | | >300mm (12in) | 96 |
| | | >100mm (4in) | 89 |
| | 30 | >760mm (30in) | 90 |
| | | >400mm (16in) | 87 |
| | | >100mm (4in) | 75 |
| | 40 | >760mm (30in) | 83 |
| | | >400mm (16in) | 80 |
| | | >100mm (4in) | 64 |
| Ductile Iron | New | | 140 |
| Plastic PVC | Average | | 140 |
| Asbestos Cement | Average | | 140 |
| Wood Stave | Average | | 120 |

3.1.1 Pipe Roughness Chart

A customized roughness nomograph for a particular water distribution system may be developed using the process illustrated in Figure 4. To obtain initial estimates of pipe roughness through field testing, it is best to divide the water distribution system into homogeneous zones based on the age and material of the associated pipes (see Figure 4a). Next, several pipes of different diameters should be tested in each zone to obtain individual pipe roughness estimates (see Figure 4b). Once a customized roughness nomograph is constructed, (see Figure 4c), it can be used to assign values of pipe roughness for the rest of the pipes in the system.

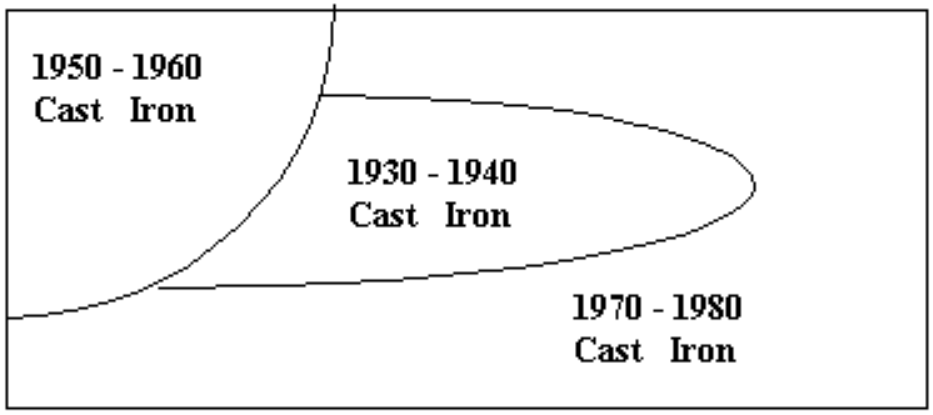



Figure 4a: Subdivide Network into Homogeneous Zones of Like Age and Material

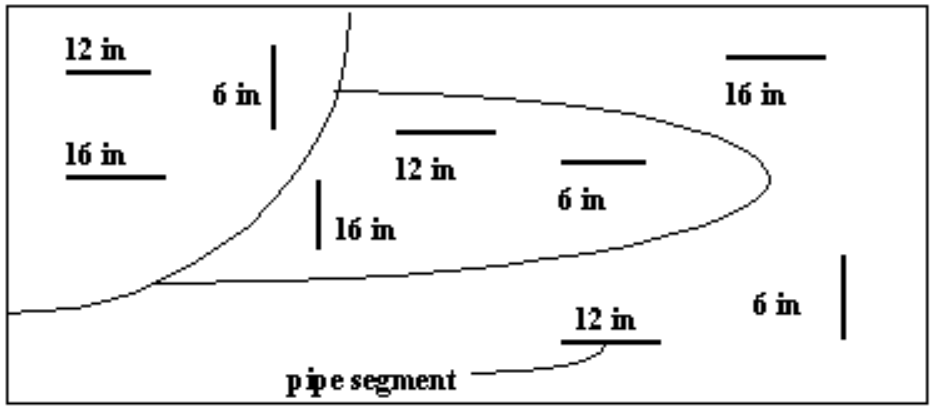



Figure 4b: Select Representative Pipes from Each Zone

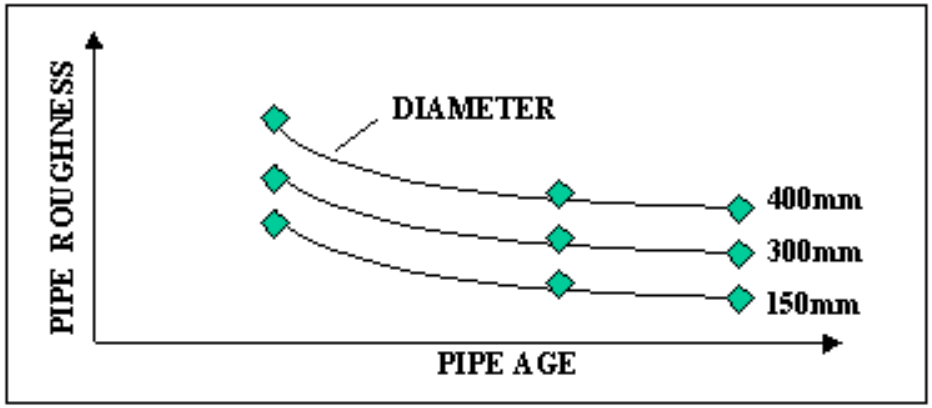



Figure 4c: Plot Associated Roughness as a Function of Pipe Diameter and Age

3.1.2 Pipe Roughness Field Estimation

Pipe roughness values may be estimated in the field by selecting a straight section of pipe that contains a minimum of three fire hydrants (see Figure 5a). When the line has been selected, pipe roughness may be estimated using one of two methods (Walski, 1984): 1) The parallel-pipe method (see Figure 5b) or 2) The two-hydrant method (see Figure 5c). In each method, the length and diameter of the test pipe are first determined. Next, the test pipe is isolated, and the flow and pressure drop are measured either through the use of a differential pressure gauge or by using two separate pressure gauges. Pipe roughness can then be approximated by a direct application of either the Hazen-Williams equation or the Darcy-Weisbach equation. In general, the parallel-pipe method is preferable for short runs and for determining minor losses around valves and fittings. For long runs of pipe, the two-gauge method is generally preferred. Also, if the water in the parallel pipe heats up or if a small leak occurs in the parallel line, it can lead to errors in the associated headloss measurements (Walski, 1985).

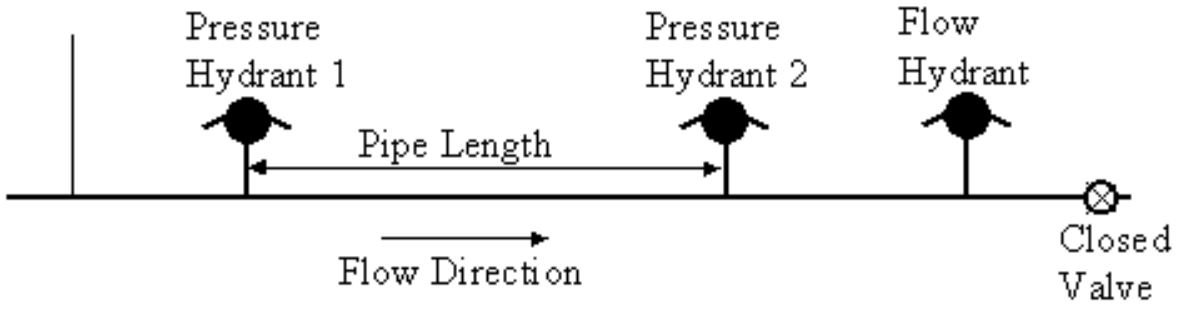



Figure 5a: Pipe Roughness Test Configuration

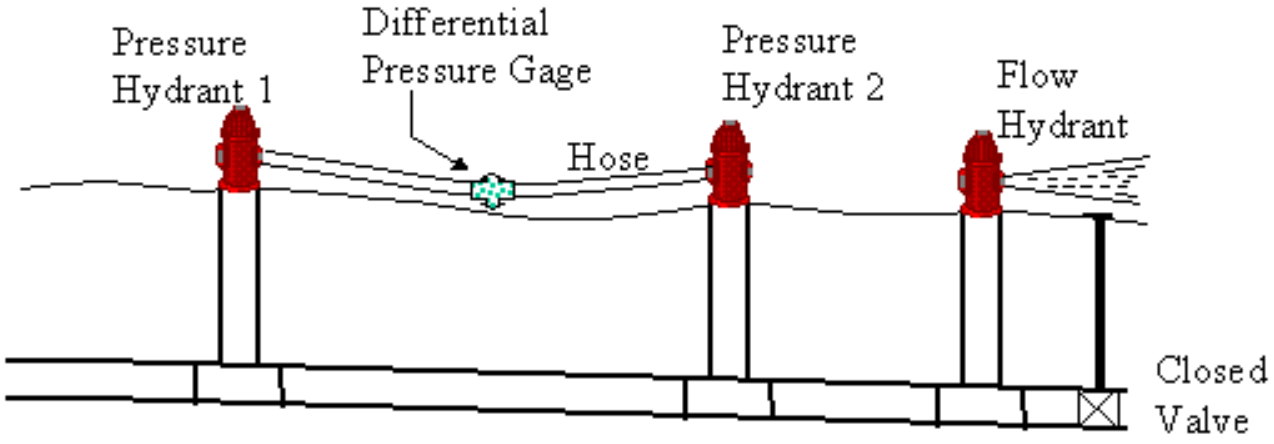



Figure 5b: Parallel Pipe Method

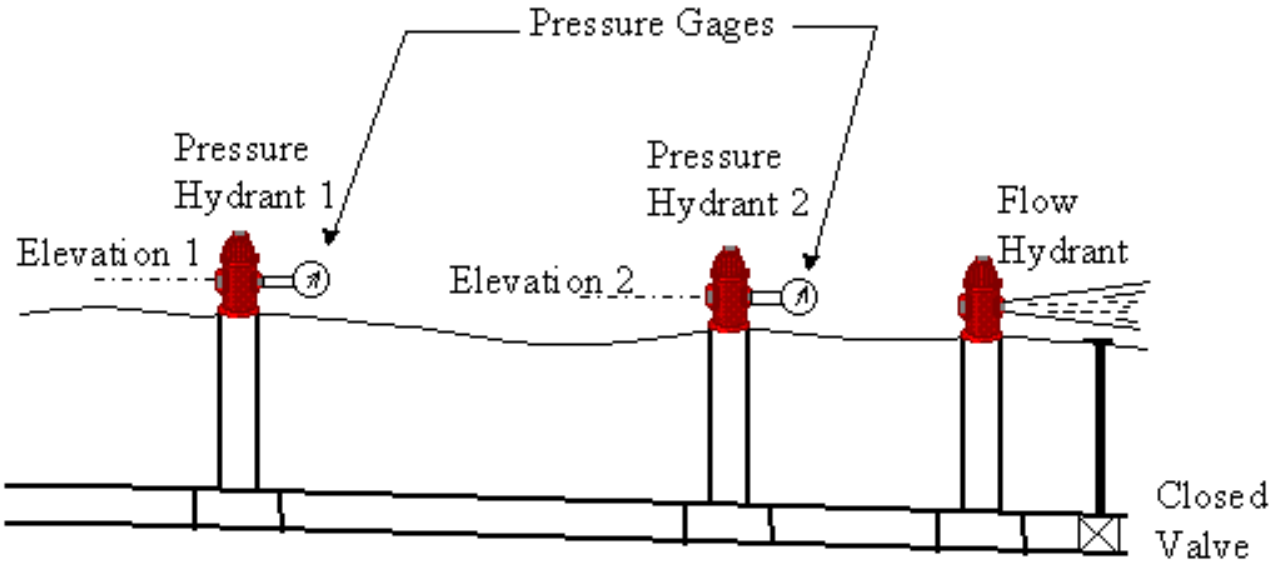



Figure 5c: Two Gage Method

3.1.2.1 The Parallel-Pipe Method

The steps involved in the application of the parallel pipe method are summarized as follows:

1) Measure the length of pipe between the two upstream hydrants (Lp) in meters.

2) Determine the diameter of the pipe (Dp) in mm. In general this should simply be the nominal diameter of the pipe. It is recognized that the actual diameter may differ from this diameter due to variations in wall thickness or the buildup of tuberculation in the pipe. However, the normal calibration practice is to incorporate the influences of variations in pipe diameter via the roughness coefficient. It should be recognized however, that although such an approach should not significantly influence the distribution of flow or headloss throughout the system it may have a significant influence on pipe velocity, which in turn could influence the results of a water quality analysis.

3) Connect the two upstream hydrants with a pair of parallel pipes, (typically a pair of fire hoses) with a differential pressure device located in between

(see Figure 5b). The differential pressure device can be a differential pressure gauge, an electronic transducer or a manometer. Walski (1984) recommends the use of an air filled manometer due to its simplicity, reliability, durability and low cost. (Note: When connecting the two hoses to the differential pressure device, make certain that there is no flow through the hoses. If there is any leak in the hoses the computed headloss for the pipe will be in error by an amount equal to the headloss through the hose).

4) Open both hydrants and check all connections to insure there are no leaks in the configuration.

5) Close the valve downstream of the last hydrant and then open the smaller nozzle on the flow hydrant to generate a constant flow through the isolated section of pipe. Make sure the discharge has reached equilibrium condition before taking flow and pressure measurements.

6) Determine the discharge Qp (l/s) from the smaller nozzle in the downstream hydrant. This is normally accomplished by measuring the discharge pressure Pd of the stream leaving the hydrant nozzle using either a hand held or nozzle mounted pitot. Once the discharge pressure Pd (in kPa) is determined it can be converted to discharge (Qp) using following relationship:

$$Q_p = C_d D_n^2 P_d^{0.5} / 900.3$$

..... eq. 1

where Dn is the nozzle diameter in mm and Cd is the nozzle discharge coefficient which is a function of the type of nozzle (see Figure 6). (Note: When working with larger mains, sometimes you can't get enough water out of the smaller nozzles to get a good pressure drop. In such cases you may need to use the larger nozzle).

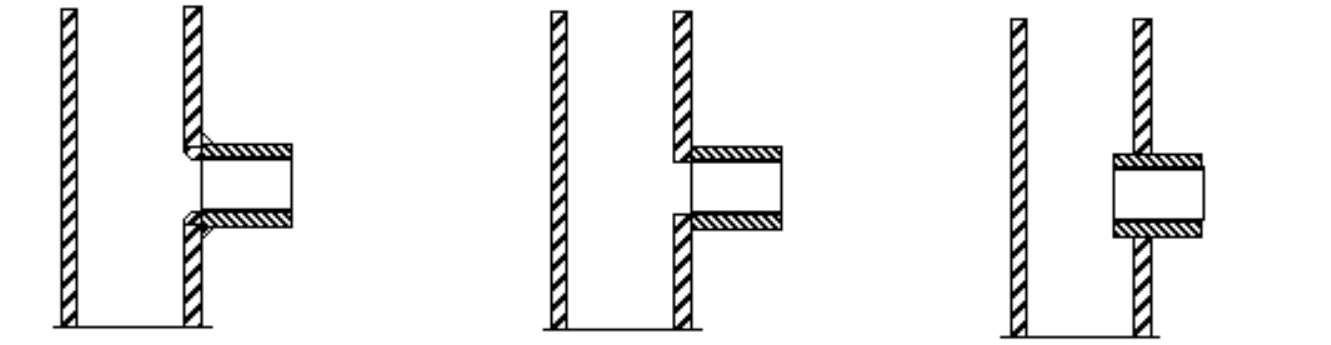



Figure 6: Hydrant Nozzle Discharge Coefficients

7) After calculating the discharge, determine the in-line flow velocity Vp (m/s) where:

$$V_p = Q_p / (\pi D_p^2 / 4)^2$$

..... eq. 2

8) After the flow through the hydrant has been determined, measure the pressure drop (p

through the isolated section of pipe by reading the differential pressure gauge. Convert the measured pressure drop in units of meters (Hp) and divided by the pipe length Lp to yield the hydraulic gradient or friction slope Sp.

$$S_p = H_p / L_p$$

..... eq. 3

9a) Once these four measured quantities have been obtained, the Hazen-Williams Roughness Factor (Cp) can then be determined using the Hazen-Williams equation as follows:

$$C_p = \frac{218 V_p}{D_p^{0.63} S_p^{0.54}}$$

..... eq. 4

9b) To calculate the actual pipe roughness (, it is first necessary to calculate the friction factor f using the Darcy-Weisbach equation as follows (Walski, 1984):

$$f = \frac{g S_p D_p}{500 V_p^2}$$

..... eq. 5

where g = gravitational acceleration constant (9.81m/sec2)

Once the friction factor has been calculated, the Reynolds number must be determined. Assuming a standard water temperature of 20oC (680 F), the Reynolds number is:

$$R = 993 V_p D_p$$

..... eq. 6

Once the friction factor f, and the Reynolds number R have been determined, they can be inserted into the Colebrook-White formula to give the pipe roughness $\varepsilon$ (mm) as:

$$\varepsilon = 3.7 D_p \left\{ \exp(-1.16\sqrt{f}) - 2.51 / R\sqrt{f} \right\}$$

..... eq. 7

3.1.2.2. The Two-hydrant Method

The two hydrant method is basically identical to the parallel pipe method with the exception that the pressure drop across the pipe is measured using a pair of static pressure gauges as shown in Figure 5c. In this case the total headloss through the pipe is the difference between the hydraulic grades at both hydrants. In order to obtain the hydraulic grade at each hydrant, the observed pressure head (m) must be added to the elevation of the reference point (the hydrant nozzle). For the two hydrant method, the head loss through the test section Hp (m) can be calculated using the following equation:

$$H_p = \frac{(P_2 - P_1)}{9.81} + (Z_2 - Z_1)$$

..... eq. 8

where P1 is the pressure reading at the upstream gauge (kPa) , Z1 is the elevation of the upstream gauge (m), P2 is the pressure reading at the downstream gauge (kPa), and Z2 is the elevation of the downstream gauge (m).

The elevation difference between the two gauges should generally be determined using a transit or a level. As a result, one should make sure to select two upstream hydrants that can be seen from a common point. This will minimize the number of turning points required in determining the elevation differences between the nozzles of the two hydrants. As an alternative to the use of a differential survey, topographic maps can sometimes be used to obtain estimates of hydrant elevations. However, topographic maps should not generally be used to estimate the elevation differences unless the contour interval is 1m or less. One hydraulic alternative to measuring the elevations directly is to simply measure the static pressure readings at both hydrants before the test and convert the observed pressure difference to the associated elevation difference (e.g. Z1 - Z2 = 2.31*[P2(static) - P1(static)]).

3.1.2.3. General Observations and Suggestions

Hydrant pressures for use in pipe roughness tests are normally measured with a Bourdon tube gauge which can be mounted to one of the discharge nozzles of the hydrant using a lightweight hydrant cap. Bourdon tube gauges come in various grades (i.e 2A, A, and B) depending upon their relative measurement error. In most cases a grade A gauge (1 percent error) is sufficient for fire flow tests. For maximum accuracy one should chose a gauge graded in 5kPa (1 psi) increments with a maximum reading less than 20% above the expected maximum pressure (McEnroe, et al., 1989). In addition, it is a good idea to use pressure snubbers in order to eliminate the transient effects in the pressure gauges. A pressure snubber is a small valve that is placed between the pressure gauge and the hydrant cap which acts as a surge inhibitor (Walski, 1984).

Before conducting a pipe roughness test, it is always a good idea to make a visual survey of the test area. When surveying the area, make sure that there is adequate drainage away from the flow hydrant. In addition, make sure you select a hydrant nozzle that will not discharge into oncoming traffic. Also, when working with hydrants that are in close proximity to traffic, it is a good idea to put up traffic signs and use traffic cones to provide a measure of safety during the test. As a further safety precaution, make sure all personal are wearing highly visible clothing. It is also a good idea to equip testing personnel with radios or walkie-talkies to help coordinate the test.

While the methods outlined previously work fairly well with smaller lines (i.e. less than 16in in diameter), their efficiency decreases as you deal with larger lines. Normally, opening hydrants just doesn't generate enough flow for meaningful head-loss determination. For such larger lines you typically have to run conduct the

headloss tests over very much longer runs of pipe and use either plant or pump station flow meters or change in tank level to determine flow (Walski, 1999).

### 3.2 Nodal Demand Distribution

The second major parameter determined in calibration analysis is the average (steady-state analysis) or temporally varying (extended-period analysis) demand to be assigned to each junction node. Initial average estimates of nodal demands can be obtained by identifying a region of influence associated with each junction node, identifying the types of demand units in the service area, and multiplying the number of each type by an associated demand factor. Alternatively, the estimate can be obtained by first identifying the area associated with each type of land use in the service area and then multiplying the area of each type by an associated demand factor. In either case, the sum of these products will provide an estimate of the demand at the junction node.

#### 3.2.1 Spatial Distribution of Demands

Initial estimates of nodal demands can be developed using various approaches depending on the nature of the data each utility has on file and how precise they want to be. One way to determine such demands is by employing the following strategy.

1. First, determine the total system demand for the day to be used in model calibration (i.e. TD). The total system demand may be obtained by performing a mass balance analysis for the system by determining the net difference between the total volume of flow which enters the system (from both pumping stations and tanks) and the total volume that leaves the system (through PRVs and tanks).

2. Second, using meter records for the day, try to assign all major metered demands (i.e. MDj where j = junction node number) by distributing the observed demands among the various junction nodes which serve the metered area. The remaining demand will be defined as the total residual demand (i.e. TRD) and may be obtained by subtracting the sum of the metered demands from the total system demand:

$$TRD = TD - \sum MD_j$$

..... eq. 9

3. Third, determine the demand service area associated with each junction node. The most common method of influence delineation is to simply bisect each pipe connected to the reference node as shown in Figure 7a.

4. Once the service areas associated with the remaining junction nodes have been determined, an initial estimate of the demand at each node should be made. This can be accomplished by first identifying the number of different types of demand units within the service area and then multiplying the number of each type by an associated demand factor (see Figure 7b). Alternatively, the estimate can be obtained by first identifying the area associated with each different type of land use within the service area and then multiplying the area of each type by an associated unit area demand factor (see Figure 7c). In either case, the sum of these products will represent an estimate of the demand at the junction node. While in theory the first approach should be more accurate the later approach can be expected to be more expedient. Estimates of unit demand factors are normally available from various water resource handbooks (Cesario, 1995). Estimates of unit area demand factors can normally be constructed for different land use categories by weighted results from repeated applications of the unit demand approach.

5. Once an initial estimate of the demand has been obtained for each junction node j (i.e. IEDj), a revised estimated demand (i.e. REDj) may be obtained using the following equation:

$$RED_j = IED_j * TRD / \quad IED_i$$

..... eq. 10

6. Once the revised demands have been obtained for each junction node, the final estimate of nodal demand can be obtained by adding together both the revised demand and the metered demand (assuming there is one) associated with each junction node:

$$D_j = NED_j + MD_j$$

..... eq. 11

3.2.2 Temporal Distribution of Demands

Time-varying estimates of model demands for use in extended-period analysis can be made in one of two ways, depending on the structure of the hydraulic model. Some models allow the user to sub-divide the demands at each junction node into different use categories, which can then be modified separately over time using demand factors for water use categories. Other models require an aggregate-use category for each node. In the latter case, spatial-temporal variations of nodal demands are obtained by lumping nodes of a given type into separate groups, which can then be modified uniformly using nodal demand factors. Initial estimates of either water use category demand factors or nodal demand factors can be obtained by examining historical meter records for various water use categories and by performing incremental mass balance calculations for the distribution system. The resulting set of temporal demand factors can then be fine tuned through subsequent model calibration.

4 Collect Calibration Data

After model parameters have been estimated, the accuracy of the model parameters can be assessed. This is done by executing the computer model using the estimated parameter values and observed boundary conditions and then comparing the model results with the results from actual field observations. Data from fire flow tests, pump station flowmeter readings, and tank telemetry data are most commonly used in such tests.

In collecting data for model calibration, it is very important to recognize the significant impact of measurement errors. For example, with regard to calibrating pipe roughness, the C factor may expressed as:

$$C = k(V + error)/(h + error)^{0.54}$$

..... eq. 12

If the magnitude of V and h are on the same order of magnitude as the associated measurement errors (for V and h) then the collected data will be essentially useless for model calibration. That is to say, virtually any value of C will provide a "reasonable" degree of model calibration (Walski, 1986). However, one can hardly expect a model to accurately predict flows and pressures for a high stress situation (i.e. large flows and velocities) if the model was calibrated using data from times when the velocities in the pipes were less than the measurement error (e.g. less than 1 ft/s). The only way to minimize this problem is to either insure

that the measurement errors are reduced or the velocity or headloss values are significantly greater than the associated measurement error. This latter condition can normally be met either using data from fire flow tests or by collecting flow or pressure readings during periods of high stress (e.g. peak hour demand periods).

### 4.1 Fire Flow Tests

Fire flow tests are useful for collecting both discharge and pressure data for use in calibrating hydraulic network models. Such tests are normally conducted using both a normal pressure gauge (for measuring both static and dynamic heads) and a pitot gauge (for use in calculating discharge). In performing a fire flow test, at least two separate hydrants are first selected for use in the data collection effort. One hydrant is identified as the pressure or residual hydrant while the remaining hydrant is identified as the flow hydrant. The general steps for performing a fire flow test may be summarized as follows (McEnroe, et al., 1989):

1. Place a pressure gauge on the residual hydrant and measure the static pressure.
2. Determine which of the discharge hydrant's outlets can be flowed with the least amount of adverse impact (flooding, traffic disruption, etc.)
3. Make sure the discharge hydrant is initially closed in order to avoid injury.
4. Remove the hydrant cap from the nozzle of the discharge hydrant to be flowed.
5. Measure the inside diameter of the nozzle and determine the type of nozzle (i.e. rounded, square edge, or protruding) in order to determine the appropriate discharge coefficient. (see Figure 6).
6. Take the necessary steps to minimize erosion or traffic impacts during the test.
7. Flow the hydrant briefly to flush sediment from the hydrant lateral and barrel.

8a. If using a clamp on pitot tube, attach the tube to the nozzle to be flowed and then slowly open the hydrant.

8b. If using a hand held pitot tube, slowly open the hydrant and then place the pitot in the center of the discharge stream being careful to align it directly into the flow.

9. Once an equilibrium flow condition has been established, make simultaneous pressure reading from both the pitot and the pressure gauge at the residual hydrant.
10. Once the readings are completed, close the discharge hydrant, remove the equipment from both hydrants and replace the hydrant caps.

In order to obtain sufficient data for an adequate model calibration it is important that data from several fire flow tests be collected. Before conducting each test, it is also important that the associated system boundary condition data be collected. This includes information on tank levels, pump status, etc. In order to obtain adequate model calibration it is normally desirable that the difference between the static and dynamic pressure readings as measured from the residual hydrant be at least 35kPa (5psi) with a preferable drop of 140kpa (20psi) (Walski, 1990a). In the event that the discharge hydrant does not allow sufficient discharge to cause such a drop it may be necessary to identify, instrument, and open additional discharge hydrants. In some instances, it may also be beneficial to use more than one residual hydrant (one near the flowed hydrant and one off the major main from the source). The information gathered from such additional hydrants can sometimes be very useful in tracking down closed valves (Walski, 1999).

### 4.2. Telemetry Data

In addition to static test data, data collected over an extended period of time (typically 24 hours) can be very useful for use in calibrating network models. The most common type of data will include flowrate data, tank water level data, and pressure data. Depending upon the level of instrumentation and telemetry associated with the system, much of the data may be already collected as part of the normal operations. For example, most systems collect and record tank levels and average pump station discharges on an hourly basis. These data are especially useful verifying the distribution of demands among the various junction nodes. If such data is available, the data should first be checked for accuracy before use in the calibration effort. If such data are not readily available, the modeler may have to install temporary pressure gauges or flowmeters in order to obtain the data. In the absence of flowmeters in lines to tanks, inflow or discharge flow rates can be inferred from incremental readings of the tank level.

### 4.3 Water Quality Data

In recent years, both conservative and non-conservative constituents have been used as tracers to determine the travel time through various parts of a water distribution system (Grayman, 1998, Cesario, A. L., et al., 1996, Kennedy, et. al., 1991). The most common type of tracer for such applications is fluoride. By controlling the injection rate at a source, typically the water treatment plant, a pulse can be induced into the flow that can then be monitored elsewhere in the system. The relative travel time from the source to the sampling point can be determined. The measured travel time thus provides another data point for use in calibrating a hydraulic network model.

Alternatively, the water distribution system can also be modeled using a water quality model such as EPANET (Rosman, 1994). In this case the water quality model is used to predict tracer concentrations at various points in the system. Since all water quality models results depend on the underlying hydraulic results, deviations between the observed and predicted concentrations can thus provide a secondary means of evaluating the adequacy of the underlying hydraulic model. For additional insights into water quality modeling and the use of such models in calibration, the reader should refer to Chapter 9.

## 5 Evaluate Model Results

In using fire flow data, the model is used to simulate the discharge from one or more fire hydrants by assigning the observed hydrant flows as nodal demands within the model. The flows and pressures predicted by the model are then compared with the corresponding observed values in an attempt to assess model accuracy. In using telemetry data, the model is used to simulate the variation of tank water levels and system pressures by simulating the operating conditions for the day over which the field data was collected. The predicted tank water levels are then compared with the observed values in an attempt to assess model accuracy. In using water quality data, the travel times (or constituent concentrations) are compared with model predictions in an attempt to assess model accuracy.

Model accuracy may be evaluated using various criteria. The most common criteria are absolute pressure difference (normally measured in psi) or relative pressure difference (measured as the ratio of the absolute pressure difference to the average pressure difference across the system). In most cases a relative pressure difference criteria is normally to be preferred. For extended period simulations, comparisons are normally made between the predicted and observed tank water levels. To a certain extent, the desired level of model calibration will be related to the intended use of the model. For example, a higher level of model calibration will normally be required for water quality analysis or an operational study as opposed to use of the model in a general planning study. Ultimately, the model should be calibrated to the extent that the associated application decisions will not be significantly affected. In the context of a design application, the model should normally be calibrated to such an extent that the resulting design values (e.g. pipe diameters, tank and pump sizes and/or locations, etc) will be the same as if the exact parameter values were used. Determination of such thresholds will frequently require the application of model sensitivity analysis (Walski,

1995).

Because of the issue of model application, it is difficult to derive a single set of criteria for a universal model calibration. From the authors' perspective, a maximum state variable (i.e. pressure grade, water level, flowrate) deviation of less than 10 percent will generally be satisfactory for most planning applications while a maximum deviation of less than 5 percent to be highly desirable for most design, operation, or water quality applications. Although no such general set of criteria have been officially developed for the United States, a set of "Performance Criteria" have been developed by the Sewers and Water Mains Committee of the Water Authorities in the United Kingdom (1989). For steady state models the criteria are:

1. Flows agree to:
    a. 5% of measured flow when flows are more than 10% of total demand.
    b. 10% of measured flow when flows are less than 10% of total demand.
2. Pressures agree to:
    a. 0.5 m (1.6ft) or 5% of headloss for 85% of test measurements.
    b. 0.75 m (2.31 ft) or 7.5% of headloss for 95% of test measurements.
    c. 2 m (6.2 ft) or 15% of headloss for 100% of test measurements.

For extended period simulation, the criteria require that three separate steady state calibrations be performed for different time periods and that the average volumetric difference between measured and predicted reservoir storage be within 5%. Additional details can be obtained directly from the report.

Deviations between results of the model application and the field observations may be caused by several factors, including: 1) erroneous model parameters (i.e. pipe roughness values and nodal demand distribution), 2) erroneous network data (i.e. pipe diameters, lengths, etc), 3) incorrect network geometry (i.e. pipes connected to the wrong nodes, etc.), 4) incorrect pressure zone boundary definitions, 5) errors in boundary conditions (i.e. incorrect PRV value settings, tank water levels, pump curves, etc.), 6) errors in historical operating records (i.e. pumps starting and stopping at incorrect times), 7) measurement equipment errors (i.e. pressure gauges not properly calibrated, etc.), and 8) measurement error (i.e. reading the wrong values from measurement instruments). The last two sources of errors can hopefully be eliminated or at least minimized by developing and implementing a careful data collection effort. Elimination of the remaining errors will frequently require the iterative application of the last three steps of the model calibration process - macro-level calibration, sensitivity, and micro-level calibration. Each of these steps is described in the following sections.

## 6 Perform Macro-level Model Calibration

In the event that one or more of the measured state variable values are different from the modeled values by an amount that is deemed to be excessive (i.e greater than 30 percent), it is likely that the cause for the difference may extend beyond errors in the estimates for either the pipe roughness values or the nodal demands. Possible causes for such differences are many but may include: 1) closed or partially closed valves, 2) inaccurate pump curves or tank telemetry data, 3) incorrect pipe sizes (e.g. 6 inch instead of 16, etc.), 4) incorrect pipe lengths, 5) incorrect network geometry, and 6) incorrect pressure zone boundaries, etc. (Walski, 1990a).

The only way to adequately address such errors is to systematically review the data associated with the model in order to insure its accuracy. In most cases, some data will be less reliable than other data. This observation provides a logical place to start in an attempt to identify the problem. Model sensitivity analysis

provides another means of identifying the source of discrepancy. For example, if it is suspected that a valve is closed, this assumption can be modeled by simply closing the line in the model and evaluate the resulting pressures. Potential errors in pump curve data may sometimes be circumvented by simulating the pumps with negative inflows set equal to observed pumps discharges (Cruickshank, and Long, 1992). This of course assumes that the error in the observed flow rates (and the induced head) are less than the errors introduced by using the pump curves. In any rate, only after the model results and the observed conditions are within some reasonable degree of correlation (usually less than 20% error) should the final step of micro-level calibration be attempted.

### 7 Perform Sensitivity Analysis

Before attempting a micro-level calibration, it is helpful to perform a sensitivity analysis of the model in order to help identify the most likely source of model error. This can be accomplished by varying the different model parameters by different amounts and then measuring the associated effect. For example, many current network models have as an analysis option the capability to make multiple simulations in which global adjustment factors can be applied to pipe roughness values or nodal demand values. By examining such results, the user can begin to identify which parameters have the most significant impact on the model results and thereby identify potential parameters for subsequent fine tuning through micro-level calibration.

### 8 Perform Micro-level Model Calibration

After the model results and the field observations are in reasonable agreement, a micro-level model calibration should be performed. As discussed previously, the two parameters adjusted during this final calibration phase will normally include pipe roughness and nodal demands. In many cases it may be useful to break the micro calibration into two separate steps: 1) steady state calibration, and 2) extended period calibration. In performing a steady state calibration the model parameters are adjusted to match pressures and flowrates associated with static observations. The normal source for such data is from fire flow tests. In an extended period calibration, the model parameters are adjusted to match time varying pressures and flows as well as tank water level trajectories. In most cases the steady state calibration will be more sensitive to changes in pipe roughness while the extended period calibration will be more sensitive to changes in the distribution of demands. As a result, one potential calibration strategy would be to first fine tune the pipe roughness parameter values using the results from fire flow tests and then try to fine tune the distribution of demands using the flow/pressure/water level telemetry data.

Historically, most attempts at model calibration have typically employed an empirical or trial and error approach. Such an approach can prove to be extremely time consuming and frustrating when dealing with most typical water systems. The level of frustration will, of course, depend somewhat on the expertise of the modeler, the size of the system, and the quantity and quality of the field data. Some of the frustration can be minimized by breaking complicated systems into smaller parts and then calibrating the model parameters using an incremental approach. Calibration of multi-tank systems can sometimes be facilitated by collecting multiple data sets with all but one of the tanks closed (Cruickshank, and Long, 1992). In recent years, several researchers have proposed different algorithms for use in automatically calibrating hydraulic network models. These techniques have been based on the use of analytical equations (Walski, 1983), simulation models (Rahal et al., 1980; Gofman and Rodeh, 1981; Ormsbee and Wood, 1986; and Boulos and Ormsbee, 1991), and optimization methods (Meredith, 1983; Coulbeck, 1984, Ormsbee, 1989; Lansey and Basnet, 1991; and Ormsbee, et al., 1992).

#### 8.1 Analytical Approaches

In general, techniques based on analytical equations require significant simplification of the network through skeletonization and the use of equivalent pipes. As a result, such techniques may only get the user close to the correct results. Conversely, both simulation and optimization approaches take advantage of using a complete model.

## 8.2. Simulation Approaches

Simulation techniques are based on the idea of solving for one or more calibration factors through the addition of one or more network equations. The additional equation or equations are used to define an additional observed boundary condition (such as fire flow discharge head). By addition of an extra equation, an additional unknown can then be determined explicitly.

The primary disadvantage of the simulation approaches is that they can only handle one set of boundary conditions at a time. For example, in applying a simulation approach to a system with three different sets of observations (all of which were obtained under different boundary conditions, i.e. different tank levels, pump status, etc.), three different results can be expected. Attempts to obtain a single calibration result will require one of two application strategies: 1) a sequential approach, or 2) an average approach. In applying the sequential approach the system is subdivided into multiple zones whose number will correspond to the number of sets of boundary conditions. In this case the first set of observations is used to obtain calibration factors for the first zone. These factors are then fixed and another set of factors is then determined for the second zone and so on. In the average approach, final calibration factors are obtained by averaging the calibration factors for each of the individual calibration applications.

## 8.3 Optimization Approaches

The primary alternative to the simulation approach is to use an optimization approach. In using an optimization approach, the calibration problem is formulated as a nonlinear optimization problem consisting of a nonlinear objective function subject to both linear and nonlinear equality and inequality constraints. Using standard mathematical notation, the associated optimization problem may be expressed as follows:

Minimize:

$$f(\mathbf{X})$$

..... eq. 13

Subject To:

$$g(\mathbf{X}) = 0$$

..... eq. 14

$$L_h \leq h(\mathbf{X}) \leq U_h$$

..... eq. 15

$$L_x \leq \mathbf{X} \leq U_x$$

..... eq. 16

where X is the vector of decision variables (pipe roughness coefficients, nodal demands, etc.), f(X) is the nonlinear objective function, g(X) is a vector of implicit

system constraints, h(X) is a vector of implicit bound constraints, and, Lx and Ux, are the lower and upper bounds on the explicit system constraints and the decision variables.

Normally, the objective function will be formulated so as to minimize the square of the differences between observed and predicted values of pressures and flows. Mathematically, this may be expressed as:

$$f(\mathbf{X}) = \alpha \sum_{j=1}^{J} (OP_j - PP_j)^2 + \beta \sum_{p=1}^{P} (OQ_p - PQ_q)^2$$

..... eq. 17

where OPj = the observed pressure at junction j, PPj = the predicted pressure at junction j, OQp = the observed flow in pipe p, PQp = the predicted flow in pipe p, and $\alpha$ and $\beta$ are normalization weights.

The implicit bound constraints on the problem may include both pressure bound constraints and flowrate bound constraints. These constraints may be used to insure that the resulting calibration does not produce unrealistic pressures or flows as a result of the model calibration process. Mathematically, for a given vector of junction pressures P these constraints can be expressed as:

$$L_p \leq \mathbf{P} \leq U_p$$

..... eq. 18

Likewise for a given vector of pipe flows Q these constraints can be expressed as:

$$L_Q \leq \mathbf{Q} \leq U_Q$$

..... eq. 19

The explicit bound constraints may be used to set limits on the explicit decision variables of the calibration problem. Normally, these variables will include (1) the roughness coefficient of each pipe, and (2) the demands at each node. For a given vector of pipe roughness coefficients C these constraints can be expressed as:

$$L_C \leq \mathbf{C} \leq U_C$$

..... eq. 20

Likewise for a given vector of nodal demands D, these constraints can be expressed as:

$$L_D \leq \mathbf{D} \leq U_D$$

..... eq. 21

The implicit system constraints include nodal conservation of mass and conservation of energy.

The nodal conservation of mass equation Fc (Q) requires that the sum of flows into or out of any junction node n minus any external demand Dj must be equal to zero. For each junction node j this may be expressed as:

$$F_c(Q) = \sum_{n \in j}^{N_j} Q_n - D_j = 0$$

..... eq. 22

where Nj = the number of pipes connected to junction node j and {j} is the set of pipes connected to junction node j.

The conservation of energy constraint Fe(Q) requires that the sum of the line loss (HLn) and the minor losses (HMn) over any path or loop k, minus any energy added to the liquid by a pump (EPn), minus the difference in grade between and two points of known energy (DEk) is equal to zero. For any loop or path k this may be expressed as:

$$F_e(Q) = \sum_{n \in k}^{N_k} (HL_n + HM_n - EP_n) - DE_k = 0$$

..... eq. 23

where Nk = the number of pipes associated with loop or path k, and {k} is the set of pipes associated with loop or path k. It should be emphasized that HLn, HMn, and EPn, are all nonlinear functions of the pipe discharge Q.

While both the implicit and explicit bound constraints have traditionally been incorporated directly into the nonlinear problem formulation, the implicit system constraints have been handled using one of two different approaches. In the first approach, the implicit system constraints are incorporated directly within the set of nonlinear equations and solved using normal nonlinear programming methods. In the second approach, the equations are removed from the optimization problem and evaluated externally using mathematical simulation (Ormsbee, 1989; Lansey and Basnet, 1991). Such an approach allows for a much smaller and more tractable optimization problem, since both sets of implicit equations (which constitute linear and nonlinear equality constraints to the original problem) can now be satisfied much more efficiently using an external simulation model (see Figure 8). The basic idea behind the approach is to use an implicit optimization algorithm to generate a vector of decision variables which are then passed to a lower level simulation model for use in evaluating all implicit system constraints. Feedback from the simulation model will include numerical values for use in identifying the status of each constraint as well as numerical results for use in evaluating the associated objective function.

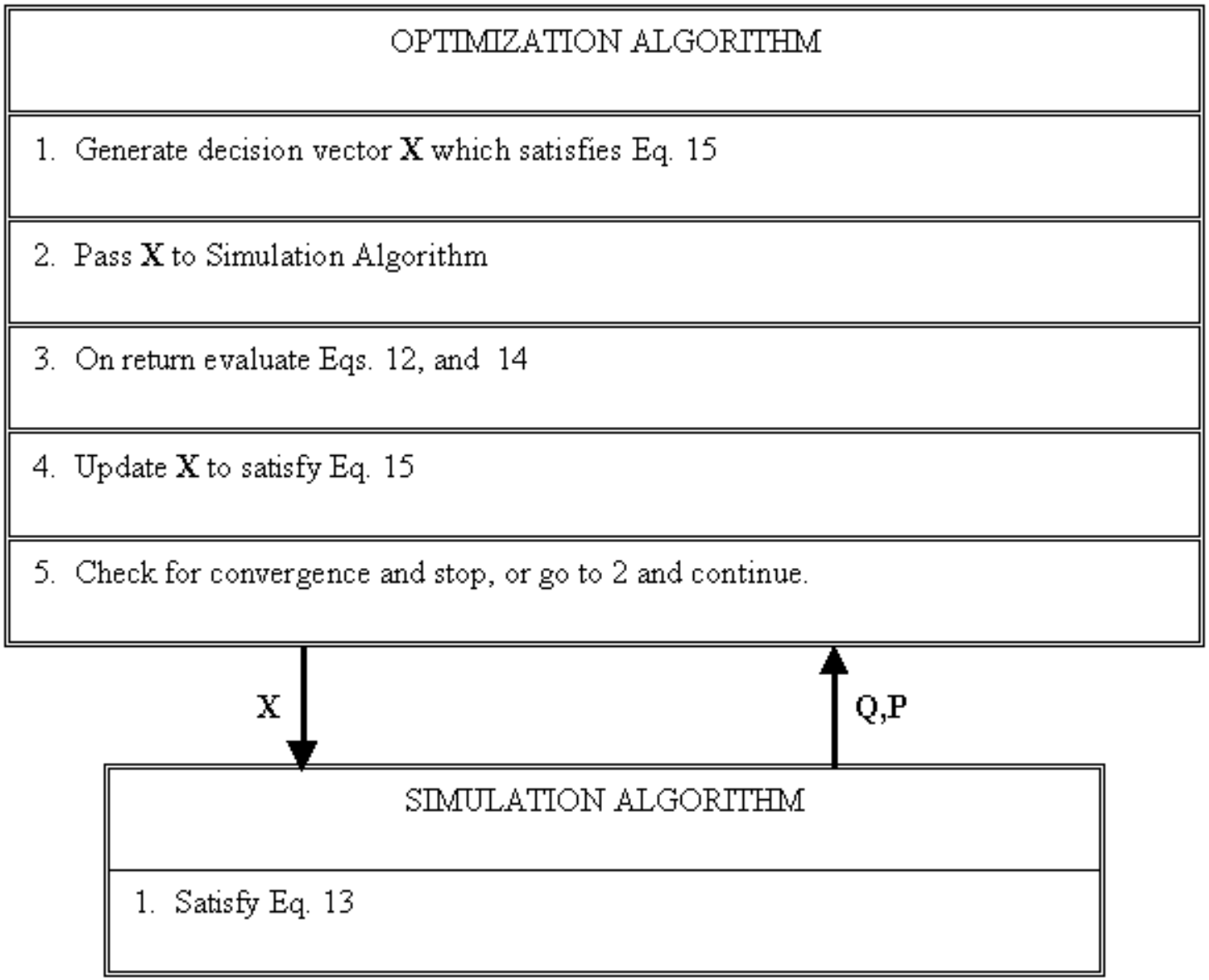

OPTIMIZATION ALGORITHM
1. Generate decision vector X which satisfies Eq. 15
2. Pass X to Simulation Algorithm
3. On return evaluate Eqs. 12, and 14
4. Update X to satisfy Eq. 15
5. Check for convergence and stop, or go to 2 and continue.
X
Q,P
SIMULATION ALGORITHM
1. Satisfy Eq. 13

Figure 8: Bi-level Computational Framework

Regardless of which approach is chosen, the resulting mathematical formulation must then be solved using some type of nonlinear optimization method. In general, three different approaches have been proposed and used: (1) gradient based methods, (2) pattern search methods, and (3) genetic optimization methods.

Gradient based methods require either first or second derivative information in order to produce improvements in the objective function. Traditionally, constraints are handled using either a penalty method or the Lagrange multiplier method (Edgar and Himmelblau, 1988). Pattern search methods employ a nonlinear heuristic that uses objective function values only in determining a sequential path through the region of search (Ormsbee, 1986, Ormsbee and Lingireddy, 1995). In general, when the objective function can be explicitly differentiated with respect to the decision variables the gradient methods are preferable to search methods. When the objective function is not an explicit function of the decision variables, as is normally the case with the current problem, then the relative advantage is not as great, although the required gradient information can still be determined numerically.

Recently, several researchers have begun to investigate the use of genetic optimization for solving such complex nonlinear optimization problems (Lingireddy et.al. 1995, Lingireddy and Ormsbee, 1998, and Savic and Walters 1995). Genetic optimization offers a significant advantage over more traditional optimization approaches in that it attempts to obtain an optimal solution by continuing to evaluate multiple solution vectors simultaneously (Goldberg, 1989). In addition, genetic optimization methods do not require gradient information. Finally, genetic optimization methods employ probabilistic transition rules as opposed to deterministic rules which has the advantage of insuring a robust solution methodology.

Genetic optimization starts with an initial population of randomly generated decision vectors. For an application to network calibration, each decision vector could consist of a subset of pipe roughness coefficients, nodal demands, etc. The final population of decision vectors is then determined through an iterative solution methodology that employs three sequential steps: 1) evaluation, 2) selection, and 3) reproduction. The evaluation phase involves the determination of the value of a fitness function (objective function) for each element (decision vector) in the current population. Based on these elevations, the algorithm then selects a subset of solutions for use in reproduction. The reproduction phase of the algorithm involves the generation of new offspring (additional decision vectors) using the selected pool of parent solutions. Reproduction is accomplished through the process of crossover whereby the numerical values of the new decision vector is determined by selecting elements from two parent decision vectors. The viability of the thus generated solutions is maintained by random mutations that are occasionally introduced into the resulting vectors. The resulting algorithm is thus able to generate a whole family of optimal solutions and thereby increase the probability of obtaining a successful model calibration.

Although optimization in general and genetic optimization in particular offer very powerful algorithms for use in calibrating a water distribution model, the user should always recognize that the utility of the algorithms are very much dependent upon the accuracy of the input data. Such algorithms can be susceptible to convergence problems when the errors in the data are significant (e.g. headloss is on the same order of magnitude as the error in headloss). In addition, because most network model calibration problems are under-specified (i.e. there are usually many more unknowns than data points), many different solutions (i.e. roughness coefficients, junction demands) can give reasonable pressures if the system is not reasonably stressed when the field data are collected.

## 9 Future Trends

With the advent and use of nonlinear optimization, it is possible to achieve some measure of success in the area of micro-level calibration. It is of course recognized that the level of success will be highly dependent upon the degree that the sources of macro-level calibration errors have first been eliminated or at least significantly reduced. While these sources of errors may not be as readily identified with conventional optimization techniques, it may be possible to develop

prescriptive tools for these problems using expert system technology. In this case general calibration rules could be developed from an experiential data base that could then be used by other modelers in an attempt to identify the most likely source of model error for a given set of system characteristics and operating conditions. Such a system could also be linked with a graphical interface and a network model to provide an interactive environment for use in model calibration.

In recent years, there has been a growing advocacy for the use of both GIS technology and SCADA system databases in model calibration. GIS technology provides an efficient way to link customer billing records with network model components for use in assigning initial estimates of nodal demands (Basford and Sevier, 1995). Such technology also provides a graphical environment for examining the network database for errors. One of the more interesting possibilities with regard to network model calibration is the development and implementation of an on-line network model through linkage of the model with an on-line SCADA system. Such a configuration provides the possibility for a continuing calibration effort in which the model is continually updated as additional data is collected through the SCADA system (Schulte and Malm, 1993).

Finally, Bush and Uber (1998) have recently developed three sensitivity-based metrics for ranking potential sampling locations for use in model calibration. Although the documented sampling application was small, the developed approach provides a potential basis for selecting improved sampling sites for improved model calibration. It is expected that this area of research will see additional activity in future years.

## 10 Summary and Conclusion

Network model calibration should always be performed before any network analysis planning and design study. A seven-step methodology for network model calibration has been proposed. Historically, one of the most difficult steps in the process has been the final adjustment of pipe roughness values and nodal demands through the process of micro-level calibration. With the advent of recent computer technology it is now possible to achieve good model calibration with a reasonable level of success. As a result, there remains little justification for failing to develop good calibrated network models before conducting network analysis. It is expected that future developments and applications of both GIS and SCADA technology, as well as optimal sampling algorithms will lead to even more efficient tools.

## 11 References

Basford, C. and Sevier, C., (1995) "Automating the Maintenance of a Hydraulic Network Model Demand Database Utilizing GIS and Customer Billing Records," Proceedings of the 1995 AWWA Computer Conference, Norfolk, VA, 197-206.

Boulos, P., and Ormsbee, L., (1991) "Explicit Network Calibration for Multiple Loading Conditions, Civil Engineering Systems, Vol 8., 153-160.

Brion, L. M., and Mays, L. W., (1991) "Methodology for Optimal Operation of Pumping Stations in Water Distribution Systems," ASCE Journal of Hydraulic Engineering, 117(11).

Bush, C.A., and Uber, J.G., (1998) "Sampling Design Methods for Water Distribution Model Calibration," ASCE Journal of Water Resources Planning and Management, 124(6). 334-344.

Cesario, L., Kroon, J.R., Grayman, W.M., and Wright, G., (1996). "New Perspectives on Calibration of Treated Water Distribution System Models." Proceedings of the AWWA Annual Conference, Toronto, Canada.

Cesario, L., (1995). Modeling, Analysis and Design of Water Distribution Systems, American Water Works Association, Denver, CO.

Coulbeck, B., (1984). "An Application of Hierachial Optimization in Calibration of Large Scale Water Networks," Optimal Control Applications and Methods, 6, 31-42.

Cruickshank, J.R & Long, S.J. (1992) Calibrating Computer Model of Distribution Systems. Proc. 1992 AWWA Computer Conf., Nashville, Tenn.

Edgar, T.F., and Himmelblau, D.M., (1988) Optimization of Chemical Processes, McGraw Hill, New York, New York, 334-342.

Gofman, E. and Rodeh, M., (1981) "Loop Equations with Unknown Pipe Characteristics," ASCE Journal of the Hydraulics Division, 107(9), 1047-1060.

Goldberg, D.E., (1989) Genetic Algorithms in Search, Optimization and Machine Learning, Addison-Wesley Pub. Co., Reading, MA.

Grayman, W.M., (1998). "Use of Trace Studies and Water Quality Models to Calibrate a Network Hydraulic Model," Esstential Hydraulics and Hydrology, Haested Press

Kennedy, M., Sarikelle, S., and Suravallop, K., (1991) "Calibrating hydraulic analyses of distribution systems using fluoride tracer studies." Journal of the AWWA, 83(7), 54-59

Lamont, P.A., (1981), "Common Pipe Flow Formulas Compared with the Theory of Roughness," Journal of the AWWA, 73(5), 274.

Lansey, K, and Basnet, C., (1991) "Parameter Estimation for Water Distribution Networks," ASCE Journal of Water Resources Planning and Management, 117(1), 126-145.

Lingireddy, S., Ormsbee, L.E. and Wood, D.J.(1995) User's Manual - KYCAL, Kentucky Network Model Calibration Program, Civil Engineering Software Center, University of Kentucky.

Lingireddy, S., and Ormsbee, L.E., (1998) "Neural Networks in Optimal Calibration of Water Distribution Systems," Artificial Neural Networks for Civil Engineers: Advanced Features and Applications. Ed. I. Flood, and N. Kartam. American Society of Civil Engineers, p277.

McEnroe, B, Chase, D., and Sharp, W., (1989) "Field Testing Water Mains to Determine Carrying Capacity," Technical Paper EL-89, Environmental Laboratory of the Army Corps of Engineers Waterways Experiment Station, Vicksburg, Mississippi.

Meredith, D. D. (1983) "Use of optimization in calibrating water distribution models," ASCE Spring Convention, Philadelphia, Pa.

Ormsbee, L.E., (1989) "Implicit Pipe Network Calibration," ASCE Journal of Water Resources Planning and Management, 115(2), 243-257.

Ormsbee, L.E., (1986) "A nonlinear heuristic for applied problems in water resources," Proc. Seventeenth Annual Modeling and Simulation Conference, University of Pittsburgh, 1117-1121.

Ormsbee, L.E., Chase, D.V., and Grayman, W., (1992) "Network Modeling for Small Water Distribution Systems," Proceedings of the AWWA 1992 Computer Conference, Nashville, TN, 15.

Ormsbee, L., Chase and D., and Sharp, W., (1991) "Water Distribution Modeling", Proceedings, 1991 AWWA Computer Conference, Houston, TX, April 14-17, 27-35.

Ormsbee, L.E. and Chase, D.V., (1988) "Hydraulic Network Calibration Using Nonlinear Programming," Proceedings of the International Symposium on Water Distribution Modeling, Lexington, Kentucky, 31-44.

Ormsbee, L.E. and Lingireddy, S., (1995) Nonlinear Heuristic for Pump Operations, Journal of Water Resources Planning and Management, American Society of Civil Engineers, 121, 4, 302-309.

Ormsbee, L.E. and Wood, D.J., (1986) "Explicit Pipe Network Calibration," ASCE Journal of Water Resources Planning and Management, 112(2), 166-182.

Rahal, C. M., Sterling, M.J.H, and Coulbeck, B., (1980), "Parameter tuning for simulation models of water distribution Networks, Proc., Institution of Civil Engineers, London, England, 69(2), 751-762.

Rossman, L., (1994) EPANET User's Manual, Drinking Water Research Division, Risk Reduction Engineering Laboratory, Cincinnati, Ohio 45268

Savic, D.A., and Walters, G.A. (1995) Genetic Algorithm Techniques for Calibrating Network Models, Report No. 95/12, 1995, Center for Systems and Control, University of Exeter, UK.

Schulte, A. M., and Malm, A. P., (1993) "Integrating Hydraulic Modeling and SCADA Systems for System Planning and Control," Journal of the American Water Works Association, 85(7), 62-66.

Walski, T.M. (1999), Personal Communication

Walski, T. M. (1995) "Standards for model calibration," Proceedings of the 1995 AWWA Computer Conference, Norfolk, VA, 55-64.

Walski, T.M. (1990a) Sherlock Holmes Meets Hardy Cross, or Model Calibration in Austin, Texas, Jour. AWWA, 82:3:34.

Walski, T. M. (1990b) Water Distribution Systems: Simulation and Sizing, Chelsea, Mich, Lewis Publishers.

Walski, T.M., (1986) "Case Study: Pipe Network Model Calibration Issues," ASCE Journal of Water Resources Planning and Management, 112(2), 238.

Walski, T.M., (1985) "Correcting Head Loss Measurements in Water Mains," Journal of Transportation Engineering, 111(1), 75.

Walski, T, M. (1984) Analysis of Water Distribution Systems, Van Nostrand Reinhold Company, New York, New York.

Walski, T. M. (1983) "Technique for Calibrating Network Models," ASCE Journal of Water Resources Planning and Management, 109(4), 360-372.

Water Authorities Association and WRc, (1989), Network Analysis - A Code of Practice, WRc, Swindon, England.

Wood, D. J., (1991) Comprehensive Computer Modeling of Pipe Distribution Networks, Civil Engineering Software Center, College of Engineering, University of Kentucky, Lexington, Kentucky.

## 1.3.7.1.2 Calibration

Calibration

See also: Optimized Calibration Data

Calibration Wizard

Calibration of Hydraulic Networks

Calibration Examples

Optimized Calibration

Calibration Demo File

---

Calibrating (bench marking) a hydraulic model is a very important step in developing a good hydraulic model. The developers of Pipe2018 have also developed advanced techniques for carrying out an effective calibration. Pipe2018 includes a very advanced state-of the-art module for optimized calibration.

Detailed information on calibration is included in the section entitled Calibration of Hydraulic Networks. Several approaches to calibration are discussed and these are illustrated with example calibrations in the section entitled Calibration Examples. This includes an illustration of the implicit approach using the **Optimized Calibration** module.

A brief description of the powerful **Optimized Calibration** objectives and approach is presented in the Optimized Calibration section and the associated Pipe2018 data requirements are outlined in the Optimized Calibration Data section.

The Calibration Wizard walks the user through the steps required to set up an Optimized Calibration.

## 1.3.7.1.3 Calibration Wizard

Calibration Wizard

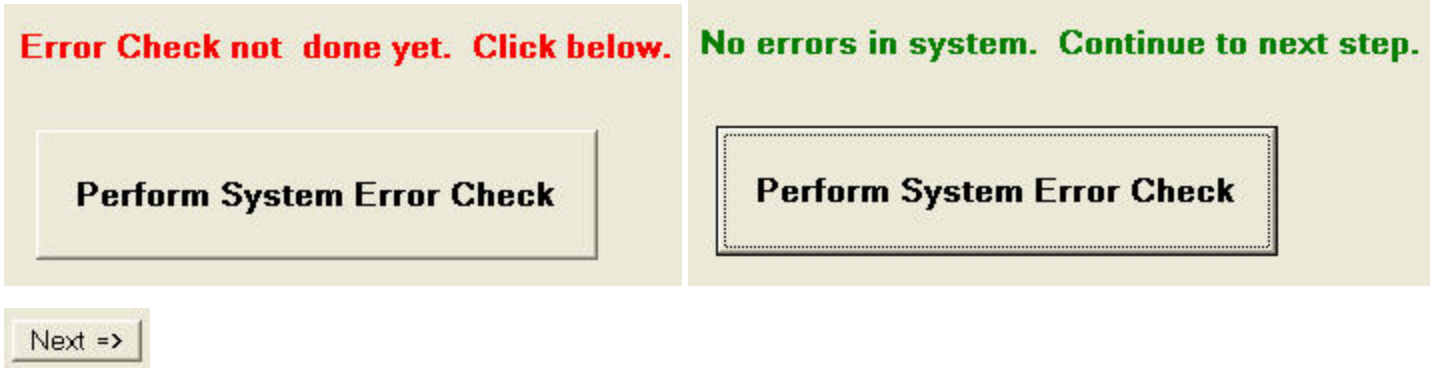
Example/Presentation

---

This tool is provided to guide the user through the Optimized Calibration process.

Important Note: In order to effectively use Optimized Calibration, it is recommended that the analysis results of the model are ideally within 5% of up-to-date, reliable field test data (e.g. static pressures). A model that is not within this range may not result in valuable optimized calibration results.

**Page 1 - Error Check**



At any time, the settings may be reset to default values by clicking:

### Page 2 - Calibration Types

**Normal Calibration** – A normal calibration will return a set of multipliers which are used to calculate new Hazen Williams roughness coefficients for each pipe in the model.

**Age Based Calibration** – An age-based calibration will return calibrated ten-year roughness values. When applied to the model, all future analyses by default use these calibrated roughness values based on current or specified year.

**Important note:** An age-based calibration may be performed only if the reference year (installation year) is provided for all pipes which are to be included in the calibration. In addition, the roughness coefficients for these pipes need to be unfixed (uncheck 'F' box next to 'Roughness' in the pipe information window).

### Page 3 - Calibration Type Checkup

**Pipe Material** - Defining the pipe material gives the Calibration Wizard more criteria on which to base the selection of pipe groups (See 'Pipe Groups' help on next Wizard page). Calibration is possible without defined material, but the diversity of the pipe groups may be limited. For instance, if pipe material is not defined, pvc and cast iron pipes may be included in the same group. The reference (assigned) roughness for pvc may be too high in the model whereas the value given to cast iron may be too low. With both types of pipe in the same group, the calibration wizard cannot differentiate between them. Since the roughness for the group as a whole is either increased or decreased, the resulting new roughness values may be exaggerated in order to reach a good result.

See Sets and Group Mode in the Help for information on how to globally assign material data to groups of pipes.

See pipe groups on the next Wizard Help page.

**Pipe Roughness** - Only pipes with unfixed roughness value may be included in the calibration process. Not all pipes must be included in order to calibrate, but it is recommended to include as many as possible.

**How to Unfix Pipe Roughness** -

Enter Group mode (Group button on left of screen).

Under Edit (Main Menu) click Select All Pipes.

In the Edit Pipe Set box, under Item to Edit, select Roughnesses.

Under Operation select Not Fixed.

Click Proceed.

**Age** - The age may be noted by entering a Reference Year (install year) for a pipe. The reference year is necessary if an age-based analysis or calibration is to be conducted. Reference year is entered in the Pipe Information window in the Other Data box. See Sets and Group mode for information on assigning values to groups of pipes.

### Page 4 - Assigning Pipe Groups

**Pipe Groups -** In the Optimized Calibration process, pipe are placed into groups. These groups will receive a either multiplier used to modify roughnesses, or a new 10-year roughness factor for age-based analysis. The Calibration Wizard will automatically divide the eligible pipes in the system into groups based on material, diameter, and rating.

The user may define groups manually if desired. Up to nine groups may be identified. These groups must be assigned an integer identifier of 1 – 9. Assign an integer to each pipe type group displayed in the Wizard. More than one pipe type may be assigned to a single group. Groups are also recorded as 'Default Calibration Group' in the pipe type table (Setups/Defaults | Pipe Type).

Pipes excluded from the calibration include pipes not assigned reference year if doing an age-based calibration or those with a fixed roughness for either calibration type. With user-defined groups, pipes may be excluded by entering a zero as the assigned group number or leaving the group number blank.

### Page 5 - Field Data and Boundry Conditions

**Field Data** – these are measurements of flow and pressure taken during a hydrant flow test or flushing event. Junctions or hydrant elements may be used to represent the location of the data collection points. It is recommended to use at least two tests for the calibration.

**Boundary Conditions** – these are the settings that describe the system during the field test. For example, if a certain supply pump was on or off during the field test, this would need to be shown along with the applicable test data. If conditions were different during the test than the baseline conditions of the model, it should be specified in the Calibration Wizard using this table.

**Use More Readings** – During a single flow test, measurements may be taken in several locations. This check box allows additional field test data to be entered.

### Page 6 - Demand Patterns

**Demand Pattern -** If, during a field test, the demands in the system were different than the baseline demands in the current model, the actual demand should be used during the Calibration process. This table is used to specify demand factors to adjust the demand to reflect conditions at the time of the test.

An existing demand pattern may be loaded.

### Page 7 - Analysis

**Advanced Analysis Options**

**Transfer data from Calibration menu** – if data has already been entered under Other Data | Calibration, this data may be imported into the Calibration Wizard. This will also import any relevant change and demand pattern data under Setups/Defaults | Change Patterns or Demand Patterns.

**Transfer data to Calibration menu** – this will export data defined in the Calibration Wizard to the calibration menu located under Other Data | Calibration. It will also transfer the boundary conditions as change data and the demand factors as demand pattern data under Setups/Default | Demand Patterns and Change Patterns. This will overwrite changes and demand pattern data already entered in these menus.

**Automatically apply calibration results** – If this is checked, the results will be applied to the model upon completion of the calibration analysis. If an age-based calibration is conducted, the 10-year roughness will be modified. An age-based roughness will be used in all future calculations. If a normal calibration is conducted, the reference roughness will be adjusted by the calculated factors.

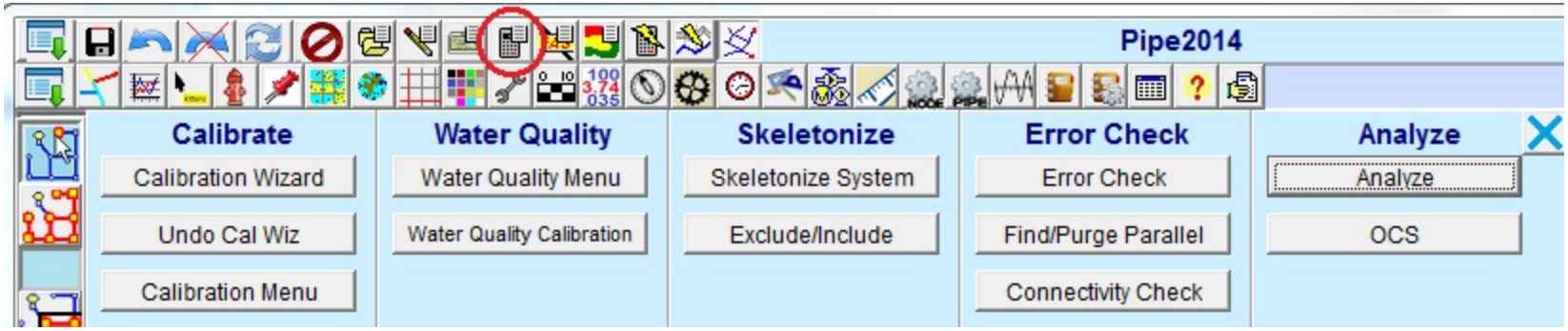
**Tolerance**

**High Roughness Tolerance** – if this box is checked the roughness bounds (which limit the magnitude of the change in pipe roughness) will be 3.0 and 0.5.

This this box is unchecked, the default values of 5.0 and 0.3 are used.

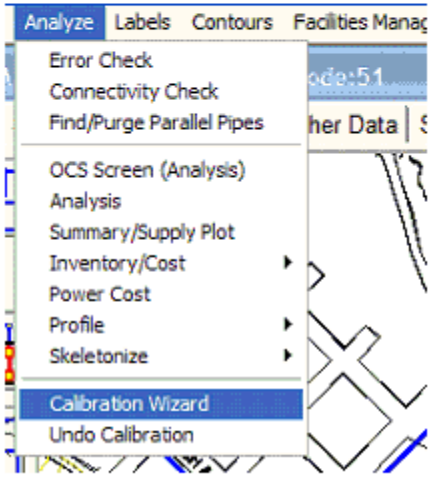
**Demand Tolerance** – The percent deviation of the total demand in the system allowable for the calibration. Depending on how much is known about the conditions in the system during the collection of field data, it may be desired to apply some tolerance to the system demand. The default is 5%.

**FireFlow Tolerance** - The percent deviation of the measured fireflow allowable for the calibration. Depending on how reliable the field test data is known to be, it may be desired to apply some tolerance to the flow measurements. The default is 10%.




## Example/Presentation

# Calibration Wizard (CW)

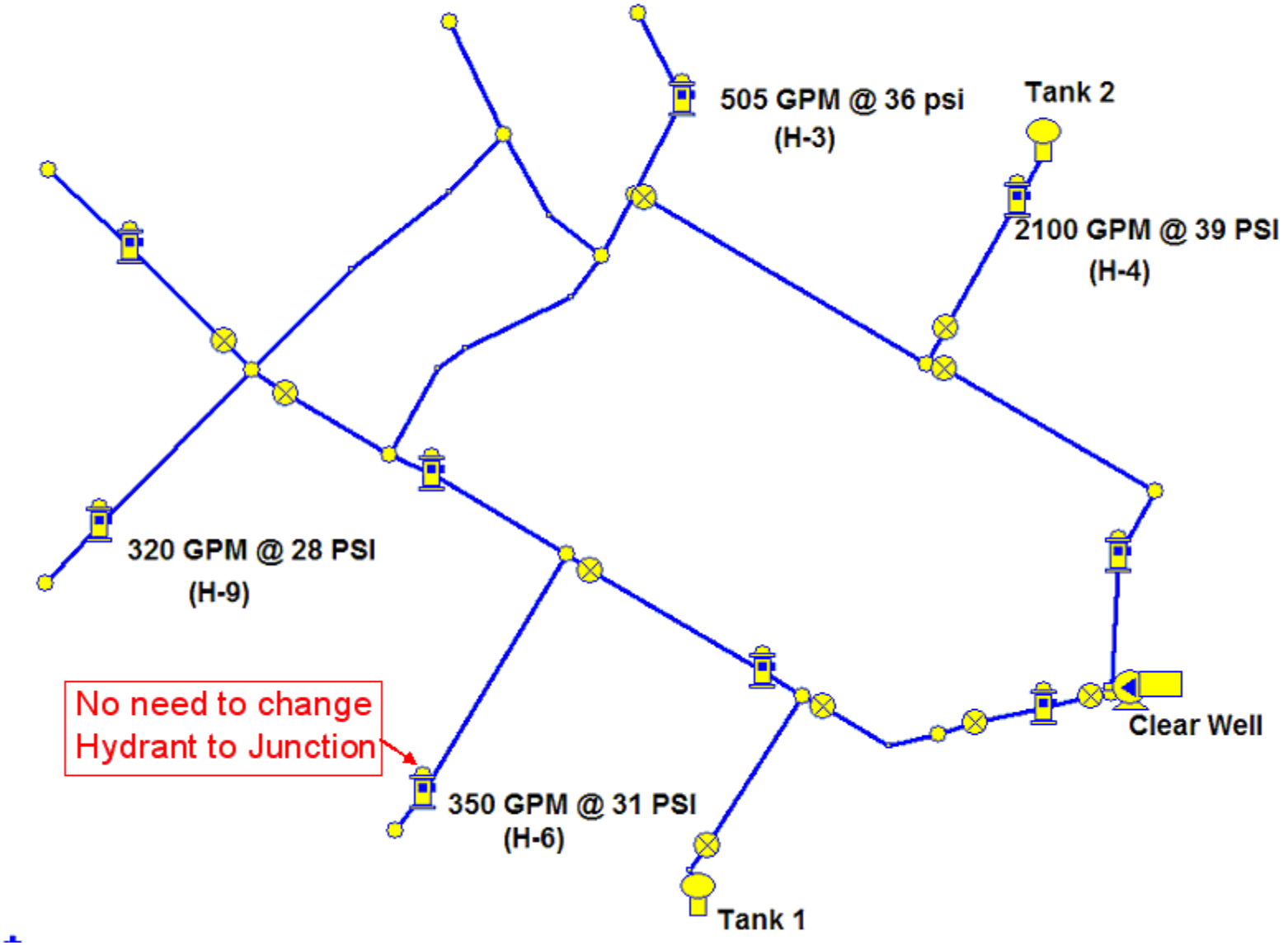



## Powerful Tool for Pipe2000 Users

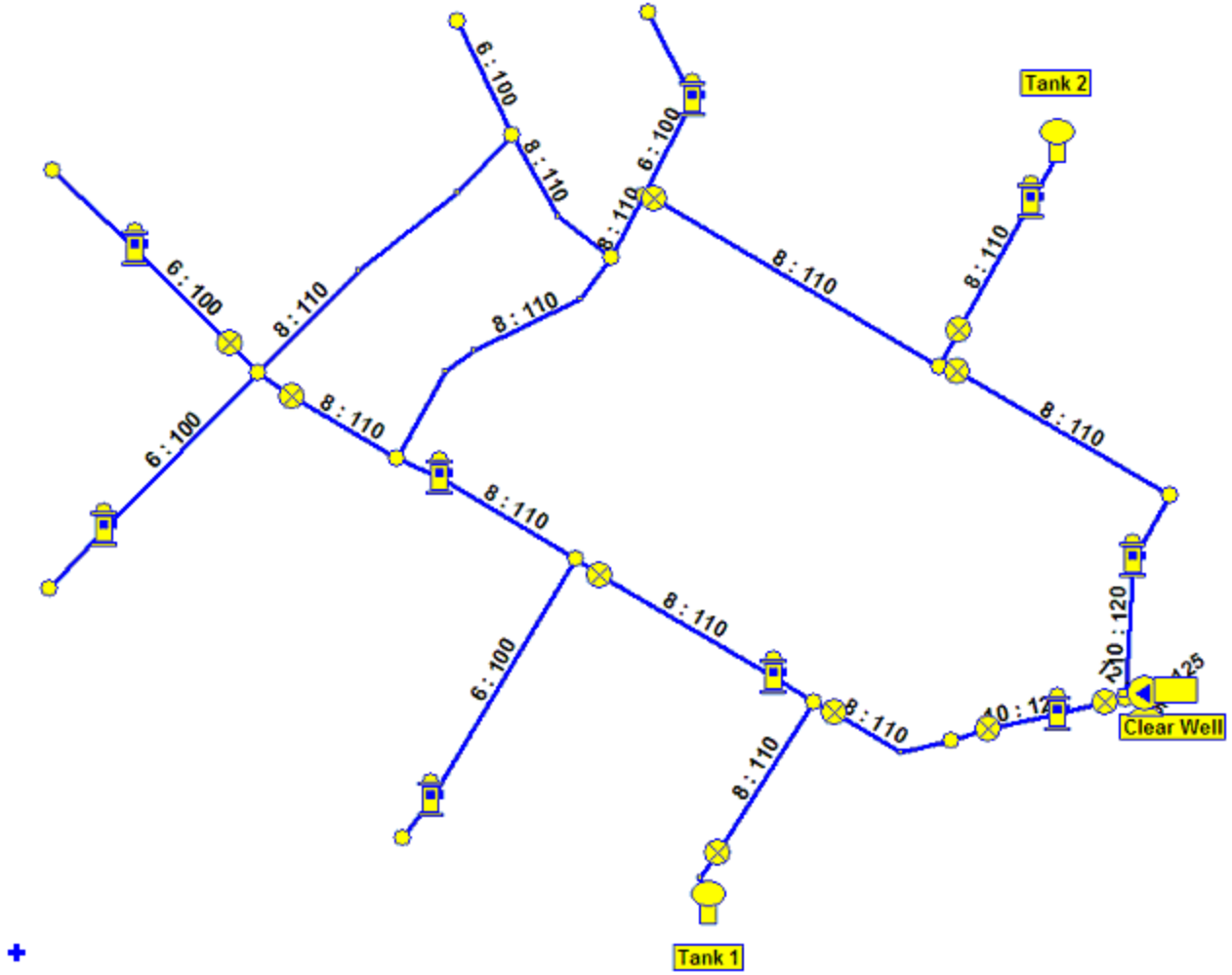
Advantages:

1) CW Simple to Use
2) Directly Uses Hydrants
3) Automatically Update Data File (Roughness)

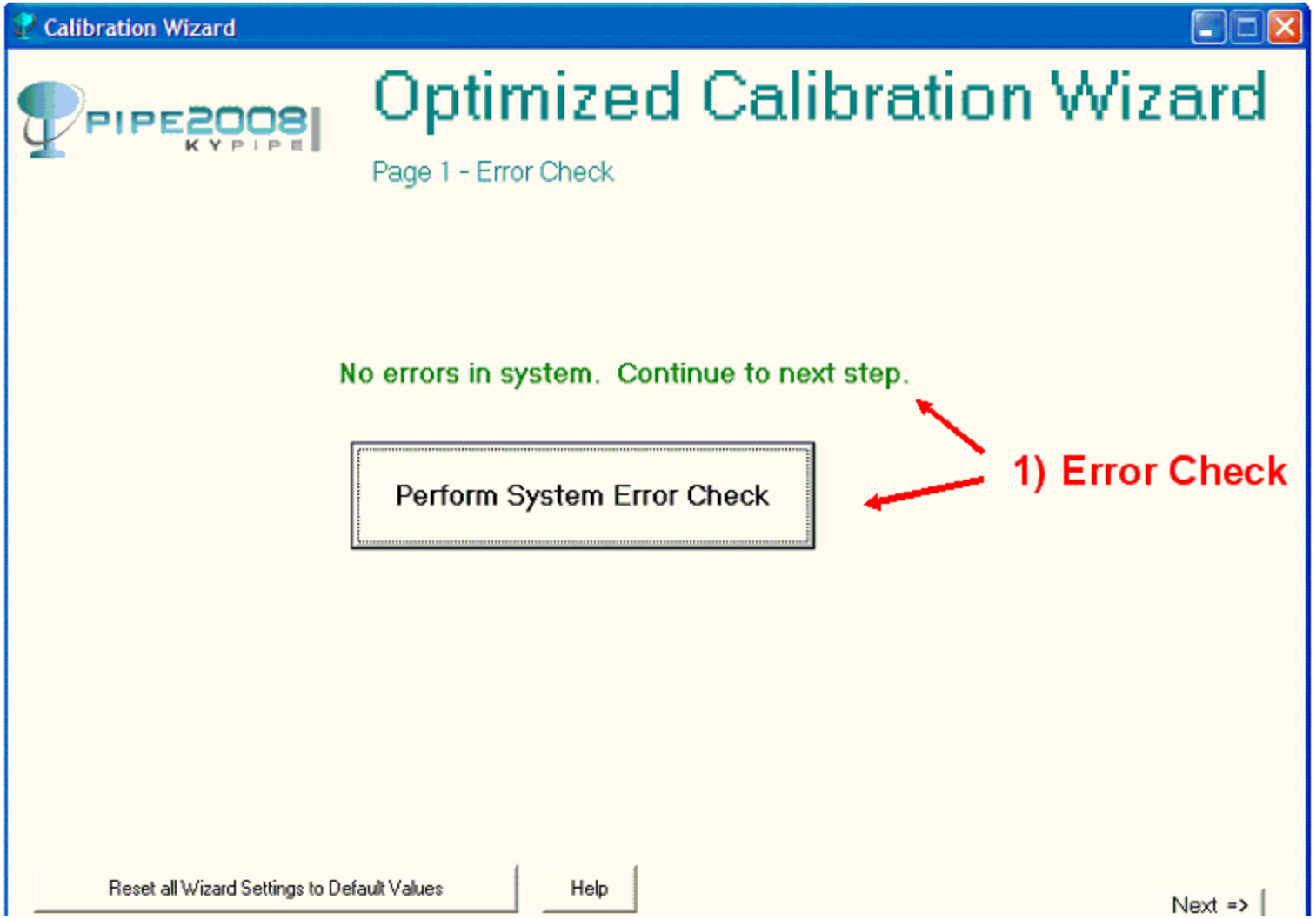
# Examples Manual - Democalibration



# Roughnesses prior to Calibration
# Best Estimates



# (1) Error Check



# 2) Define Pipe Groups

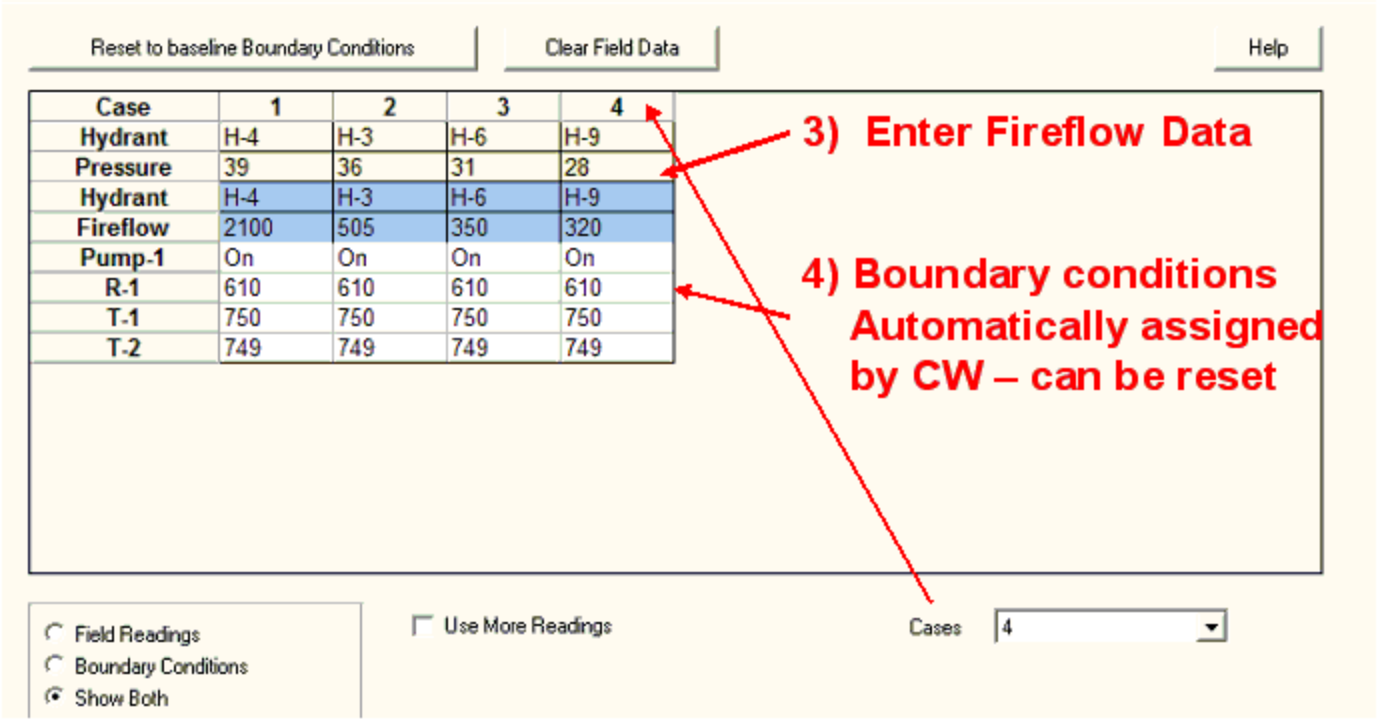
| Pipe Type | # of Pipes | Ref Roughness | Suggested Group | User Assigned Group |
|---|---|---|---|---|
| 6 - di - 150 | 5 | 100 | 1 | |
| 8 - di - 150 | 12 | 110 | 2 | |
| 10 - di - 150 | 2 | 115 | 3 | |
| 12 - di - 150 | 2 | 120 | 4 | |

**2) CW automatically picks groups based on material & diameter**

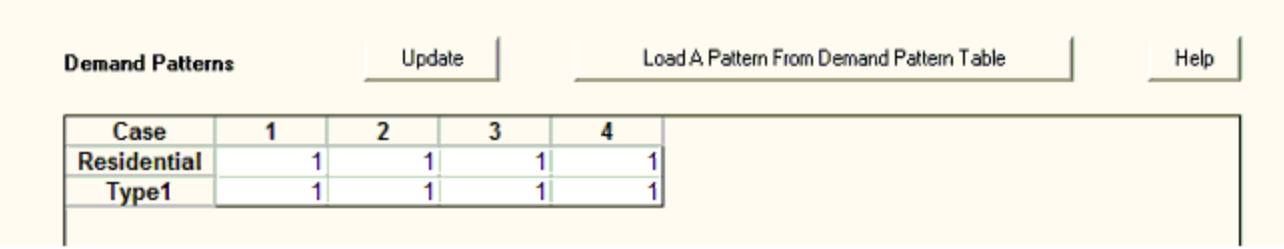
**Calibration determines Roughness Adjustment Factor for each group**

Number of pipes excluded from Calibration: 0

# (3) Fireflow Data
# (4) Boundary Conditions

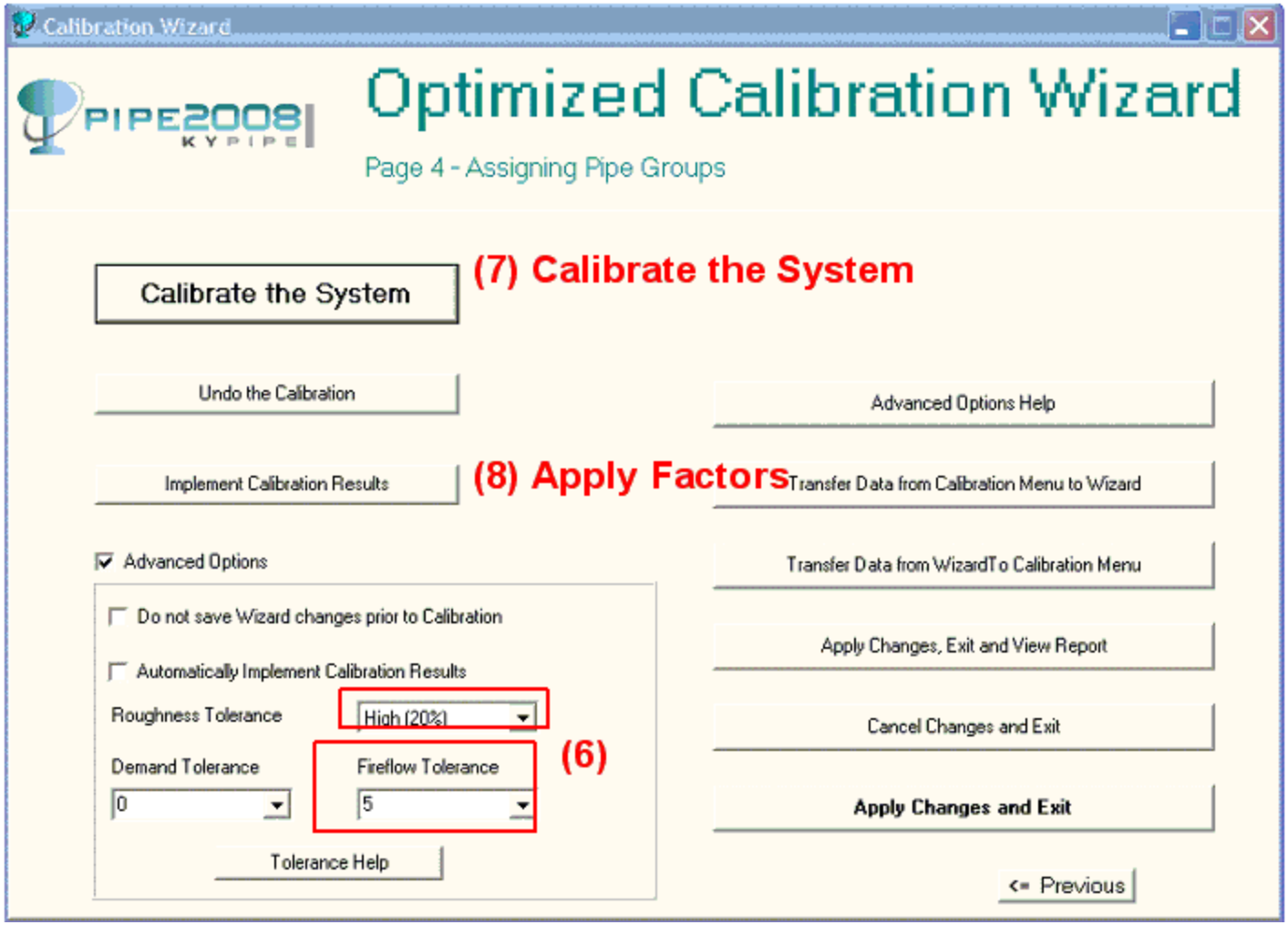


# (5) Demand Patterns



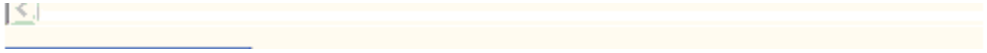

**5) Demand Patterns**
**Automatically assigned**
**by CW – can be reset**

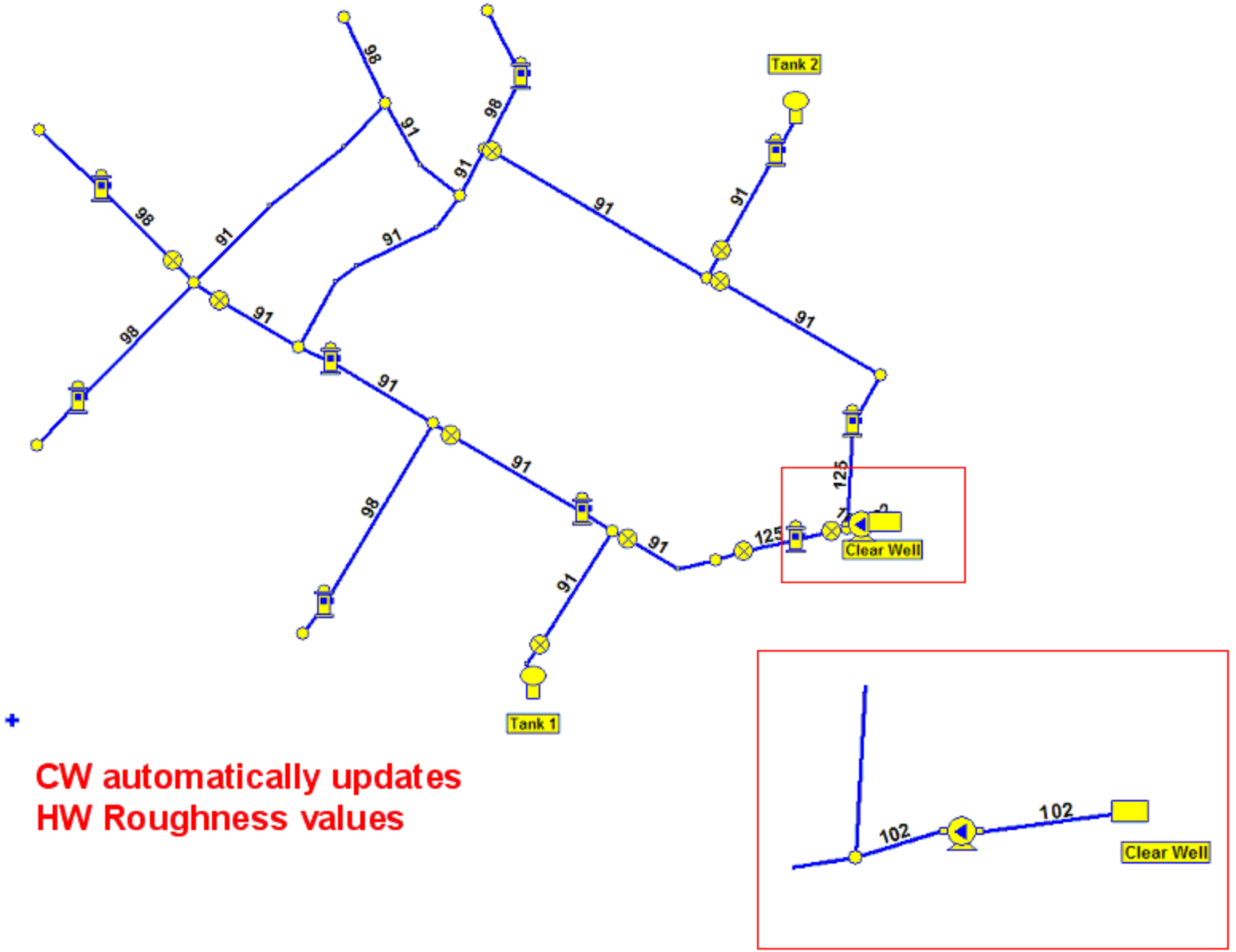
# 6) Set Tolerances and Options



# Pipe2008 Calibration Report

```
Percent Deviation between MEASURED and TARGET Values =   1.699


OPTIMAL values for the Decision variables:

Hazen William coefficients  for group number   1
                are adjusted by a factor of       0.98 [ 1.20< > 0.80]
Hazen William coefficients  for group number   2
                are adjusted by a factor of       0.83 [ 1.20< > 0.80]
Hazen William coefficients  for group number   3
                are adjusted by a factor of       1.05 [ 1.20< > 0.80]
Hazen William coefficients  for group number   4
                are adjusted by a factor of       0.81 [ 1.20< > 0.80]

No demand adjustments are made.
Demand Tolerance is meant for re-distributing
demands among nodes of diff demand types, keeping
the total demand constant. There must be at least
TWO types of demands to use this feature.

Junction (Fire) Flow(s) for Change  1 are DECREASED by  1.13%
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  4.35%


Measured and Target pressures (psi or kPa):

   TEST        NODE       MEASURED        OPTIMAL
   CASE       NUMBER      PRESSURE       PRESSURE
 -----------------------------------------------
     1          H-4         39.0           39.0
     2          H-3         36.0           33.6
     3          H-6         31.0           31.1
     4          H-9         28.0           27.9
```




Tank 2
Tank 1
Clear Well
98
91
125
102
CW automatically updates
HW Roughness values

## 1.3.7.1.4 Calibration Examples

See also Calibration of Hydraulic Networks

- Optimized Calibration Data
- Calibration Wizard
- Optimized Calibration

Using Pipe2018 to Calibrate a Water Distribution Model

In general, there are three different ways that Pipe2018 can be used to calibrate a water distribution system. These include 1) an Empirical Approach (trial and error), 2) an Explicit Approach (using the constraint feature), and 3) an Implicit Approach (using optimization). Although each of these methods are discussed and illustrated in the following sections, it should be stressed that the implicit method will normally be the recommended method, especially for larger systems or systems involving several different field observations. This powerful feature is based on genetic optimization (GA) technology and can be used to produce an optimally calibrated system with minimal effort. Calibrating real water distribution systems using the empirical approach can be very frustrating except for the most simple system. The explicit approach can usually be successfully applied to smaller systems with one or two field observations, but unfortunately, the method cannot constrain or restrict the resultant parameter values to predetermined limits. As a result, the method may produce unrealistic parameter values that may not be valid for boundary conditions other than those observed during the associated field test.

To illustrate the application of each technique, a simple 16-pipe system has been prepared and saved with the appropriate data for each application (see Figure 1). The associated three files are named: empirical.p2k, explicit.p2k, and implicit.p2k. In calibrating this system, two fire-flow tests have been conducted. The first fire-flow test was conducted at junction 3 during a slack demand period (i.e. tanks full at 210 feet and demands equal to 50% of the average day demands). During the fire-flow test a flow of 1750 gpm was observed at junction 3 with a residual pressure of 14.4 psi. This situation is modeled by assigning a total demand of 2000 gpm at junction 3 which represents the sum of the fire-flow and the base flow (i.e. 2000 gpm = 1750 gpm + 500 gpm*0.5 slack demand factor). The second fire-flow test was conducted at junction 7 during a peak demand period (i.e. tanks empty at 190 feet and demands equal to 200% of the average day demands). During

the fire-flow test, a flow of 900 gpm was observed at junction 7 with a residual pressure of 15.4 psi. This situation is modeled by assigning a total demand of 1500 at junction 7 which represents the sum of the observed fire-flow and the base flow (i.e. 1500 gpm = 900 gpm + 300 pgm*2.0 peak demand factor).

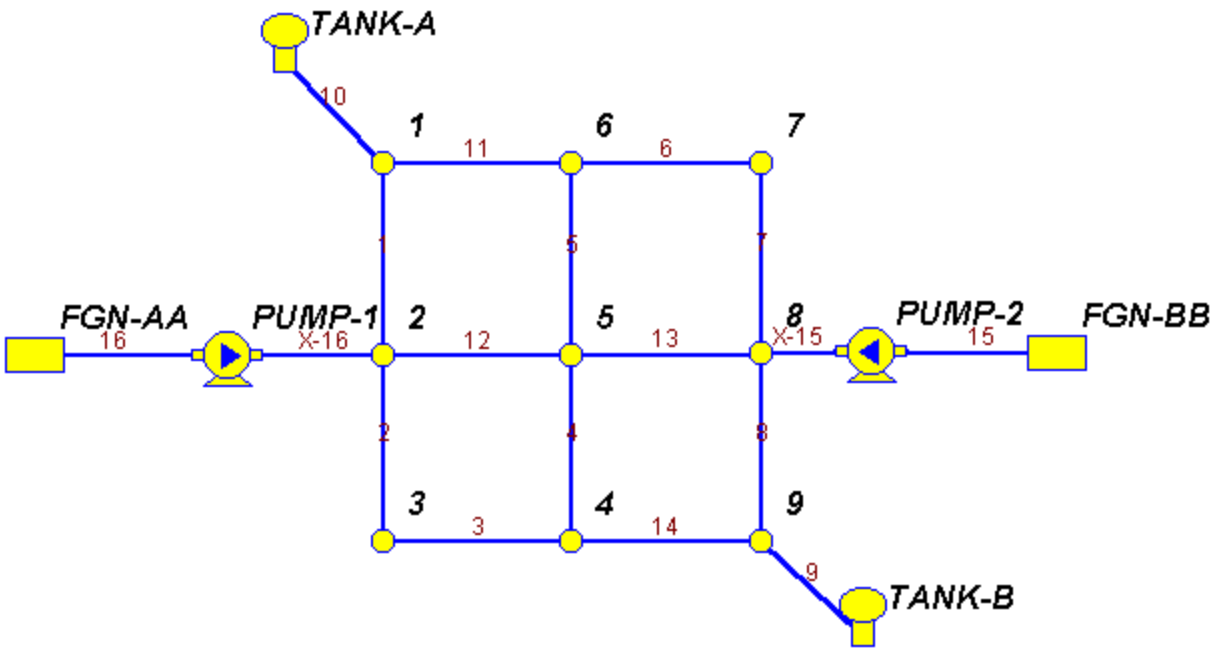



Figure 1: Example System

1. Empirical Approach

In using an empirical approach, the modeler tries to calibrate the model by systematically changing different model parameters (e.g. pipe roughness, nodal demands, etc.) until the model results (e.g. pressures and flows) approach those obtained in the field at various times and under various loading or boundary conditions. Comparisons between the values obtained in the field and those predicted by the model can be made by displaying the pressures or flows for a particular pipe or junction node via the Map environment or from examining the results from the Report screen. Changes to the model parameters may be made through the Pipe2018 Map environment or via the Pipe and Node tables that may be accessed from the Table button in the Map environment. Global changes to demands and global changes to pipe roughness can also be made through the Demand Patterns and Change Patterns menus respectively (see Figures 2 and 3).

KYnetic Demand Pattern icon:

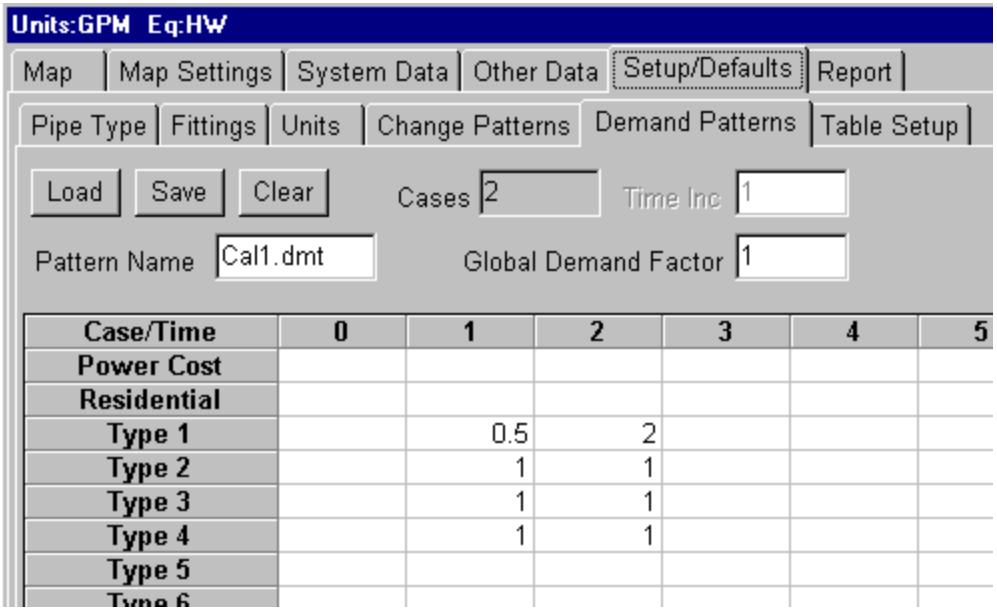



Figure 2. Demand Patterns Menu

KYnetic Pipe Change Table icon:

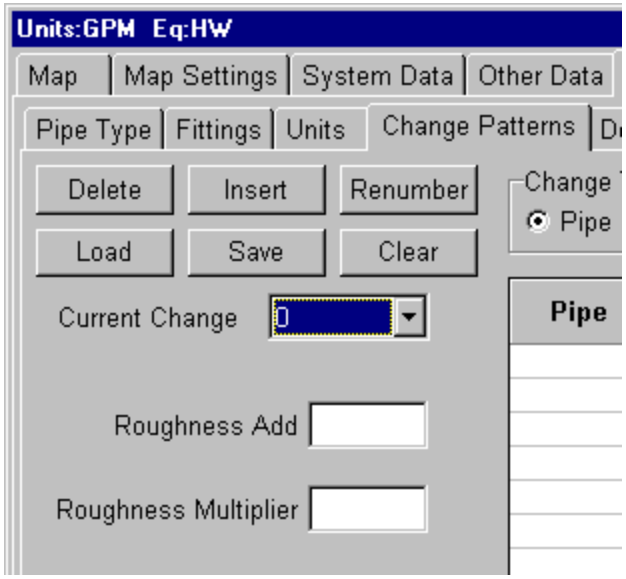



Figure 3. Change Patterns Menu

In order to properly calibrate the model, the modeler must precisely specify the boundary conditions associated with the system at the time of the field measurements were made. In calibrating a water distribution model using Pipe2018 this is handled using the **Demand Patterns** and **Change Patterns** menus which are accessed from the icons or **Setup/Defaults** (Classic) menu. In using these menus, each time/case will correspond to a particular point in time when one or more field observations were made. The Demand Patterns menu can be used to specify the total demand at the time a particular set of field measurements were made. Likewise, the **Change Patterns** menu can be used to specify the status of existing components (e.g. pumps, lines, etc.) as well as the values of particular units (e.g. tanks, junctions, etc.) at the time the same particular set of field measurements were made. For example, suppose that a particular model is being calibrated using two sets of data that were collected at two different points in time. In this case, the modeler would input the boundary conditions for the first observation as Time/Case 1 and the boundary conditions for the second observation as Time/Case 2. Suppose for example that Time/Case 1 corresponded to a slack demand period (with a global demand factor of 0.5) and Time/Case 2 corresponded to a peak demand period (with a global demand factor of 2.0). Typing in the data as shown in Figure 2 previously would reflect these conditions. Suppose in addition, that Time/Case 1 has a total demand of 2000 gpm at junction 3 (base demand plus fire flow demand) with both corresponding tanks at full (i.e. grade = 210), and Time/Case 2 has a total demand of 1500 gpm (base demand plus fire flow demand) at junction 7 with corresponding tanks empty (i.e. grade = 190). This situation can be represented using the **Change Pattern** menu as shown in Figure 4 below. In this particular case, the only changes in boundary conditions were related to node data. If each Time/Case has an associated change in the pipe data, then this difference would be reflected by separate entries in the Pipe Change Type of the Change Pattern menu.

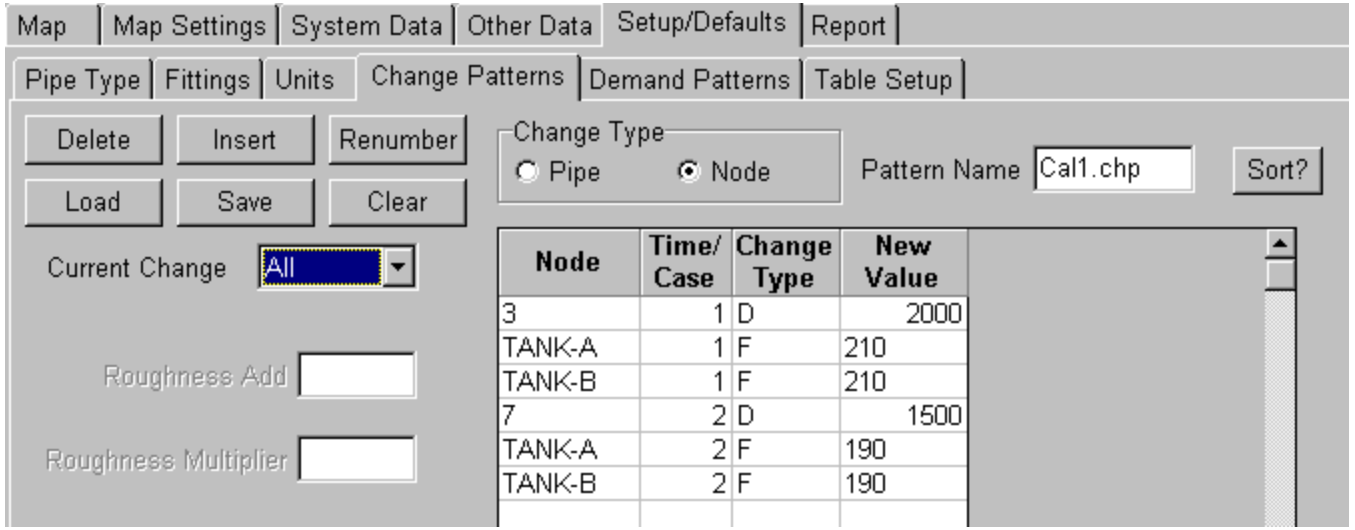



Figure 4. Boundary Conditions for Example System

Once the user has input the data associated with each field observation, Pipe2018 may be executed and the results from the model compared with the observed conditions. In the event the resulting values are not judged to be within an acceptable tolerance, the model parameters (i.e. pipe roughness and/or nodal demands) may be adjusted and the model re-run until the model results are deemed acceptable.

2. Explicit Approach

In using an explicit approach, the modeler can use the advanced parameter determination feature (use of Constraints) to calibrate the model by having the model automatically adjust pipe roughness or nodal demands to explicitly match observed pressures at one or more junction nodes in the system. As with the empirical approach, the modeler must precisely specify the boundary conditions associated with the system at the time the field measurements were made. In the case where all field data were collected at the same time, the associated boundary conditions may be specified via the node and pipe menus within the Map environment. In the event that the field observations were taken at different points in time (as is the normal case and the case of the example problem), the boundary conditions may be specified using the Demand Patterns (see Figure 2) and the Change Patterns (see Figure 4) menus which are accessed from the Setup/Defaults menu. As before, each time/case will correspond to a particular point in time when one or more field observations were made. For the example network, the slack demand and peak demand boundary conditions associated with fire flow tests 1 and 2 are input as global demand factors of 0.5 (slack demand) and 2.0 (peak demand) for cases 1 and 2 respectively (see Figure 2). The tank levels associated with both conditions (tanks full at 210 feet for slack conditions, and

tanks empty at 190 feet for peak conditions) would be set up as before in Figure 4. In this case the demands assigned at junction 3 (Q=2000 gpm) for case 1 and the demands assigned at junction 7 (Q=1500) for case 2 represent the combined base demand and fire flow demand at each junction at the time of the test. (i.e. Q3 = 2000 = 1750 fire flow + 500 base demand*0.5 demand factor, Q7 = 1500 = 900 fire flow demand + 300 based demand*2.0 demand factor).

In using the explicit approach to calibrate a water distribution system with multiple observations, two approaches are possible: 1) the Sequential Approach, and 2) the Average Approach. In both approaches, the pipes (or nodes) are divided and lumped into separate calibration groups, with each calibration group corresponding to a particular pressure observation (e.g. constraint). This is accomplished by assigning each pipe (or node) to a specific constraint group via the constraint menu as shown in Figure 5.

Once the pipes (or nodes) are assigned to different constraint groups, separate pressure constraints may be developed for each field observation using the constraint menu. Constraint menus for the example problem for both field observations are shown in Figures 6 and 7. In order to enforce a particular constraint, the user must first turn the constraint on (by clicking on the "Apply this Constraint" box with the mouse) so that the check mark appears. The constraint can then be run by simply analyzing the system via the Analyze System menu.

KYnetic Data Table icon:

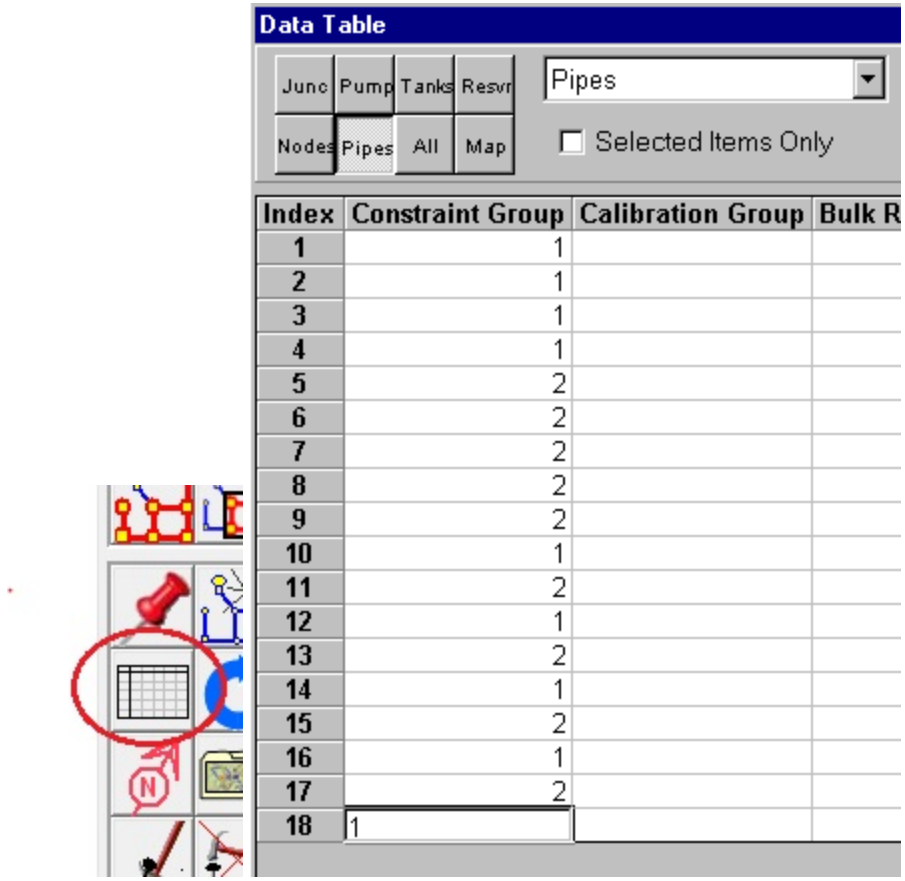



Figure 5. Constraint Groups

KYnetic Design Tools icon:

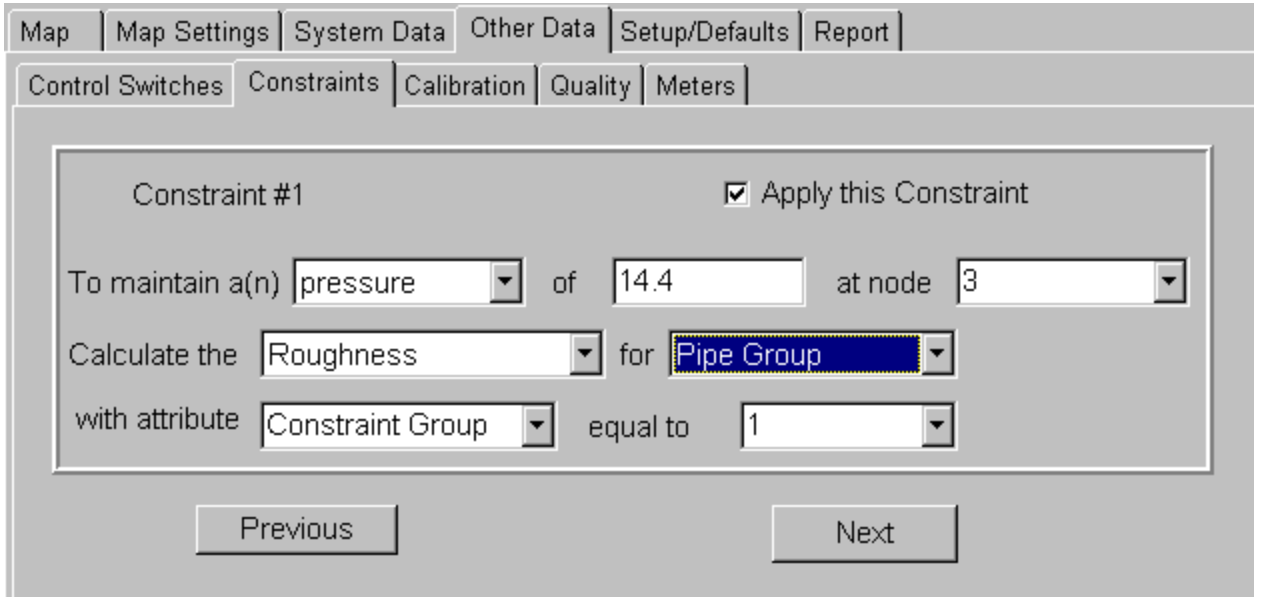



Figure 6. Constraint Menu for Fire Flow Test One

(Pressure of 14.4 observed at Junction 3)

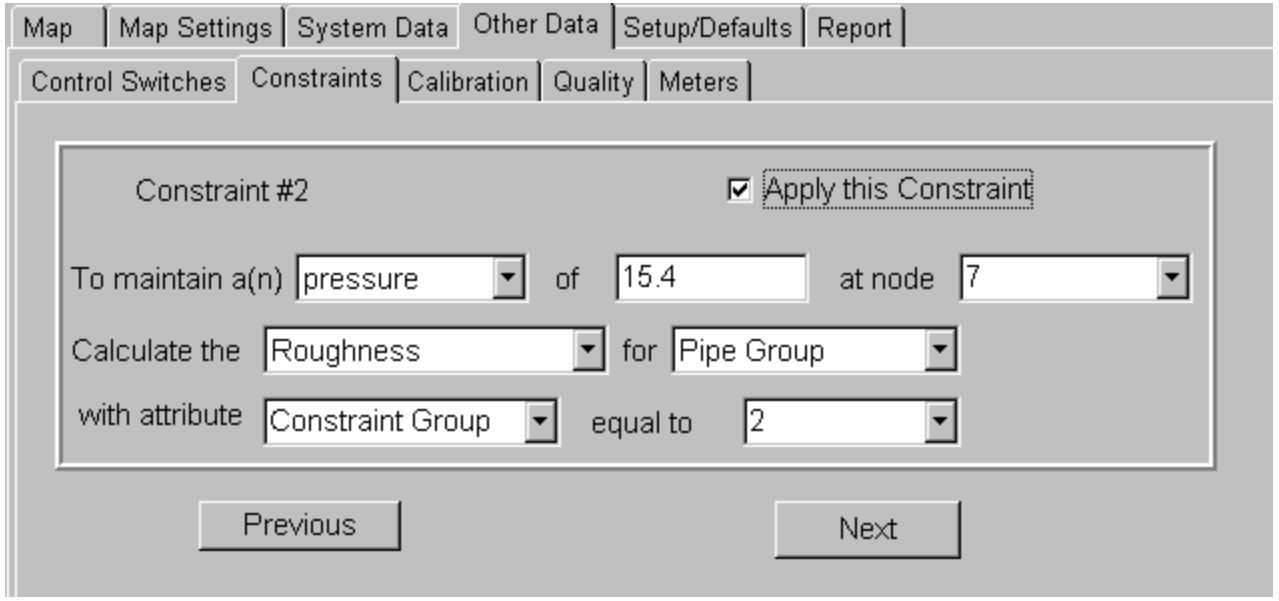



Figure 7. Constraint Menu for Fire Flow Test One

(Pressure of 15.4 observed at Junction 7)

2.1. Sequential Approach

In using the sequential approach, different groups of pipes are assigned to a different constraint group as illustrated for the example problem in Figure 5. Each constraint group will be associated with one of the constraints in either Figure 6 or Figure 7. That is, the pipes assigned to constraint group 1 will be adjusted in an attempt to meet the conditions of constraint 1 and the pipes assigned to constraint group 2 will be adjusted in an attempt to meet the conditions of constraint 2. Once the pipes (or nodes) have been assigned to different constraint groups, the first constraint is run (e.g. Figure 6) and the new pipe roughness values for the first group of pipes is obtained (see Figure 8). [Note: In running the second constraint, make sure that the first constraint is de-activated so that the associated "Apply this Constraint" box does not have a check mark in it]. Once these values are obtained, they should then be used to change the original values for the pipes associated with the first constraint group (either via the Map environment or via the Table menu). Once these changes have been made, the second constraint is run (e.g. Figure 7) and the new pipe roughness values for the second group of pipes is obtained (see Figure 9). [Note: In running the second constraint, make sure that the first constraint is de-activated so that the associated Apply this Constraint box does not have a check mark in it]. While in theory, such an approach should lead to an improved set of calibration factors for the entire system, the process of sequential changing the roughness values of individual constraint groups can modify the hydraulic conditions associated with the earlier calibrations and thus introduce errors into the previously calibrated data sets. One way to minimize this problem is to continue to repeat the process by alternating between updating and fixing the parameters associated with one constraint while then adjusting the remaining parameters through application of the second constraint. Eventually, an improved set of parameters should be obtained.

It should be noted that in the current example problem, only two constraints have been considered. However, there is nothing to prevent the process from being extended to problems with more than one constraint. In this case, the first constraint is enforced while all remaining constraints are relaxed. Next, the second constraint is enforced with again all remaining constraints are relaxed (including the first one). This process is then repeated until all constraints have been applied. At the end, the process can be repeated again as with the two constraint example.

```
C H A N G E S   F O R   N E X T   S I M U L A T I O N  (Change Number =   1 )


JUNCTION DEMANDS CHANGED - PLEASE SEE RESULTS TABLE

TANK at node     TANK-A  has a new HGL of                210.000
TANK at node     TANK-B  has a new HGL of                210.000

RESULTS OBTAINED AFTER   5 TRIALS: ACCURACY =      .00014


      P A R A M E T E R   R E S U L T S

     CONSTRAINT     PARAMETER       PIPE/NODE     PARAMETER
       NUMBER         TYPE            NAME         VALUE
     ------------------------------------------------------
          1         ROUGHNESS              1        88.917
          1         ROUGHNESS              2        88.917
          1         ROUGHNESS              3        88.917
          1         ROUGHNESS              4        88.917
          1         ROUGHNESS             10        88.917
          1         ROUGHNESS             12        88.917
          1         ROUGHNESS             14        88.917
          1         ROUGHNESS             16        88.917
          1         ROUGHNESS            X-16       88.917

     CONSTRAINT NO.  1 REQUIRES A FACTOR =   .889
```

Figure 8. Sequential Results for Case 1

```
C H A N G E S   F O R   N E X T   S I M U L A T I O N  (Change Number =   2 )

JUNCTION DEMANDS CHANGED - PLEASE SEE RESULTS TABLE

TANK at node     TANK-A  has a new HGL of                 190.000
TANK at node     TANK-B  has a new HGL of                 190.000

RESULTS OBTAINED AFTER   6 TRIALS: ACCURACY =      .00118

        P A R A M E T E R   R E S U L T S

     CONSTRAINT     PARAMETER       PIPE/NODE     PARAMETER
       NUMBER         TYPE             NAME        VALUE
     ---------------------------------------------------
          1         ROUGHNESS              5        89.710
          1         ROUGHNESS              6        89.710
          1         ROUGHNESS              7        89.710
          1         ROUGHNESS              8        89.710
          1         ROUGHNESS              9        89.710
          1         ROUGHNESS             11        89.710
          1         ROUGHNESS             13        89.710
          1         ROUGHNESS             15        89.710
          1         ROUGHNESS            X-15       89.710

     CONSTRAINT NO.  1 REQUIRES A FACTOR =   .897
```

Figure 9. Sequential Results for Case 2

2.2. Average Approach

Instead of first subdividing the pipes into multiple calibration groups with each group associated with a particular constraint, one may sequentially apply each individual constraint to all the pipes in the system. In this case, one would obtain a separate global roughness factor for each constraint. For the example problem this would mean that all pipes would be calibrated twice, once for the first constraint and once for the second constraint. In the end, a final parameter value is obtained by simply averaging the individual calibration values from each constraint application. The constraint menus for application of the average approach to the example problem are shown in Figures 10 and 11. Results for the example problem are shown in Figures 12 and 13. These results were obtained by first running Pipe2018 with the first constraint in force and then re-running Pipe2018 with the second constraint in force. From these results a final roughness coefficient of 89 was obtained by averaging the values of 89 and 90 obtained from application of constraints 1 and 2 respectively.

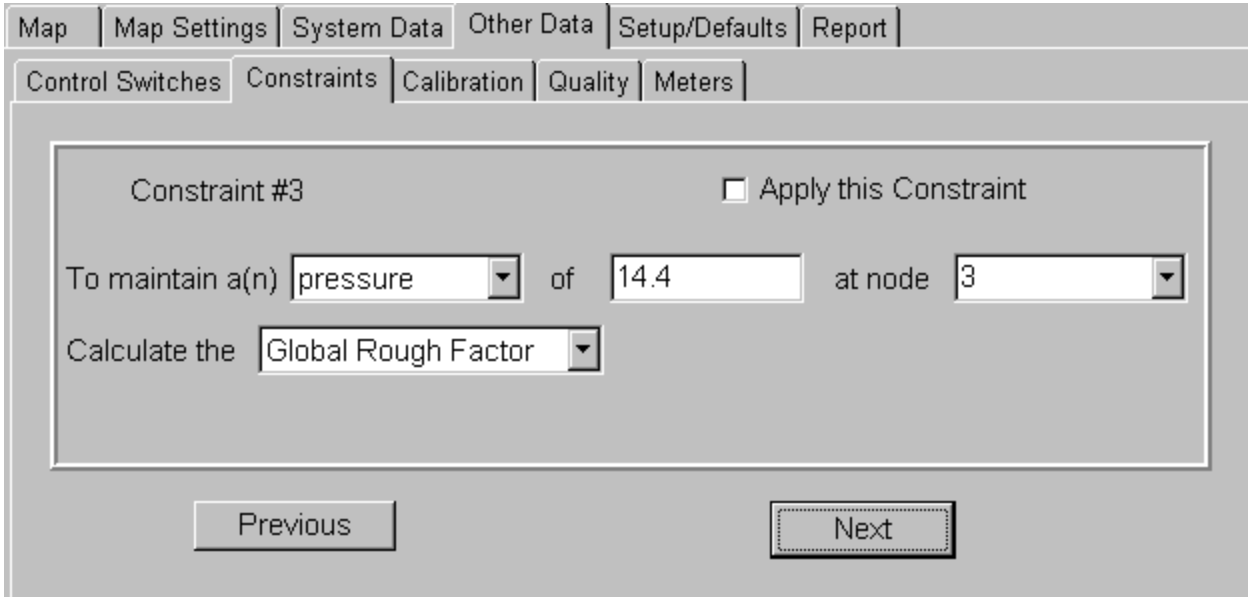



Figure 10. Constraint Menu for Fire-Flow Test One (Average Approach)

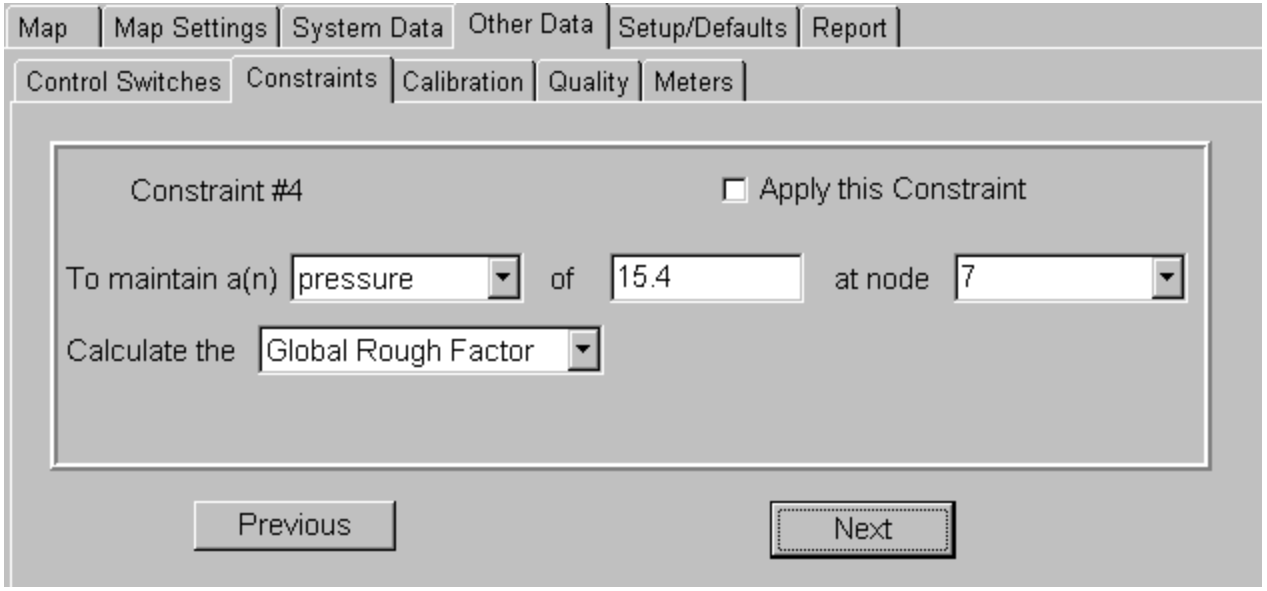



Figure 11. Constraint Menu for Fire-Flow Test Two (Average Approach)

P A R A M E T E R  R E S U L T S

| CONSTRAINT NUMBER | PARAMETER TYPE | PIPE/NODE NAME | PARAMETER VALUE |
|---|---|---|---|
| 1 | ROUGHNESS | 1 | 88.987 |
| 1 | ROUGHNESS | 2 | 88.987 |
| 1 | ROUGHNESS | 3 | 88.987 |
| 1 | ROUGHNESS | 4 | 88.987 |
| 1 | ROUGHNESS | 5 | 88.987 |
| 1 | ROUGHNESS | 6 | 88.987 |
| 1 | ROUGHNESS | 7 | 88.987 |
| 1 | ROUGHNESS | 8 | 88.987 |
| 1 | ROUGHNESS | 9 | 88.987 |

Figure 12. Average Results for Case 1

P A R A M E T E R  R E S U L T S

| CONSTRAINT NUMBER | PARAMETER TYPE | PIPE/NODE NAME | PARAMETER VALUE |
|---|---|---|---|
| 1 | ROUGHNESS | 1 | 89.925 |
| 1 | ROUGHNESS | 2 | 89.925 |
| 1 | ROUGHNESS | 3 | 89.925 |
| 1 | ROUGHNESS | 4 | 89.925 |
| 1 | ROUGHNESS | 5 | 89.925 |
| 1 | ROUGHNESS | 6 | 89.925 |
| 1 | ROUGHNESS | 7 | 89.925 |
| 1 | ROUGHNESS | 8 | 89.925 |
| 1 | ROUGHNESS | 9 | 89.925 |

Figure 13. Average Results for Case 2

## 3.0 Implicit Approach (Optimized Calibration)

In using an implicit approach, the modeler can use the built in calibration (optimization) algorithm associated with Pipe2018 to automatically adjust selected model parameters (e.g. pipe roughness, system demand) to match observed field conditions (e.g. pressures and flows). In order to use the built-in calibration algorithm, the user must first prepare the necessary calibration data. Similar with the explicit approach, the user first assigns different groups of pipes (or nodes) to separate calibration groups. In this case each calibration group will correspond to an individual field observation (e.g. a set of pressures or flows that were all measured at the same time and under the same boundary conditions). As with the previous example problem, two separate field observations are used: 1) results from a fire flow test at junction 3 under slack demand conditions and 2) results from a fire flow test at junction 7 under peak demand conditions. In applying the calibration program to the example problem, half of the pipes are assigned to calibration group 1 (as associated with fire flow test 1) and half of the pipes are assigned to calibration group 2 (and associated with fire flow test 2) - see Figure 14. As with the explicit approach, the Demand Patterns and Change Patterns menus are then used to specify boundary conditions associated with each individual field observation (i.e. either pressure or flow). For the example problem, the Demand Patterns menu is configured just like for the explicit application as shown in Figure 5. The Change Patterns menus for the example problem are shown in Figures 15 and 16. However, in the implicit case (as distinguished from the explicit case as shown in Figure 6), the individual fire flow data are not input in the Change Patterns menu. Instead, these values are input directly in a separate calibration menu (i.e Figure 17).

Data Table

Junc | Pump | Tanks | Resvr | Nodes | Pipes | All | Map

Pipes

Selected Items Only

Primary Data

User Data

| Index | Name | Constraint Group | Calibration Group | Bulk Rate | Wa |
|---|---|---|---|---|---|
| 1 | 1 | | 1 | | |
| 2 | 2 | | 2 | | |
| 3 | 3 | | 2 | | |
| 4 | 4 | | 2 | | |
| 5 | 5 | | 2 | | |
| 6 | 6 | | 2 | | |
| 7 | 7 | | 2 | | |
| 8 | 8 | | 2 | | |
| 9 | 9 | | 1 | | |
| 10 | 10 | | 1 | | |
| 11 | 11 | | 2 | | |
| 12 | 12 | | 1 | | |
| 13 | 13 | | 1 | | |
| 14 | 14 | | 2 | | |
| 15 | 15 | | 2 | | |
| 16 | 16 | | 2 | | |
| 17 | X-15 | | 2 | | |
| 18 | X-16 | | 2 | | |

Figure 14. Calibration Groups for Example Problem

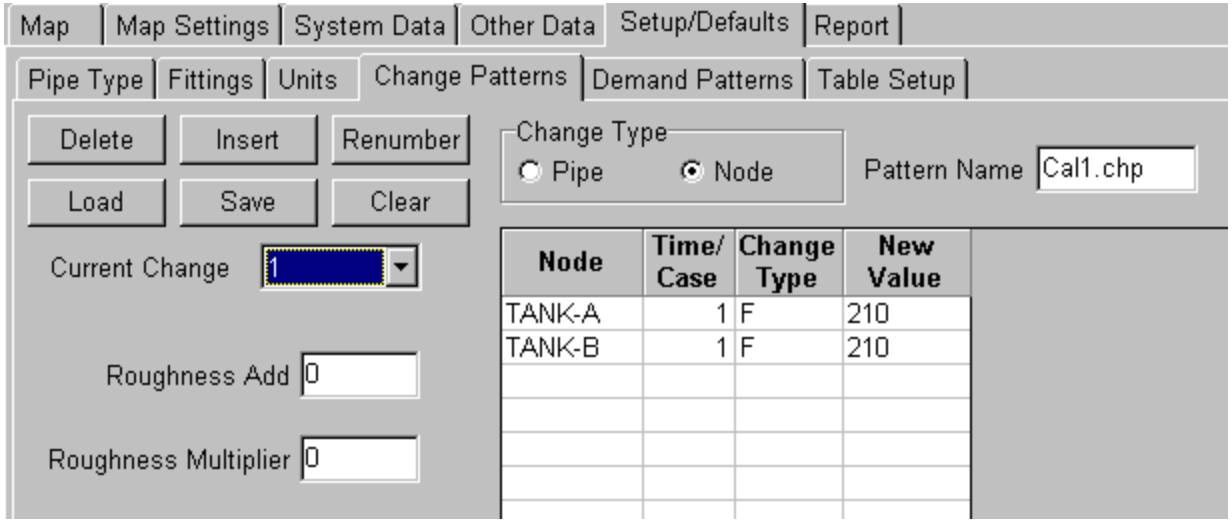



Figure 15. Change Patterns Menus for Case 1 for Example Problem

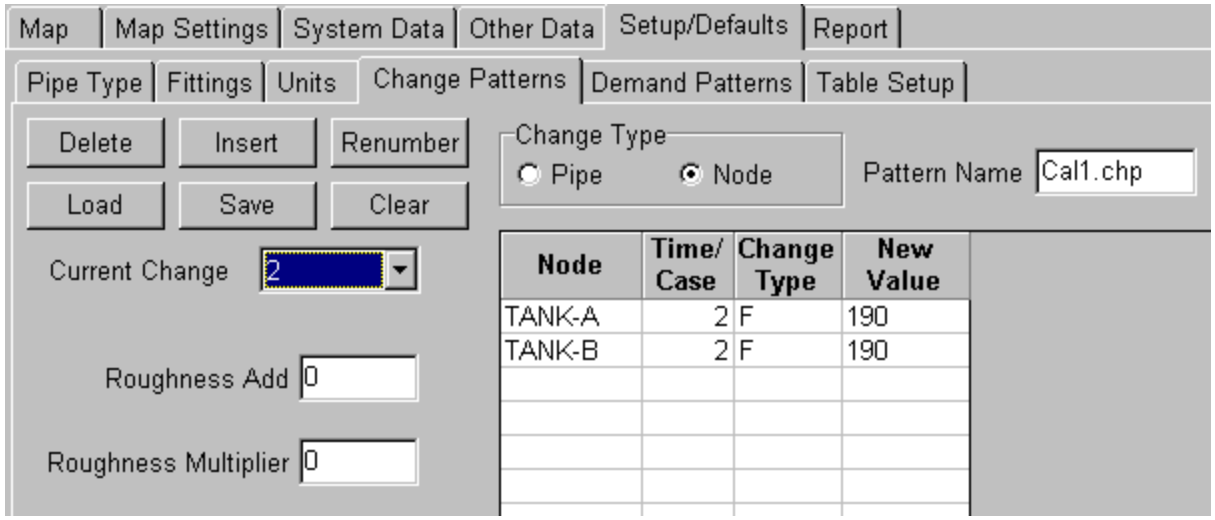



Figure 16. Change Patterns Menus for Case 2 for Example Problem

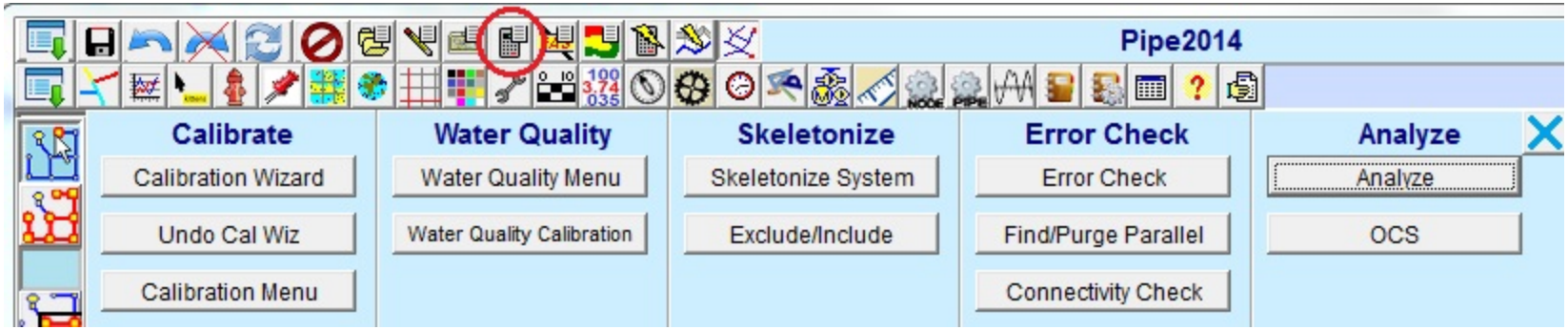
## 3.1 Optimized Calibration Data (Calibration Menu)

*** **See Optimized Calibration Data**

Once the calibration groups have been assigned and the associated boundary condition data input via the Demand Patterns and Change Patterns menus, the user is ready to input the remaining data required to perform the model calibration. This data is input via the Calibration menu which may be accessed from the Other Data menu as shown in Figure 17. The data values displayed in the Figure 17 correspond to the calibration settings for the Example Program. As can be seen from Figure 17, the Calibration menu is divided into six submenus. Above the six submenus are several input cells that are used to specify additional variables associated with the calibration process. Each of these items are discussed below:

### 3.1.1 Pipe Attribute

The first input cell is identified as the Attribute used for Pipe Type. The default entry is calibration group. This entry is used to tell the program how to distinguish one group of pipes from another in the subsequent pipe roughness bounds. Instead of using the normal calibration groups (as discussed previously), the user could use the constraint grouping associations (as in the explicit approach) to designate the associated calibration groups.




Use Calibration Menu button shown above.

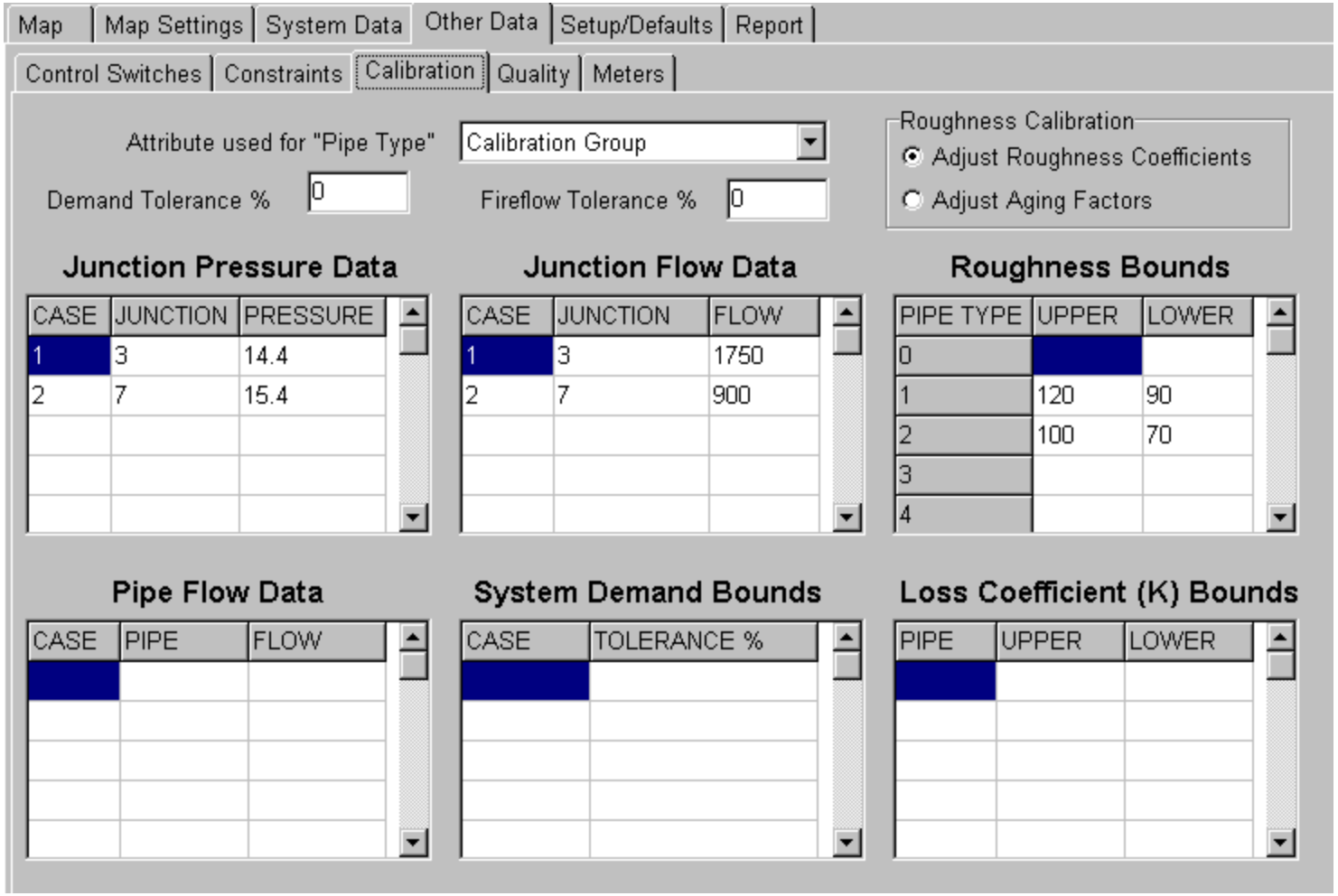



Figure 17. Calibration Menu.

3.1.2 Demand Tolerance %

The Demand Tolerance % cell is provided to allow the user to specify a calibration tolerance associated with the total system demand. When a non-zero value is specified, the calibration algorithm will attempt to make adjustments to the total system demand (within the specified tolerance) in an attempt to decrease the

deviation between the observed and model predicted state values (e.g. pressures and flows). Normally, it will be expected that the tolerance will be zero, that is, the system demand is completely known. However, in some situations, there may be some uncertainty associated with the system demand measurements and in that case the uncertainty may be taken into account via the demand tolerance. As an example, if the user were to specify a system demand of 1 MGD with a tolerance of 5% then the calibration algorithm would allow the total system demand to vary between .95 MGD and 1.05 MGD during the calibration process.

### 3.1.3 Fireflow Tolerance %

The Fireflow Tolerance % cell is provided to allow the user to specify a calibration tolerance associated with each individual "fire-flow" observation. When a non-zero value is specified, the calibration algorithm will attempt to make adjustments (within the specified tolerance) to any "fire-flow" values (i.e. as specified in the Junction Flow Data in an attempt to decrease the deviation between the observed and model predicted state values (e.g. pressures and flows).

### 3.1.4 Roughness Calibration

The Roughness Calibration menu is used to specify whether one wants to calibrate the pipe roughness coefficients (the default), or individual aging factors. In order to determine individual aging factors (for an associated group of pipes), the user must first specify the pipe roughness for each pipe (in the regular pipe data) on the basis of observed values 10 years ago. The program will then determine the associated aging factors that will produce the existing field observations (e.g. pressures and flows). Once these factors are obtained, they can then be used to predict future roughness values.

### 3.1.5 Junction Pressures Menu

The Junction Pressures menu is used to specify observed pressures associated with selected junction nodes. The first column of the menu is used to identify which set of change data the junction pressure is to be associated with. The second column is used to specify the selected junction number while the third column is used to specify the observed pressure (psi or kpa). The user may specify up to four different pressure observations per set of change data. For the example problem, two separate pressure readings were obtained, one for each different set of boundary conditions. As a consequence, the junction pressure menu contains two separate pressure readings with each one associated with a different change (or boundary condition).

### 3.1.6 Junction Flow Data Menu

The Junction Flow Data menu is used to specify the observed flowrates associated with the junction pressures that are input in the Junction Pressure menu. It should be emphasized that the junctions input in the junction flow menu do not have to correspond to the junctions input in the pressure menu. For example, fire flow test data for a particular field observation may involve a measured flow from one junction with a residual pressure measured at another junction. In addition, there does not have to be a one to one correspondence between the number of junction nodes in each menu. For example, a user may input an observed pressure from a single junction node with fire flows from multiple junctions. Conversely, a user may specify flow from a single junction with pressures measured at

multiple junctions.

As with the Junction Pressures Menu, the first column is used to identify which set of change data the junction flow is to be associated with. The user may specify up to four different flow observations per set of change data. The second column is used to specify the selected junction number while the third column is used to specify the observed flowrate. For the example problem, one flow reading was obtained for each of the observed pressures recorded in the Junction Pressure Menu.

### 3.1.7. Roughness Bounds Menu

Once the various field observation data has been input, the user may specify bounds or limits on the values that the decision variables (i.e. pipe roughness or nodal demands) may assume. The Roughness Bounds menu is used for setting bounds on the values of the roughness coefficient associated with each pipe group. The first column of the menu is used to identify the number of the particular Pipe Type group. The next two columns are then used to specify both upper and lower bounds for the pipe roughness coefficient associated with that pipe group may assume. Bounds may be expressed in terms of actual roughness values (i.e. 80 to 120). For the example problem, upper and lower values of 120-90; and 100-70 were assumed for pipe type groups 1 and 2 respectively.

### 3.1.8. Pipe Flow Data Menu

In addition to the use of junction pressures, the user may also elect to use the calibration model to adjust the model parameters to match observed flowrates in specified pipes. This data may be entered using the Pipe Flow Data menu. As with the previous menus, the Pipe Flow Data menu has three columns. The first column is used to identify which set of change data the pipe flow is to be associated with. The user may specify up to four different flow observations per set of change data. The second column is used to specify the selected pipe number while the third column is used to specify the observed flowrate. For the example problem, no pipe flow observations were obtained and as a result, no values are included in the Pipe Flow Data menu.

### 3.1.9. System Demand Bounds Menu

The System Demand Bounds menu is used to set the total system demand for each set of boundary conditions (i.e. each change data set). In the event that the demand is left blank, the program will determine the total system demand on the basis of the sum of the initial nodal demands along with whatever demand adjustments are made in the change data. The first column in the System Demand Bounds menu is used to specify the set of boundary conditions (i.e. change set) to be associated with the total system demand that is to be entered in the second column. A global tolerance value for these values may be specified in the Demand Tolerance % Cell as discussed in section 3.1.2.

### 3.1.10 Loss Coefficient (K) Bounds

The Loss Coefficient Bounds menu is used for setting bounds on the values of minor loss coefficients associated with a pipe. The first column of the menu is use to identify the number of the particular pipe. The next two columns are then used to specify both upper and lower bounds for the pipe roughness coefficient associated with that pipe may assume. These parameters may be used in addition to or as an alternative to adjustments to pipe roughness coefficients.

3.2 Model Application

Once the necessary data are input to the associated calibration menus, the calibration analysis can be run by selecting the Analyze option from the top command bar. When the Analysis Setup menu appears, the user should select "System Calibration" from among the available Analysis Types. Once this is done, the calibration analysis may be launched by depressing the Analyze button with the mouse (see Figure 18). Upon execution of the program, a dialogue box will appear in which the status of the calibration runs will be displayed. Once the calibration is complete, a message box will appear on the screen (see Figure 19) indicating that the calibration is completed. An exit code of 0 indicates that the program terminated normally. At this point, the user should click on "Yes" to exit the termination menu and to return to the normal menu environment. The results of the calibration may now be examined via the Report environment which may be accessed using the Report tab.

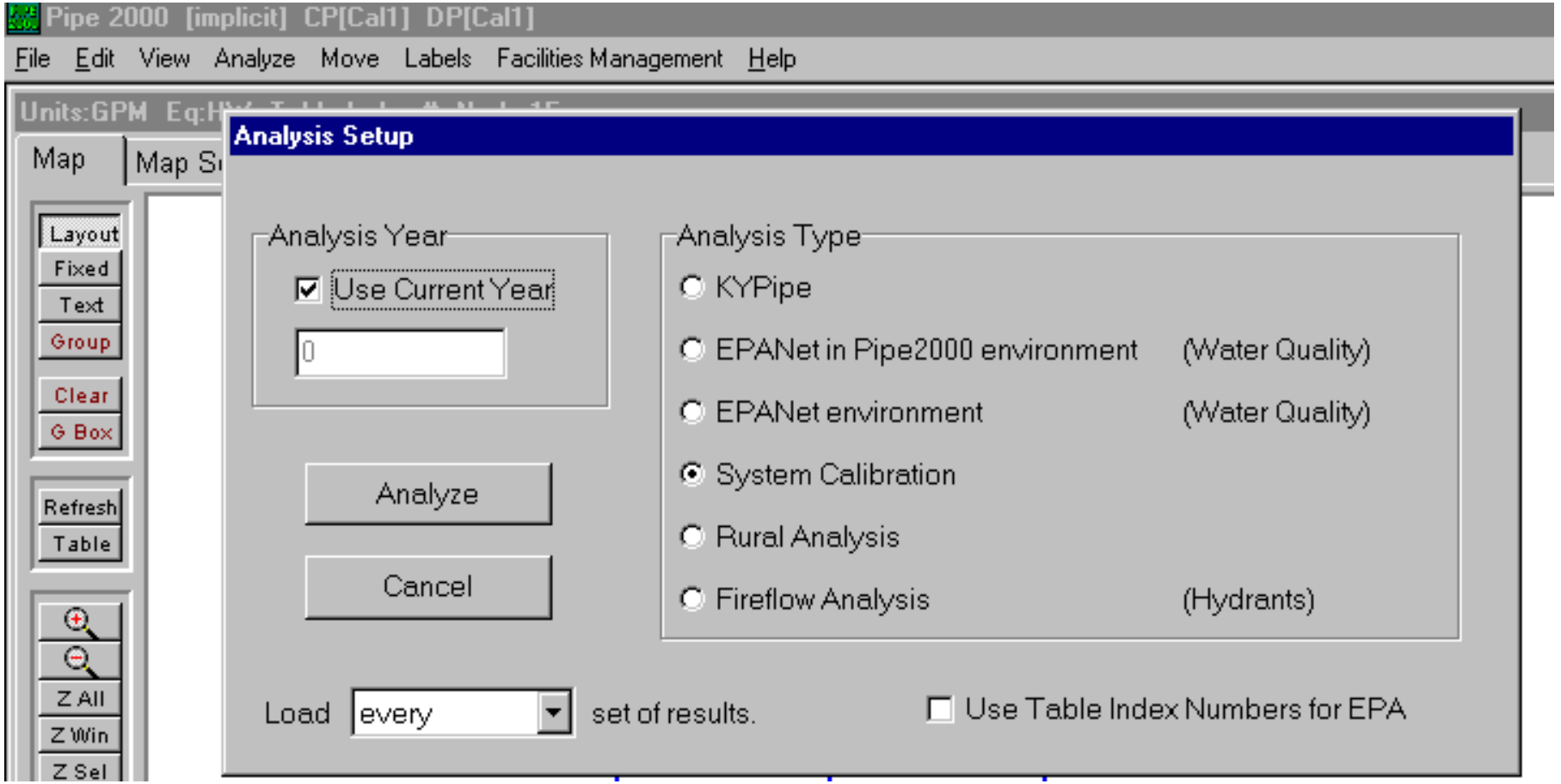



Figure 18. Analysis Setup Menu

Figure 19. Calibration Termination Menu

3.3 Calibration Results

The calibration results for the example problem are shown in Figure 20. As can be seen from the Figure, the calibration results in a roughness coefficient of 120 for pipe group (type) 1 and a roughness coefficient of 90 for pipe group (type) 2. Use of these values of pipe roughness result in values of 14.4 and 15.5 psi (optimal pressures) at junctions 3 and 7 respectively. These values obviously compare very favorably with the actual measured values of 14.4 and 15.4 psi.

```
SUMMARY OF RESULTS :
--------------------

 Percent Deviation between MEASURED and TARGET Values =     .393

 OPTIMAL values for the Decision variables:

 Hazen William coefficients: for group number    1 =  120.  [120.0< > 90.0]
 Hazen William coefficients: for group number    2 =   90.  [100.0< > 70.0]

 Measured and Target pressures (psi or kPa):

   TEST        NODE       MEASURED         OPTIMAL
   CASE       NUMBER      PRESSURE        PRESSURE
 ---------------------------------------------------
     1            3         14.4            14.4
     2            7         15.4            15.5

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

Figure 20. Calibration Results for Example Problem

### 1.3.7.1.5 Optimized Calibration

Optimized Calibration

See also Calibration of Hydraulic Networks

- Calibration Examples
- Optimized Calibration Data

---

What is calibration?

Calibration is a necessary step of tuning the Pipe2018 model to match your physical piping system. This step is necessary because, without calibration, results obtained by computer simulation of your system can differ from actual field readings of your system (frequently by as much as 40% !). A optimally calibrated model will typically produce results within 2% of field measurements. The most frequent cause of differences in the readings is due to inaccurate values for pipe roughness values. The pipe roughness values represent the amount of resistance to flow caused by the interior surface of a pipe. These values for new pipes are very accurate, but values for older pipes are usually extracted from statistical tables that predict how much different pipe materials will change with age. The problem is that the tables are based upon average conditions, and the differing levels of acidity and contaminants in the fluids in individual systems will cause the actual roughness values to greatly differ from the charts. The compilation of the errors in roughness values can easily translate to large errors in system results. In many cases an uncalibrated model provides results not much better than an educated guess.

The traditional approach (empirical) is what we call the GTS iteration method. You first GUESS what the correct roughness values are, you then TRY them in the model, then you SWEAR when the results are still significantly different. This approach is ineffective, not just because it is time consuming, but also because it usually results in an inaccurate calibration. Even if you manage to find roughness values that exactly match your field readings under one set of conditions, you may find that when you close a valve or change the speed of a pump that the model is again producing grossly incorrect results. In order to accurately calibrate a model, the calibration routine must solve the simultaneous equations to ensure that the model is correctly adjusted to match the physical system under all conditions.

Pipe2018 uses an advanced optimization method (implicit) based upon the genetic algorithm approach to optimally adjust pipe roughnesses, valve settings, tank levels, demand distribution, and other data to provide a calibrated model. The program minimizes the difference between observed field data (usually fire flow test data) and model predictions considering all test data simultaneously to provide the best calibration possible. The program directly utilizes the KYPIPE data file with a small amount of additional data (Calibration Data). Pipe2018 can save you a tremendous amount of time and produce better models through optimum calibration. So significant is calibration that two KYPipe developers were awarded the best technical paper of the year (1997) by the AWWA for their paper on the topic.

## 1.3.7.1.6 Optimized Calibration Data

Optimized Calibration Data

Optimized Calibration Example

See also Calibration Wizard

Calibration Examples

Calibration of Hydraulic Networks

Optimized Calibration

---

## Setting Up the Optimized Calibration Run

**What is a Optimized Calibration? O**ptimized calibration adjusts the roughnesses and valve settings (within the bounds you specify) to minimize the differences between model calculations and measured field data (hydrant flows, pressures and pipe flows).

**What field data is required?** Several field measurement tests are required. Each of these tests consists of measured pressures at (or near) junction nodes, pipe flows and hydrant flows. The boundary conditions for each test should be recorded (demands, tank levels, pump and valve status). If your hydrants are not located near to an existing junction node then you should add a junction node in your Pipe2018 model at that location.

**How is the Calibration Data Set Up?** Each field test represents a case. A Pipe2018 run must first be set up where the boundary conditions for each field test (case) is represented by change data. Thus, case 2 represents the boundary conditions (demands, tank levels, pump and valve status) for that field test. The actual field measurements for each test (case) (pressures, hydrant flows and pipe flows) are entered on the Calibration Data Screen shown below in the boxes labeled Junction Pressure Data, Junction Flow Data and Pipe Flow Data. The case number for all the field data is entered so the measured data will be associated with the corresponding boundary conditions. A detailed description of each entry on the Calibration Data Screen is presented below.

**What are Pipe Groups and how are they assigned ?** Each Pipe Type Group referred to in the Roughness Bounds data represents a group of pipes with some common properties such as material, size and/or age. You can assign up to 10 groups and each pipe in a particular group will have their roughnesses adjusted in the same manner (to the same new value or by the same multiplier). The best way to select a pipe group is to go into Group Mode and use the Set Selector feature. Using this feature you can easily select all the 6 inch lines or all the 6 inch PVC pipes or all the pipes with assigned roughnesses between 90-100, etc. It is important to choose logical groups to get a good calibration. Once you select a group then use the Edit Group feature to assign that group a unique Calibration Group number (0-9). When you set the Roughness Bounds the Pipe Type (group) which can range from 0-9 will correspond to your Calibration Group assignments. Make sure the attribute selected for “Pipe Type” is set to Calibration Group which is the the default (top of Calibration Data screen).

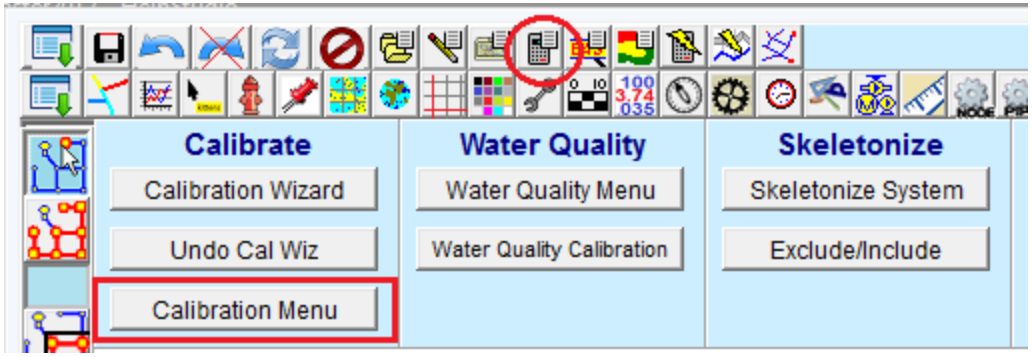
Some Important Considerations

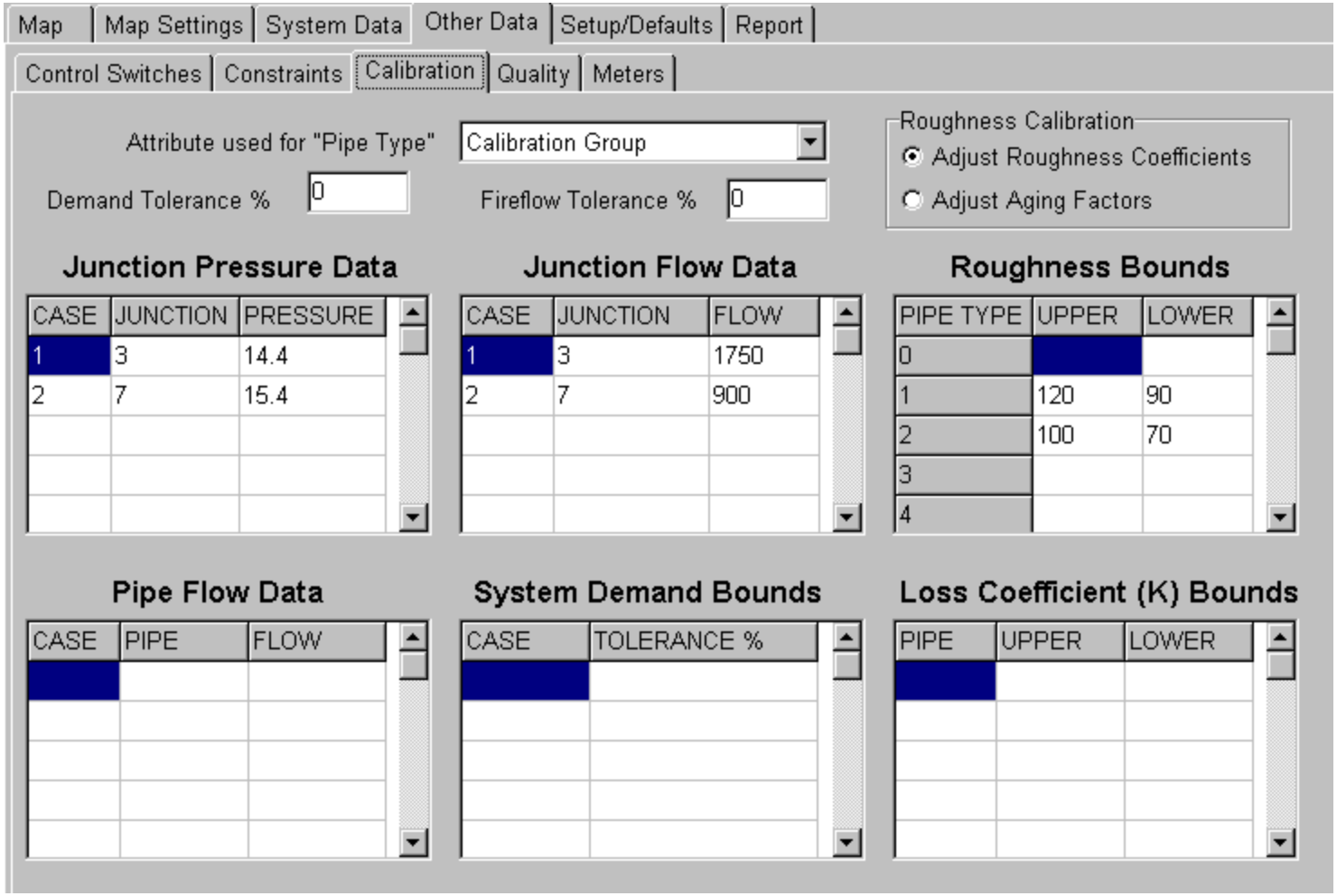
1. The roughness bounds can be absolute bounds (such as 80-120) where the optimization module will find the single best value for the roughness for all the pipes in that group. However, if you enter values for the Roughness bounds from 0-2, then this will be considered a multiplier and will multiply the

assigned roughnesses in that group by a factor within the bounds specified. The advantage of using a multiplier is that the pipes will retain roughnesses that reflect differences based on the judgment applied when the initial (uncalibrated) roughnesses were assigned. For example, if 2 pipes in the same pipe group were initially assigned roughness of 90 and 110 (because of age differences), they will be adjusted to a single new value within the bound (say 93) if the absolute bound is applied. However, if a multiplier is used the adjusted values may end up as 81 and 99 (for a multiplier of 0.9) still reflecting the difference in roughness factored into the initial assignment.

2. If you carry out a calibration and all but one or two field measurements are in good agreement, you may want to repeat the calibration without using this data. The poor agreement may be an indication that the data is flawed.

3. Before you run the optimized calibration, but after you have set up the calibration run, which includes Change Data that reflects the different test case boundary conditions, you may wish to make a run to determine how well the uncalibrated model predicts the field test results. This will also establish the improvement due to the calibration. To do this, just add the hydrant flows as demand changes to the Change Data for each test. The calculated junction pressure and pipe flows may then be compared to the corresponding field measurements. After calibration, these same results will be compared for the calibrated model.






**Attribute used for "Pipe Type"** - The default entry is 'Calibration Group'. This entry is used to tell the program how to distinguish one group of pipes from another in the subsequent pipe roughness bounds. Instead of using the normal calibration groups (as discussed previously), the user could use the constraint grouping associations (as in the explicit approach) to designate the associated calibration groups.

**Demand Tolerance %** - The Demand Tolerance % cell is provided to allow the user to specify a calibration tolerance associated with the total system demand. When a non-zero value is specified, the calibration algorithm will attempt to make adjustments to the total system demand (within the specified tolerance) in an attempt to decrease the deviation between the observed and model predicted state values (e.g. pressures and flows). Normally, it will be expected that the tolerance will be zero, that is, the system demand is completely known. However, in some situations, there may be some uncertainty associated with the system demand measurements and in that case the uncertainty may be taken into account via the demand tolerance. As an example, if the user were to specify a system

demand of 1 MGD with a tolerance of 5% then the calibration algorithm would allow the total system demand to vary between .95 MGD and 1.05 MGD during the calibration process.

**Fireflow Tolerance %** - The Fireflow Tolerance % cell is provided to allow the user to specify a calibration tolerance associated with each individual "fire-flow" observation. When a non-zero value is specified, the calibration algorithm will attempt to make adjustments (within the specified tolerance) to any "fire-flow" values (i.e. as specified in the Junction Flow Data in an attempt to decrease the deviation between the observed and model predicted state values (e.g. pressures and flows).

**Roughness Calibration** - The Roughness Calibration menu is used to specify whether one wants to calibrate the pipe roughness coefficients (the default), or individual aging factors. In order to determine individual aging factors (for an associated group of pipes), the user must first specify the pipe roughness for each pipe (in the regular pipe data) on the basis of observed values 10 years ago. The program will then determine the associated aging factors that will produce the existing field observations (e.g. pressures and flows). Once these factors are obtained, they can then be used to predict future roughness values.

**Junction Pressures Data** - The Junction Pressures menu is used to specify observed pressures associated with selected junction nodes. The first column of the menu is used to identify which set of change data the junction pressure is to be associated with. The second column is used to specify the selected junction number while the third column is used to specify the observed pressure (psi or kpa). The user may specify up to four different pressure observations per set of change data. For the example problem, two separate pressure readings were obtained, one for each different set of boundary conditions. As a consequence, the junction pressure menu contains two separate pressure readings with each one associated with a different change (or boundary condition).

**Junction Flow Data** - The Junction Flow Data menu is used to specify the observed flowrates associated with the junction pressures that are input in the Junction Pressure menu. It should be emphasized that the junctions input in the junction flow menu do not have to correspond to the junctions input in the pressure menu. For example, fire flow test data for a particular field observation may involve a measured flow from one junction with a residual pressure measured at another junction. In addition, there does not have to be a one to one correspondence between the number of junction nodes in each menu. For example, a user may input an observed pressure from a single junction node with fire flows from multiple junctions. Conversely, a user may specify flow from a single junction with pressures measured at multiple junctions.

As with the Junction Pressures Menu, the first column is used to identify which set of change data the junction flow is to be associated with. The user may specify up to four different flow observations per set of change data. The second column is used to specify the selected junction number while the third column is used to specify the observed flowrate. For the example problem, one flow reading was obtained for each of the observed pressures recorded in the Junction Pressure Menu.

**Roughness Bounds** - Once the various field observation data has been input, the user may specify bounds or limits on the values that the decision variables (i.e. pipe roughness or nodal demands) may assume. The Roughness Bounds menu is used for setting bounds on the values of the roughness coefficient associated with each pipe group. The first column of the menu is used to identify the number of the particular Pipe Type group. The next two columns are then used to specify both upper and lower bounds for the pipe roughness coefficient associated with that pipe group. Bounds may be expressed in terms of actual Hazen Williams roughness values, 40-140, or if a number less than 0.5 is used, it will be treated as a multiplier. For the example problem, upper and lower values of 120-90; and 100-70 were assumed for pipe type groups 1 and 2 respectively.

**Pipe Flow Data** - In addition to the use of junction pressures, the user may also elect to use the calibration model to adjust the model parameters to match observed flowrates in specified pipes. The data usually comes from a pipe containing a flow meter. This data may be entered using the Pipe Flow Data menu. As with the previous menus, the Pipe Flow Data menu has three columns. The first column is used to identify which set of change data the pipe flow is to be associated with. The user may specify up to four different flow observations per set of change data. The second column is used to specify the selected pipe number while the third column is used to specify the observed flowrate. For the example problem, no pipe flow observations were obtained and as a result, no values are included in the Pipe Flow Data menu.

**System Demand Bounds** - The System Demand Bounds menu is used to set the total system demand for each set of boundary conditions (i.e. each change data set). In the event that the demand is left blank, the program will determine the total system demand on the basis of the sum of the initial nodal demands along with whatever demand adjustments are made in the change data. The first column in the System Demand Bounds menu is used to specify the set of boundary conditions (i.e. change set) to be associated with the total system demand that is to be entered in the second column. A global tolerance value for these values may be specified in the Demand Tolerance % Cell as discussed in section 3.1.2 of "Calibration of Hydraulic Networks."

**Loss Coefficient (K) Bounds** - The Loss Coefficient Bounds menu is used for setting bounds on the values of minor loss coefficients associated with a pipe. The first column of the menu is used to identify the number of the particular pipe. The next two columns are then used to specify both upper and lower bounds for the pipe roughness coefficient associated with that pipe, 0-100. These parameters may be used in addition to or as an alternative to adjustments to pipe roughness coefficients.

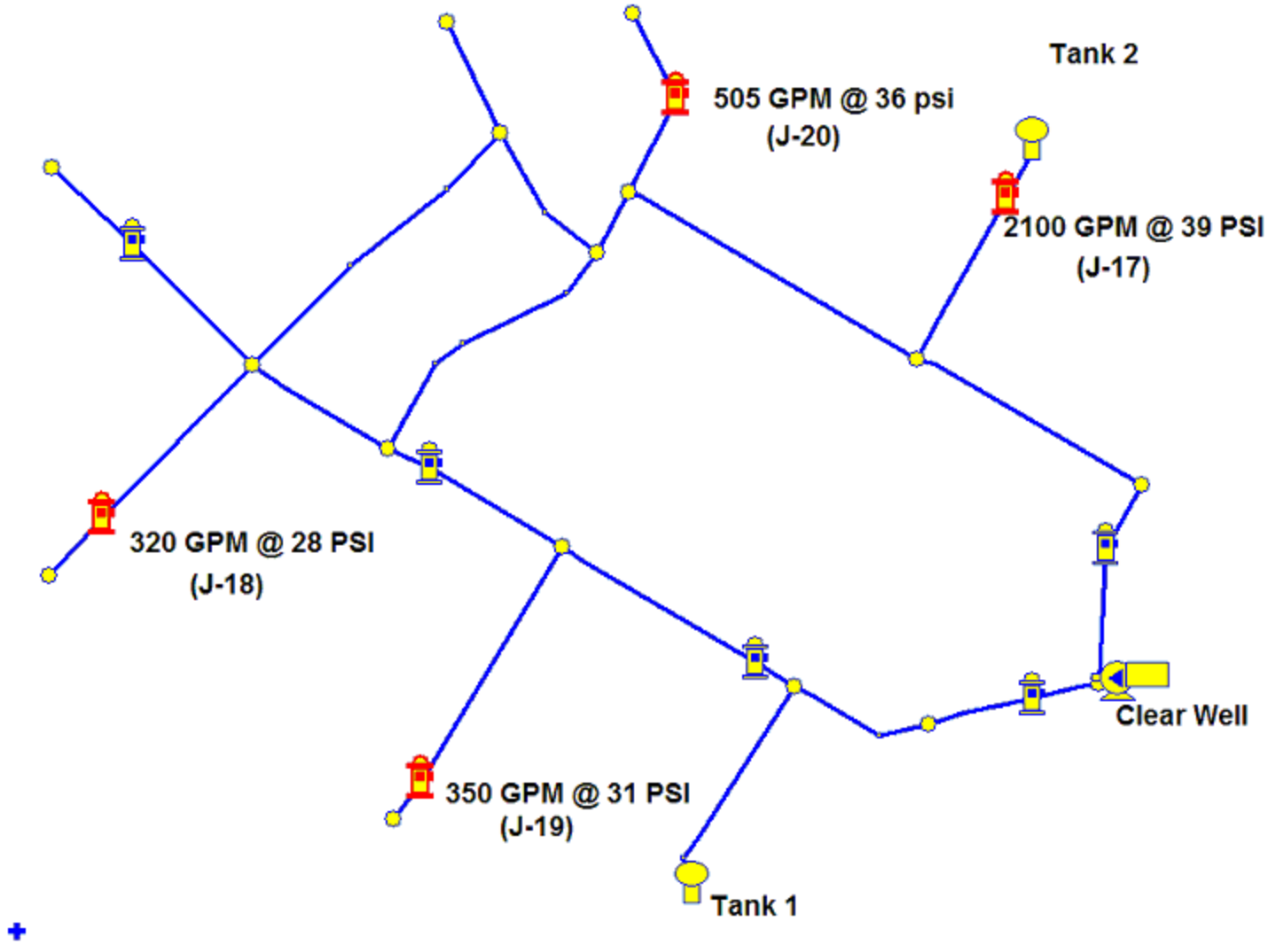
To set up a Calibration Group, see Sets and Groups

**Optimized Calibration Example presentation**

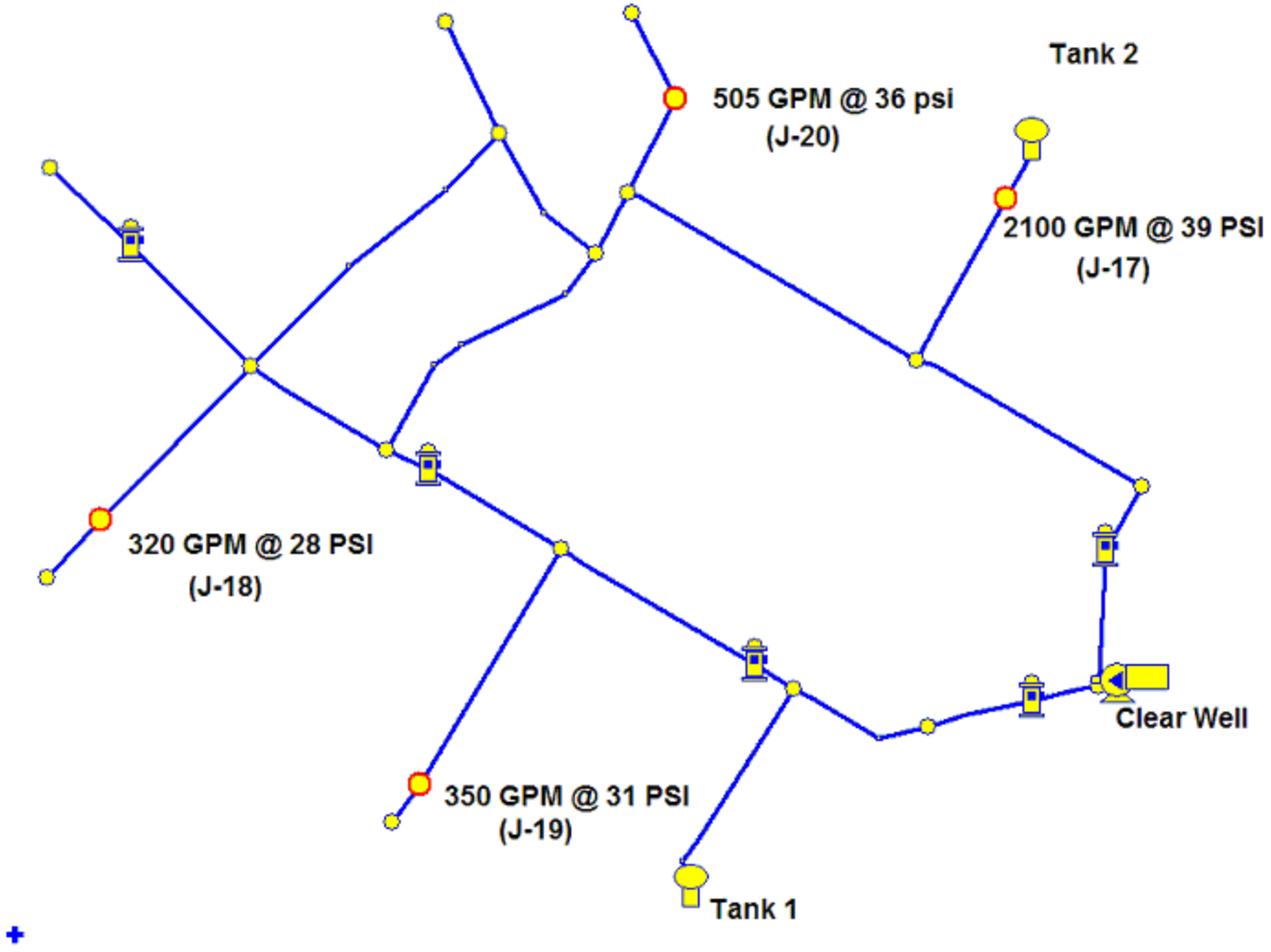
# System Calibration

1) Utilize fire flow data
   - Static pressure
   - Residual pressure and flow
   - Pipe flows
2) Adjust pipe roughness, valve settings
3) Perform Pipe2000 GA Optimization

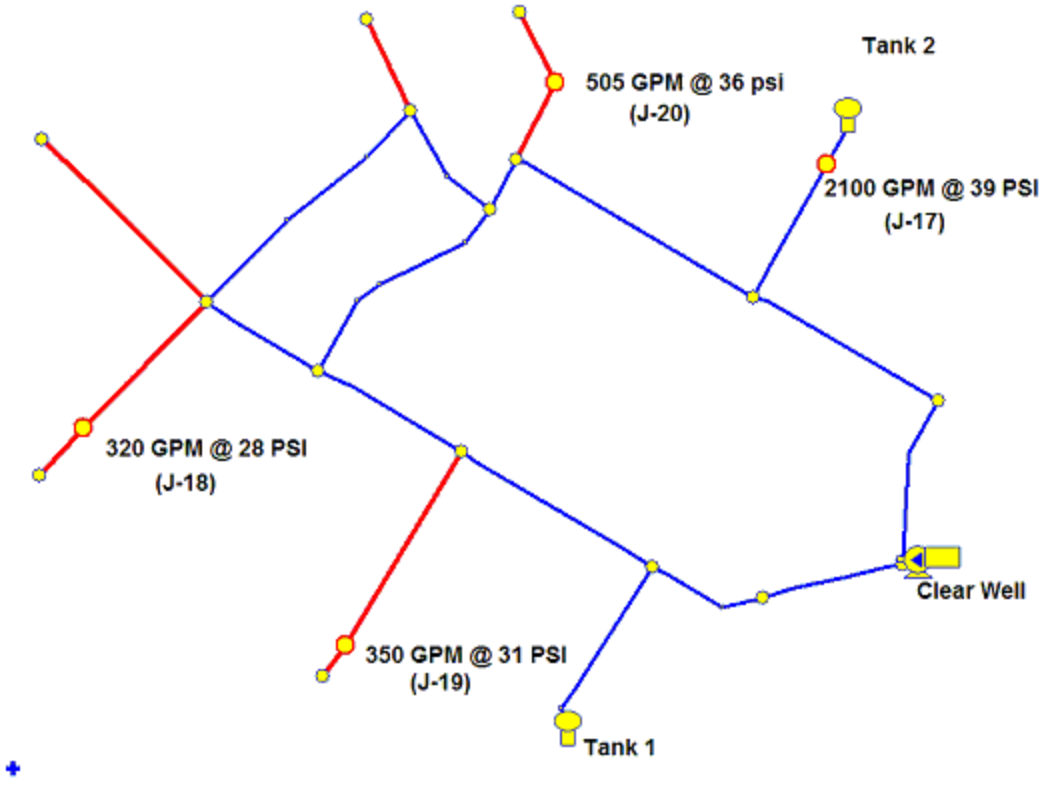
# Fire Flow Data for Calibration

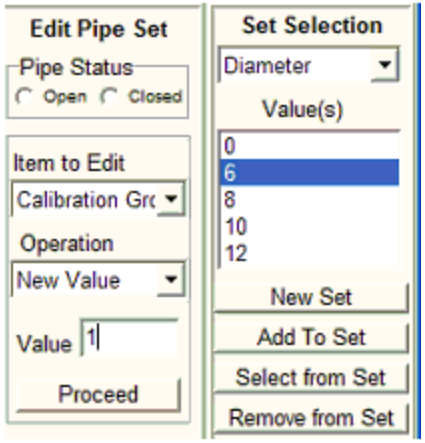


# Change Test Hydrants to Junctions

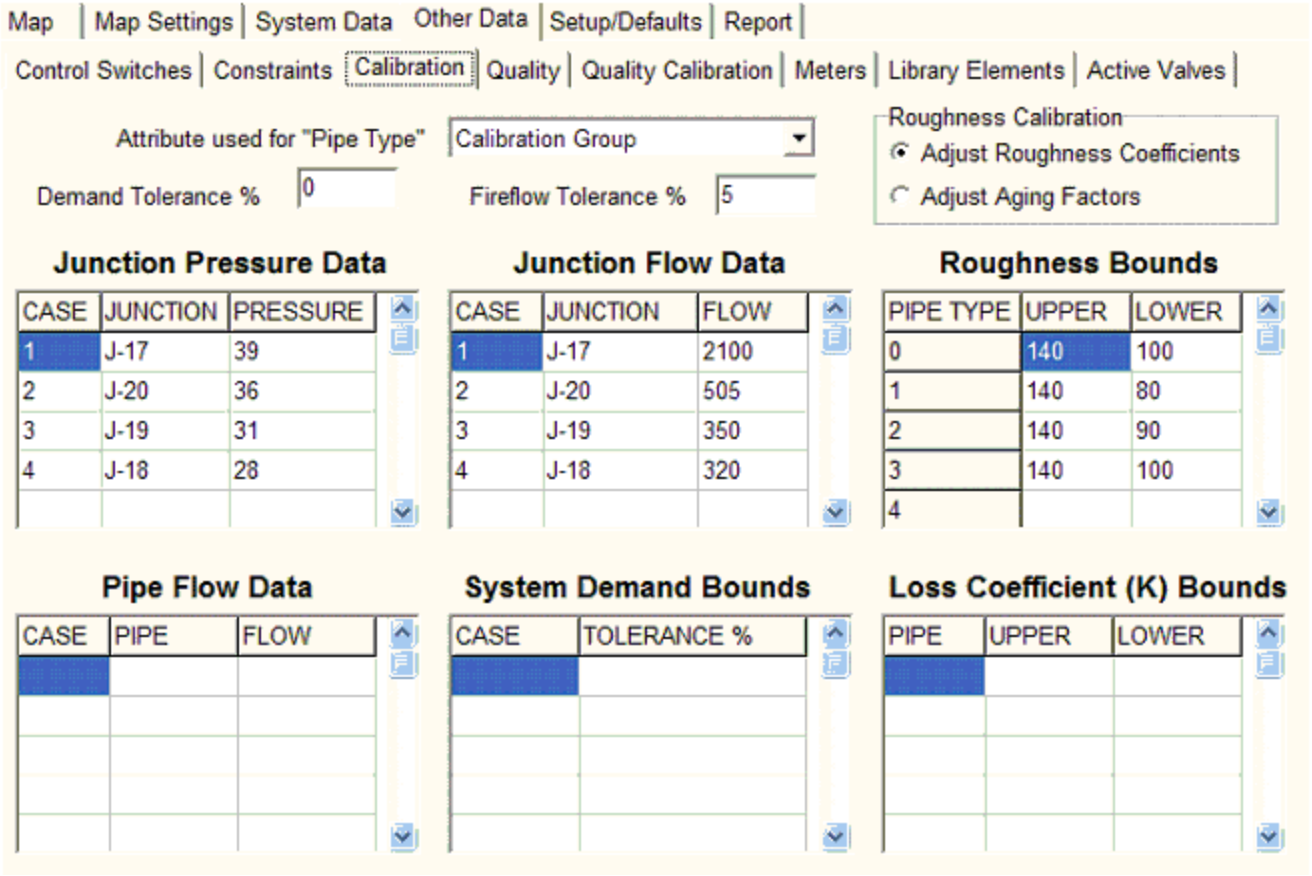


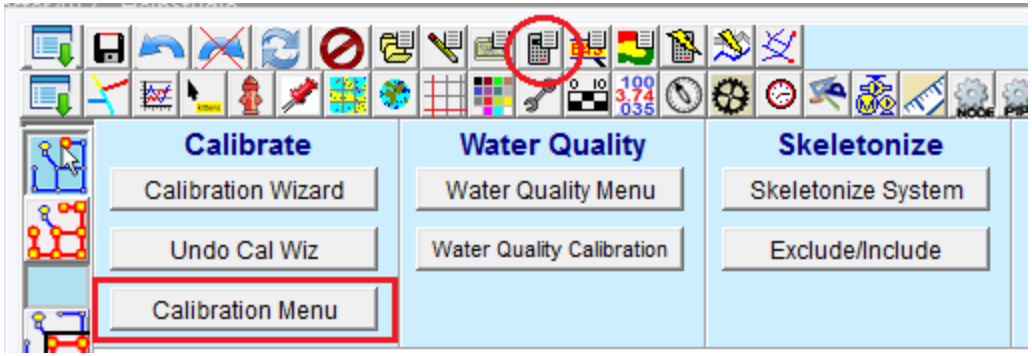
# Select Pipe (calibration) Groups



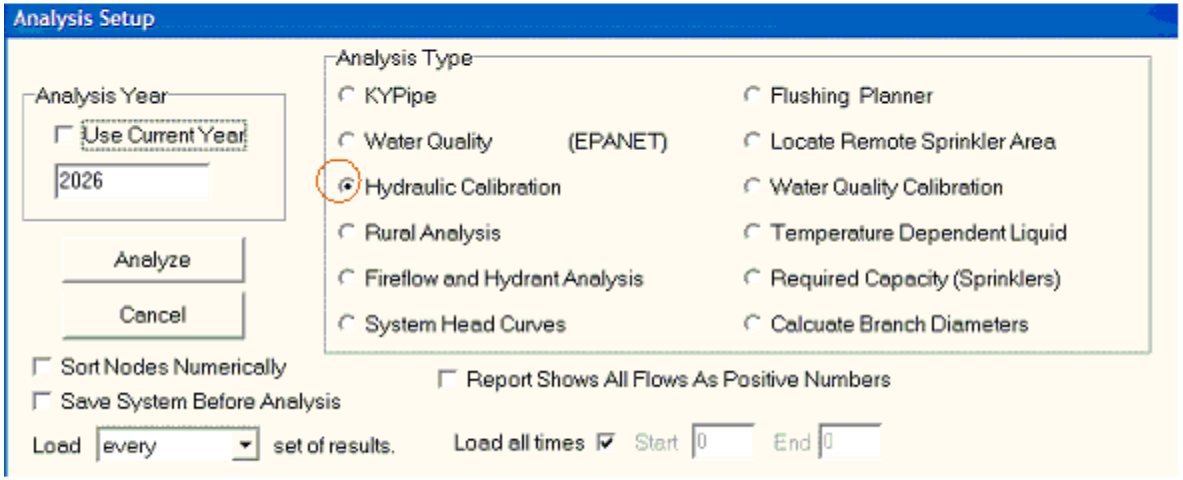



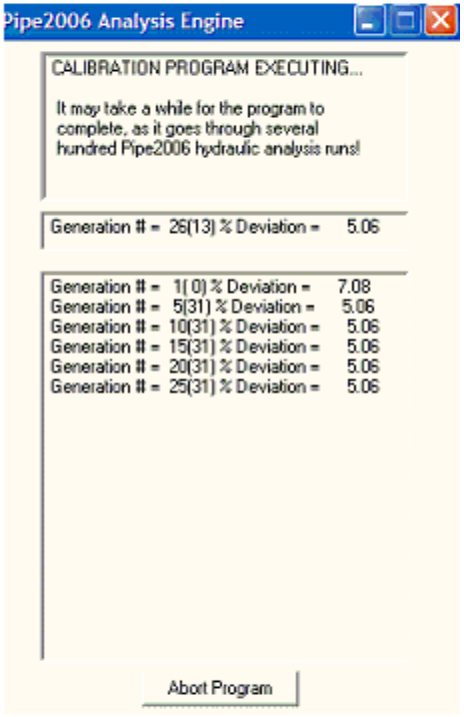
# Provide Calibration Data




Calibrate
Calibration Wizard
Undo Cal Wiz
Calibration Menu
Water Quality
Water Quality Menu
Water Quality Calibration
Skeletonize
Skeletonize System
Exclude/Include

# Run Hydraulic Calibration (GA)



# Results – 5% Fire Flow Tolerance

```
SUMMARY OF RESULTS :
--------------------


 Percent Deviation between MEASURED and TARGET Values =   1.442


 OPTIMAL values for the Decision variables:

 Hazen William coefficients: for group number    0 =  100.  [140.0< >100.0]
 Hazen William coefficients: for group number    1 =  107.  [140.0< > 80.0]
 Hazen William coefficients: for group number    2 =   92.  [140.0< > 90.0]
 Hazen William coefficients: for group number    3 =  104.  [140.0< >100.0]

 No demand adjustments are made.
 Demand Tolerance is meant for re-distributing
 demands among nodes of diff demand types, keeping
 the total demand constant. There must be at least
 TWO types of demands to use this feature.

 Junction (Fire) Flow(s) for Change  1 are INCREASED by  3.71%
 Junction (Fire) Flow(s) for Change  2 are DECREASED by  5.00%
 Junction (Fire) Flow(s) for Change  3 are DECREASED by  2.42%
 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           39.1
      2          J-20         36.0           34.3
      3          J-19         31.0           31.1
      4          J-18         28.0           28.4
```

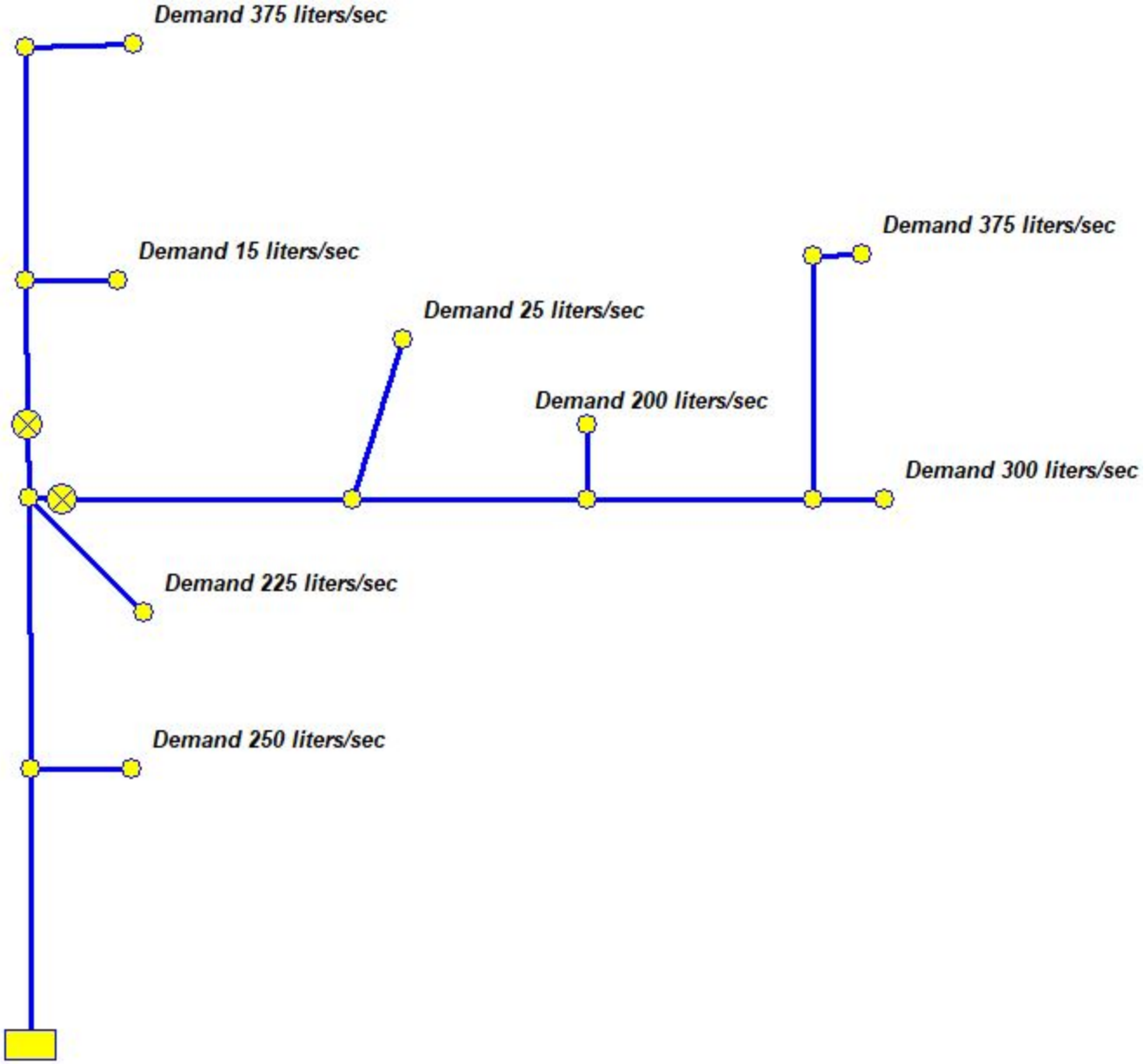
# Results – 0% Fire Flow Tolerance

```
SUMMARY OF RESULTS :
--------------------


 Percent Deviation between MEASURED and TARGET Values =   5.060


 OPTIMAL values for the Decision variables:

 Hazen William coefficients: for group number    0 =  100.  [140.0< >100.0]
 Hazen William coefficients: for group number    1 =  121.  [140.0< > 80.0]
 Hazen William coefficients: for group number    2 =  101.  [140.0< > 90.0]
 Hazen William coefficients: for group number    3 =  100.  [140.0< >100.0]



 Measured and Target pressures (psi or kPa):

    TEST         NODE       MEASURED         OPTIMAL
    CASE        NUMBER      PRESSURE        PRESSURE
  -------------------------------------------------
      1          J-17         39.0            40.6
      2          J-20         36.0            33.9
      3          J-19         31.0            31.0
      4          J-18         28.0            32.2

 Date & Time:  Sun Sep 24 14:00:04 2006
```

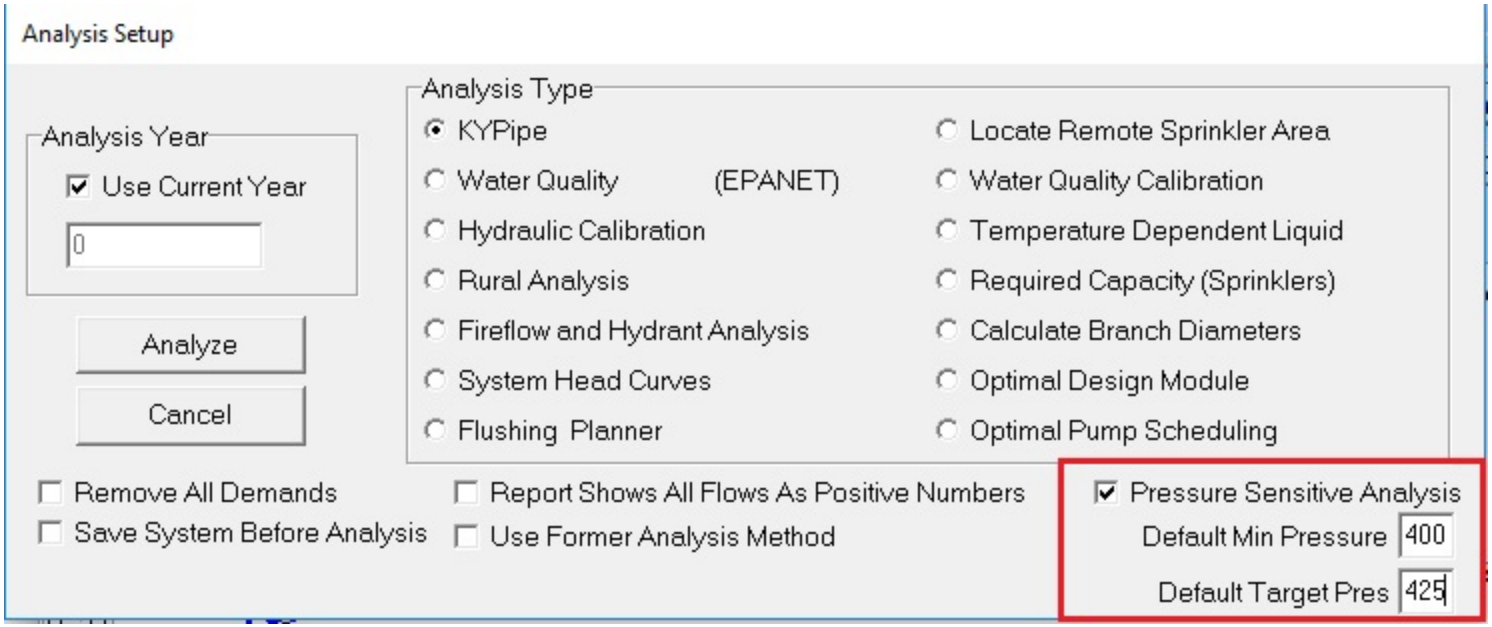
### 1.3.7.2 Pressure Sensitive Analysis

Demands in a system can be evaluated to see if a minimum and target pressure can be achieved.

Here is an example system with expected demands:

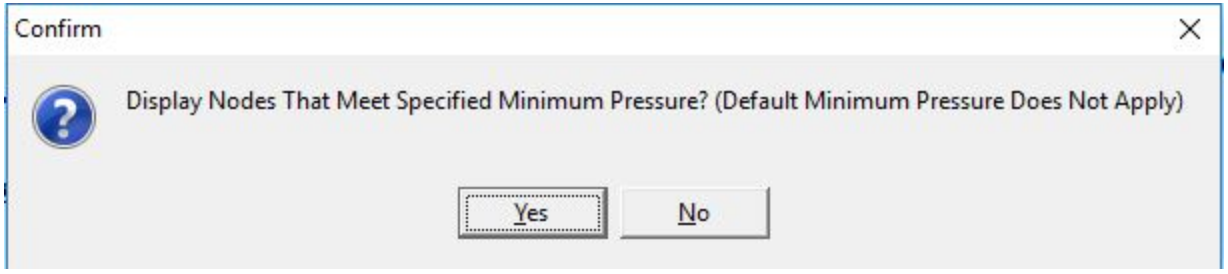

Demand 375 liters/sec
Demand 15 liters/sec
Demand 375 liters/sec
Demand 25 liters/sec
Demand 200 liters/sec
Demand 300 liters/sec
Demand 225 liters/sec
Demand 250 liters/sec

The Pressure Sensitive Demand analysis option is selected and set up:



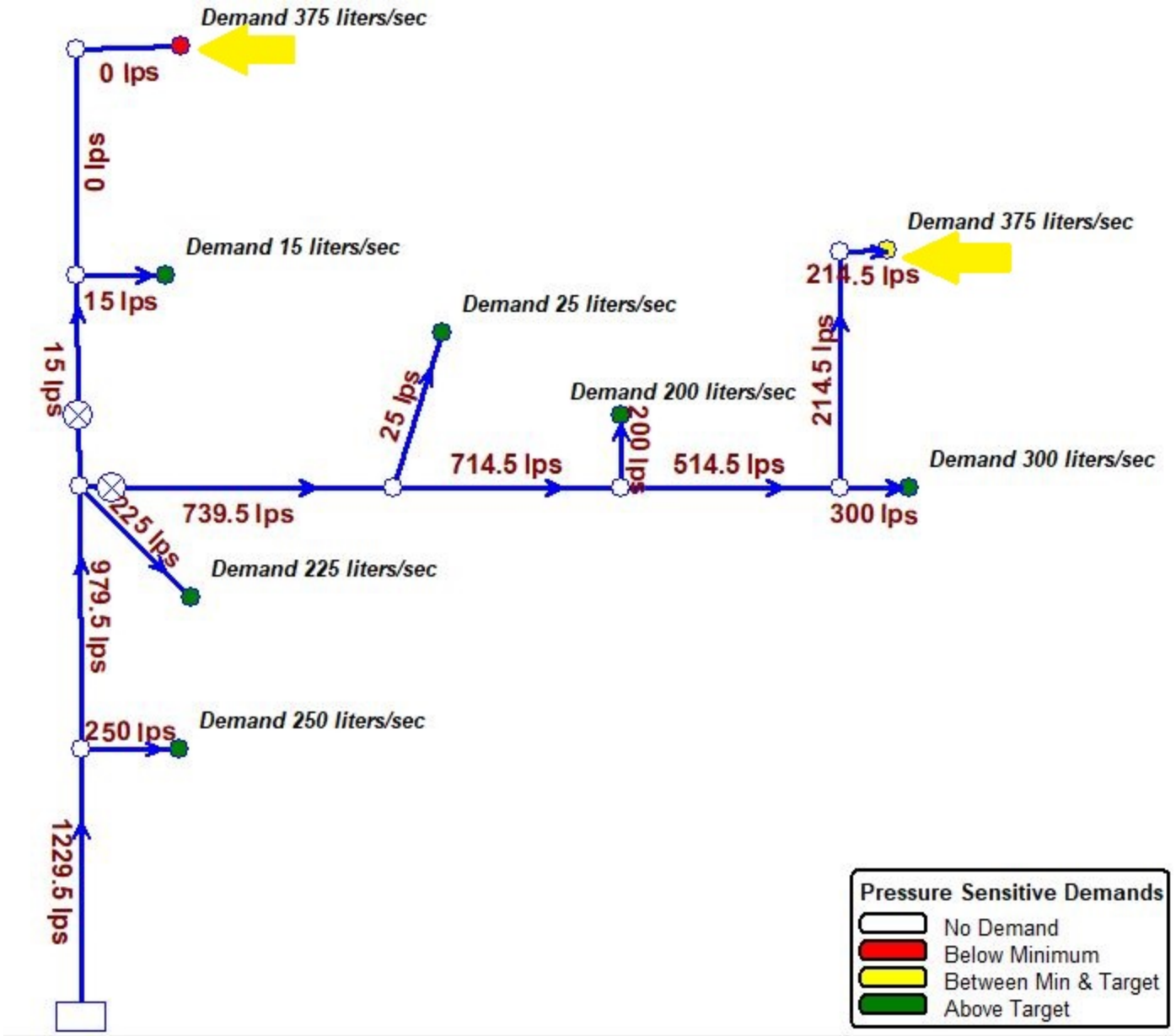

A Minimum Pressure and a Target Pressure are identified. Then the analysis is run.

An option to emphasize (color-code) the results is given:



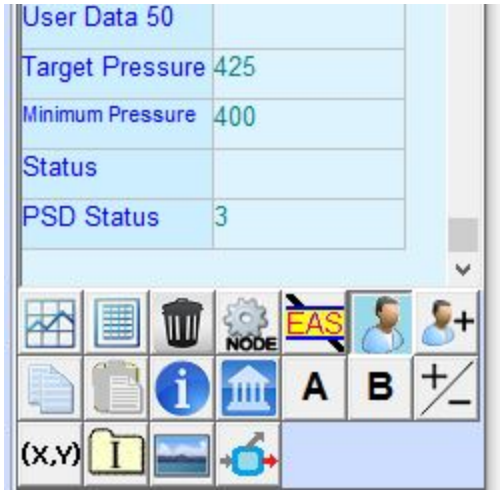

The results may be emphasized as shown below. In this example one node is below the minimum pressure so the flow is set to zero. Another node falls between the target and minimum pressure, so the flow rate shown is the amount that can be attained without falling below the minimum pressure, but is less than the original Demand value.

Note that Pipe Result A labels for flow are also displayed in this image:

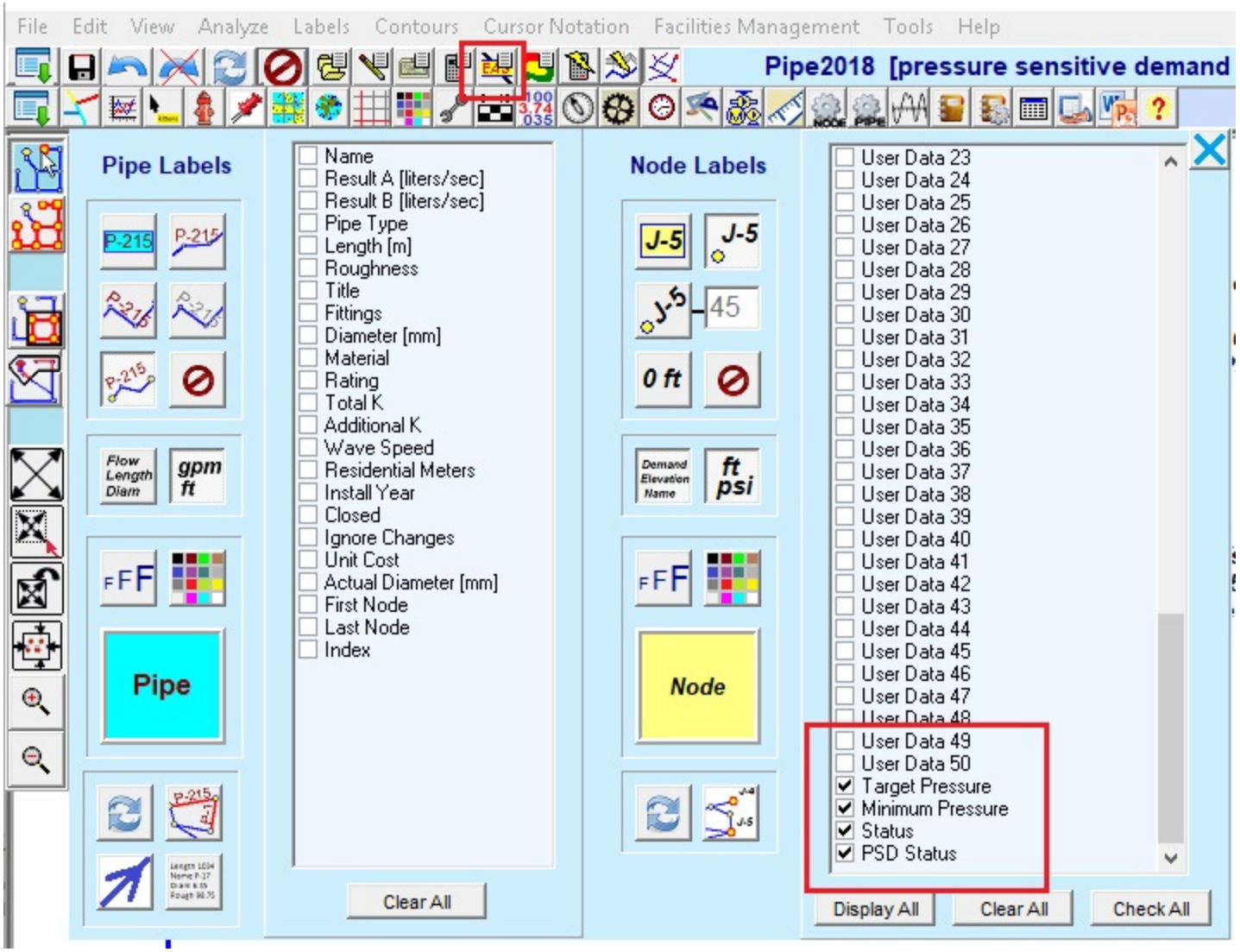

Demand 375 liters/sec
0 lps
0 lps
Demand 15 liters/sec
15 lps
15 lps
Demand 25 liters/sec
25 lps
Demand 200 liters/sec
200 lps
714.5 lps
514.5 lps
Demand 375 liters/sec
214.5 lps
214.5 lps
Demand 300 liters/sec
300 lps
739.5 lps
225 lps
Demand 225 liters/sec
979.5 lps
250 lps
Demand 250 liters/sec
1229.5 lps
Pressure Sensitive Demands
No Demand
Below Minimum
Between Min & Target
Above Target

Optional settings:

Target and Minimum Pressures may be specified at individual nodes using User Data attibutes called Target Pressure and Minimum Pressure as shown:



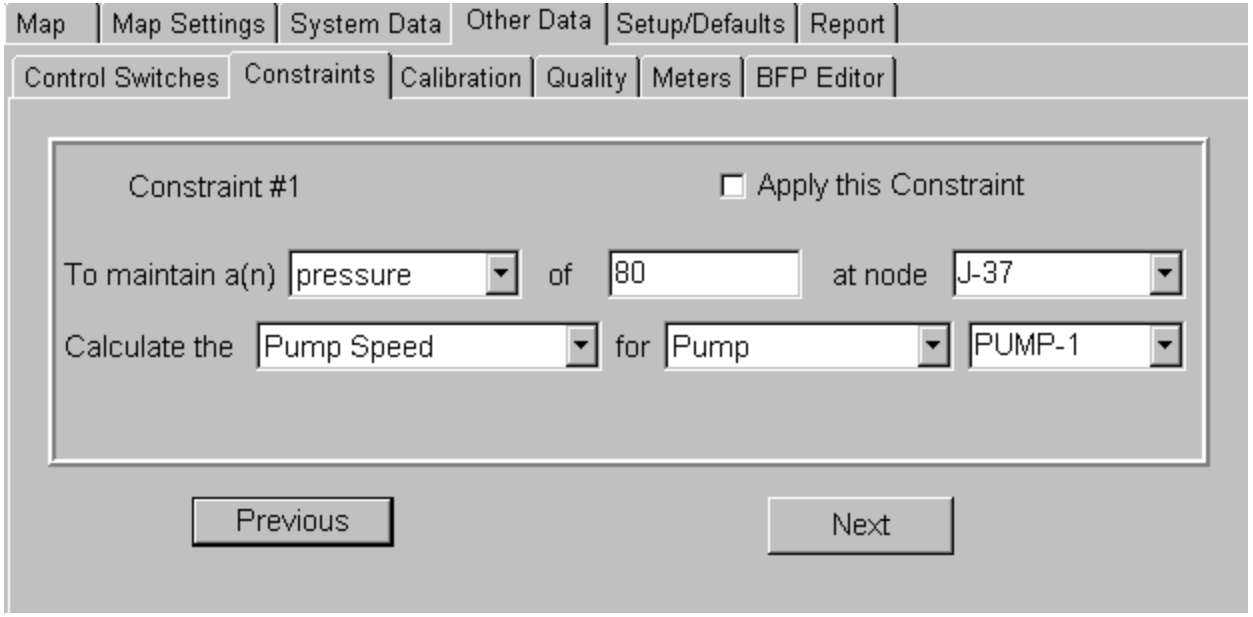

Additionally, labels existing for Target Pressure, Minimum Pressure, and node Status (Below, Between, or Above Minimum Pressure) .




## 1.3.7.3 Constraints Data

Constraints Data

KYnetic Design Tools menu icon:




See the Constraints Video on the KYPipe web site.

---

## What are constraints?

The ability of the KYPIPE engine to carry out direct parameter calculations provides a major step forward for network modeling. With this capability you can rapidly carry out a variety of calculations which previously required a repetitious trial and evaluation approach or were not even practical to attempt. Constraints refers to the extremely powerful KYPIPE modeling capability to calculate a variety of design or operating parameters to exactly produce a designated pressure at the designated node. To utilize this capability the following steps are required.

1. Select units (pressure, head, or HGL)

2. Provide values to be maintained.

3. Select a junction node where pressure (Head or HGL) is to be maintained.

4. Choose the parameter to be calculated from the following;

pump speed

pump power

Tank / Reservoir Setting (HGL)

PRV Setting (HGL or pressure)

Valve Setting (loss coefficient)

Pipe Diameter

Pipe Roughness (or global factor)

Demand (or global factor)

5. Choose an appropriate pipe, node, or group for the parameter calculation.

For example, you can directly calculate the pump power required for Pump-1 to maintain a pressure of 80 psi at junction J-37. More than one constant can be set up. **Warning:** you may attempt to set up constraint(s) which are not feasible and the solution will not be attained.

If a group is selected then two additional entries are required

1. Group name (usually constraint group)

2. Attributes for items to be used.

Pipe2018 provides the capability to set up and recall groups and a Constraint. Group data entry is provided for all nodes and pipes. You should assign a common integer to any groups you wish to access. This provides a convenient means of identifying groups for setting up constraints.

See also Direct Parameter Calculation

Constraints

Sets and Groups

For more detail on constraints please see Direct Parameter Calculation - Constraints.

## 1.3.7.4 Cost and Inventory Calculations

Cost and Inventory Calculations

Power Cost

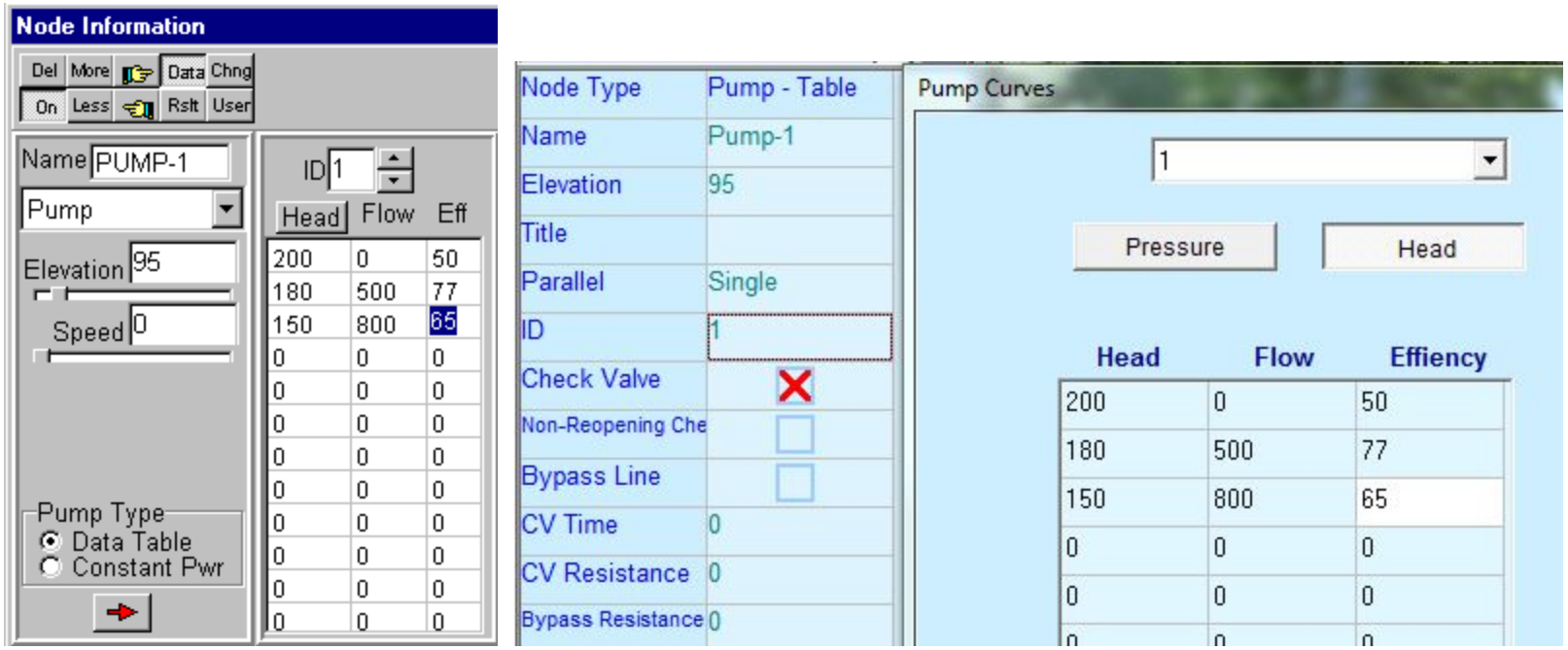
Inventory/Cost

See the Cost Video on the KYPipe web site.

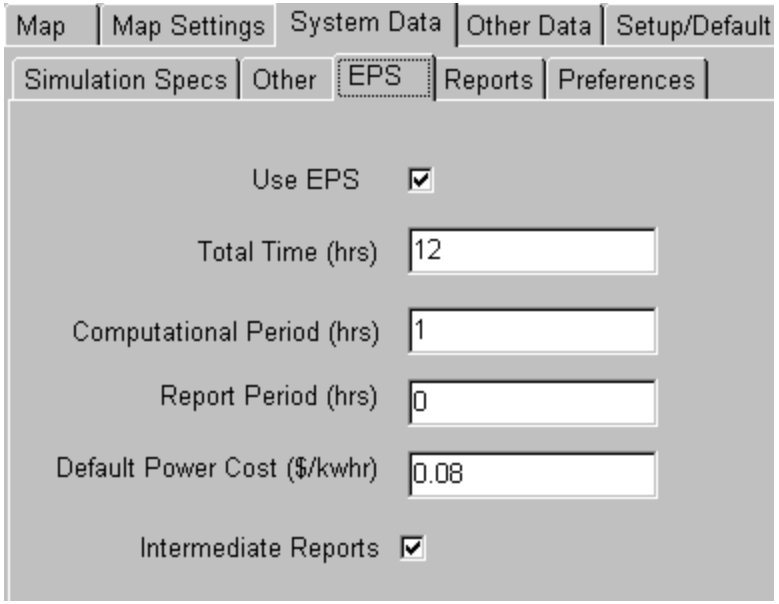
---

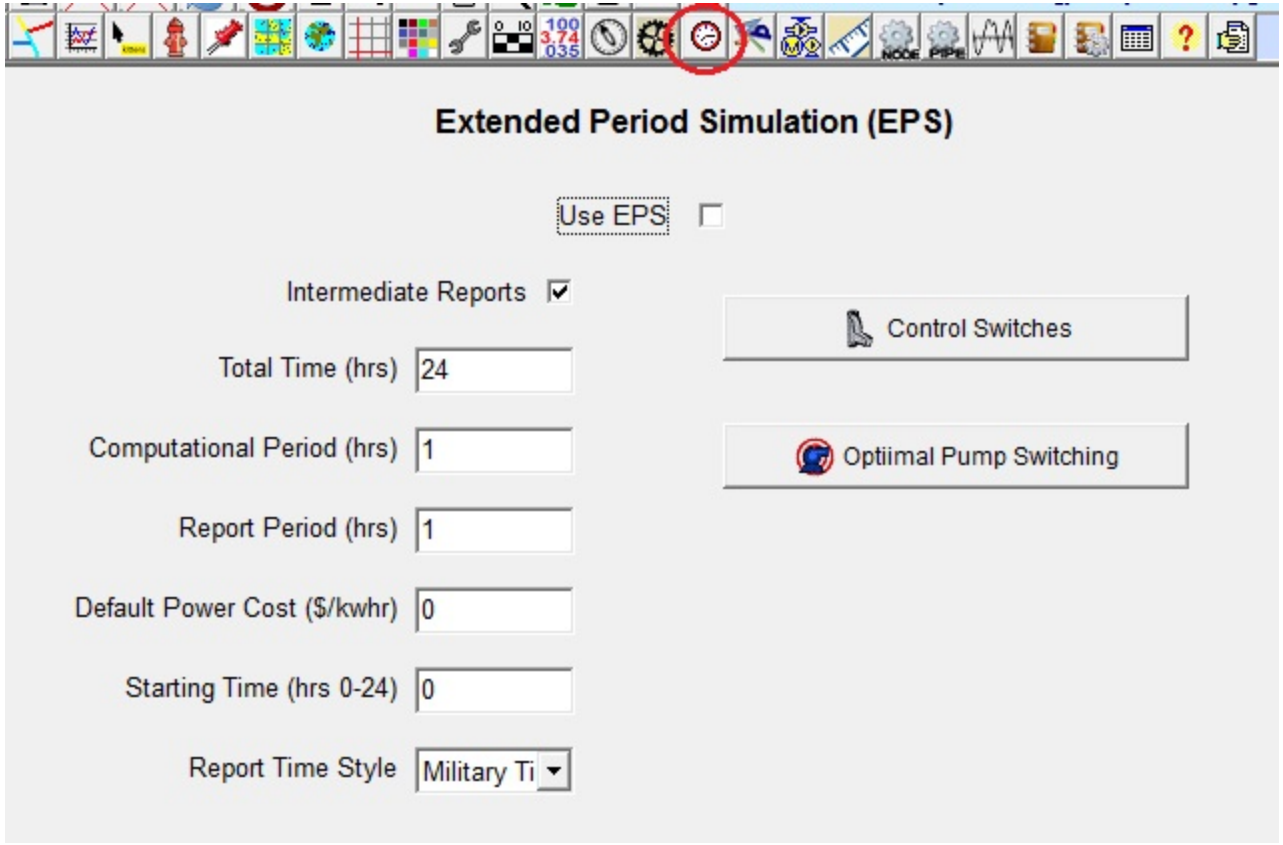
### Power Cost

A Power Cost calculation may be set up as part of an Extended Period Simulation (EPS). To calculate the power cost for pump operations, select (highlight) the pump of interest. In the Node Information window, a percent Efficiency may be entered along with the ID data describing the pump.




Under the System Data, EPS tabs, a default power cost ($/kwhr) is specified. The Use EPS box must be checked. Other EPS specifications are also entered for the simulation.

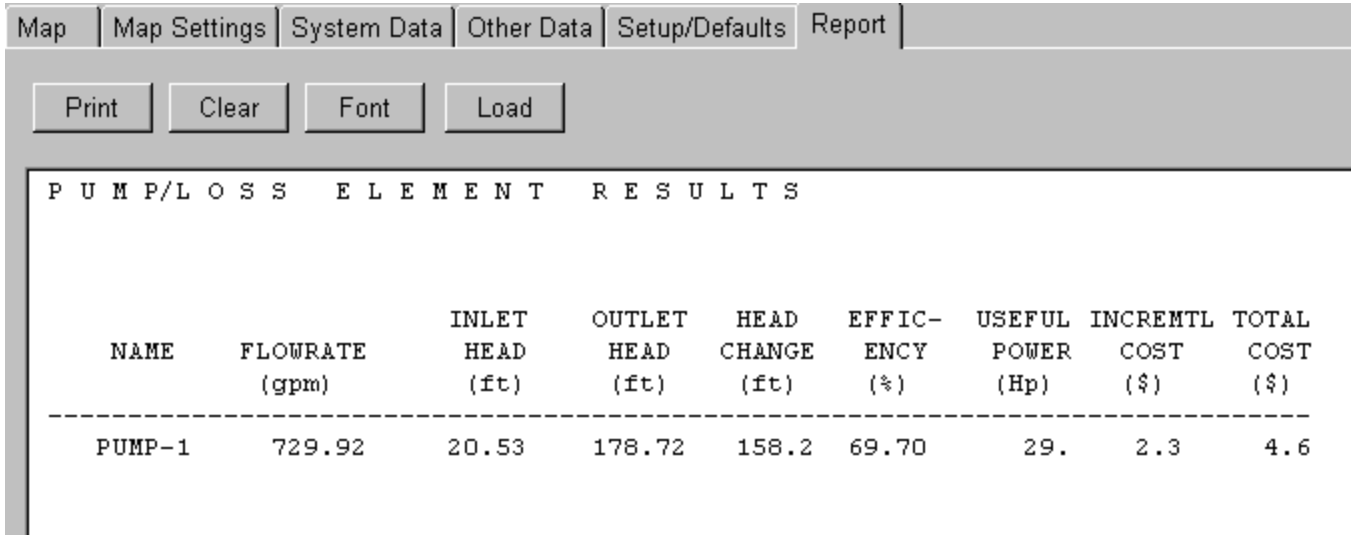

Map
Map Settings
System Data
Other Data
Setup/Default
Simulation Specs
Other
EPS
Reports
Preferences
Use EPS
Total Time (hrs)
12
Computational Period (hrs)
1
Report Period (hrs)
0
Default Power Cost ($/kwhr)
0.08
Intermediate Reports




A Power Cost analysis is automatically conducted with an EPS simulation when a cost is specified. Therefore, to see Power Cost calculations results, select Analyze under Analyze in the Main Menu.

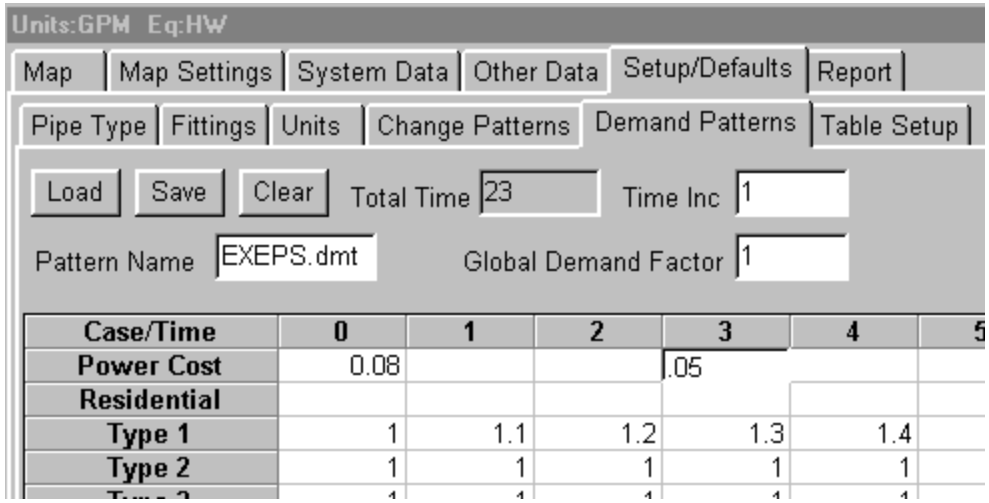
The results may then be reviewed in the Report (under the Report tab). The cost for each computational period (incremental cost), cumulative total cost, and the total cost for the simulation are all reported.

KYnetic Report icon:




Variable rate power costs (for different time periods or computational periods) may be simulated by specifying the rate data under the Setup/Defaults, Demand Patterns tabs. A row called Power Cost is provided for this data.

KYnetic Demand Pattern icon:

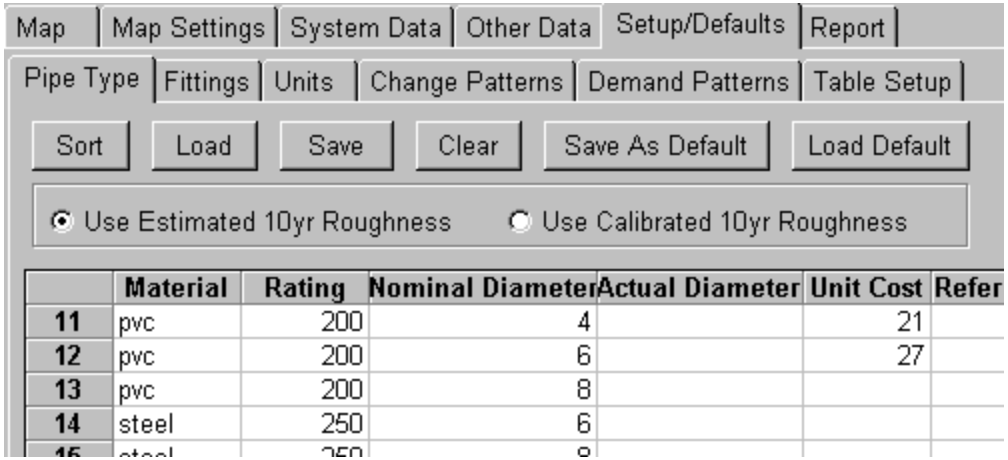



## Inventory/Cost

**Specify the material/installation cost for pipes, nodes and fittings. The create an Inventory to see the total cost.**

**Pipes -** A Unit Cost may be specified under the Setup/Defaults, Pipe Type tabs for each type of pipe in the system. If English units have been selected for the model, this value is cost per foot of pipe. For SI units, this is cost per meter.

KYnetic Components menu icon:

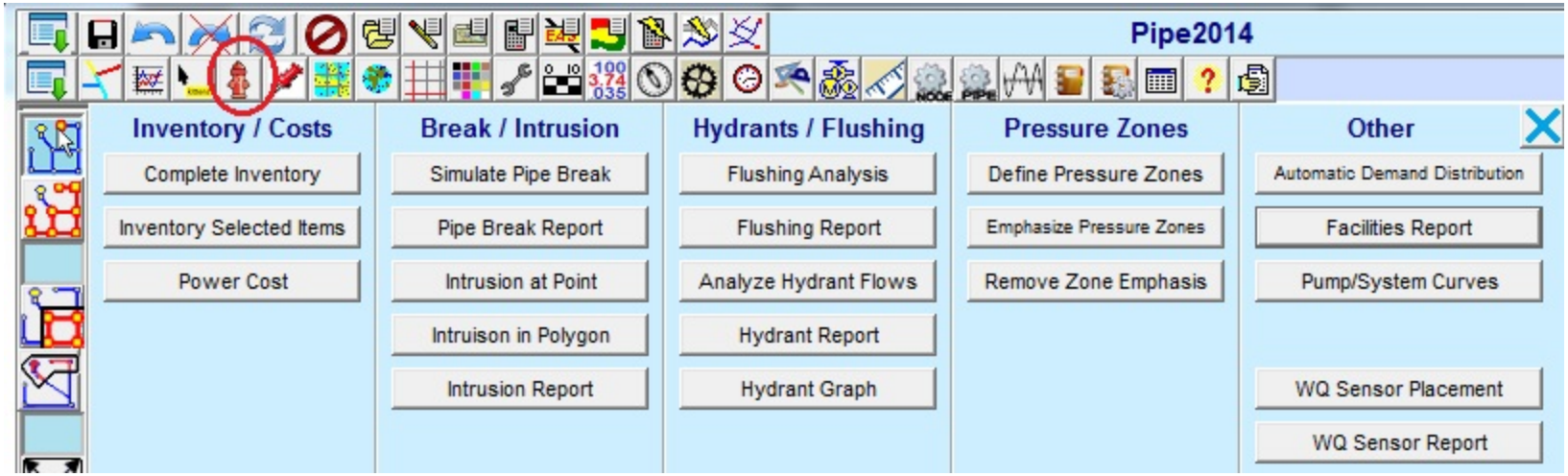



| | Material | Rating | Nominal Diameter | Actual Diameter | Unit Cost | Refer |
|---|---|---|---|---|---|---|
| 11 | pvc | 200 | 4 | | 21 | |
| 12 | pvc | 200 | 6 | | 27 | |
| 13 | pvc | 200 | 8 | | | |
| 14 | steel | 250 | 6 | | | |

**Nodes** - Create a User Data item called "Cost". Displaying User Data in the Node Info window, click on the node and input the total cost for that device (Pipe2014 and later versions).

**Fittings** - In the Fittings Table, (under Setup/Default in the Classic Interface and the Components menu in KYnetic) input a total cost for the fitting (Pipe2014 and later versions).

To run the analysis, select Facilities Management icon (KYnetic) or Inventory/Cost under Analyze in the Main Menu (Classic).




The results along with an inventory report are then viewed in the Report under the Report tab.

## Inventory/Cost Summary

| Pipe Type | Number | Total Length | Cost/Unit | Total Cost |
|---|---|---|---|---|
| pcv - 150 - 6 | 5 | 12016 | 12.00 | 144192.00 |
| pcv - 150 - 8 | 12 | 26915 | 15.00 | 403725.00 |
| pcv - 150 - 10 | 2 | 3795 | 18.00 | 68310.00 |
| pcv - 150 - 12 | 2 | 482 | 22.00 | 10604.00 |
| Total | 21 | 43208 | 14.51 | 626831.00 |

## Fittings Summary

| | Fittings Symbol | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|
| Pipe Type | 2 | 6 | 8 | C | K | S | i | m | p | --- | other |
| pcv - 150 - 8 | 2 | 2 | 2 | 0 | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| pcv - 150 - 12 | 0 | 0 | 0 | 0 | 1 | 0 | 1 | 1 | 0 | 0 | 0 |

Symbol Key

2 - 90Elbow, standard
6 - Tee std, along run, branch blanked
8 - Tee elbow usage, entering branch
K - GlbV open (Globe, bevel seat)
i - Meter, disk
m - Ent.straight (Entrance)

```
----------------------
    Device Summary
----------------------

16 junction nodes
2 tanks
1 resevervoirs
1 pumps
9 hydrants
10 on/off valves
8 intermediate nodes
```

## 1.3.7.5 Direct Parameter Calculation

Direct Parameter Calculation - Constraints

See also the Constraints Video on the KYPipe web site.

See also Constraint Data

---

The current KYPIPE engine provides a fast and accurate calculation of a variety of design, operating and calibration parameters for pipe distribution networks. Pipe system parameters are calculated to exactly satisfy stated pressure requirements at designated locations in the piping system for a range of operating conditions. This offers a basis for determining the "optimum" values for the various design, operating and calibration parameters in the sense that the parameters are calculated to just meet the specified pressure conditions. This will allow you to provide sound decision making and to conceive and evaluate efficient and reliable alternatives or recommendations with reference to suggested or required system performance.

With the addition of this capability KYPIPE becomes a comprehensive distribution network analyzer. It has widespread applications

associated with the design, operation, and calibration of pipe distribution networks. It allows a wide variety of pipe system parameters and any of their combinations to be determined while meeting specified system performance criteria. These criteria represent specified pressure requirements at designated junction nodes throughout the distribution network for specified operating conditions. The parameters that can be considered may be divided into design, operating and calibration parameters, although there may be some overlap in these designations. The parameters include:

1. Design parameters such as: pipe diameter, pump power, pump head, storage level, and valve characteristics.

2. Operating parameters such as: pump speed, pressure regulating valve setting, control valve setting, and flow or pressure specifications.

3. Calibration parameters such as: pipe roughness, node demands, and minor loss coefficients.

There are three ways in which system parameter values can be calculated using KYPIPE. For each designated pressure condition, one of the following calculations can be made:

1. Calculation of a single value for a single designated parameter. For example, the calculation of the diameter of a designated pipe needed to just meet a specified pressure condition.

2. Calculation of a single value for a multiple designated parameter. For example, the calculation of a single value of HGL to be used as the setting for a number of pressure regulating valves which will just meet a pressure specification in the regulated region.

3. Calculation of a global factor change for a designated parameter. For example, a percentage change in all the pipe roughness values for a group of designated pipes which will just meet a specified pressure based on a field test.

Parameter calculation requires a "one-to-one" relation between the selection of parameters for evaluation and the specification of pressure constraints. That is, one designated pressure specification allows the determination of one pipe system parameter. However, through the use of global factors and grouping of parameters, considerable flexibility in choosing network variables is possible.

Combinations of the numerous parameter types and the three methods for calculating the parameter values provide a very powerful and general approach for defining the network variables to be determined.

**General Approach**

KYPIPE provides a direct calculation of the values of a variety of system parameters which exactly meet the stated specifications. The objective is to simultaneously satisfy the network conservation laws and the pressure specifications imposed. This approach involves adding equations and corresponding unknowns to the full set of flow continuity and energy equations describing the network hydraulics. The added equations describe the specified pressure requirements (pressure constraints) and the added unknowns represent the system parameters to be determined. The augmented system of equations is then recasted analytically in terms of pipe flow rates and indeterminate pipe system parameters.

Various pressure specifications representing desired performance conditions can be defined. For each defined specification, an additional energy equation is incorporated into the equation set for the piping system. Each additional equation allows the explicit calculation of one parameter. The solution, which is determined from a continuous variable space, is optimal in the sense that the calculated parameters are those required to exactly meet the stated pressure requirements. It is assumed that the basic network geometry is fixed, along with the location of the basic network components. Any number of pressure specifications and, thus, equations may be added. Each added specification will allow the explicit determination of an additional parameter. There is no restriction on the number of additional pressure specifications and corresponding parameter calculations as long as a one-to-one relation is maintained. That is, the following identity, which is derived from Eq. 1, must hold:

$$p + d = j + \mu + f + c - z \quad (1a)$$

in which d designates the number of parameters to be determined; and c is the number of pressure constraints. In addition, a single pipe cannot be assigned more than one indeterminate parameter. For example, it is not possible to solve for the diameter and roughness of a particular pipe required to meet two pressure specifications. Eq. 1a ensures the assembly of as many equations as there are unknowns and, therefore, should always be verified. The augmented system of equations can then be solved for network flow distribution plus the additional specified system parameters.

**Pressure Constraints**

KYPIPE explicitly determines the value of selected pipe system parameters to exactly satisfy one or more stated pressure requirements (constraints) for given network operating conditions. Pressure requirements can be specified at designated critical locations throughout the distribution network. The critical locations often correspond to junction nodes where pressures are maximum or minimum. Any junction node in the system can be pressure constrained.

**Pipe System Parameters**

A variety of pipe system parameters and any of their combinations can be utilized as decision variables for direct calculations to exactly meet stated pressure requirements. The parameters include:

**1** **pump speed** - Pump speed may be calculated for pumps described by three points of operating data. Using homologous units this data can be modified to represent the operation of the pump at other speeds and provide an expression for the pump head-flow curve at various speeds (Equ. 2d). Based on the calculated operating point the required pump speed may be determined.

**2** **pump power** - For pipes originally containing no pump or a pump described by useful power, the useful power can be selected as a decision variable for direct calculation. The useful power, P , refers to the actual power which is transformed into an increase in pressure head and kinetic energy of the liquid as it passes through the pump and was previously given as:

$$P_u = \frac{E_p Q \gamma}{CON}$$

where $\gamma$ is the density of the fluid and CON is a conversion term which equals 550 for English units (horsepower) and 1.0 for SI units (kilowatts). The calculation of this parameter is particularly useful for a preliminary design when the specific operating characteristics of the pump are not known. The flowrate, Q, and pump head, Ep, will also be calculated for the operating point.

**3** **FGN setting** - The setting (head) for any FGN may be selected as a decision variable. This application will normally be utilized to determine the water level in storage facilities for various operating conditions.

4 **PRV settings** - The setting (head) for a single or group of PRV's may be selected as a decision variable. The ability to determine these settings for various operating conditions is essential for efficient operation of systems with several pressure zones.

5 **minor loss coefficient** - These losses are included by using the concept of a minor loss and can be expressed as previously presented as:

in which hLM is the concentrated headloss at the component; ΣM is the combined minor loss coefficient for the pipe section which is a non-dimensional term; g is the acceleration of gravity; A is the pipe cross-sectional area; and Q is the volumetric flow rate. The term ΣM can be selected as a decision variable for direct calculation.

**control valve setting** - The setting for a control valve in a particular pipe section required to meet a specified pressure condition can be determined as follows. The combined minor loss coefficient, Sum K, for the pipe section is designated as a decision variable for direct calculation. From the calculated value of Sum K, the minor loss coefficient for the valve may be obtained. If the pressure-flow relation for the valve is adequately handled by the orifice relation, then a ratio of the open area to the fully open area for the control valve can be calculated.

6 **pipe diameter** - The inside diameter of a pipe section can be selected as a decision variable for direct calculation. The calculated value, Dc, will, in general, not be an available nominal pipe size. Once this calculation is made, you can select the actual design pipe size in one of following ways:

1. Select the next largest nominal diameter.

2. Determine the lengths of sections of a series pipe of the next smallest and next largest nominal pipe equivalent to the calculated value, Dc.

3. Determine the smallest nominal diameter of a pipe parallel to the original pipe which provides a capacity equal or greater than Dc.

4. Determine the lengths of sections of a series pipe installed parallel to the original pipe with a capacity equal to Dc.

For each pipe diameter calculated, subsequent calculations may be carried out to determine nominal diameters for each of the above options.

7 **pipe roughness** - The roughness of a pipe section can be selected as a decision variable for direct calculation. Here the pipe roughness refers to the Hazen Williams roughness coefficient for the pipe section, i.e., C factor. The calculation of this parameter is particularly useful for network model calibration when the initial estimates of C factors are not fairly well defined. The C factor values can be adjusted to improve agreement between predicted and measured values of pressure for known operating conditions. This capability is limited to analysis carried out using the Hazen Williams head loss equation.

**8a external demands at junction nodes** - Node demands required to meet observed or stated conditions of pressure can be designated as decision variables for direct calculations. This is particularly useful for calibrating or fine tuning network models when small variations in the demand distributions are acceptable. This variable can also be used to determine the flowrate required to satisfy a specified pressure constraint as noted below.

**8b flow limiting control device (pressure sustaining valve)** - A direct calculation of the magnitude of flow, which can be allowed to exit a distribution system such that a specified pressure condition will be maintained, can be made as follows. The location at which the flow exits the system is denoted as a junction node and the external demand at that location is designated as a decision variable for direct calculation. This is especially useful for analyzing fire flow conditions. Also, the flow requirement for a flow control valve to meet a specified pressure can be directly calculated by simultaneously computing the demand and inflow at adjacent nodes separated by a closed pipe.

**Selection of Decision Variables (Parameters) for Calculation**

There are three ways in which pipe system parameter values can be calculated using KYPIPE. For each designated pressure specification, one of the following calculations can be made:

1. Calculation of a single value for a single designated parameter. For example, the calculation of the setting of a control valve needed to just meet a specified pressure constraint.

2. Calculation of a single value for a parameter applied to a group of variables. For example, the calculation of a single value of HGL to be used as the setting for a number of pressure regulating valves, which will just meet the minimum pressure specification in the regulated region. This is accomplished by initially setting all the original values for the calculated parameters in the input data file to a single value.

3. Calculation of a global multiplying factor change for a designated parameter. For example, a percentage change in all the roughnesses for a group of designated pipes, which will just meet a specified pressure constraint. For this application the original values for the calculated parameters can differ. If the original values are identical this is the same as (2).

Considerable flexibility in the adjustment of network parameters is allowed when using a global multiplying factor as a decision variable. This factor, which will adjust all or a group of selected network parameters, can be computed in order to satisfy the pressure specification imposed. When more than one pressure specification is designated, various pipe system parameters can be grouped into several types such that the sum of the groups equals the number of specified pressure constraints. A different global factor for each group can then be calculated. This factor is used to adjust all decision variables included in its respective group. Each group will consist of a set of pipes with one indeterminate system characteristic, which may differ from one group to the other. In addition, selected pipes may be excluded from these groups and would, thus, be kept unaffected. For example, one group may consist of the roughness of all pipes older than a stated age, and a second group may consist of all the node demands which represent industrial consumption.

Combinations of the numerous parameter types and the three methods for calculating the parameter values provide a very powerful and general approach for defining the decision variables to be determined to simultaneously meet designated pressure specifications, and it is possible to accommodate most practical situations. This will allow the practicing engineer to form and evaluate efficient and reliable recommendations regarding suggested system behavior.

The choices for the parameters which can be designated as decision variables are summarized below:

1 pump speed

2 pump power

3 FGN settings (HGL)

4 PRV settings (HGL)

5 minor loss coefficients (valve setting)

6 pipe diameter

7 pipe roughness

8 demand - flow requirements.

**Special Considerations**

There are a number of special considerations which should be reviewed before carrying out parameter calculations with KYPIPE. Errors may result if these requirements are not considered.

1. KYPIPE requires a one-to-one zone relation between the selection of parameters for evaluation and the specification of pressure constraints. That is, each selected pipe system parameter and associated pressure constraint must belong to the same pressure zone. For example, it is not possible to solve for a diameter of a pipe in a particular pressure zone to meet a pressure specification at a junction node in a different pressure zone.

2. A pipe section, which is designated to be closed (valve shut), must not contain an unknown parameter for direct calculation. For example, it is not possible to solve for a pipe diameter, which is required to meet a specified pressure condition, if this pipe section is coded to be closed in the original data or closes due to changing conditions as the simulation proceeds. Thus, considerable attention must be given when selecting a pipe section, which contains a pump or a check valve, for parameter evaluation. The check valve allows flow only in the direction specified by the user (first to second node input in the original data file). If conditions exist for flow reversal, the valve shuts and the line closes and a selection for a parameter for that line is not possible.

3. If a pipe section in a branched area of the network with no terminating FGN node is to be selected as a decision variable for direct calculation, then the pressure at a terminating junction must be designated as a pressure constraint. For example, it is not possible to solve for the diameter of a pipe section, which is connected to a single junction node, unless the pressure at that node is specified.

4. When an external demand is selected for direct calculation, then the junction node selected must contain a non-zero external demand. For example, it is not possible to solve for a demand for nodes with demands initially set to zero. This is because a factor which multiplies the initial demand is calculated.

5. You can not select a node adjacent to the first FGN in the data set for a pressure constraint designation. This will produce an

error.

**Non Feasible Situations For Parameter Calculations**

There are a number of non feasible situations which will produce a situation where the solutions will not converge or the equations can not be solved as indicated by a computer message. The possibility of encountering a non feasible situation increases as more pressure constraints and parameter calculations are added.

**1. Due to Network Geometry**

Non feasible situations due to network geometry occur because the parameters chosen are positioned such that they can not independently or uniquely control the pressures set by the pressure constraints. Several examples are:

a) Two decision variable parameters specified for a single pipe. For this situation no unique solution exists. For example, it is not possible to determine the diameter and roughness of a particular pipe to meet two pressure constraints anywhere in the system.

b) Different decision variables specified for pipes in series or parallel. For example, a determination of two different pump speeds for parallel pumps is not acceptable because no unique solution exists.

c) A pipe with a decision variable and the corresponding pressure specified node must be in the same pressure zone. For example, it is not possible to calculate the diameter of a particular pipe section, which is in the main zone, to meet a pressure specification in a separate zone.

d) A pipe with a decision variable in a branching pipe section of the network which does not terminate with a junction node which is pressure specified. For this situation no solution exists.

Determining if the above requirements are met usually can be verified by inspection of the network geometry.

**2. Due to Network Hydraulics**

Non feasible situations due to the network hydraulics are much more difficult to anticipate because these situations depend on the flow distribution. Such situations essentially occur because the decision variables are unable to control the specified pressures for the baseline conditions. Some illustrations of the types of conditions which lead to this situation follow:

a) No value of the decision variable will meet the pressure specification. For example, a pipe with a diameter decision variable can be closed and the pressure specification is exceeded. Therefore no solution exists for the diameter which will meet the pressure constraint.

b) The pressure at a pressure specified node is unaffected by the value of the decision variable. For example, the setting of a throttle valve in a line leading from a storage tank can not affect the pressure at a node where none of the supplied flow originates from that storage tank.

There are a number of similar situations which are non feasible due to network hydraulics. The possibility of encountering this type of non feasible condition increases as more decision variables are considered. If you encounter situations which can not be handled you should modify your pressure constraint or parameter designations or both. Parameter calculation provides a powerful capability but even experienced users may occasionally encounter non feasible situations. These are not errors in the usual sense and normally require only trying other variations to obtain useful results.

SEE ALSO: Introduction to Modeling
Method of Analysis
Model Simplification
Model Calibration
Pipe System Model Geometry
Pipe System Components
Pressure and Flow Specifications
Multiple Scenarios - Changes

Direct Parameter Calculations - Constraints

## 1.3.7.6 Network Analysis

Network Analysis

**See also Analyze (Main Menu)**

---

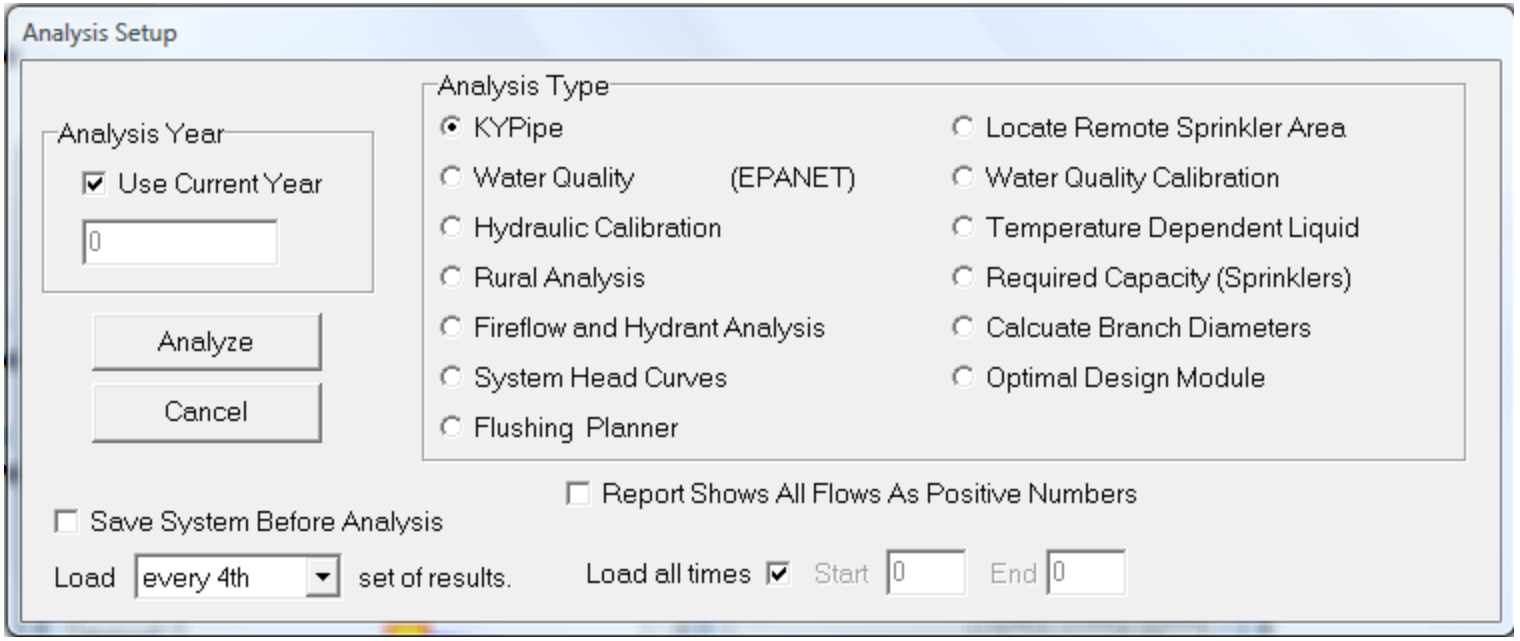
Once the system layout and data entry is complete, you are ready to perform the analysis. Prior to attempting the analysis, you should perform an Error Check.

Click on Analyze (Main Menu) and select Error Check

Note and correct all errors which are flagged. When the Error Check provides the message No Errors you are ready to proceed with the analysis.

Click on Analyze in the main menu and select Analyze.

An Analysis Setup pop up menu appears.




Select the Analysis Year (use current year or enter a different year) and Analysis Type and then click on Analyze. Note that the Analysis Type selection must be one which is available for your data file and Pipe2018 configuration.

You can now review results. It is recommended that the tabulated output (click Report tab) be reviewed first to check for any errors flagged by the analysis engine and review the general nature of the results. Then proceed to develop and print presentations of the results (pressure/flow labels, contours, color code, etc.).

## 1.3.7.7 Operational Control Settings

Operational Control Settings

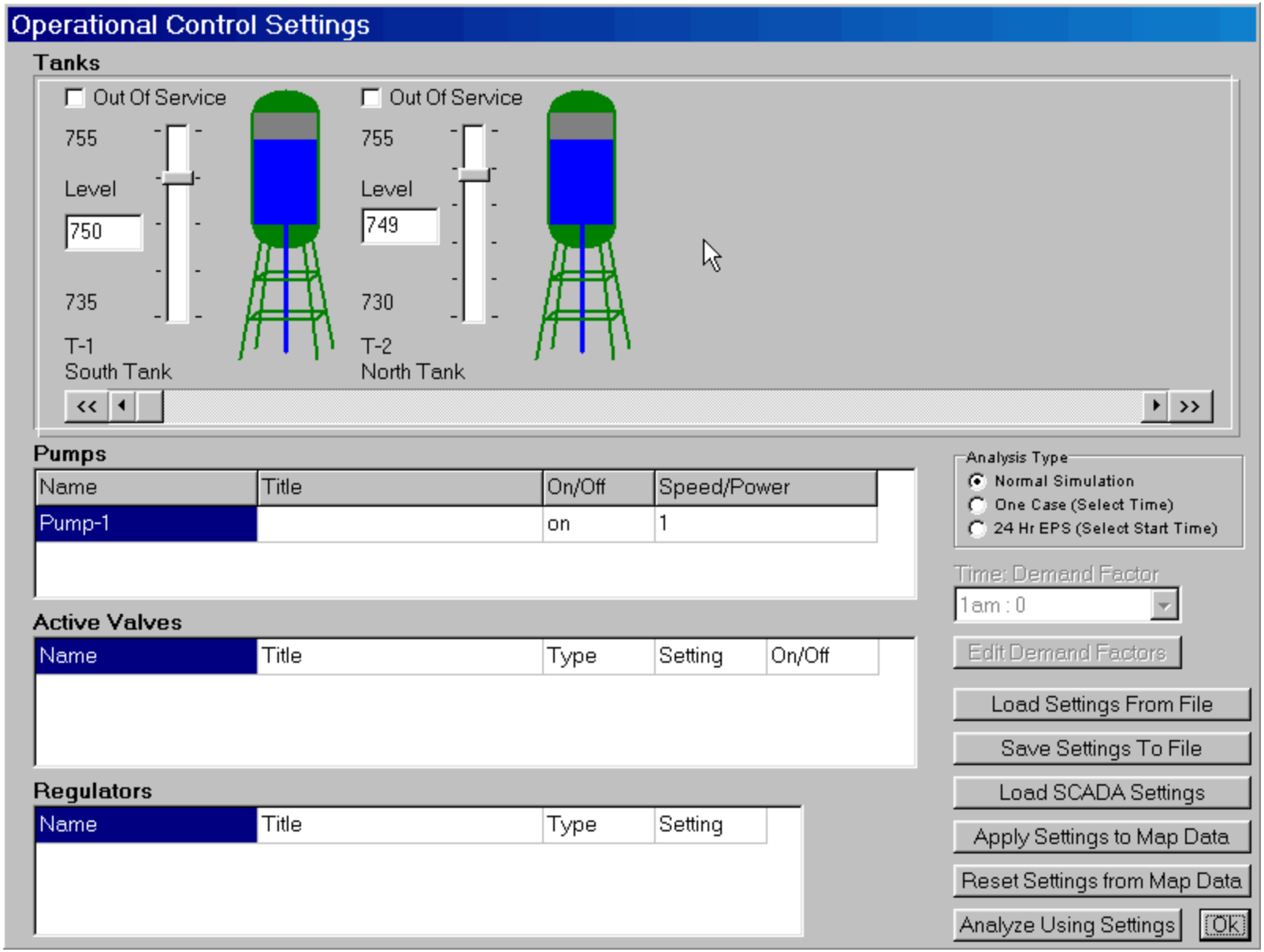
---

### Operational Control Settings Screen

This screen provides some very advanced capabilities for reviewing and modifying settings which affect the operation of the system and launching an analysis using these settings.

Important Note: When analyzing using this feature, only the settings in the Operational Control Settings screen are considered. The Change Data and Demand Pattern Data entered into the main data file are not recognized. Therefore, for example, when running an EPS using the Operational Control Settings screen, remember to use the Edit Demand Factors button to set up demand data if desired.

Click on Analyze | OCS Screen (Analysis) and the following screen appears.


Operational Control Settings
Tanks
Out Of Service
755
Level
750
735
T-1
South Tank
Out Of Service
755
Level
749
730
T-2
North Tank
Pumps
Name
Title
On/Off
Speed/Power
Pump-1
on
1
Active Valves
Name
Title
Type
Setting
On/Off
Regulators
Name
Title
Type
Setting
Analysis Type
Normal Simulation
One Case (Select Time)
24 Hr EPS (Select Start Time)
Time: Demand Factor
1am : 0
Edit Demand Factors
Load Settings From File
Save Settings To File
Load SCADA Settings
Apply Settings to Map Data
Reset Settings from Map Data
Analyze Using Settings
Ok

Within this screen, the following settings can be viewed and modified if desired.

**Tank level and on/off status**
**Pump speed (or power) and on/off status**
**Valve setting (% open) and on/off status**
**Regulator setting**

Once the settings have been reviewed and modified (if desired), a variety of hydraulic analyses can be launched. This is controlled by selecting Analysis Type and then clicking the button, Analyze System Using Settings

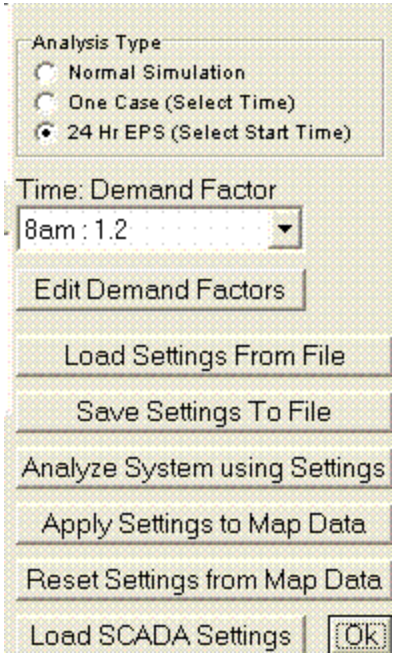
The Analysis Types (KYPipe) are:

**1) Normal Simulation: The simulation (regular or EPS) specified for this data file**
**2) One Case (Select Time) A regular simulation using the GDF (gobal demand factor) for the time shown in the Time:Demand Factor box.**
**3) 24hr EPS (Select Starting Time) A 24 hour EPS starting at the time shown in the Time:Demand Factor box**

The Time:Demand Factor list (KYPipe) for options 2 and 3 is defined and can be edited by clicking on Edit Demand Factors. The following box appears:

**You can load and save 24 hour Demand Factor files (.dmd extension). One example file is provided (AWWA.dmd) which has the values shown above. These values are provided by AWWA. Important note: When an EPS analysis is run through the OCS screen, demand factors in this table are global, that is, they are applied to all demands in the system regardless of the demand type.**

**There are several additional user options which are provided using the selection shown below.**




**Load Settings From File and Save Settings to File - The setting can be saved or loaded using a .ocs extension.**

**Apply Settings to My Data - The basic data file will be modified using the settings shown.**

**Reset Settings from Map Data - This will over write any modified settings to those in the Map data.**

**Using SCADA data to update settings - The option Load SCADA Settings allows you to interface your model with SCADA data and use this data to update your settings. You can then launch an hydraulic analysis based on the current time to obtain real time results. The SCADA data is assumed to reside in a file called SCADA.ocs which resides in the Pipe2018 directory. The data file format is**

**a Node Name**
**b on/off**
**c setting**

**repeat for each setting**

**for example:**

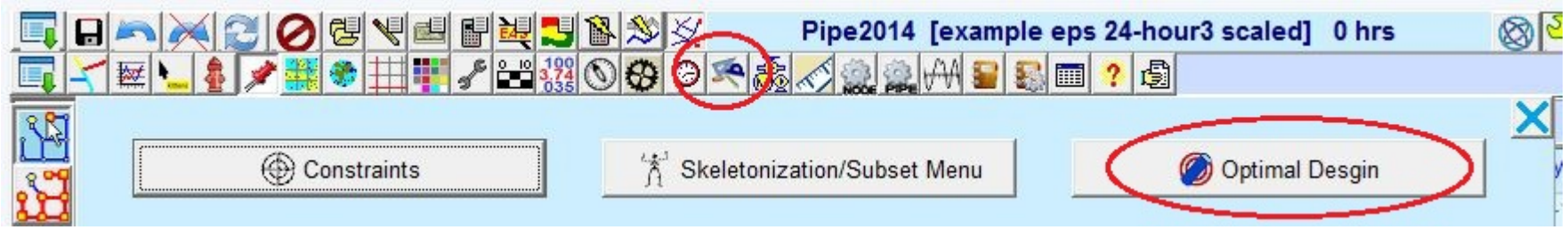
**T-1**
**on**
**735**
**Pump - 3**
**off**
**0**

**This file will set tank T-1 to 735 feet and set pump Pump-1 to off.**

## 1.3.7.8 Optimal Design

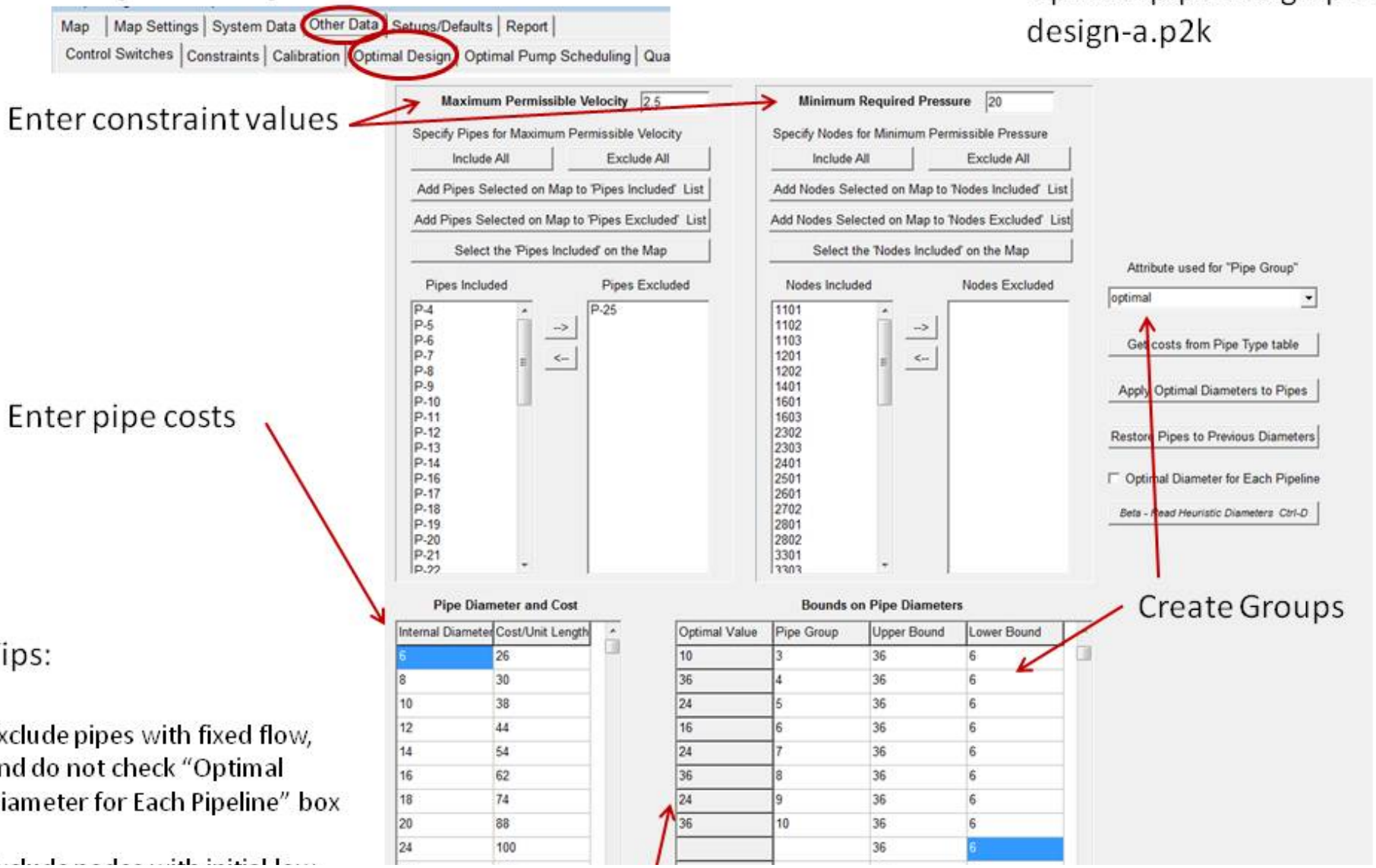
Based on Pipe install cost, maximum velocity and minimum pressure, Pipe Optimization feature will determine best diameter of pipe and lowest cost for installation.

Some example p2k files are available to demonstrate the feature.



# Pipe Optimization

Optimal pipe design.p2k
design-a.p2k




Tips:

Exclude pipes with fixed flow, and do not check "Optimal Diameter for Each Pipeline" box

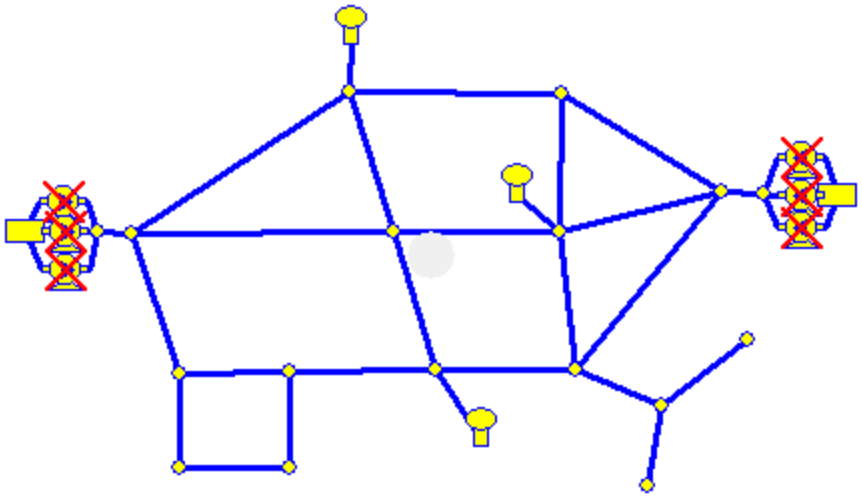
Pipe Diameter and Cost values must be input in increasing values.

It is best of Lower Bound values agree with lowest Internal Diameter values in the Pipe Diameter and Cost table.

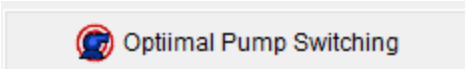
Though KYPIPE development team has published many papers on optimal design of water distribution systems over the last two decades, the real need for an optimal design module was felt only recently. Much of the research on optimal design was academically oriented, too far away from reality, and very few utilities could make use of the technology even after spending hundreds of thousands of dollars. Most of the industry needs were addressed by our unique and built-in "direct parameter calculations" feature that allows for optimal sizing of a single or a group of pipe diameters to meet the specified pressure requirements. However, the recent advanced modeling concepts (use hydrant flow data in the form of pressure supplies in lieu of a complete distribution network model) introduced by KYPIPE for designing distribution systems extensions (new sub-divisions etc) enables the use of optimal design technology in a more meaningful manner. This was the driving factor for releasing the optimal design module now instead of 12 years ago along with genetic algorithm based optimal calibration module. The optimal design module allows for sizing diameters of individual pipes or several group of pipes minimizing the overall cost of the project while satisfying pressure, flow and velocity constraints. The standard Pipe2018 GUI features allow for several different easy and convenient ways to define the input data (unit cost of pipes, pipe groups, velocity and pressure constraints etc) for the design module. The module generates tabulated text results summarizing the input data and optimal design diameters and project costs and also allows for auto updating of designed diameters. The module can be used not only for extensions and modifications to existing pipe distribution networks but also for designing new distribution systems (branched as well as looped) or complete redesigning of an existing distribution system.

Tip: The efficacy of optimization models drop as the number of decision variables increase. If a model is set up to do obtain optimized diameters for each pipeline in the model, this can be a lot of decision variables. Genetic algorithms and other nonlinear optimization methods fail when the number of decision variables exceed 30 or so. One way to reduce the number of decision variables is to group the pipes. Also, a user can remove certain pipes from the decision variable set by not including them in the groups. The option "Optimal diameters for each pipe" can be UNchecked.

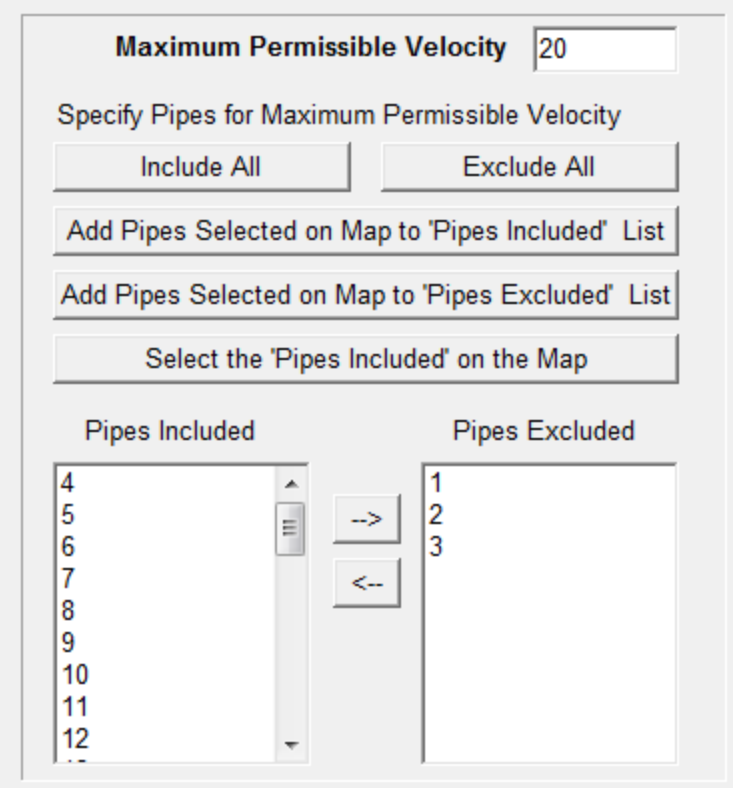
## 1.3.7.9 Optimal Pump Scheduling



Using the above model, the Optimal Pump Scheduling module is demonstrated. The model is set up as a 24-hour Extended Period Simulation or EPS. A Change Pattern or Demand Pattern may be applied, but no pump control (change data) is used for pumps to be optimized. A default Pump Power Cost should be specified under EPS. Time varying power cost may be specified under Demand Pattern.

Under EPS click the Optimal Pump Switching button 

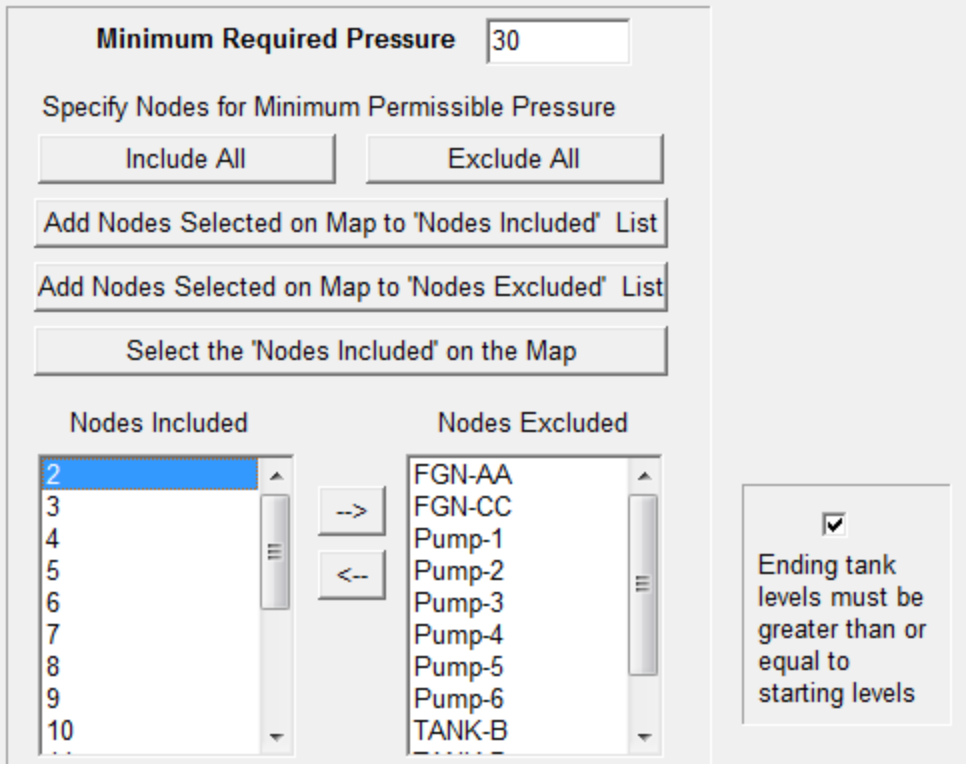
:




The maximum desired velocity may be specified. The pipes that are constrained by the specified velocity will be displayed in the list to the left, excluded pipes are on the right.

**Add Pipes Selected on Map to "Pipes Included/Excluded" List** - See Sets and Group Mode for instructions on how to select a group of pipes. Once pipes are selected, one of these buttons may be used to populate either list.

**Select the "Pipes Included" on the Map** - will cause all pipes in the 'Pipes Included' list to be highlighted in Group Mode when the map screen is viewed.

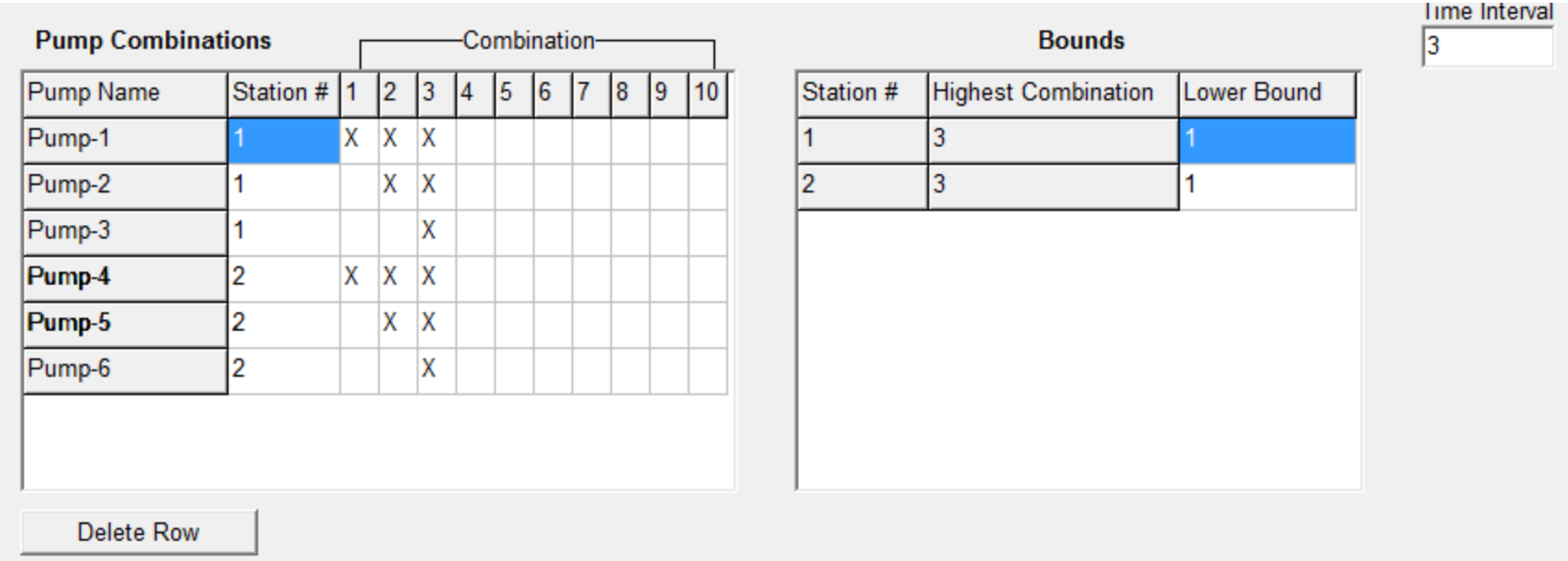



The minimum desired pressure may be specified. The nodes that are constrained by the specified pressure will be displayed in the list to the left, excluded nodes are on the right.

**Add Nodes Selected on Map to "Nodes Included/Excluded" List** - See Sets and Group Mode for instructions on how to select a group of nodes. Once nodes are selected, one of these buttons may be used to populate either list. For example, nodes on suction side pump may NOT have sufficient pressure to meet the pressure constraint no matter what the optimal pump schedule is and it may be best to exclude them.

**Select the "Nodes Included" on the Map** - will cause all nodes in the 'Nodes Included' list to be highlighted in Group Mode when the map screen is viewed.

Check **Ending tank levels must be greater than or equal to starting levels** to set the condition of the tanks at the end of the Extended Period Simulation.




**Time Interval** – Controls the number of decision variables for the optimization problem. A smaller time interval would increase the total number of decision variables and may reduce efficiency of the optimization algorithm. In addition, a smaller time interval may also lead to excessive on/off conditions for pumps. Larger time interval will reduce the number of decision variables but the pumps may be forced to run longer times than needed for optimal solution. A 3 to 6 hour time intervals would be ideal and the user may try different time intervals within this range to find a better solution.

All the pumps in the system will appear in the list. If a pump is to be excluded from the optimization, click on it and use the **Delete Row** button to remove it.

**Station #** - assign a Station number to each pump. Each Station Number will appear in the **Bounds** table. To refresh the Bounds table, click on the blank space below.

**Combination** - if the pumps may operate simultaneously, specify the possible combinations by clicking on the matrix. The total number of pumps in any combination will be reflected in the Bounds table. To refresh the Bounds table, click on the black space below.

**Bounds**

**Highest Combination** – this is the largest number/capacity of simultaneously operating pumps for this station. This number is automatically set by the number of combinations specified in the Combination matrix.

**Lower Bound** – this may be manually set by the user. It is the minimum number of pumps simultaneously operating for this station.

To run the Optimization, go to Analyze | Analysis and select Optimized Pump Scheduling.

The Optimized Pump Schedule program will make an EPS run based on the optimal pump scheduling results as the final analysis. The program asks the user whether to apply the results to the p2k model. If the new schedule is applied, a series of change cases are created and applied to the model. The new p2k model will have the

pump on/off status data updated automatically according to the optimization.

Now run the regular Extended Period Simulation (Analyze | Analyze, select KYPipe, and click Analyze). The Total Pump Power Cost may be viewed in the Report.

To remove the optimization, go to Edit Node Changes and click the Clear button. The Optimization may also be saved and reloaded at another time.

A bit about pump schedule optimization. Pump station optimization modules are greatly affected by the total number of decision variables - there are all possible combination of pumps and duration of pumping you can imagine which at times might run into several dozens and in some cases several hundreds! The efficiency of an optimization model reduces drastically with the increasing number of decision variables. Ideally, there should not be more than 20 decision variable for most of the optimization techniques to succeed in consistently obtaining a good optimal solution. Some vendors may claim that they have a great optimization tool but based on our extensive research in this area, they are all cursed by the dimensionality. We have come up with a unique way of combining the pump and operating time into simplified decision variables that in turn reduces the solution space thereby leading to better optimal solutions.

Using this method, for example, a decision variable of 3.42 represents the pump combination 3 running for 42% of pump schedule interval and therefore pump combination 2 (one combination lower than the previous pump combination) running for 58% of pump schedule interval.

Continuing this example, if there are 3 pumps at a station and if all pumps are of same capacity: the pump ranks could be rank-1 (only pump-1 operates, however, it could be pump-2 or pump-3 as well if they are of same size), rank-2 (pump-1 and pump-2 operate at any given time) and rank-3 (pump-1, pump-2 and pump-3 operates at any give time).

If all three pumps are different, then there could be many possible combinations. For example, rank-1 could be pump-1, rank-2 could be pump-2, rank-3 could be pump-3, rank-4 could be pump1- and pump-2, rank-5 could be pump-1 and pump-3 etc.

In case of traditional optimization formulation, the number of decision variables would have been (even for the case of all same size pumps) 3 * number of pump schedule time periods. This is just at one station. With the new formulation, the number of decision variables for each pump station is 1 * number of pump schedule time intervals in a 24- hour period irrespective of whether the pumps are of same size or different size thereby drastically reducing the total number of decision variable and hence increasing the efficiency of optimization technique.

For the case of equal capacity pumps, if the optimal decision variable is say 3.42, what it implies is pump combination 3 runs for 42% of time and pump combination 2 runs for 58% of time. This results in pump-1 and pump-2 running for 100% of the time (as they are part of both combination 2 and 3) and pump-3 runs only for 42% of time. The time here represents the pump schedule time interval and is usually 3 to 4 hours to minimize too frequent switching on and off of pumps.

The upper bound on pump combination permits the user to limit the number of pumps to be operational at any given time.

## 1.3.7.10 Optimal Pump Scheduling Algorithms

# Optimal Pump Scheduling For Water Distribution Systems

Lindell Ormsbee[1], Srini Lingireddy[2], Donald Chase[3]

**Abstract:** Three different mathematical formulations for the scheduling of pumps to minimize energy consumption while still satisfying operational constraints are presented. Each formulation requires the solution of a highly nonlinear optimization problem that requires the iterative solution of an external numerical model for evaluating and satisfying the operational constraints. Two different algorithms for solving the problem are examined.

## 1 Introduction

Perhaps one of the most important, yet most widely overlooked components of the urban infrastructure in the United States, is the public water supply system. However, over the last several decades and in particular since 9/11, many utilities are beginning to move to the use of sophisticated computer technologies to not only provide critical real time information about the state of their systems, but also to improve their overall operations. One such area of focus has been in the optimal scheduling of pumps so as to minimize cost and improve operational conditions.

Researchers have been exploring the area of optimal pump operations for several decades [11]. In its most general form, the classical non-linear constrained optimization problem as applied to pump operations may be formulated as follows:

| | | |
|---|---|---|
| Minimize or Maximize: | F(X : q, p) | (1) |
| Subject to: | q(X) = 0 | (2) |
| | p(X)>/=0 | (3) |
| | Xmin < X < Xmax | (4) |

where F(X:q,p) represents the objective function (in terms of cost) to be minimized, q(X) represents the explicit system constraints (conservation of mass and energy in terms of flow q), and p(X) represents the implicit bound constraints (minimum pressures) to be satisfied. Finally, $X_{min}$ and $X_{max}$ represent explicit bound constraints on the decision variables of the optimal control formulation (duration of pump operations). The system constraints can be represented explicitly by the use of a simplified inductive model of the water distribution system [15] or a deductive model [10]. The exact form of the objective function and the associated decisions variables depend upon the nature of the problem formulation.

---

[1]Director, Kentucky Water Resources Research Institute, University of Kentucky, Lexington, Kentucky 40506-01907, lormsbee@engr.uky.edu

[2]Fluid Hammer Consultancy Services (P) Ltd, #102, Sri SaiBalaji Apts, Plot #54, Arunodaya Colony, Madhapur, Hyderabad 500 081, INDIA, srini@fluidhammer.com

[3]Department of Civil and Environmental Engineering and Engineering Mechanics, University of Dayton, 300 College Park, Dayton, OH 45469, Donald.Chase@notes.udayton.edu

## 2 System Constraints

Because of the complexity of the system constraints, they are normally extracted out of the formal optimization formation and handled externally through use of a simulation program as shown in Figure 1. When using this structure, a vector of decision variables X is selected that explicitly satisfies the explicit bound constraints. This vector is then passed to the simulation program where the system constraints are satisfied and a vector of resulting pipe flowrates q and junction pressures p are determined. Once determined these values are passed back to the optimization program in order to check for feasibility of the implicit bound constraints (i.e. minimum pressures - p), and if feasible, the resulting value of the objective function associated with the decision vector X.

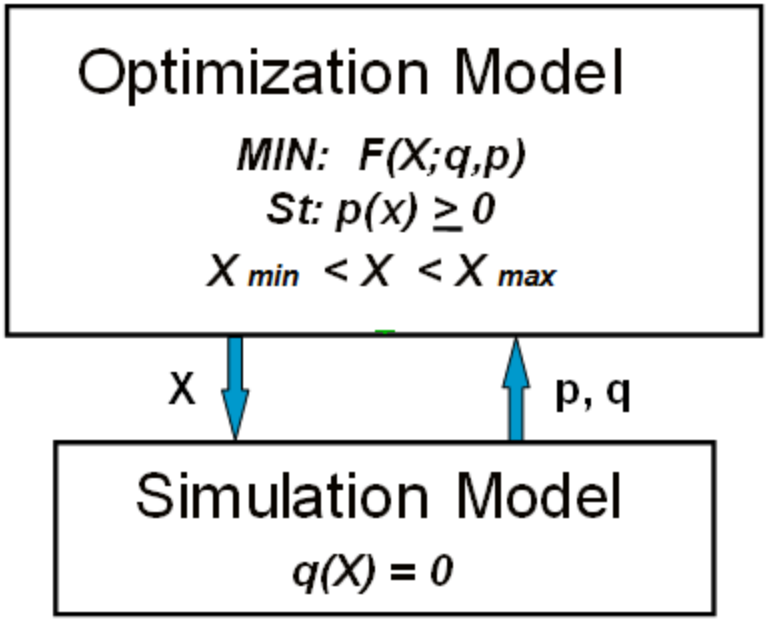



Figure 1. Problem Disaggregation Structure

The system constraints for this type of problem can be illustrated by considering the simple pipe distribution system shown in Figure 2. This system contains two tanks, five pipes, three junction nodes and one pump. The direction of flow in each of the pipes is indicated by the arrow on the line segment. In this case, a typical problem might be to determine the operating times of the pumps at the pump station over the course of an operating period (e.g. a day) so as to meet the varying demands $M_t$ at junction node 3 while maintaining adequate operating pressures at each junction node (which is related to insuring that the water in the tank never drops below a critical level).

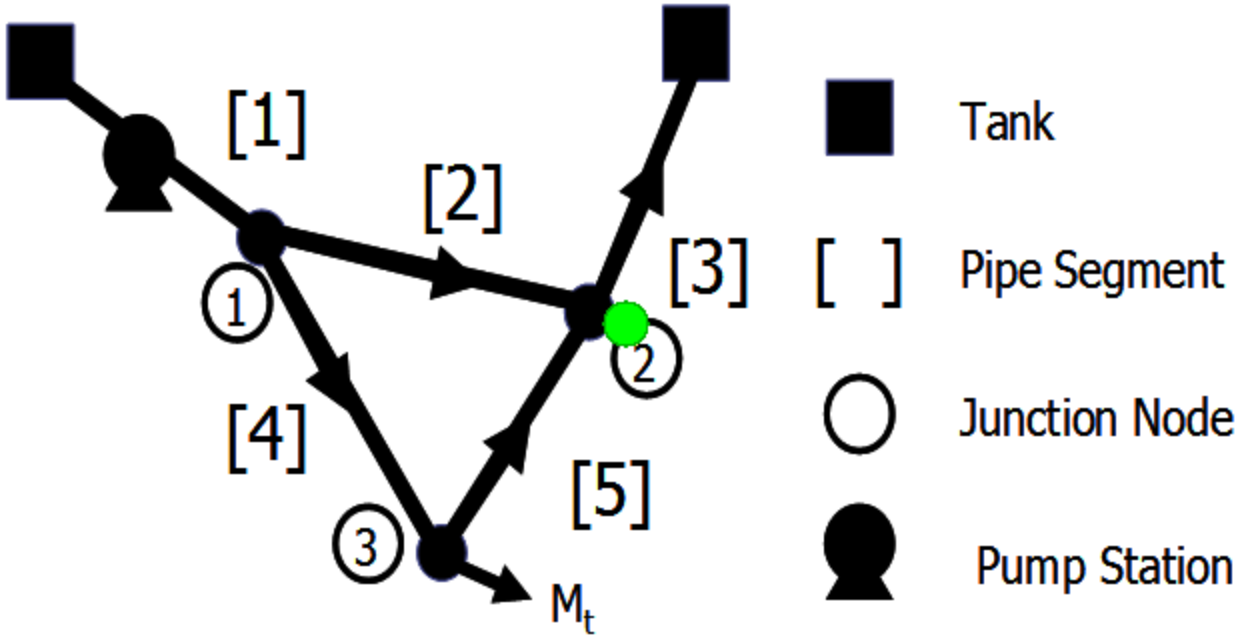



Figure 2. Example Pipe Network

The majority of pumps used in the water industry are constant speed pumps, in which the discharge is a function of the water levels in the tank. During those times in which the pump station discharge exceeds Mt, the excess will fill the tank. During those times in which the pump station discharge is less than Mt, then the tank will drain. The decision of what pumps to turn on and how long they should run will be driven by both the nodal demand Mt and the tank level (which controls the associated junction pressures). In many cases, pump stations will consist of different size pumps with different flow versus pressure relationships. Thus, the decision of which pump or combination of pumps to run during a particular period can become rather complicated. This is further complicated by the fact that the discharges of multiple pumps cannot be simply added together. Instead, the combined flows are a nonlinear function of the energy loss through the system (as determined by a system energy curve) and the associated tank level. For example, consider the situation of two pumps as shown in Figure 3. Each pump will have a different discharge versus pressure curve (known as a pump characteristic curve) as shown in the accompanying graph. The actual discharge that a particular pump will produce is determined by the intersection point of the characteristic curve and the system energy loss curve. When pump A is operating, the discharge is Qa. When pump B is operating the discharge is Qb. However, when both pumps are operating, the discharge is Qa+b. Technically, the system energy loss curve will shift up or down depending upon the water level in the tank, so things can become somewhat complicated.

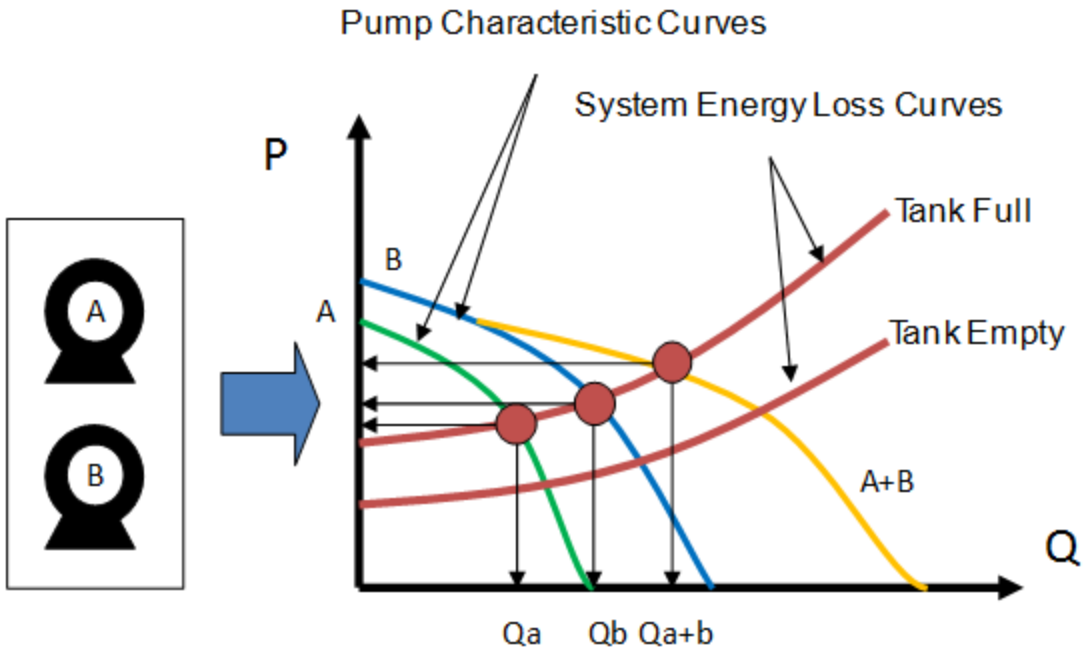



Figure 3. Pump Characteristic Curves and System Energy Loss Curves

The system energy loss curve represents a conceptual approximation of the flow versus pressure relationship that exists in a real water distribution network. In actual practice, this relationship must be determined by solving a series of nonlinear energy equations that are used to describe the physics of the system. For example, if we use the simple system shown in figure 2, then we are required to write one energy equation for each loop or energy path that exists in the network. An energy path is simply a path or series of pipes that connect any two tanks (technically any two points in the system with a known boundary condition – i.e. water level). For this simple system, the energy equation for the simple loop may be written as:

$$K_2 Q_2^{1.852} - K_4 Q_4^{1.852} - K_5 Q_5^{1.852} = 0 \qquad (5)$$

Likewise the energy equation for the path may be written as:

$$K_1 Q_1^{1.852} - Z_P / Q_1 + K_2 Q_2^{1.852} + K_3 Q_3^{1.852} = \Delta E \qquad (6)$$

Where $Q_i$ is the flowrate in pipe i, $Z_p$ coefficient which is a function of the horsepower of the pump, $\Delta E$ is the difference in elevation between the water levels of the two tanks, and $K_i$ is an energy loss term for each pipe that may be expressed as

$$\backslash\ K_i = \frac{4.73 L_i}{C_i^{1.852} D_i^{4.87}} \qquad (7)$$

where L = the pipe length in feet, D = the pipe diameter in feet, and C = an empirical roughness coefficient that depends on pipe age and pipe material (e.g. C = 130 for a new ductile iron pipe).

Since equations 5 and 6 are nonlinear in terms of Q, they cannot be solved directly but they may be solved iteratively using a Taylor's series approximation. For our simple system, the resulting equations become:

$$1.852[K_2 Q_2^{.852} - K_4 Q_4^{.852} - K_5 Q_5^{.852}]\Delta Q = K_2 Q_2^{1.852} - K_4 Q_4^{1.852} - K_5 Q_5^{1.852} \qquad (8)$$

$$[1.852 K_1 Q_1^{.852} + Z/Q_1^{2} + 1.852 K_2 Q_2^{.852} + 1.852 K_3 Q_3^{.852}]\Delta Q =$$

$$K_1 Q_1^{1.852} - Zp/Q_1 + K_2 Q_2^{1.852} + K_3 Q_3^{1.852} + \Delta E \qquad (9)$$

If we express equations 8 and 9 in matrix notation, then the set of equations can be written as:

$$\begin{bmatrix} G_I(\mathbf{Q}_I) & 0 \\ 0 & G_{II}(\mathbf{Q}_{II}) \end{bmatrix} \begin{bmatrix} \Delta Q_I \\ \Delta Q_{II} \end{bmatrix} = \begin{bmatrix} -F_I(\mathbf{Q}_I) \\ -F_{II}(\mathbf{Q}_{II}) \end{bmatrix}$$

Figure 4. Matrix structure for simple network example

where $G_I$ and $F_I$ represents the function on the right hand side and left hand side of equation 8 respectively, and $Q_I$ represents the vector of flows. Likewise, $G_{II}$ and $F_{II}$ represent the functions on the right hand side and left hand side of equation 9 respectively, and $Q_{II}$ represents the vector of flows. Reducing further, the equations can now be expressed as:

$$[G]\Delta Q = \{F\} \quad (10)$$

Thus, for this simple example, each time the simulation model is accessed by the optimization model, initial estimates of the flows in each pipe must be generated so that conservation of mass is preserved at each junction node (e.g. $Q_1 - Q_2 - Q_4 = 0$). Once this is done, the K coefficients are determined and substituted into the equations. Next, flow adjustment factors (i.e.$\Delta Q$) are calculated for each loop or path as follows:

$$\Delta Q = [G]^{-1}\{F\} \quad (11)$$

Once these are determined, each of the individual flows in each pipe may be determined using a recursive equation. For example, for pipe 1 the flows are updated as follows:

$$Q_{1i+1} = Q_{1i} + \Delta Q_I \quad (12)$$

For pipes that are common to more than one equation (e.g. pipe 2), the recursive equation is expressed as follows:

$$Q_{2i+1} = Q_{2i} + \Delta Q_I + \Delta Q_{II} \quad (13)$$

Once all the flows have been updated, they are then used to update the K coefficients and the process is repeated until the $\Delta Q$'s converge to zero.

It should be recognized that most water distribution systems are much more complicated than the simple network shown in Figure 2. Further, in simulating the performance of such a system over time (e.g. a day), the previous solution methodology must be repeated for each time step in the simulation. In this case, the flows determined at the beginning of each time step are then used to update the boundary conditions for use in solving for the flows at the next time step. In order to minimize the error associated with this type of Euler integration, a maximum time step of one hour may be necessary. When applied to typical water distribution system, the computational requirements associated with each "function call" from the optimization model can become significant.

## 3 Objective Function

The objective of the extended period operation problem is to minimize the total energy consumption charges associated with operating a set of pumps over the course of an operating horizon, while simultaneously satisfying any service and reliability- related requirements of the system. In a typical water

distribution system, the energy consumption costs incurred by pumping water depends mainly on the rate at which water is pumped, the associated pump head, the duration of pumping, the unit cost of electricity, and the different combined efficiencies of various pump combinations. Mathematically the objective function may be expressed as:

$$\text{Minimize } Z = \left(\frac{0.746\gamma}{550}\right)\sum_{t=1}^{T} R_t \sum_{i=1}^{I}\left[\frac{Q_{t,i}H_{t,i}X_{t,i}}{e_{t,i}}\right]$$

Minimize (14)

Where: Z = the total energy cost to be minimized ($)

$Q_{t,i}$ = the average flowrate associated with pump i during time t (cfs)

$H_{t,i}$ = the average head associated with pump i during time t (ft)

note: the term “head” is a common term in pump hydraulics used to express pressure in terms of an equivalent height of water.

$X_{t,i}$ = the duration of the time pump i is operating during interval t (hr)

$e_{t,i}$ = the average wire to water efficiency associated with pump i during time t

$R_t$ = the electric rate during time t ($/Kw-hr)

$\gamma$ = the specific weight of water ($lb/ft^3$)

I = the total number of pumps included in the optimization

T = the total number of time intervals in the operating horizon

For a given network configuration and an associated set of initial boundary conditions (including the vector of initial tank levels E and the vector of system demand loadings M), the average discharge $Q_i$, pump head $H_i$, and pump efficiency $e_i$, associated with a particular pump i can be expressed as a function of the characteristics of the pump itself plus the characteristics of another pump which may be operating during the same time periods as pump i. Since the set of pumps operating during a particular period, t, can be explicitly defined by the duration of time each pump in the set is operating, (i.e. if $X_{t,i} = 0$ then pump i is not operating during time period t and if $X_{t,i} > 0$ then the pump is operating during period t), then the pump discharge, the pump head, and the pump efficiency can be expressed as implicit functions of the vector of total pump durations for a particular time interval [12]. As a result, equation 14 may now be expressed as:

$$\text{Min } Z = \sum_{t=1}^{T} \sum_{i=1}^{I} f\left[ Q_{t,i}(\mathbf{X}_t, \mathbf{M}_t, \mathbf{E}), \mathbf{H}_{t,i}(\mathbf{X}_{t,s}, \mathbf{M}_t, \mathbf{E}_t), \mathbf{e}_{t,i}(\mathbf{X}_{t,s}, \mathbf{M}_{t,i}, \mathbf{E}_{t,i}), \mathbf{X}_{t,s}, \mathbf{R}_t \right] \quad (15)$$

As a result, the objective function can be expressed solely in terms of the vector of the individual pump operating times. As will be discussed in the following sections, the exact nature of the pump operating times will be dependent upon the problem formulation.

## 2 Implicit Scheduling Formulation

Historically, the optimal pump scheduling problem has been either formulated as an implicit control problem or an explicit control problem. In the implicit formulation, pump station discharge, supply pressure, or tank water levels are treated as the decision variables of the optimal control problem. In the explicit formulation, the actual pump operating times (either individually or compositely) are treated as the decision variables.

The implicit formulation will typically require the solution of two sub-problems. The first sub-problem involves determining an optimal decision trajectory. The optimal decision trajectory can be defined as that series of pump station discharges, supply pressures, or tank water levels which, over the course of the operating horizon, result in a minimal total operating cost.

The second sub-problem involves determining the specific pump operations which produce the optimal decision trajectory. The difficulty associated with finding specific pump combinations is compounded by the fact that many combinations capable of producing the desired decision trajectory may exist. Furthermore, from all the possible combinations capable of producing the desired trajectory, the combination which results in minimal operating costs must be found.

Historically, the implicit approach has been used to develop control strategies for single tank systems. In such cases, the tank water level has been used as the implicit decision variable and the resulting formulation is typically solved using dynamic programming [7], [8], [13], [16]. Due to the curse of dimensionality, dynamic programming solutions are normally restricted to problems involving no more than three storage tanks. For systems with several tanks, researchers have either employed dynamic programming along with an associated site specific spatial decomposition scheme [3], [21] or nonlinear programming along with an alternative implicit decision variable [9].

## 3 Discrete Explicit Pump Scheduling

In the discrete explicit approach, the actual times of the operation of each pump are treated as the decision variables [6], [14]. In this case, the objective function may be characterized in terms of operational costs and the associated state variables (such as flow or pressure) can then be modeled using a much more robust model of the water distribution system that can be linked to the optimization algorithm via iterative subroutine calls [19].

### 3.1 Restricted Formulation.

The restricted formulation of time as a decision variable was originally proposed by Chase and Ormsbee [5] and later applied by Chase [4] and Brion [2]. In this approach, a pump is forced to begin operating at the beginning of a pre-determined timer interval (e.g. every four hours), in effect, restricting the time a pump can begin operating. The decision variable in this case is the duration of time each pump operates during a particular time interval. For example, suppose decision variable $X_{3,2}$ has a value of 2.50 and a four hour time interval is used. Under the restricted approach, pump 2 would be turned on at the beginning of time interval #3 and turned off 2.5 hours into the interval. In other words if the operating horizon begins at midnight, pump 2 would be placed

on line at 8:00 a.m. and turned off at 10:30 a.m. The values taken on by the decision variables are bounded between 0 (pump off) and Δt (duration of time interval). Figure 5 provides a pictorial representation of the decision variables in the restricted approach.

As can be seen from Figure 5, the total number of decision variables required to solve the optimal control problem is equal to the product of the number of pumps and the number of time intervals making up the operating horizon. As the size of the time interval is decreased, the closer the problem becomes to one of continuous pump operation. Unfortunately a decrease in the size of the time interval results in a proportional increase in the number of time intervals which, in turn, causes an increase in the number of decision variables. An increase in the number of decision variables can be undesirable since the computation time required to solve the optimal control problem will increase.

3.2 Unrestricted Formulation.

In an effort to reduce the number of decision variables, yet continue to pose the problem within the framework of an explicit time model, the restricted formulation can be modified to yield an unrestricted formulation. In the unrestricted formulation, the decision variables become the starting and ending time for the pumps. Figure 6 shows a pictorial representation of the decision variables for the unrestricted formulation.

Unlike the restricted formulation, in the unrestricted formulation there are fewer conditions as to when a pump may start or stop. Instead of forcing a pump to begin operating at the start of a particular time interval, pumps are allowed to start operating at any time during the operating horizon. Likewise, instead of shutting down a pump during a given time interval perhaps only to have it operating again at the start of the next time step, under the unrestricted approach a pump may stop operating at any time during the operating horizon. Consequently, feasible values of the decision variables for the unrestricted approach are bounded between 0 (pump off) and T where T is the operating horizon, usually 24 hours.

3.3 Discussion of Unrestricted and Restricted Formulations

A nice feature of the unrestricted approach is that it more closely replicates actual pump operation. In other words, system operators will typically place pumps on line in response to certain system parameters such as a low tank level, low pressure, an increase in system demands, a fire, etc. Such changes in system indices and/or system loadings are not likely to occur at the start of a pre-defined time interval, unless of course, the time interval is extremely small.

The restricted formulation allows pumps to operate several times per day. This can be accommodated in the unrestricted formulation by increasing the number of decision variables. In this case, a pair of decision variables are associated with each pump duty cycle. If a pump is allowed to operate only once per day, then there is one pump duty cycle per day and the number of decision variables is twice the number of pumps, i.e. one decision variable for the starting time and one for the ending time. If all pumps are allowed to operate twice per day, then there are two pump duty cycles for each pump. Following the convention that a pair of decision variables is associated with each pump duty cycle, then the total number of decision variables would be four times the number of pumps.

In general, the number of decision variables associated with the unrestricted formulation can be found using the following equation:

$$\mathrm{NDV} = \sum_{i=1}^{I} (2 * DC_i) \tag{16}$$

where NDV is the total number of decision variables in the unrestricted explicit formulation, $DC_i$, is the number of daily duty cycles allowed for pump i, and I is the number of pumps included in the optimization.

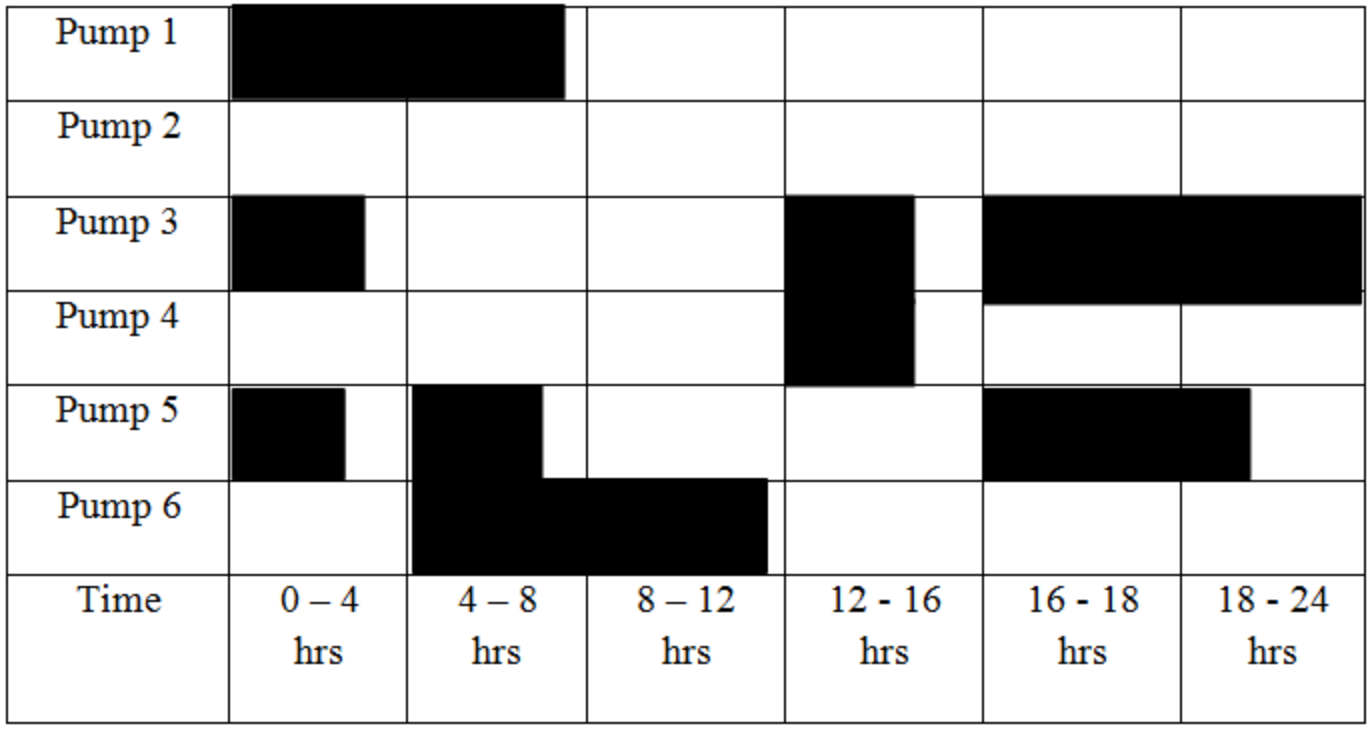



Figure 5 Schematic Representation of Decision Variables for the Restricted Approach

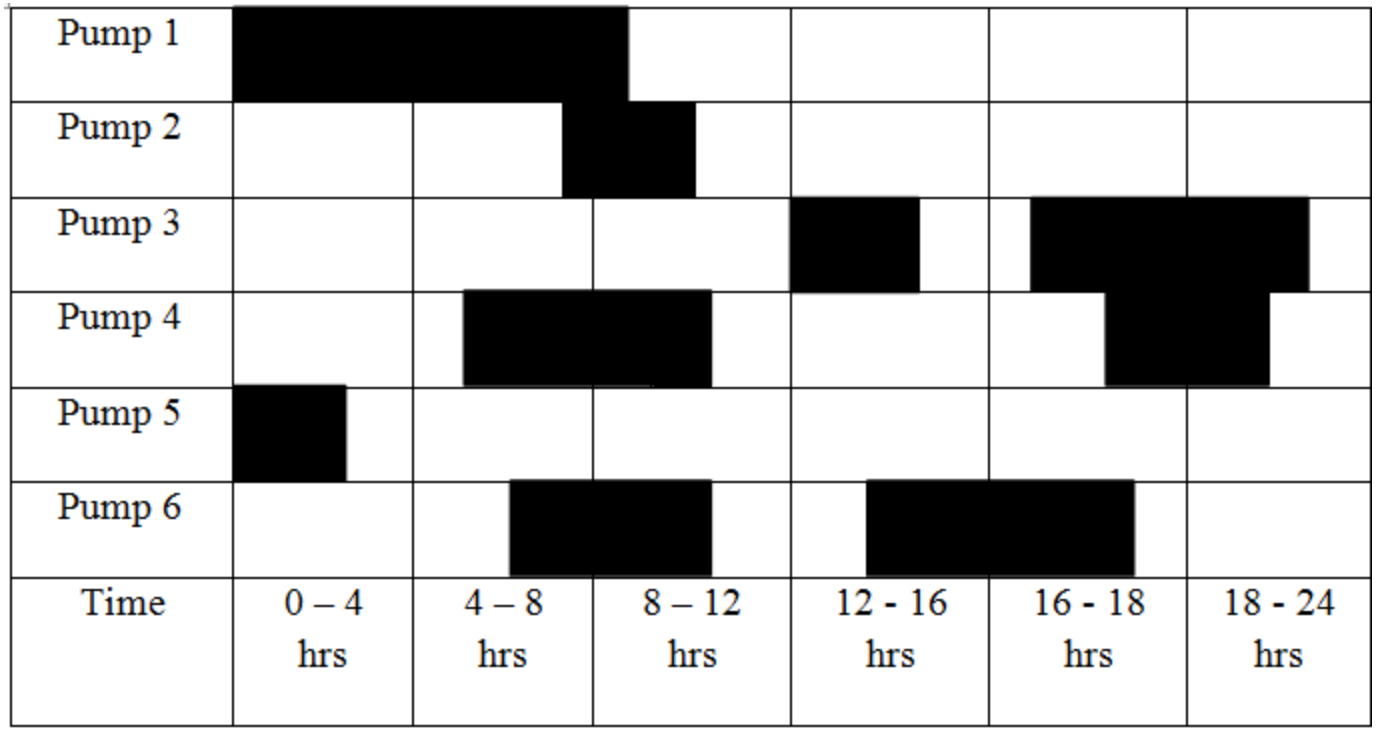



Figure 6. Schematic Representation of Decision Variables for the Unrestricted Approach

With the unrestricted formulation, additional bound constraints are needed to assure that the starting time for a pump is less than the stopping time for the same pump. If multiple duty cycles are allowed, then constraints are needed to insure that the starting time for particular pump duty cycle is greater than the stopping time for the pump's previous duty cycle. The additional constraints required for the unrestricted formulation can be expressed with the following equations:

$$\mathbf{X}_{i+1} - \mathbf{X}_{i} > 0 \quad \text{i= 1, 2.....I} \qquad (17)$$

$$\mathbf{X}_{n,I} - \mathbf{X}_{n-1,i+I} < 0 \quad \text{i= 1, 2.....I; n= 2, 3.....DC} \qquad (18)$$

Where $\mathbf{X}_i$ is the starting time for pump i, $\mathbf{X}_{i+1}$ is the stopping time for pump i, $\mathbf{X}_{n,i}$ is the starting time for pump I for the nth duty cycle, and $\mathbf{X}_{ni+I}$ is the

stopping time for pump i during the nth duty cycle. Such constraints are not necessary for the restricted approach since the decision variable is the duration of time a pump operates and since the formulation keeps pump operation within a single time interval.

Two separate explicit control formulations have been proposed: 1) a restricted formulation and 2) an unrestricted formulation. Under the restricted formulation, pumps are forced to begin operating at the start of a time step and the value of the decision variable is equal to the duration of time the pump operates. Under the unrestricted formulation, pumps are allowed to start and stop operating at any time within the operating horizon and the decision variables represent the actual starting and stopping times for each pump duty cycle. With respect to the restricted formulation, the unrestricted approach allows more freedom of operation and more closely resembles actual pump operation. However, the unrestricted approach requires the use of additional constraints to regulate pump operation within practical limits.

As a result, the addition of new constraints results in a slight modification of the optimal control problem originally shown in equations 1-4. The modified optimal control problem for the explicit formulation may now be expressed as:

$$\text{Minimize or Maximize:} \quad F(X) \tag{19}$$

$$\text{Subject to:} \quad G_j(X) \le 0 \qquad \text{j= 1, 2, 3....J} \tag{20}$$

$$H_k(X) = 0 \qquad \text{k= 1, 2, 3....K} \tag{21}$$

$$U_m(X) \le 0 \qquad \text{m= 1, 2, 3....M} \tag{22}$$

$$X_{\min} \le X \le X_{\max} \tag{23}$$

where all the terms are the same as before and U(X) represent bound constraints used to keep pump starting and stopping times within practical limits when using the unrestricted approach. If constraints (20) – (23) are included directly in the optimization problem as binding constraints, then the result for most real problems would be a large scale nonlinear problem. The scale of the problem can be reduced by handling the constraints in the following manner. The implicit system constraints, H(X), are solved through the use of a simulation model [20]. Depending upon the size of the mathematical model describing the water distribution systems, accommodating the implicit constraints through the use of a simulation model can substantially reduce the number of constraints which would otherwise be solved by the optimization algorithm. The implicit bound

constraints, G(X), and the bound constraints associated with the unrestricted approach, U(X), can be included in the objective function as a penalty term so as to allow solution using an unconstrained optimization method. Alternatively, depending upon the type of solution algorithm employed, they may also be explicitly satisfied outside of the algorithm. Finally, the explicit bound constraints can be used to assign values of decision variables when the variables exceed their bounds. For example, if during the course of the optimization a value taken on by a decision variable is greater than the upper bound of the variable, $X_{max}$, then the value of the decision variable is set equal to the upper bound. Similarly if a value of a decision variable is less than the lower bound, $X_{min}$, then the value of that variable is set equal to its lower bound.

**4 Composite Explicit Pump Scheduling**

As discussed previously, the exact form of the decision variable will be dependent upon whether a restricted or an unrestricted formulation is employed. In applying the restricted approach, the normal operating horizon (typically 24 hours) is divided in to T separate time intervals (e.g. 4 hours) and the pump operating time for each pump in each time interval is determined. In the unrestricted approach, a specific number of pump duty cycles is specified for each pump and the beginning and ending times of each duty cycle is determined. For distribution systems with multiple pump stations, and with each pump station containing numerous pumps, both formulations can result in an excessive number of decision variables. One way to significantly reduce the total number of decision variables would be to develop a single decision variable for each pump station for each time interval that relates the particular set of pumps in operation during that period. Such a formulation can be obtained by ordering the various available pump combinations associated with each pump station on the basis of unit cost. A single decision variable can then be developed for each pump station s and each time interval t of the form $X_{st}$ = II.CC where II = is an integer and corresponds to the identification number of the pump combination that operates CC% percent of the time interval. By definition, it will also be assumed that combination II-1 operates the remaining (1-CC) percent of the time interval. (The combination in which II = 0 corresponds to the null combination or the decision to run no pumps). Modification of the original objective function to accommodate the proposed formulation yields the following objective function:

$$\text{Min } Z = \sum_{t=1}^{T} \sum_{i \in S}^{I_S} f\left[ Q_{t,i}(\mathbf{X}_{t,s}, \mathbf{M}_t, \mathbf{E}_t), \mathbf{H}_{t,i}(\mathbf{X}_{t,s}, \mathbf{M}_t, \mathbf{E}_t), \mathbf{e}_{t,i}(\mathbf{X}_{t,s}, \mathbf{M}_{t,i}, \mathbf{E}_{t,i}), \mathbf{X}_{t,s}, \mathbf{R}_t \right] \qquad (24)$$

This objective function as expressed in Eq. 24 is subject to the same three sets of constraints as with the previous explicit formulations. These include: (1) a set of implicit system constraints, (2) a set of implicit bound constraints, and (3) a set of explicit decision variable constraints. While both the implicit system and bound constraints will have an identical form as before, the explicit decision variable constraints are different for the new formulation as a result of the use of a new set of decision variables. In this case the decision variable for each pump station for a particular time interval will be restricted between a lower value of zero (corresponding to no pumps in operation) and an upper value related to the maximum number of pump combinations available for that pump station.

In applying the proposed algorithm to a specific distribution system, the desired operating horizon (typically 24 hours) is once again divided into a discrete set of time intervals. A separate decision variable for each time interval is then assigned to each pump station. To initiate the algorithm a separate vector for each time interval is randomly generated or explicitly specified which contains the values of the decision variables for each pump station in the system. As a result, any potential solution will consist of a set of N vectors where N – the number of time intervals which constitute the operating horizon. To insure a feasible solution, the initially specified or generated set of decision vectors must satisfy the explicit bound constraints.

Similar to the previous explicit control formulations, the proposed composite formulation also uses a disaggregated solution methodology. That is, once an

initial set of decision vectors is obtained, it is then passed down to a network simulation model [20] for use in explicitly satisfying the implicit system constraints and for use in evaluating the implicit bound constraints. The values of the resulting state variables (i.e. flowrate, pressure, kilowatt consumption, etc.) are then passed back to the optimization algorithm for use in quantifying the objective function and identifying any violations in the implicit bound constraints. This information is then used to generate an improved set of decision vectors which automatically satisfies the explicit bound constraints and which seeks to minimize the objective function. Once generated, the improved set of decision vectors is then passed back down to the simulation algorithm for subsequent evaluation. This process is then repeated until a specified level of algorithmic convergence is obtained.

To permit increased model flexibility, the computational time interval used in the simulation model is not restricted to be equal to the interval associated with the decision variables. For example, if a 12 hour time interval is used for the decision variables a much smaller time step (i.e. 2 hour, 4 hour, etc.) can be used as the time interval in the simulation model.

## 5. Solution Methodologies

Various optimization methods have been employed by researchers in solving the discrete explicit pump scheduling problem. These methods have varied from traditional gradient based methods to more exotic evolutionary methods (e.g. genetic algorithms). More recently Tufail [17] and Ormsbee and Tufail [19] have proposed using a modified and improved version of a relatively straightforward direct search method [1]. The approach, called the Shuffled Box Complex Method, has a potential advantage over genetic algorithms in that both implicit and explicit constraints may be handled directly without the use of a penalty formulation. Similar to genetic algorithms, their method also pursues an optimal solution along multiple simultaneous search paths, thereby improving the efficiency of the original method.

While the composite explicit pump scheduling problem can also be solved using the same algorithms discussed above, Ormsbee and Reddy [12] found that the unique nature of the problem yields itself to solution via a simple heuristic. Because of the nature of the selected decision variable, the least cost solution to the unconstrained objective function (i.e. Eq. 24) is explicitly known. That is, the least cost solution is one in which no pumps are operated. As a result, the optimization problem reduces to one of finding the set of decision vectors which produce solutions on the composite constraint boundaries that are as close to the origin of the n dimensional solution space as possible. For a given initial set of decision vectors an improved set may be obtained by simply contracting the scalar values in the vectors toward the origin. In the event that a contraction results in a constraint violation, then the vector that produced the constraint violation can be subsequently expanded until a feasible solution is obtained. By continuing to bisect the resulting search vector, a set of decision variables can be obtained which will result in a solution which lies just on the constraint boundary. In the event the initial solution violates an implicit bound constraint, then the solution may be expanded away from the origin along the resulting search direction until a feasible solution is obtained. Due to the nature of the explicit bound constraints, a feasible solution will eventually be found.

It should be recognized that application of such an approach will only result in the best solution that lies on the search direction located between the given initial solution and the origin of the dimensional solution space. However, additional feasible solutions may be obtained by simply replicating the methodology using additional sets of initial decision vectors. Once the final set of feasible solutions is obtained an “optimal” solution may be obtained by simply selecting the best solution from among the resulting feasible solutions.

More recently, Ormsbee and Bradley [17] combined the general search heuristic with a genetic algorithm (GA) by using the GA to continue to generate a new set of potentially feasible solutions. Once again, the algorithm is restricted to that class of problems where the unconstrained optimal solution is known (e.g. where all the pump operating times are zero). In this case, once the new population of solutions is generated, each solution is checked for feasibility. Those solutions that are not feasible are projected up to the constraint boundary from below while those solutions that are feasible are now projected down to the constraint boundary from above (Figure 7). This will then insure that each offspring in the current population is both feasible and locally optimal (e.g. relative to the vector that originates from the origin of the solution space). Once these solutions are generated, the GA is now engaged and a new population is generated (Figure 8). Once again, some of these solutions will be feasible and some will be infeasible. As before, both sets of solutions are projected to the constraint boundary, and the process is continued (Figure 9). Eventually, the GA will drive the solutions to that region on the constraint boundary that is closest to the origin of the solution space and thereby yield an optimal solution. (Figure 10).

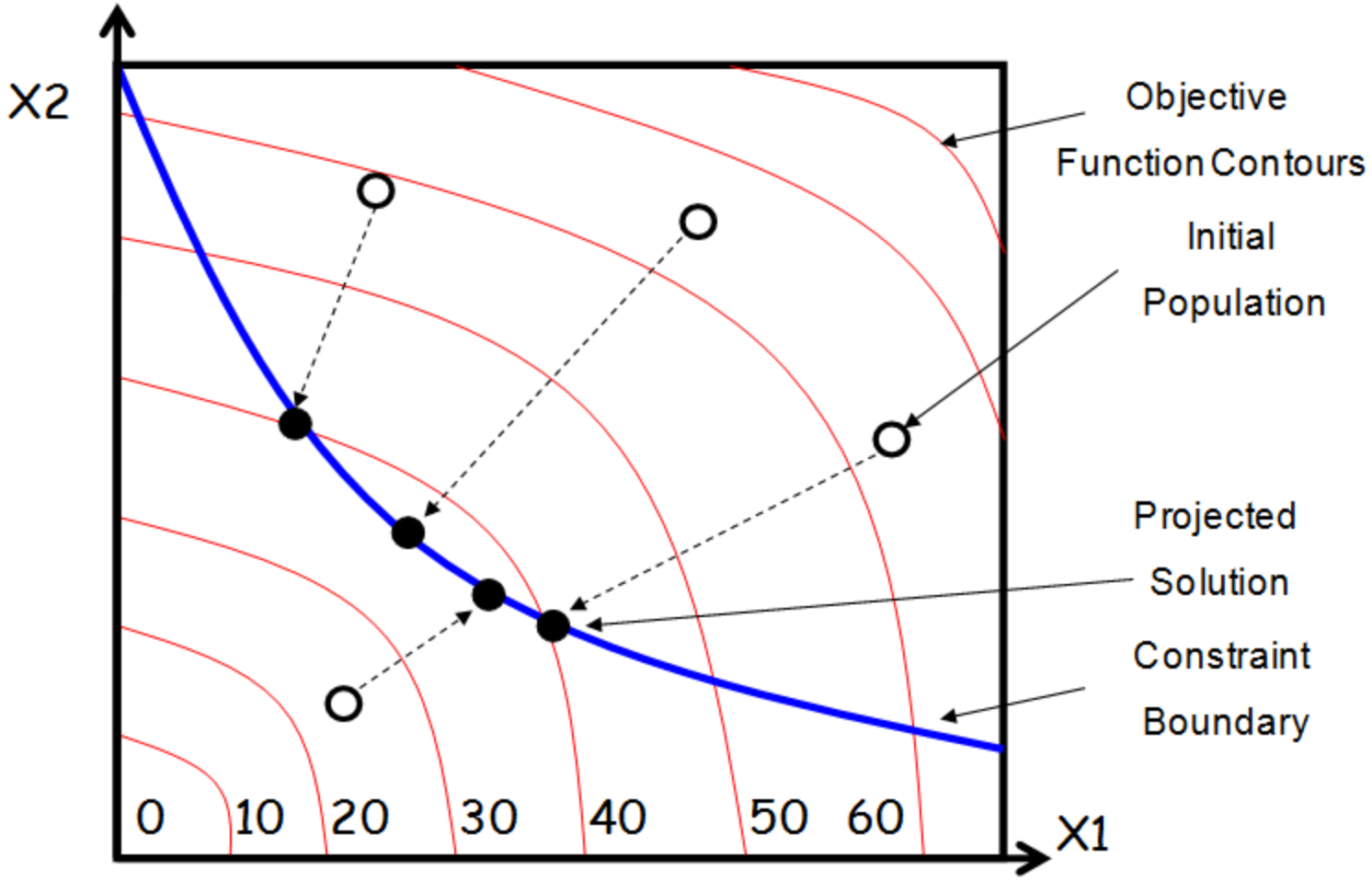



Figure 7. Projection of Initial Solutions to Constraint Boundary (2 dimensional example)

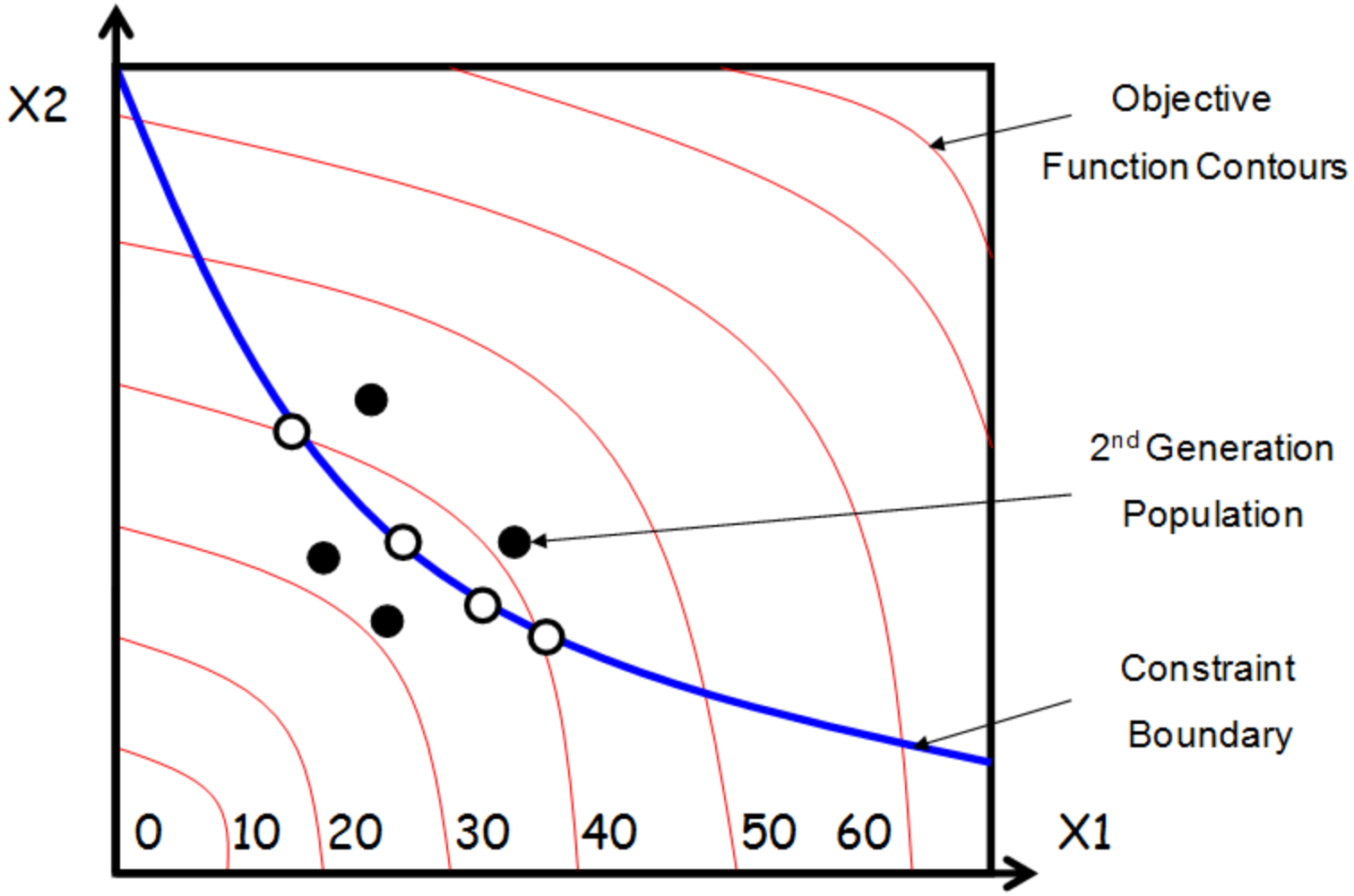



Figure 8. Generation of New Population (2 dimensional example)

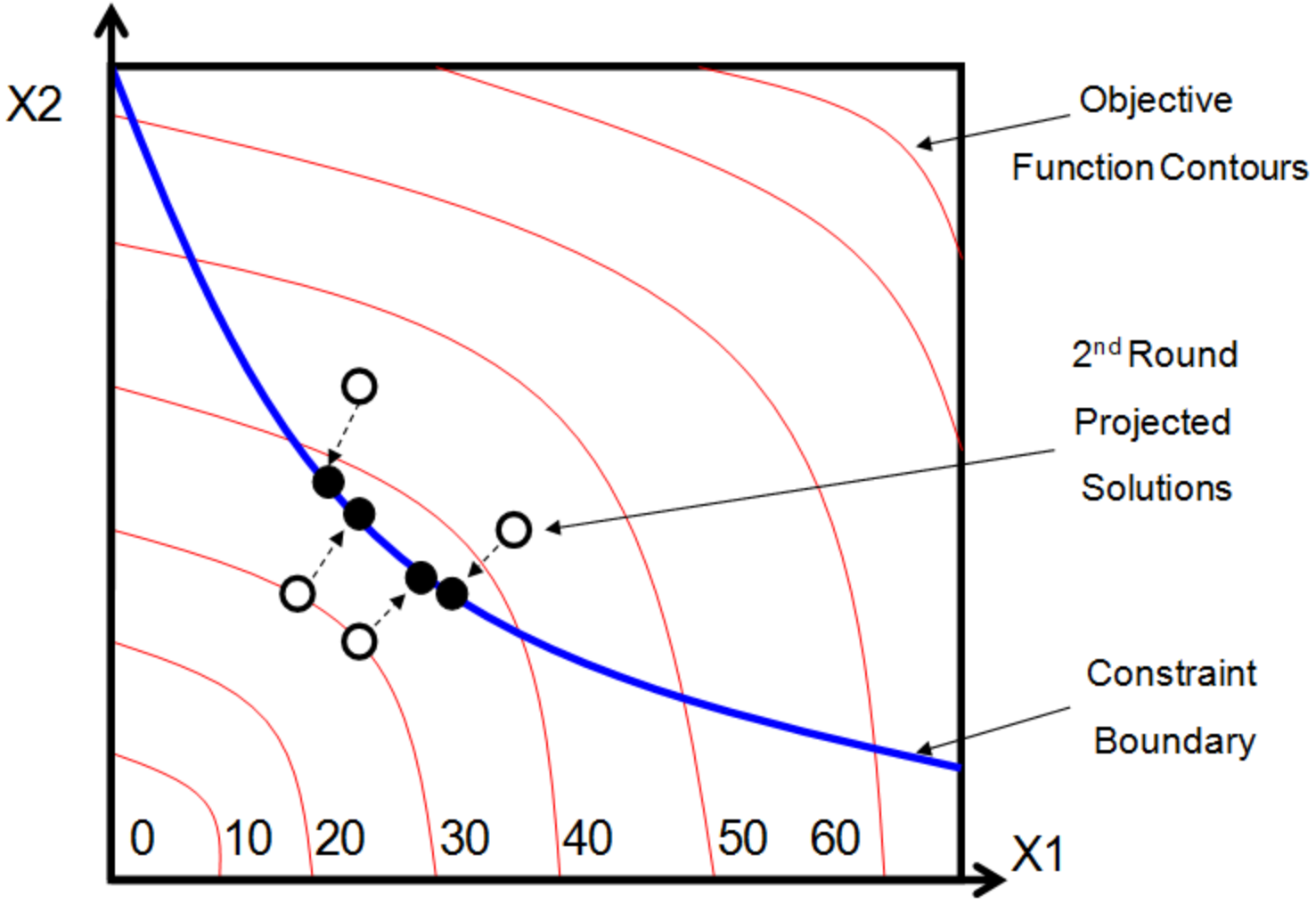



Figure 9. Projection of 2nd Round Solutions to Constraint Boundary (2 dimensional example)

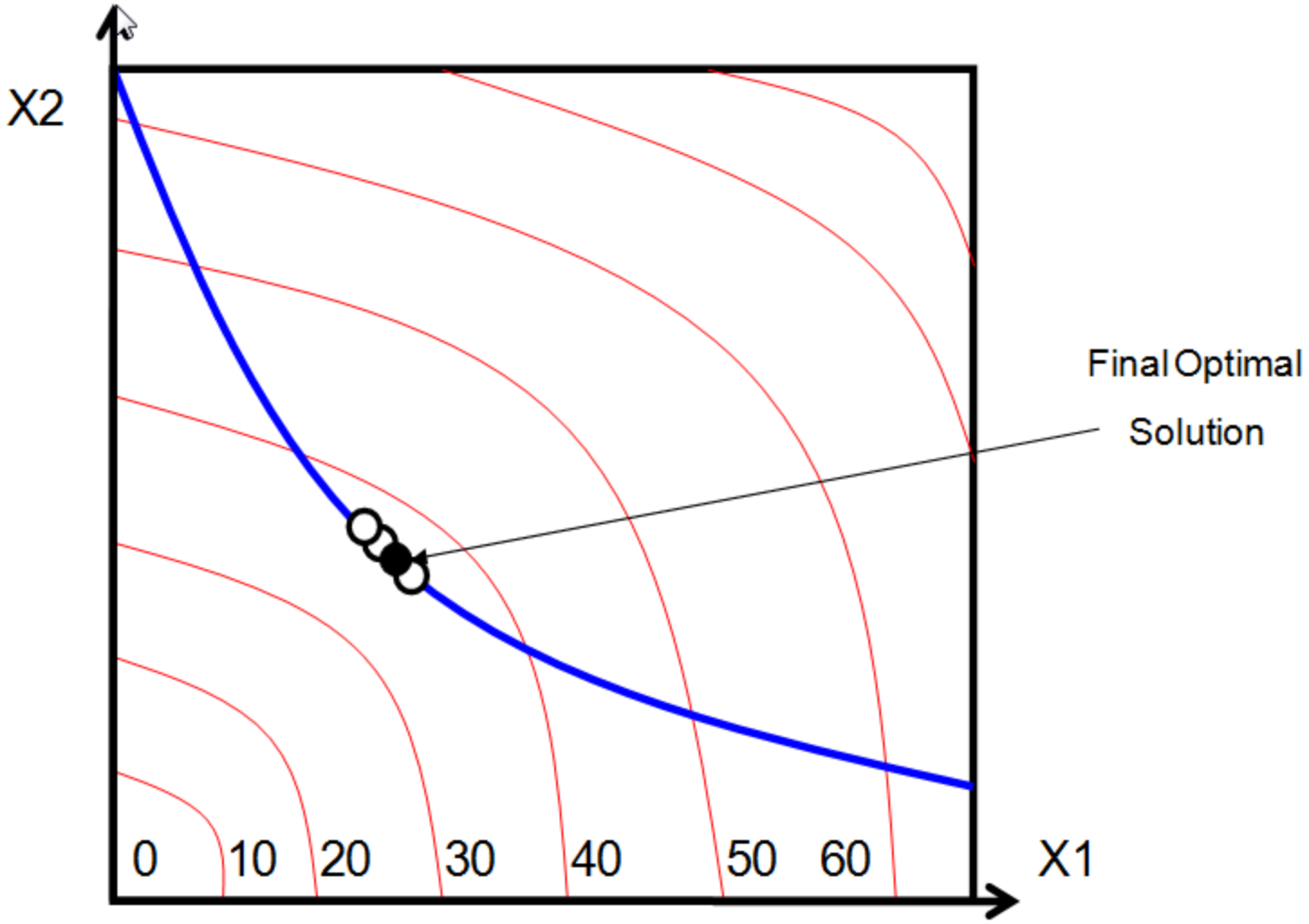



Figure 10. Collapse of Nth Round Solutions to Optimal Solution (2 dimensional example)

**6 Summary**

Three different explicit formulations of the optimal pump scheduling problem have been presented. The resulting formulations may be solved by using either unconstrained methods along with penalty terms or constrained methods that explicitly incorporate the constraints via the mechanisms involved in the algorithm. In either case, the implicit system constraints can be solved directly using an external simulation program which is linked to the optimization algorithm via subroutine calls. Given the complexity of the system being modeled (i.e. water distribution systems), it is felt that the provided formulations as well as the explicitly constrained GA may have applicability to other complex scheduling problems as well.

## 7 References


1. Box, M. J. A new method for constrained optimization and a comparison with other methods. *Computer Journal,* (8), 42-52 (1965).

2. Brion, L., M., *Methodology for Optimal Pumping Stations in Water Distribution Systems,*" Ph.D., Dissertation, University of Texas at Austin (1990).

3. Carpentier, P., and G. Cohen, "Decomposition, Coordination and Aggregation in the Optimal Control of a Large Water Supply Network," *Proc. of the 9th Triennial IFAC World Congress,* Budapest, pp. 3207-3212, (1984).

4. Chase, D., V., "*A Computer Program for Optimal Control of Water Supply Pump Stations: Developing and Testing,"* USACERL Technical Report N-90/14, U.S. Army Corps of Engineers, Construction Engineering Research Laboratory, Champaign, Illinois, (1990).

5. Chase, D., and L. Ormsbee, "Optimal Pump Operation of Water Distribution Systems with Multiple Storage Tanks," *Proceedings of the Amer. Water Works Association. Computer Specialty Conf*., Denver, CO, pp. 205-214 (1989).

6. Chase, D., and Ormsbee, L., "An Alternate Formulation of Time as a Decision Variable To Facilitate Real-Time Operation of Water Supply Systems", *Proceedings of the 18th Annual Conference of the ASCE Water Resources Planning and Management Division*, New Orleans, LA, pp. 923-927 (1991).

7. Coulbeck, B., "Optimisation of Water Networks," *Transactions of the Institute of Measurement and Control*, Vol. 6, No. 5, pp. 271-279 (1984).

8. DeMoyer, R., and L. Horowitz, *A Systems Approach to Water Distribution Modeling and Control,* Lexington Books, pp. 143 (1975).

9. Lansey, K., and Q. Zhong, "A Methodology for Optimal Control of Pump Stations," *Proceedings of the ASCE Water Resources Planning and Management Specialty Conference,* Fort Worth, TX (1990).

10. Ormsbee, L.E. (1985). "OPNET: A nonlinear algorithm for Hydraulic Networks", *Computer Applications in Water Resources, American Society of Civil Engineers,* New York, New York, pp. 739-748 (1985).

11. Ormsbee, L.E., and Lansey, K.E., "Optimal Operation of Water Distribution Systems, *ASCE Journal of Water Resources Planning and Management*, Vol. 120, No. 2, pp. 237-252 (1994).

12. Ormsbee, L.E., and Lingireddy, S. L., "Nonlinear Heuristic for Pump Operations," *ASCE Journal of Water Resources Planning and Management*, Vol. 121, No 4, pp. 302-309 (1995).

13. Ormsbee, L.E., Walski, T., Chase, D., Sharp, W., "Methodology for Improving Pump Operation Efficiency," ASCE Journal of Water Resources

Planning and Management, Vol., 115, No. 2, pp. 148-164 (1989).

14. Ormsbee, L., Reddy, S., and Chase, D., "Comparison of Three Nonlinear Control Algorithms for the Optimal Operation of Water Supply Pumping Systems, *Proceedings of the International Conference on Integrated Computer Applications for Water Supply and Distribution*, Leicester, England (1993).

15. Ormsbee, L., and Reddy, S., "Pumping System Control Using Genetic Optimization and Neural Networks," *Proceedings of 7th IFAC/IFORS/IMACS Symposium on Large Scale Systems: Theory and Applications*, London, England, (1995).

16. Sabet, M., and O. Helweg, "Cost Effective Operation of Urban Water Supply System Using Dynamic Programming," *Water Resources Bulletin*, AWRA, Vol. 21, No. 1, pp. 75-81 (1985).

17. Teegavarapu, R., Bradley, S., and Ormsbee, L., "Probabilistic Goal Driven Watershed Management", *Proceeding of the World Environmental and Water Resources Congress,* Tampa, May 15-19, (2007).

18. Tufail, M., *Optimal Water Quality Management Strategies for Urban Watersheds Using Macro-Level Simulation Models Linked with Evolutionary Algorithms,* Submitted in Partial Fulfillment of the Doctor of Philosophy Degree, University of Kentucky, Lexington, Kentucky, 40506 (2006).

19. Tufail, M., and Ormsbee, L.E., "Optimal Water Quality Management Strategies for Urban Watersheds Using Macro-Level Simulation and Optimization Models, *ASCE Journal of Water Resources Management* (To be published in 2009).

20. Wood, D. J., User's Manual – Computer Analysis of Flow in Pipe Networks Including Extended Period Simulations, Department of Civil Engineering, University of Kentucky, Lexington, KY, (1980)

21. Zessler, U., and U. Shamir, "Optimal Operation of Water Distribution Systems," *ASCE Journal of Water Resources Planning and Management*, Vol. 115, No. 6, pp. 735-752 (1989).

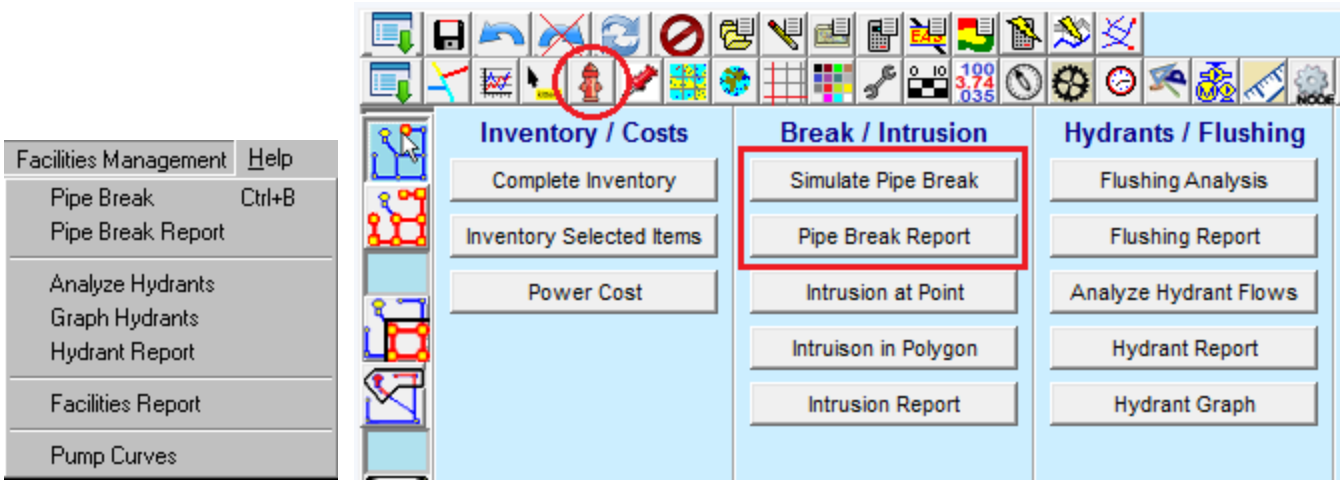
## 1.3.7.11 Pipe Break

Pipe Break Simulation

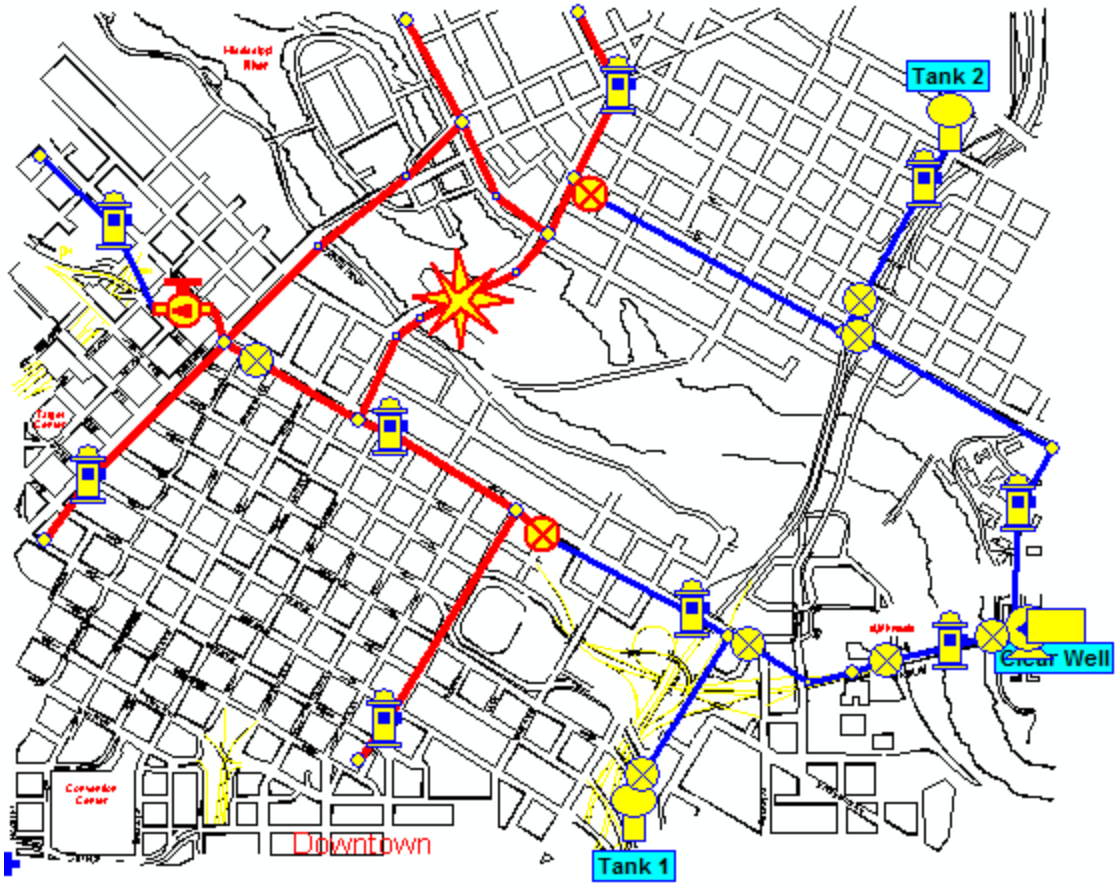
---

See also Pipe Break in Surge Analysis

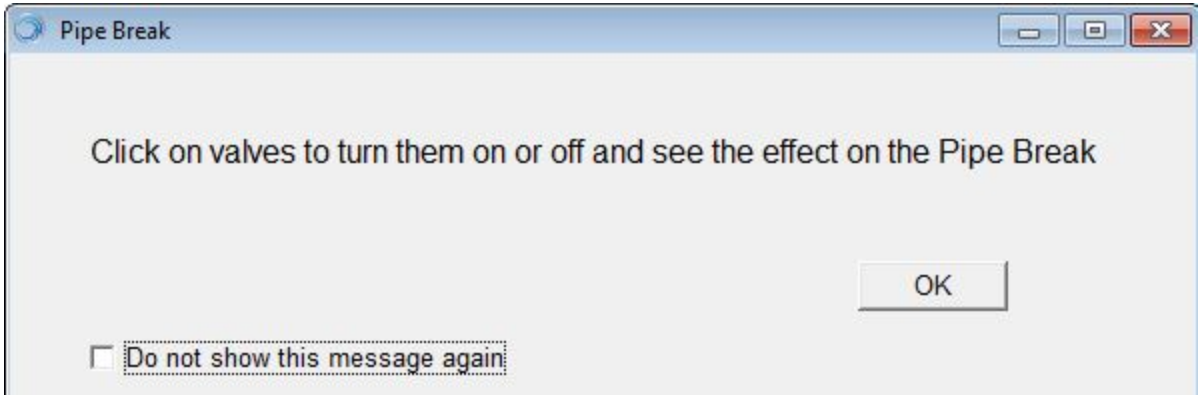
To simulate a pipe break, click on Facilities Management on the Main Menu at the top of the screen. Select Pipe Break.




A special cursor symbol will appear and a pipe may be selected for the simulation. The pipes affected by the break and the on/off or active valves which must be closed to isolate the break will become highlighted.




Note: After simulation, valves may be clicked to open or close. Pipe Break area will be extended or limited accordingly.




Results may also be viewed in the Pipe Break Report. Once the pipe break has been simulated, click Facilities Management in the Main Menu again. Select Pipe Break Report. A report appears as follows:

```
     ------------------------------
       Pipe2000 Pipe Break Report
     ------------------------------


System Name: DEMOREG.P2K
Date: 4/1/2009
Broken Pipe: P-17
Volume of Pipe Break: 6393.929 ft^3


The following valves must be turned off:

1: AV-1
   Address: SE corner of 1st and Washington

2: V-5
   Address: Washington at Metrodome main gate

3: V-6
   Image File: 0|1|1.25876010781671|C:\images\J_VALVEM.BMP
   last maintained: 5/2/2000
   Manufacturer: valco
   Model: gate sr9
   Install Date: 8/2/1989
   Address: SE corner of University and Central Ave.
```

The addresses which appear are Node Title entries (see Node Images). Click Map to return to the map screen. To clear the pipe break simulation from the Map screen, click Clear on the vertical toolbar on the left of the Map screen.

The Volume of the liquid loss due to the pipe break is also included at the top of the report

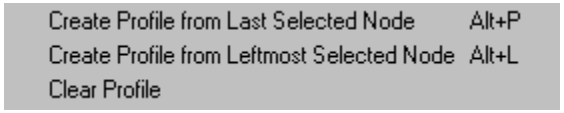
## 1.3.7.12 Profile

These instructions are only for the "Static" or original Profile feature.

For the current or "Animated" Profile, see Animated Profile

See also how to Create a Profile path.

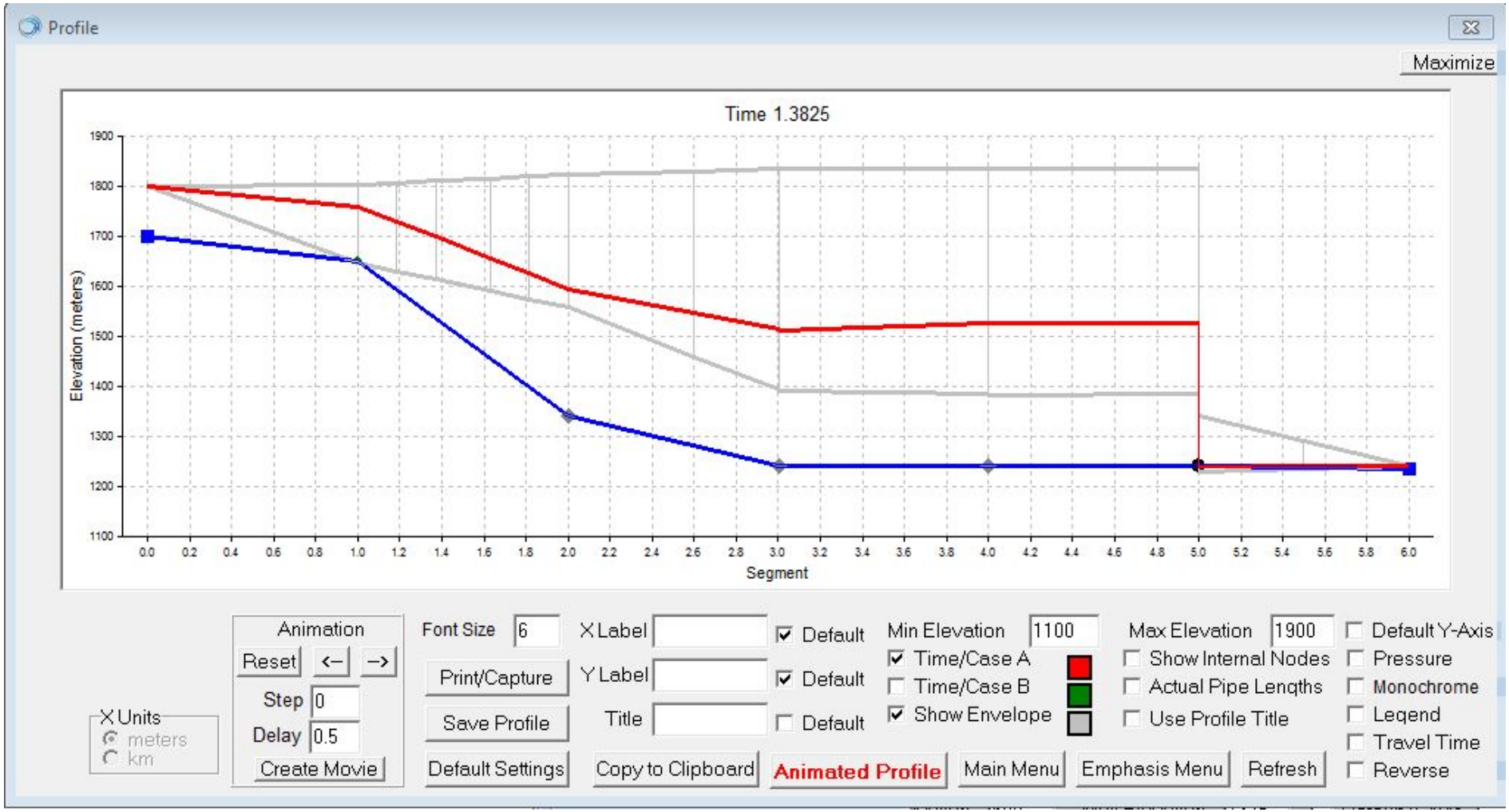
The Pipe2008 and earlier versions profile function allows the user to select two or more nodes in a system and create an elevation and hydraulic grade line graph vs. pipe distance which may be displayed and printed. To use this function, a starting and ending node must first be selected. Nodes in between may also be highlighted to specify a specific path. This is done within Group Mode. Once the nodes are selected, click on Analyze in the main menu bar and select Profile. The following three options will appear:




Create Profile from Last Selected Node will draw the profile from the last node highlighted and back along the specified pipe path.

Create Profile from Leftmost Selected Node will draw the profile from the node that is furthest to the left in the map screen and along the specified pipe path from there.

Clear Profile unselects all highlighted nodes.
Upon choosing one of the first two options, a graph similar to the following will appear:




Several display options exist which the user may specify at this point.

Animation - For a system with several cases or times, will display each result in turn.

Step - allows the user to skip sets of results, e.g., if 2 in entered, the animation will display every other result.

Delay - the amount of time in seconds that each results set is displayed.

Create Movie - If a file has multiple cases, a movie can be created of the animate function based on the Step and Delay defined in the Animation menu. This AVI may be played on certain movie players. There is an option (under System Data/Preferences) to create bitmaps instead of an AVI file. The bitmaps will be in a folder called 'Movie' which you must create first within your Pipe2018 folder.

**Note: AVI movies can be created faster if:**
* **You have more memory**
* **The step size is larger**
* **The Pipe2018 Window is made smaller**

Font Size allows the user to set the font size of the profile labels

Print/Capture - Save to various formats or print to printer. Also, Add to Presentation (Pipe2014 and later versions)

Save Profile - allows the user to save up to 10 profiles. To open a saved profile, select Analyze (main menu in the Map screen) and click on the desired profile..

Default Settings will reset all options to the default settings

X Label user-specified label for x axis

Y Label user-specified label for y axis

Title user-specified title for the profile

Copy to Clipboard copies the profile to the clipbaord.

Min Elevation and Max Elevation allow the user to set the y-axis limits.

Time/Case A, and Time/Case B allow the user to turn the hydraulic grade line portion of the graph on and off, displaying results A or B, which are selected in the map screen prior to creating the profile.

Show Internal Nodes includes all internal nodes in the graph.

Use Profile Title overrides the title entered in the Title field with the name given to a profile when saved.

Show Envelope allows the user to show the hydraulic grade line envelope.

Actual Pipe Lengths allows the user to toggle the x-axis between displaying each pipe link in equal sections for easy reference or displaying the pipelinks in proportional lengths.

Reverse reverses the x-axis of the graph.

Default Y-axis chooses an appropriate y-axis range for the current profile.

Pressure toggles the profile to display pressure instead of the default hydraulic grade line.

Monochrome causes the graph to be displayed in shades of grey.

Legend - Creates a legend which includes references all items displayed on the profile

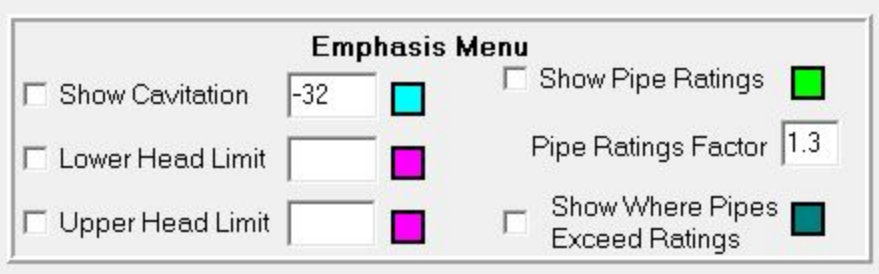
Travel Time - Calculates and displays the travel time through the profiled pipe section for the selected time/case A.

Refresh may be used to redraw the graph when changes are selected.

Lower Head Limit and Upper Head Limit may be used to add an upper and lower head line to the graph for the user's reference.

Show Cavitation - If the current result being displayed fall below the cavitation level which is specified in this field. Note it does not use the cavitation limit set under Simulation Specs.

**Emphasis Menu**




Show Caviation - emphasize where along the pipeline cavitation occurs using this color.

Lower/Upper Head Limit - a head line will be drawn at the level specified in this field when the box is checked.

Show Pipe Ratings - The pipe rating defined in the Pipe Type Table or in the Data Table is in units of psi or kPa. Checking this box will show the rating limit on the profile.

Pipe Ratings Factor - multiplies the pipe rating by this number.

Show Where Pipes Exceed Rating - Emphasizes the pipe in this color where rating is exceeded.

The symbols used in graphing are as follows:

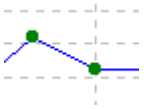
junction node

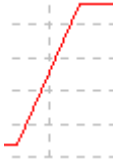
device, such as PRV

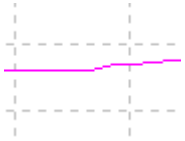
pump

tank or reservoir

 internal node


plot of elevation vs. pipe distance


plot of hydraulic grade line (or pressure) vs. pipe distance


plot of upper and lower head limit

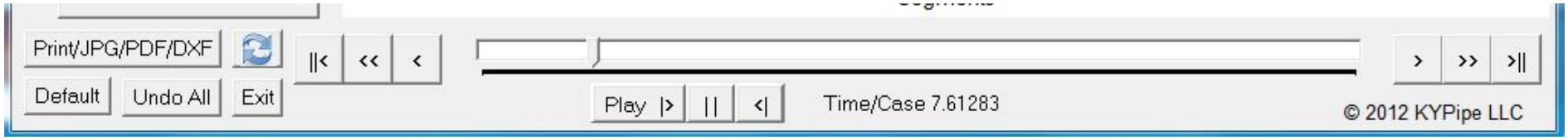
## 1.3.7.13 Profile Animated

Pipe2010 and later versions include animated profiles. The former profile feature is still available and may be specified as the default profile function. To use the former profile go to Preferences . In the Map/Layout section, check the box which says "Non-Animated Profiles". If this box is unchecked then the Animate Profile will be the default.

See also How to Create a Profile Path.

Below is an image of a typical profile. The system depicted is a Surge analysis of a system for which a pump has tripped and a bladder tank has been employed to provide surge protection.

Pipe Animation 1.7
Style
Normal
Simple
Static
Appearance
Axis
Labels
Envelope
Animation
AVI
Emphasis
Water Size (50%)
Pipe Size (75%)
PipeColor
Gray
Cyan
Blue
256 Color Gray
Cavitation Pressure
-14
Hydrant Size
On/Off Valve Size
Other Size
SDO Size
SDO Shape
Animate
Symbol
Valve Size
Pump Size
Elevation (ft)
Segments
psi
47
20



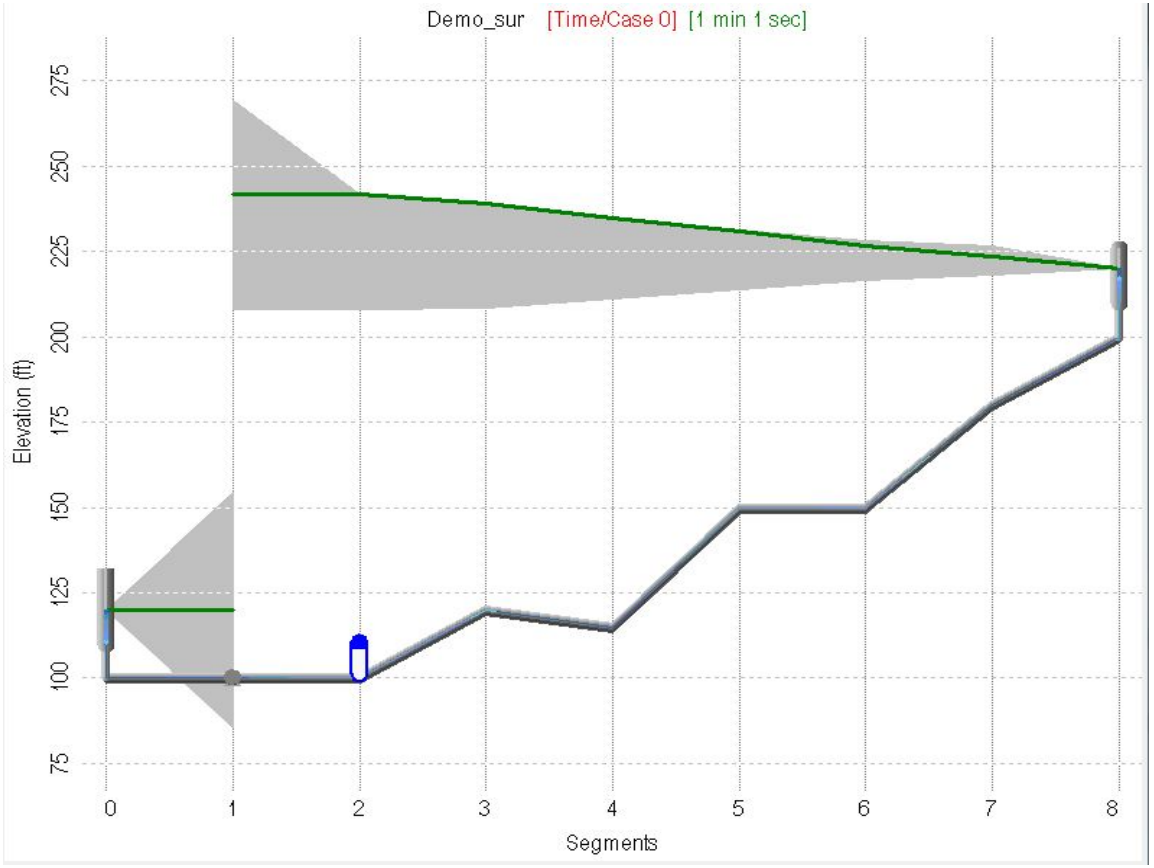

There are many options for controlling the appearance and function of the Animate Profile

Normal/Simple/Static

Normal - as shown above.

Simple - appears a more simple-lined version shown below.  Ideal for inclusion in reports.

Static - reverts to the original profile feature




Simple Profile

Appearance

(**Simple Mode - Symbols on/off**) - show or hide the symbols for devices.

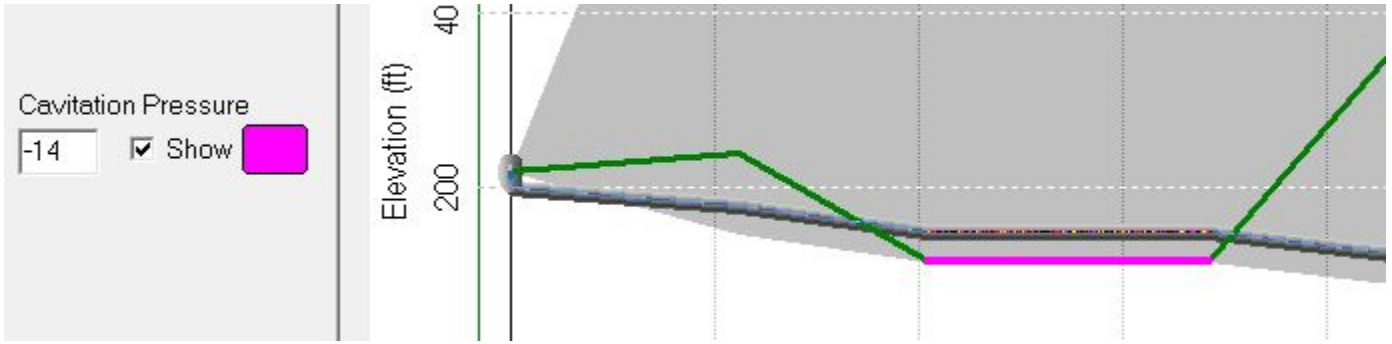
Water Size - in animating water flow, color-gradiated, moving segments are used to indicate direction of flow. The number of segments can be increased or decreased.

Pipe Size - the width of the pipe can be made narrower or wider. The width of the pipe also affects the appearance of the pumps, tanks, reservoirs and surge devices.

Pipe Color - the four color choices include a 256 color grey which provides a preview of what an AVI of the animation will look like. The file size of the

generated AVI will be much smaller in 256 colors.

Cavitation Pressure - cavitation is depicted in the animation as black space (with colored dots) in the interior of the pipeline. The value entered for Cavitation Pressure will determine when and where cavitation is indicated in the profile. A cavitation **Emphasis** line can be displayed and the color selected.




Element sizes - Many of the elements included in the pipeline may be sized using these individual scale. A'0' makes the element invisible. Important note: The size of the pipe (see Pipe Size above) will also affect the size of certain elements, pumps, tanks, reservoirs and surge devices

Axis

X-Axis Type - Choosing "Equal Segments" will divide the pipe equally between elements in the pipeline regardless of the distance between the elements. This makes the elements most easily viewable in the profile. "Actual Pipe Length" uses the horizontal pipe length. "Graphcial Length" displays the angled length of each pipe segment.

X-Axis Units - ft (m) or miles (km). Only applicable when Graphical or Actual type X-Axis is used.

Y Axis may be defined using the default which will fit the head envelope within the viewed area or the user may uncheck the default box and enter max and min y axis data.

For the following options, if the selection is not reflected in the profile, use the Refresh button at the bottom of the Profile menu.

Show lines at nodes - vertical lines where each node is located in the pipeline

Lines at X divisions - vertical lines at each distance division or pipe segment

Volume Graph - for Surge protection devices, a graph of changing volume is added

Pressure - Profile is plotted to show pressure envelope instead of Hydraulic Grade line

Show Ground - show or remove green ground color

Show Sky - show or remove blue sky color

Reverse Profile - switch the pipeline ends, but will not take effect until the profile window is closed and a new profile is generated.

**Show Axis Lines** - adds a line at the axes

**Box Around Graph** - draws a box around the profile

Labels

Font - to modify the font, click on the Font area and a font setup menu will appear.

Top Label - Pressure, head, HGL results, **Name, and X Coordinate** or name for each node may be displayed as text across the top of the profile screen by selecting one of these options. Note if "pressure" option is checked under Axis, head and HGL will not be available.

Bottom Label - Velocity and flow results or name for each pipe may be displayed as test across the bottom of the profile by selecting one of these options.

Show Units - determines whether the units for the Top Label and Bottom Label are displayed.

Title - may be added to the top of the profile

Show Title - used to turn title off and on.

Show Time Case - if checked the time/case currently represented in the profile is displayed along with a title, if specified.

Show Travel Time - travel time of water in pipeline from beginning of simulation based on velocity and pipelength in each pipe segment.

Envelope

Colors - the number of colors used to gradiate the head envelope may be specified. Up to seven colors may be used. To change the individual colors click on the color box and a color palette will appear. The range for each color is automatically determined based on the number of colors chosen, unless the Automatic Values box is unchecked.

Key - a Y-axis key for pressure results may be displayed or hidden using this check box.

Style/Lighten - are used to modify the appearance of the head envelope.

Default Colors - resets the envelope gradient colors

Automatic Values - the gradient pressure values are automatically assigned unless this box is unchecked. If unchecked Auto Fill and Range Fill options become available. See below

Auto Fill - if Automatic Values is unchecked, this option can be used to automatically assign pressure values to the number of gradients selected.

Range Fill - if Automatic Values is unchecked, an upper pressure value may be entered into the bottom pressure field and a lower pressure value in the top pressure field. Then this button is used to fill in the pressure fields in between with equal intervals.

Show HGL Line - if this box is unchecked the HGL profile is not shown within the envelope.

Animation

Animate Water - The profile may be depicted as a solid pipe or as a cut-away with animated water flow/

Water Speed - this determines the speed at which the water animation moves to depict flow.

Total Cycle Time - sets the time it takes to animate the simulation from start to finish, one cycle.

Cycle - determines how the animation cycles, "Loop" goes from beginning to end of the analysis repeatedly, "None" goes through the cycle once and then stops, "Rock" goes from beginning to end then cycles backwards to the beginning repeatedly.

Surge Animation - within closed surge tanks and bladder tanks, the range of the volume of air calculated in the analysis may be depicted to scale (using the "From Zero to Max" options) or the measured range can 'fill' the animated device (using the "From Min to Max" option). In other words, the "From Min to Max" option 'stretches the actual range across the entire animated device.

Return to Static Profile - initiates the former method of profiling. To set the default profile method, go to System Data | Preferences. Under Map / Layout, use the "Unanimated Profiles" check box.

AVI

Animation Type - the four option affect the file size and quality of the captured animation

Time Range - the animation may be created for the entire simulation or just for the flow and conditions at a single time.

Frame Rate - This affects the size and quality of the AVI that is created.

Pixel Height/Width or Screen Size - the AVI may be sized using the specified number of pixels or according to the size of the window.

AVI Lenth / Use Cycle Time - If the Use Cycle Time box is checked the time specified under Animation/Total Cycle Time will be used. If it is unchecked, then a time length for the AVI may be specified.

Estimated Size - this is the size the created AVI will have.

Create AVI - Make a movie based on the specified settings.

Emphasis

Lower Head Limit - enter a value which will be plotted on the profile. Used to show where the pressure results fall below a certain value. You may change the color by clicking the color block.

Upper Head Limit - enter a value which will be plotted on the profile. Used to show where the pressure results fall above a certain value. You may change the color by clicking the color block.

Show Pipe Ratings - if the pipe data includes Rating values, these value may be plotted on the profile. You may change the color by clicking the color block.

Pipe Ratings Factor - multiplies all pipe ratings by this value and then plots the new pipe rating. You may change the color by clicking the color block.

Show Where Pipes Exceed Rating - Changes the color of the HGL/Pressure results line to indicate where the pressure is greater than the plotted pipe ratings line. You may change the color by clicking the color block.

Previous - Shows the results line for the analysis run immediately prior to the current profiles analysis. You may change the color by clicking the color block.

Previous Envelope - Shows the head envelope lines for the analysis run immediately prior to the current profiles analysis. You may change the color by clicking the color block.

**Case B and Case C** - the pressure/HGL line for the selected case can be displayed along with the current case (selected in the Results Selector bar of the program window).

Show Legend - displays a key for any of the above emphasis options which are turned on.

Some Profile Emphasis Options

File Menu

Copy to Clipboard - an image of the current time is copied and may be pasted into another application.

Load - Load a previously saved profile

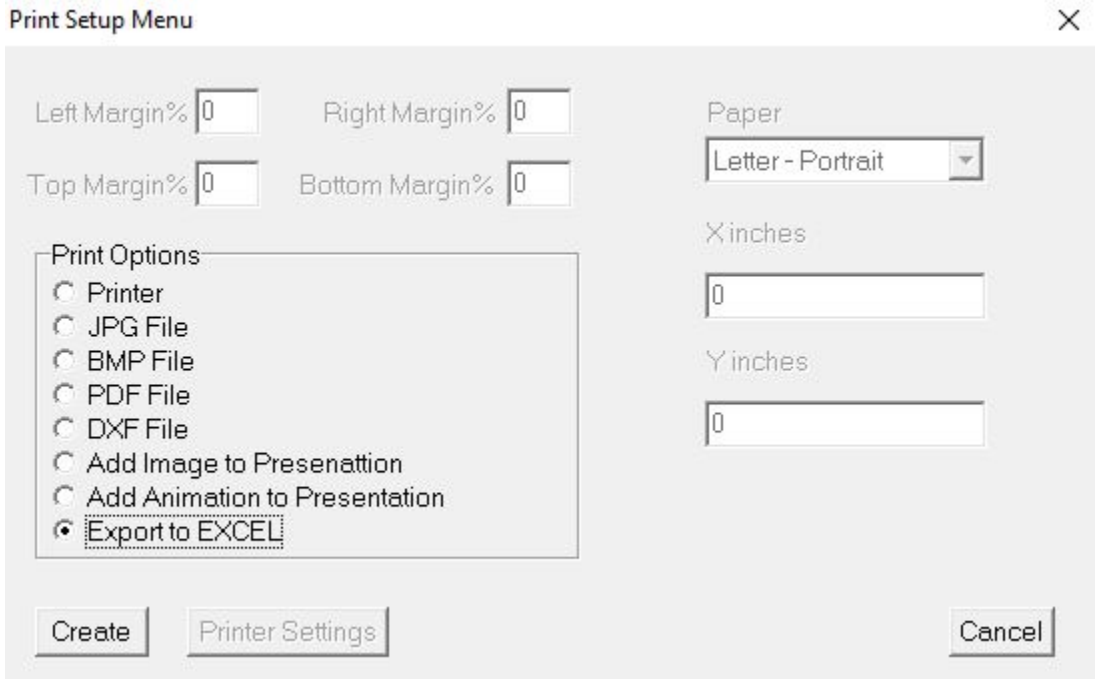
Save/Save As - Save current profile.  Saved profiles have a ".PRO" file extension.

Exit Without Saving

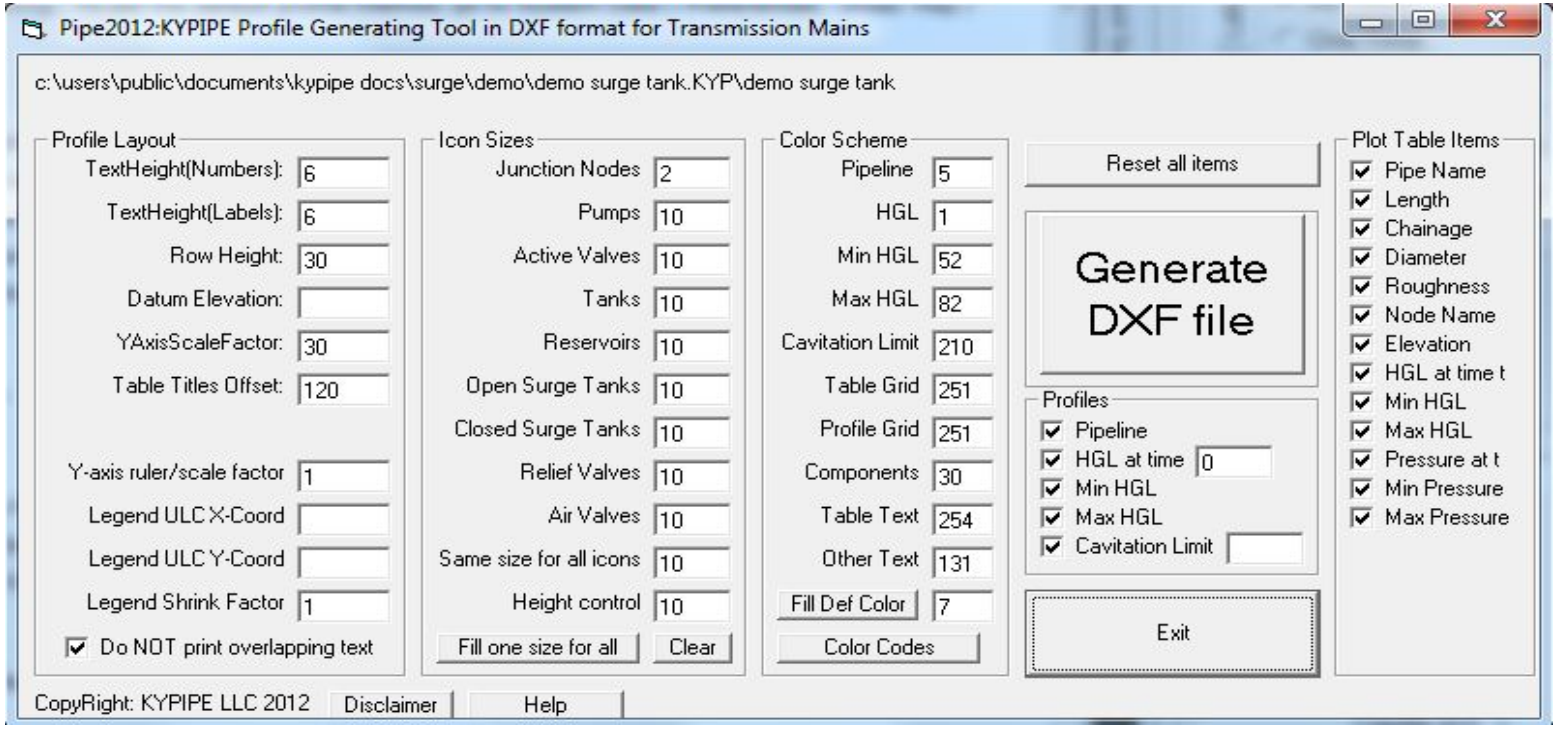
Exit and Add to Presentation - add current profile to Presentation Builder

Load "Previous" - add to the current profile the envelope and results plot for a previously saved profile.

Print/JPG/PDF/DXF - open a print menu with the following options, which includes **Export to Excel** and **DXF File**:



"DXF File" opens the following export menu:




.

Refresh - refreshes the profile so any change that has just been made will be displayed.

Note, all changes to the profile are automatically saved for that model file upon exiting. They will be used next time the profile for this model is generated.

Default - returns the profile to the default settings.

Undo All - cancel all changes made during the current profile session.

Exit - exits the profile animator

These buttons are used to select the case/time displayed on the profile. The outside buttons move the to beginning or end of the simulation, the double arrows select every fourth case/time, and the single arrows select each case/time.

Slider - may be used to select a case anywhere in the simulation by clicking and dragging the bar.

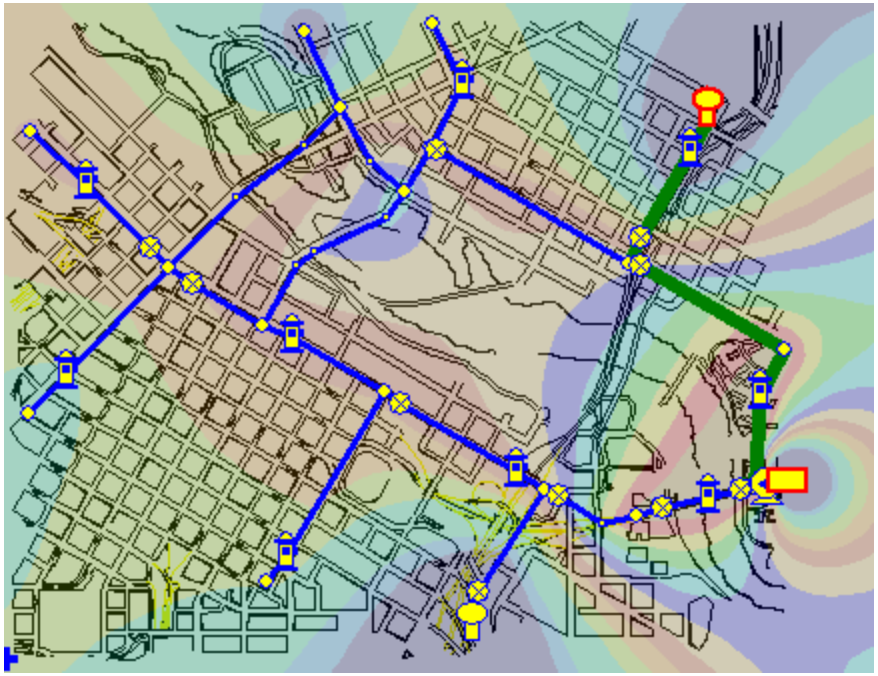
These three buttons are used to play the profile animation to the left, to the right, or pause the animation.

### 1.3.7.14 Creating Profiles

# Creating Profiles

## Custom Profile Path

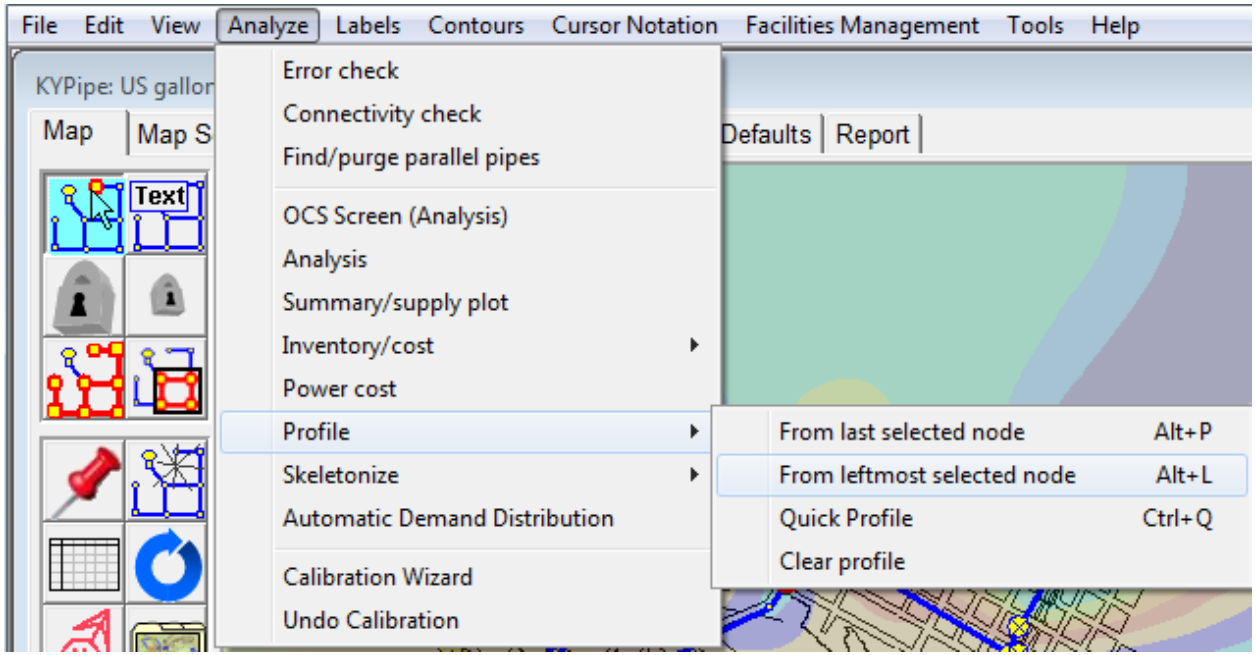
To select a pipe profile path, go into Group mode . Select the starting and ending node of the profile path.



Then choose to create the path from the “Last” selected node or the “Left” (leftmost) selected node.

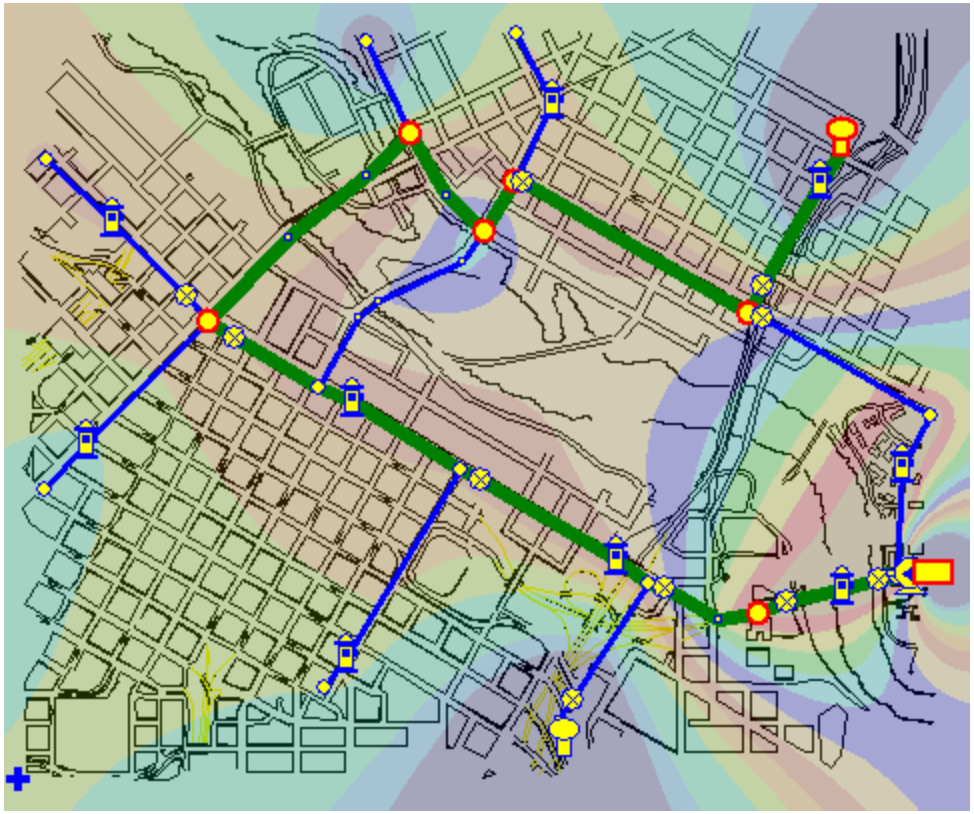
KYnetic - go to Tables, Graphs, and Profiles menu. Click Left or Last button:

Classic – Go to Analyze – Profiles and select the “leftmost” or “last” option:




## Complex Paths

If the system has loops and the desired path is not the shortest between the two nodes, multiple nodes along the path may be selected.



IMPORTANT NOTE: If selecting more than two nodes in group mode, make certain one of the two nodes which are at the **end of your path is clicked on LAST** or the path will not be able to be drawn.

## Custom Profile Path in TranSurge

If it is desired to make a custom path in TranSurge, toggle to Surge

.

Once in Surge, follow the steps described above. Select nodes in Group mode, then go to Tables, Graphs, and Profiles menu and select the Left or Last button. Once complete, toggle back to TranSurge.

## Quick Profiles

KYnetic

Classic – go to Analyze – Profile – Quick Profile.

To determine the Quick Profile path, the program will consider for starting and ending points all reservoirs, tanks, and branched ends. Internally it will draw the shortest path from the leftmost node that is considered to the rightmost considered node.

**Longest Path Option**

In the Preferences menu (KYnetic , Classic – System Data / Preferences) , an option exists to ensure that the Quick Profile is drawn on the longest path available path given certain criteria. Considering Reservoirs, Tanks, and Branched ends (same as with standard Quick Profile) for the starting and ending points of the profile, the program draws the most direct paths between all of these considered nodes. Then the longest path of them all is selected for the profile.

Note this process can take a while to complete on bigger systems.

## 1.3.7.15 Pump and System Curves

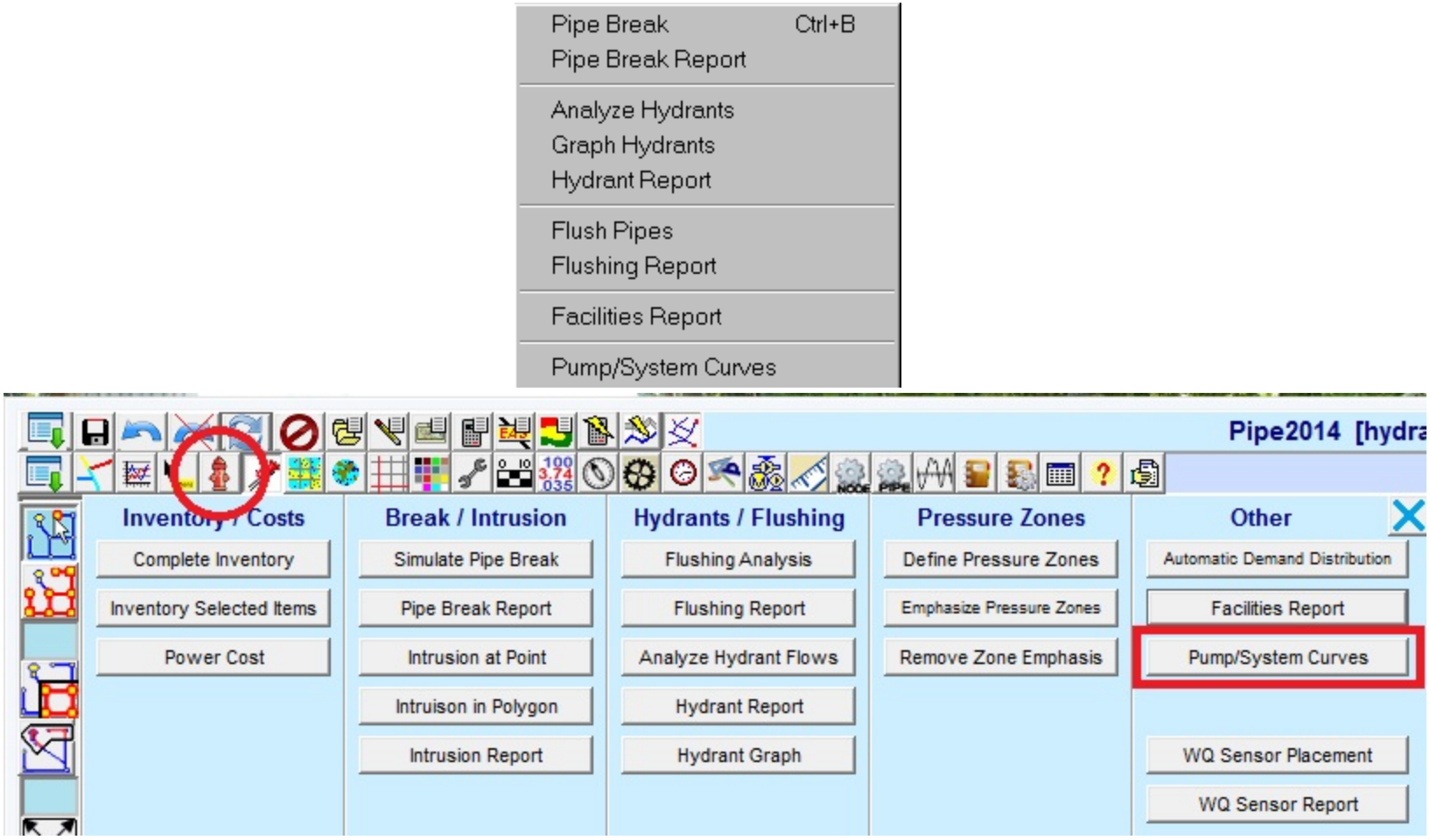
Pump and System Curves

---

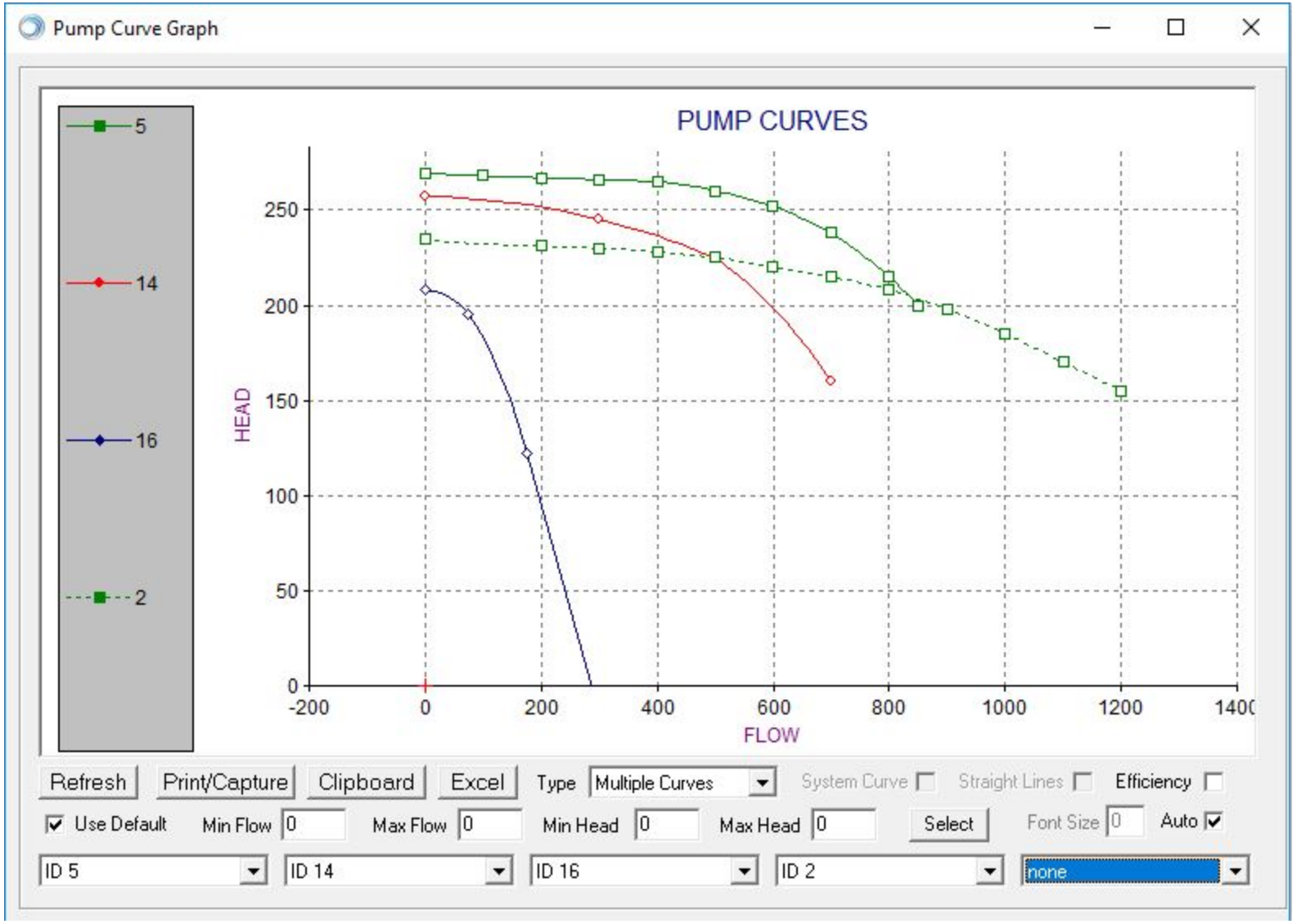
# Pump Curves

For information about entering Pump data, see Pump Data.

Pipe2018 has the ability to provide a plot of pump curves (head/flow data). Pump curve data is data entered by the user. A plot of this data is readily available by clicking on Facilities Management in the Main Menu and selecting Pump Curves.




The following graph appears:




The graph of the pump curve represents the actual data used in the KYPipe calculation.

Refresh - refreshes the graph

Print/Capture - prints the graph or captures to JPG, BMP, or PDF. Also can be used to add the plot to a Presentation.

Clipboard – copies the image of the graph to the clipboard for pasting into other applications

**Excel - (added in Pipe2018)** exports the pump data point to and Excel file.

Type - multiple curve option, see below.

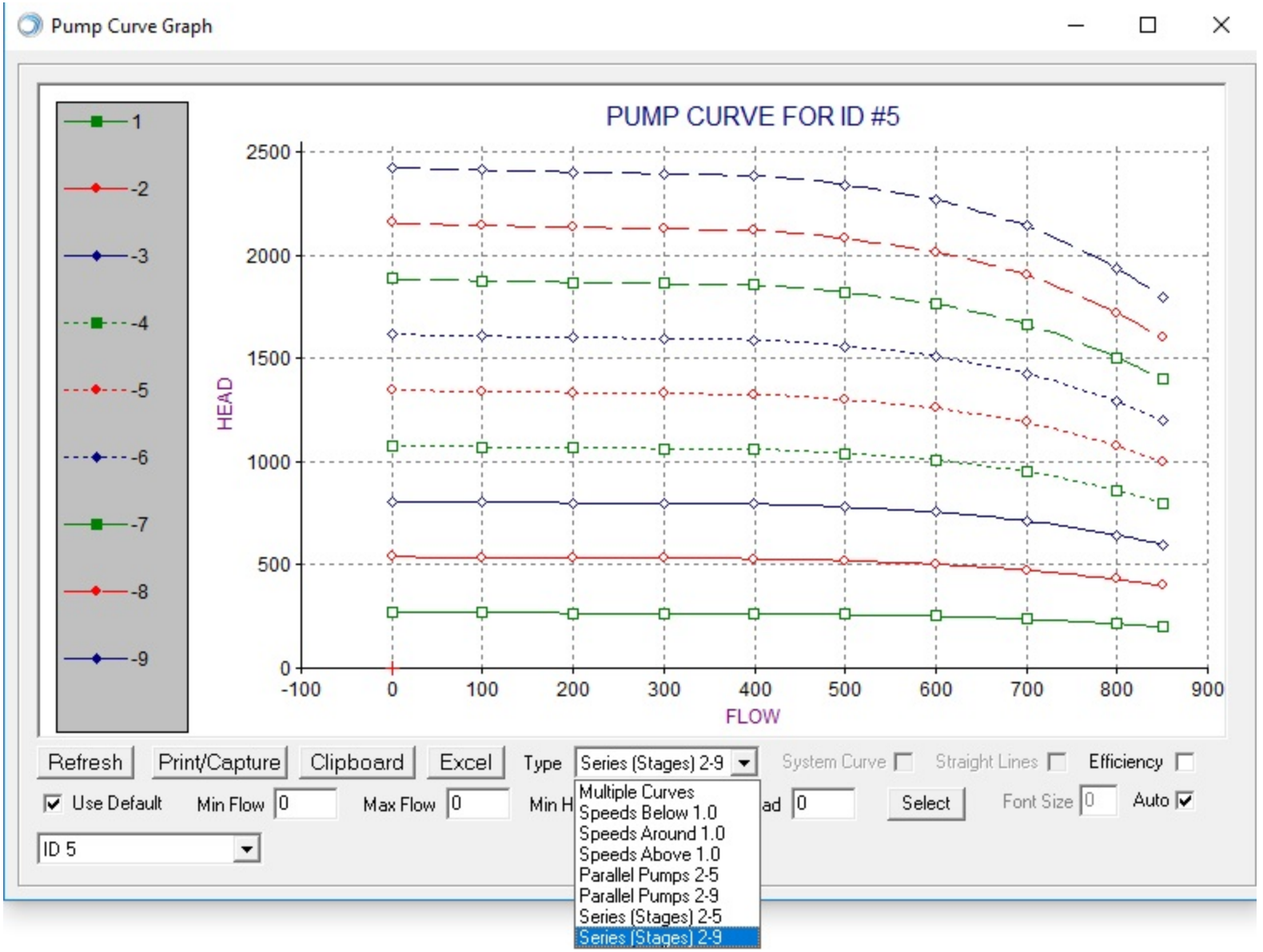
System Curve - displays the calculated system curve. See below. See also System Curve for Surge

Efficiency - when this box is checked, the efficiency of the selected pump is graphed

Font Size - uncheck the Auto box in order to customized the font size.

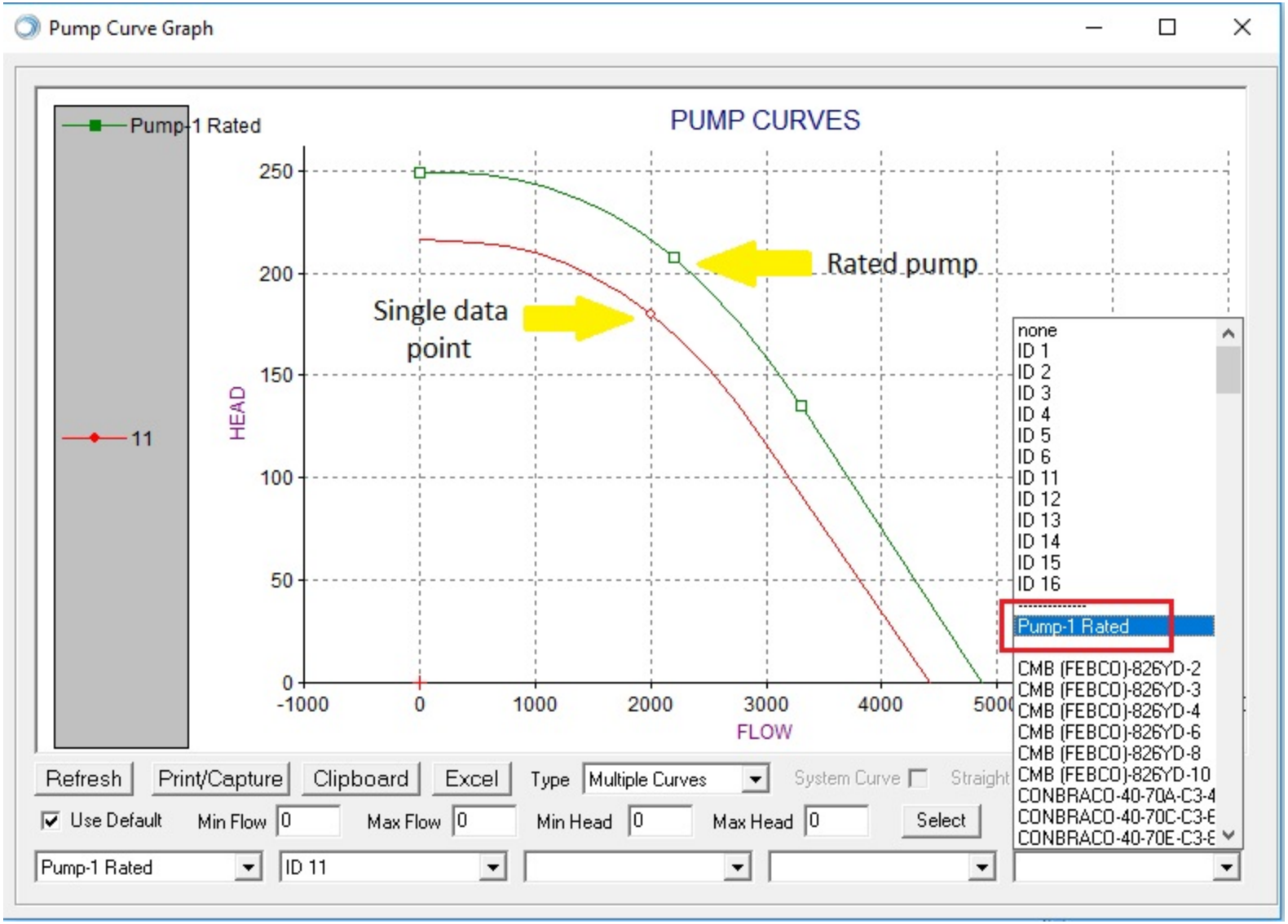
Max/Min - a minumum and maximum head and flow can be specified for the graph if the Use Default box is unchecked.

**Type** - There are several graph **Type** options for displayed the performance for mulitple combinations of a single pump curve. Curves up to 9 in parallel and series was added in Pipe2018. Select the curve in the first drop down selector then set the combination in the Type drop-down selector as shown:


Pump Curve Graph
PUMP CURVE FOR ID #5
1
-2
-3
-4
-5
-6
-7
-8
-9
HEAD
2500
2000
1500
1000
500
0
-100
0
100
200
300
400
500
600
700
800
900
FLOW
Refresh
Print/Capture
Clipboard
Excel
Type
Series (Stages) 2-9
System Curve
Straight Lines
Efficiency
Use Default
Min Flow
0
Max Flow
0
Min H
ad
0
Select
Font Size
0
Auto
ID 5
Multiple Curves
Speeds Below 1.0
Speeds Around 1.0
Speeds Above 1.0
Parallel Pumps 2-5
Parallel Pumps 2-9
Series (Stages) 2-5
Series (Stages) 2-9

**Rated Pump Curves and ID with single data point.**

As of Pipe2018, Rated Pumps and pump IDs with a single data point may also be plotted as shown below.  Rated pumps appear below ID list as indicated.

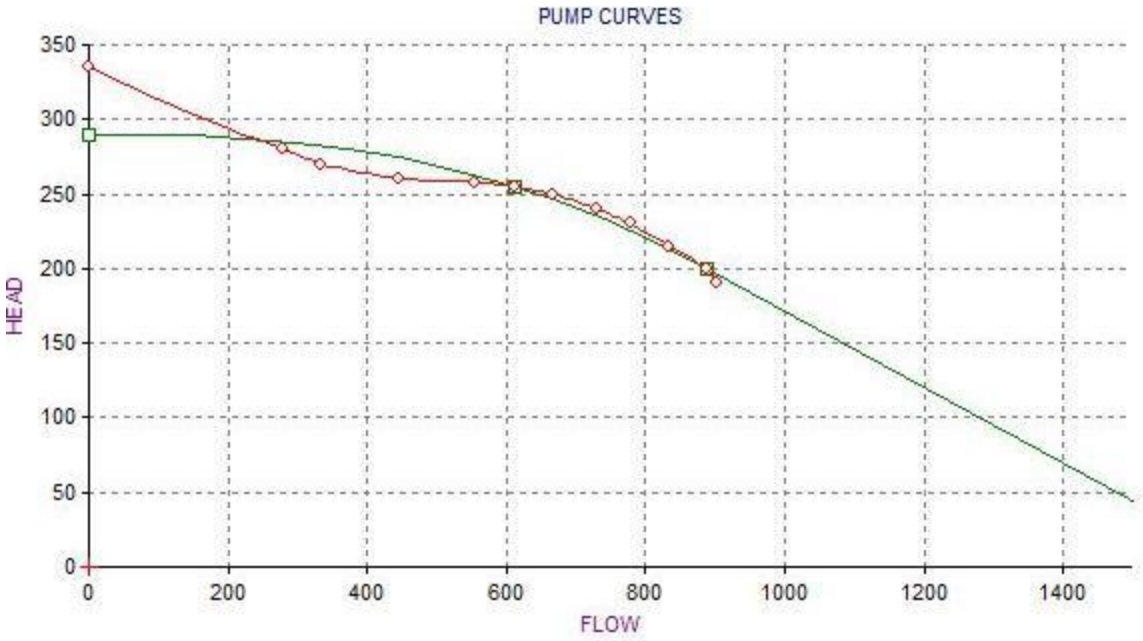



**Convergence Issues Due to Pump Curve Shape**

Convergence problems (maximum number of trials or accuracy not attained errors) can occur when using a pump curve which is not concave downward

throughout its entire range. For example, the curve shown in red has a substantial region where the curve is concave upward. This particular curve would not allow a correct solution to be computed. The curve in green is concave downward throughout its entire range and will produce a good solution. Note that even though the two curves characteristics are quite different in the range up to 600 gpm they are very similar beyond 600 gpm. Thus solutions beyond 600 gpm will closely match the operating conditions for either curve.

If you encounter a problem while using a curve with concave upward regions (like the red curve) you should replace the curve with one which is concave downward throughout and closely matches the initial and final steady state operation




## System Curves

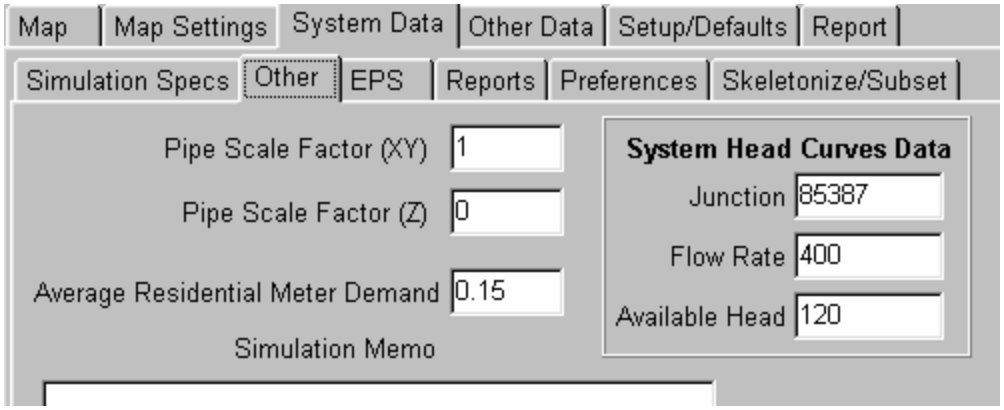
Available for Surge with release Pipe2018

A system curve is a set of head/flow data which describes the performance at a given node in a piping system. A system curve is useful, for instance, in determining the maximum flow the system can handle based on the rating of the pipes and is useful for determining the pump requirements and sizing a pump for that location. A tank, reservoir, or sprinkler needs to be in the region of where the system curve calculation is taking place. The system curve calculation functions by forcing flow into that region and so there needs to be a fixed grade node where this flow can go.

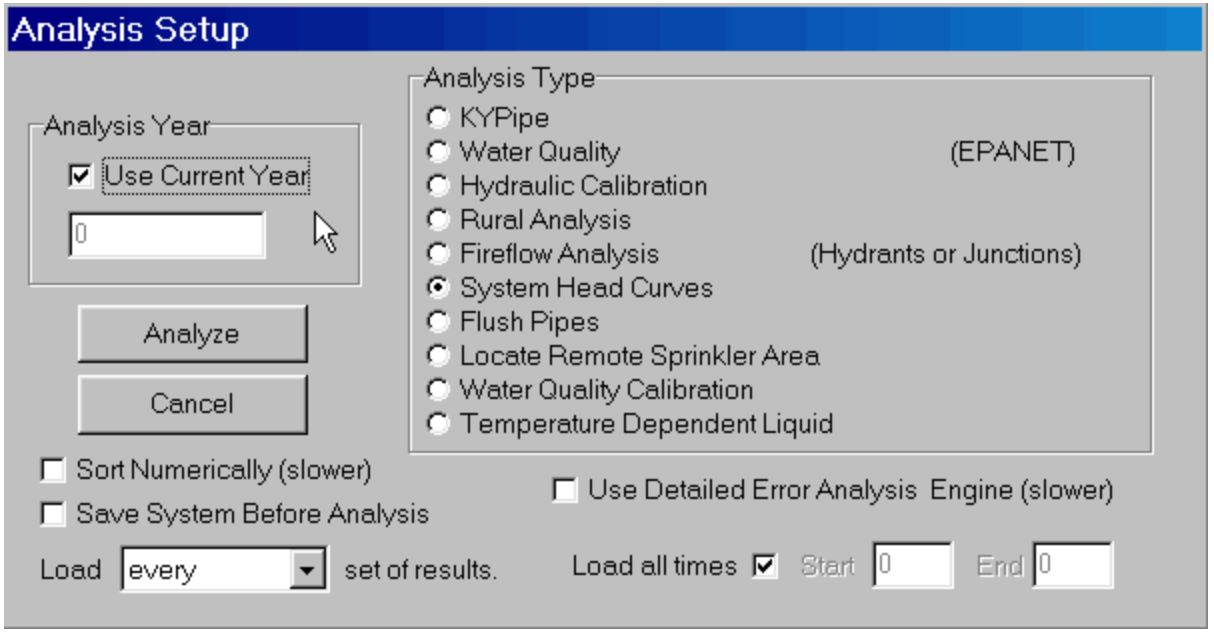
To obtain a system curve, first choose the node at which the curve is to be generated and enter the **Junction** name, the **Flow Rate** which is desired at this junction, and **Available Head** under **System Data|Other**, in the **System Head Curves Data** box as shown below. The **Available Head** is equivalent to the head that will be available on the suction side of a pump at this location and it is recommended to enter this value if the system curve is calculated near a reservoir or some other constant head location.

**Important:** Make sure that no flow is coming from a pump or supply upstream of the designated node. This is normally accomplished by turning upstream pumps OFF.

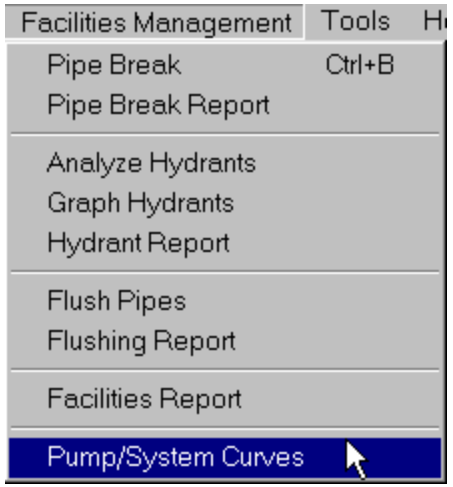
KYnetic Other menu icon:

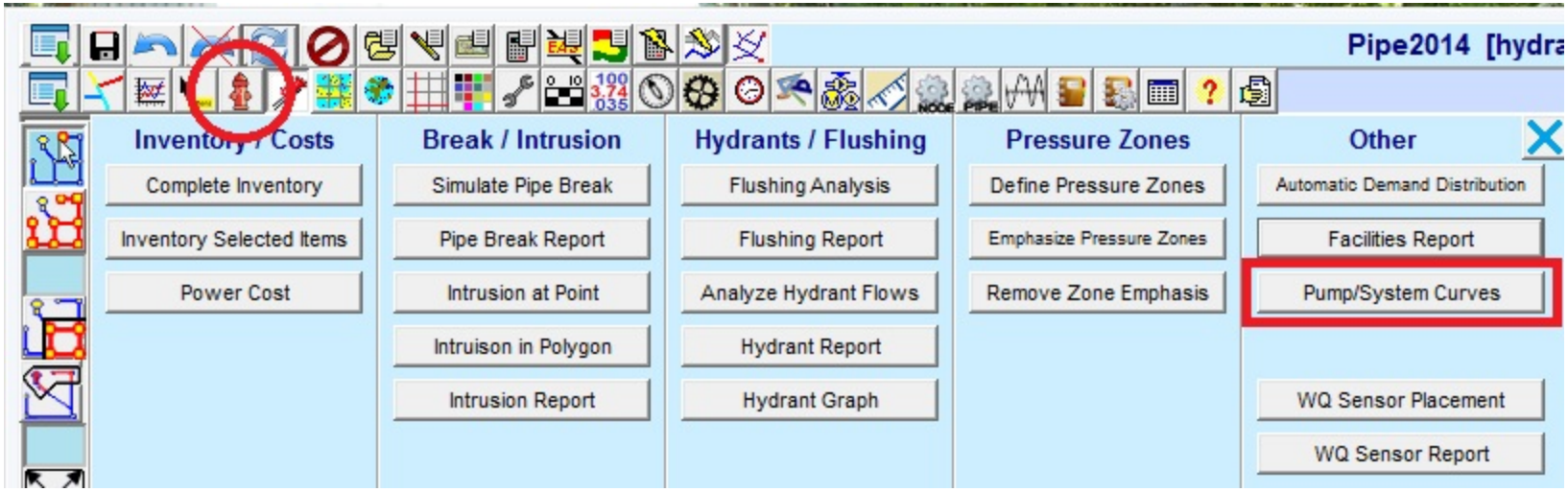



Then analyze the system using the **System Head Curves** under **Analysis Type** as shown.

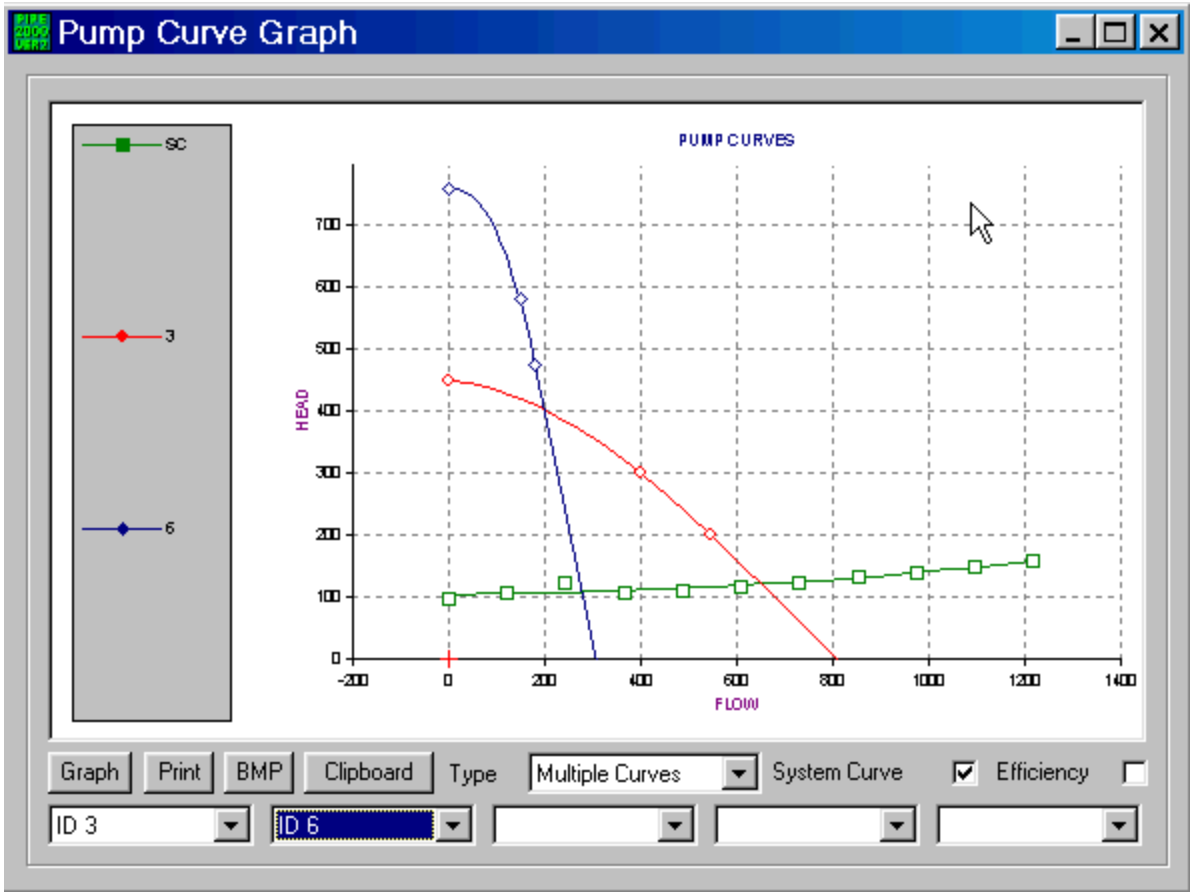



After the analysis has been carried out, the system curve is viewed using the Pump/System Curves option under Facilities Management.





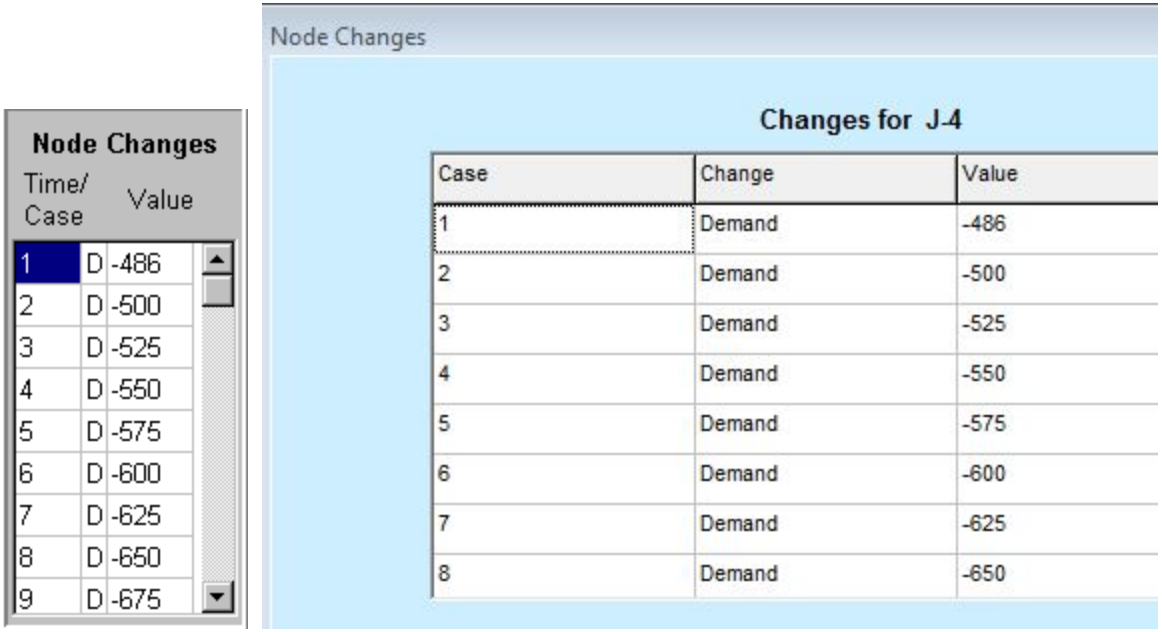

The system curve will appear and the user may choose any other pump in the system to compare to the system curve. The user could also create a new pump curve to compare to the system curve by creating a new pump ID (click on an existing pump, give it a new ID table number and enter pump curve data points then change ID number back to original ID if desired for modeling purposes). See Pump Curves above for other options.




Setting up a system curve manually (indirectly).

With a few easy steps, a model may be set up to generate system curve data. This data may then be entered as an ID as described in the Pump Data section and graphed as described above. To generate a system curve, use the following steps:

1. At the point which the system curve is to be generated, create a node or use an existing node and enter a series of inflows (negative demands). This process is done using Change data as described in the Node Change Box section and illustrated as below.



**Node Changes**

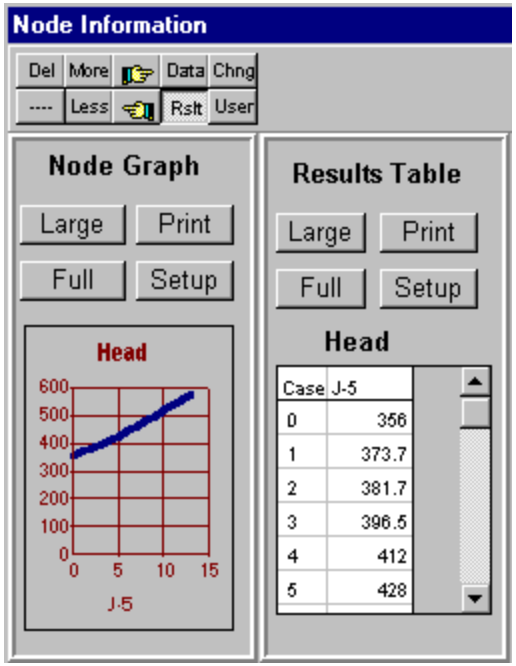
| Time/ Case | | Value |
|---|---|---|
| 1 | D | -486 |
| 2 | D | -500 |
| 3 | D | -525 |
| 4 | D | -550 |
| 5 | D | -575 |
| 6 | D | -600 |
| 7 | D | -625 |
| 8 | D | -650 |
| 9 | D | -675 |

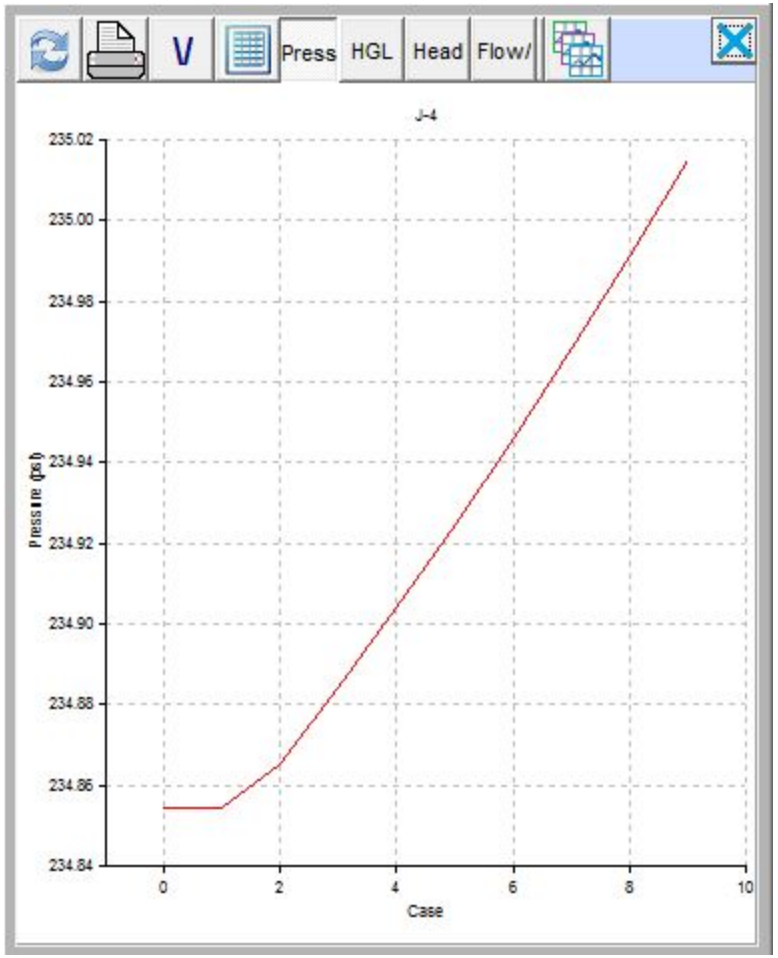
Node Changes

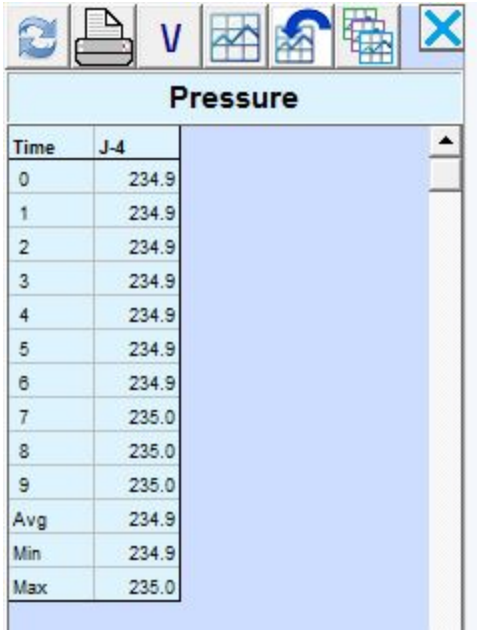
**Changes for J-4**

| Case | Change | Value |
|---|---|---|
| 1 | Demand | -486 |
| 2 | Demand | -500 |
| 3 | Demand | -525 |
| 4 | Demand | -550 |
| 5 | Demand | -575 |
| 6 | Demand | -600 |
| 7 | Demand | -625 |
| 8 | Demand | -650 |

2. Run an analysis of the system. Obtain a plot or table of the resulting heads (click the Rslt button in the Node Information Window) at the node where the negative demands were imposed. This data in conjunction with the flow data entered (as a positive value) makes up the system curve.


Node Information
Del
More
Data
Chng
----
Less
Rslt
User
Node Graph
Large
Print
Full
Setup
Head
600
500
400
300
200
100
0
0
5
10
15
J-5
Results Table
Large
Print
Full
Setup
Head
Case J-5
0 356
1 373.7
2 381.7
3 396.5
4 412
5 428

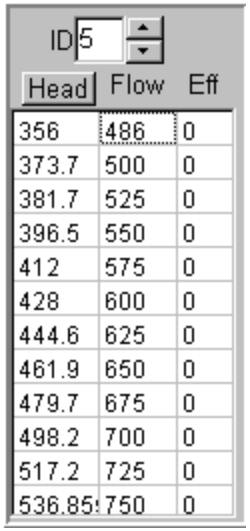

V
Press
HGL
Head
Flow/
J-4
235.02
235.00
234.98
234.96
234.94
234.92
234.90
234.88
234.86
234.84
Pressure (psi)
0
2
4
6
8
10
Case



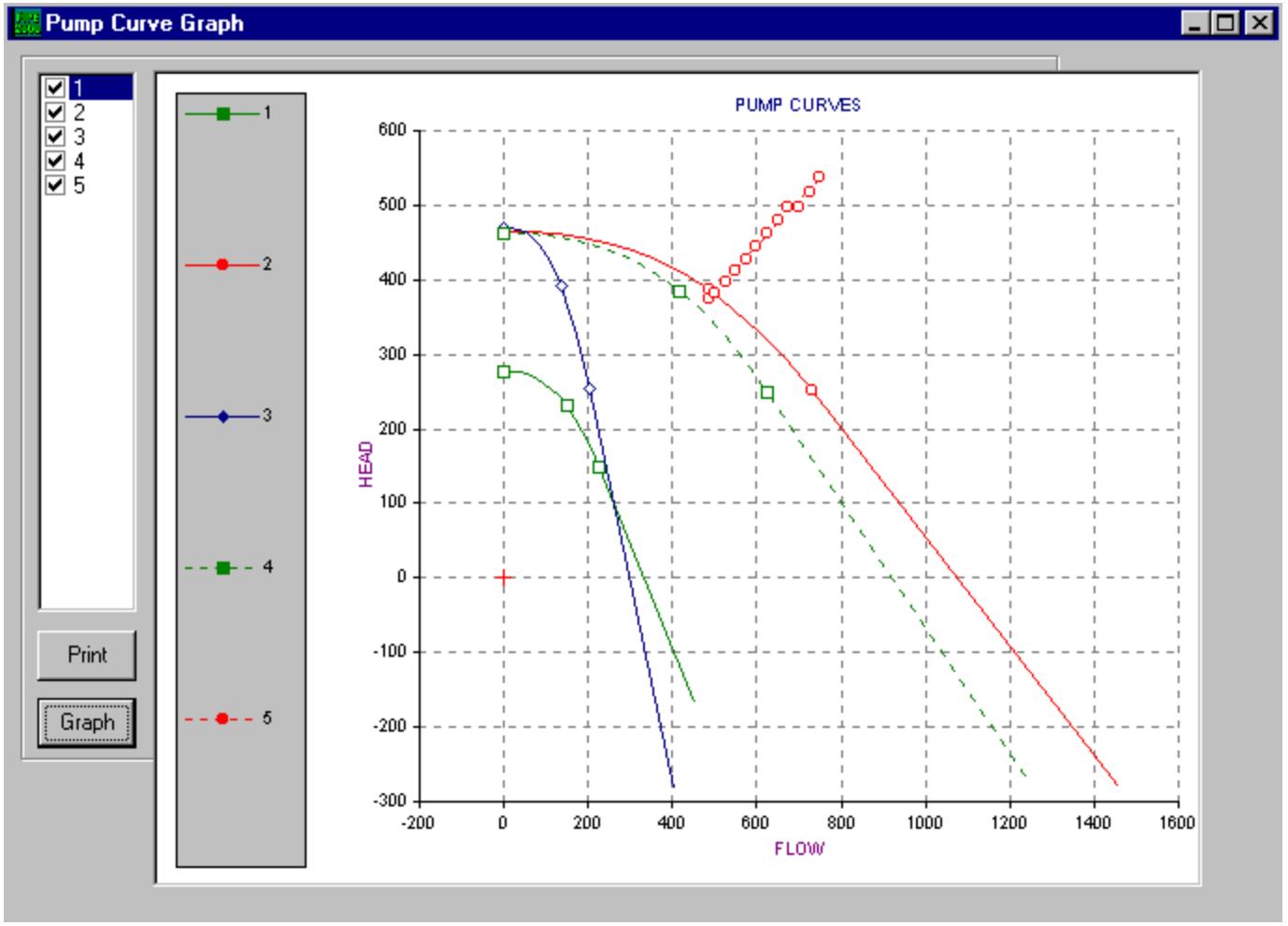
Pressure

| Time | J-4 |
|---|---|
| 0 | 234.9 |
| 1 | 234.9 |
| 2 | 234.9 |
| 3 | 234.9 |
| 4 | 234.9 |
| 5 | 234.9 |
| 6 | 234.9 |
| 7 | 235.0 |
| 8 | 235.0 |
| 9 | 235.0 |
| Avg | 234.9 |
| Min | 234.9 |
| Max | 235.0 |

3. This data may now be entered as head/flow data for an ID in the Node Information Window and graphed as shown below (select a pump node with Data Table selected as the pump type and access the ID table).



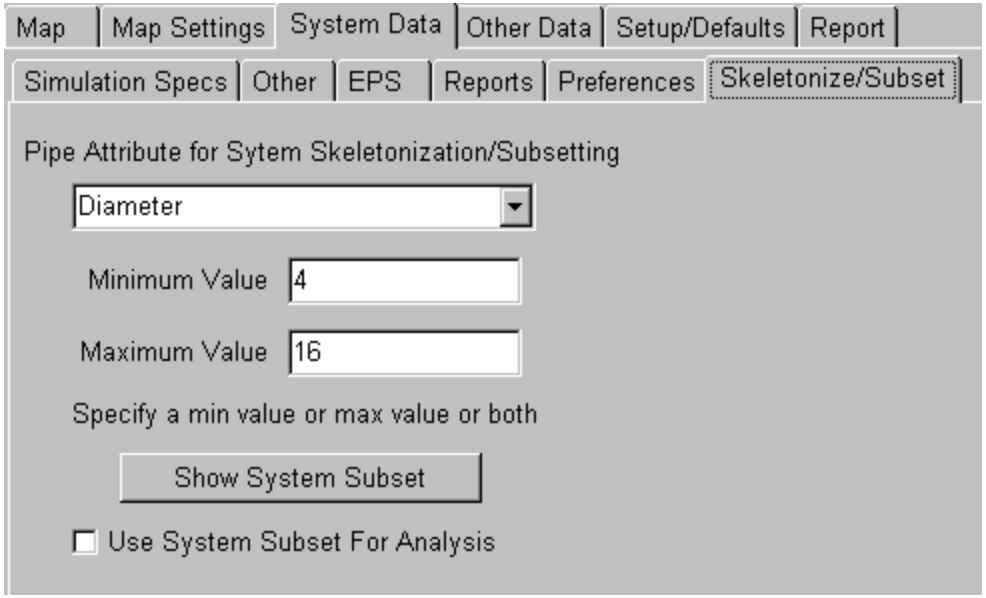

4. This data can now be plotted with other selected ID's to display the system curve and appropriate pump curves to see where the two intersect.


Pump Curve Graph
1
2
3
4
5
Print
Graph
PUMP CURVES
HEAD
FLOW
600
500
400
300
200
100
0
-100
-200
-300
-200
0
200
400
600
800
1000
1200
1400
1600

## 1.3.7.16 Skeletonize/Subset

Skeletonize/Subset for Surge models

In KYnetic use Skeletonize icon:




---

Pipe2018 has the ability to analyze a user-specified subset, excluding all pipes, elements, and demands outside the subset. It also has the ability to Skeletonize the system, which is minimizing the total number of pipes, but keeping the system the same hydraulically by maintaining all demands and major elements (tanks, pumps. etc.) in the system. The Skeletonize/Subset tab is used to create a subset for analysis. For skeletonizing (minimizing), see Skeletonize.

The Subset feature is similar to the Group Mode function in that it is used to select a subset of pipes. However, this particular selected pipe group may be used to conduct an analysis of only the specified portion of the system. This is especially important when a Surge analysis is being conducted on a system. To conduct a complete analysis, but only show a portion of the output in the Report, see Selected Output.

To select a pipe group, the user first selects the Pipe Attribute for System Skeletonization/Subsetting using the drop-down selector box. Then the Minimum Value and Maximum Value for that attribute are entered. One or the other or both Values may be entered. By clicking on Show System Subset, the user may view the highlighted subset. The Use System Subset for Analysis checkbox is used to specify that the selected portion of the system is to be for subsequent analyses. If the box remains unchecked, a complete analysis will be conducted.

## 1.3.7.17 Skeletonize

Skeletonize

---

See also Skeletonize/Subset

### Skeletonization Module

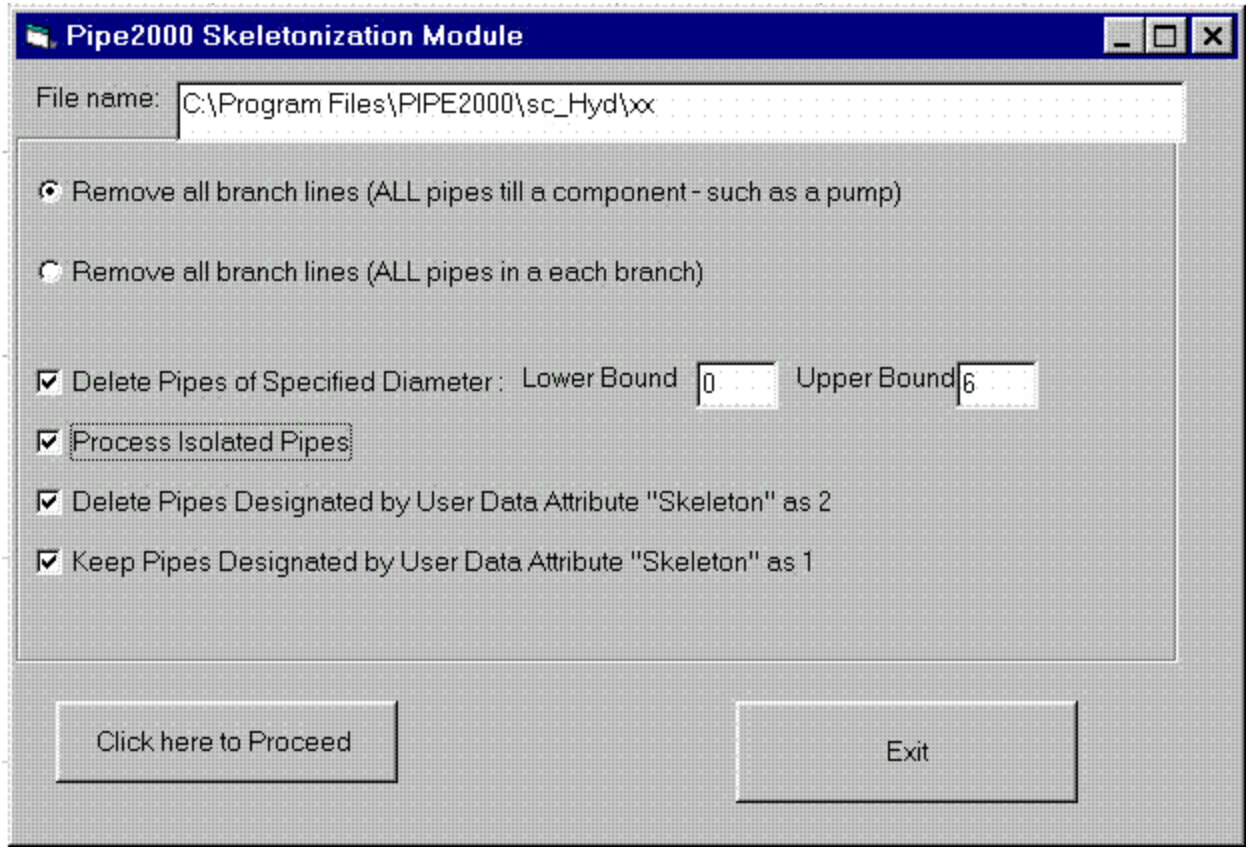
This Pipe2018 module skeletonizes (minimizes) a pipe system while maintaining the total system demand. The principal features include

1. Removing branch lines

2. Removing pipes equal to and smaller than a designated size

When pipes are removed the demands are moved to the skeletonized model. Unless specified, pumps, tanks, reservoirs and regulators will not be removed. The user can also designate specific pipes to be removed or to be retained by defining a pipe user data item (skeleton) as 2 (remove pipes) or 1 (retain pipes).

To skeletonize a system select Analyze | Skeletonize and provide a name for the skeletonized model. The following menu will appear.




Make a selection such as the ones shown and proceed.

Once the skeletonized model is obtained, it is a good idea to compare the performance to the original model, using an EPS if possible.

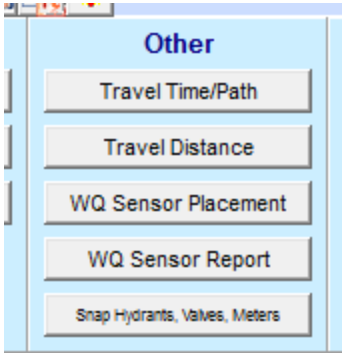
## 1.3.7.18 Travel Time Path and Distance

**Travel Time/Path and Distance**

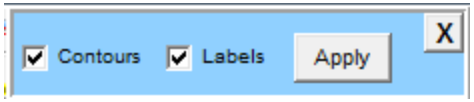
**Travel Distance**

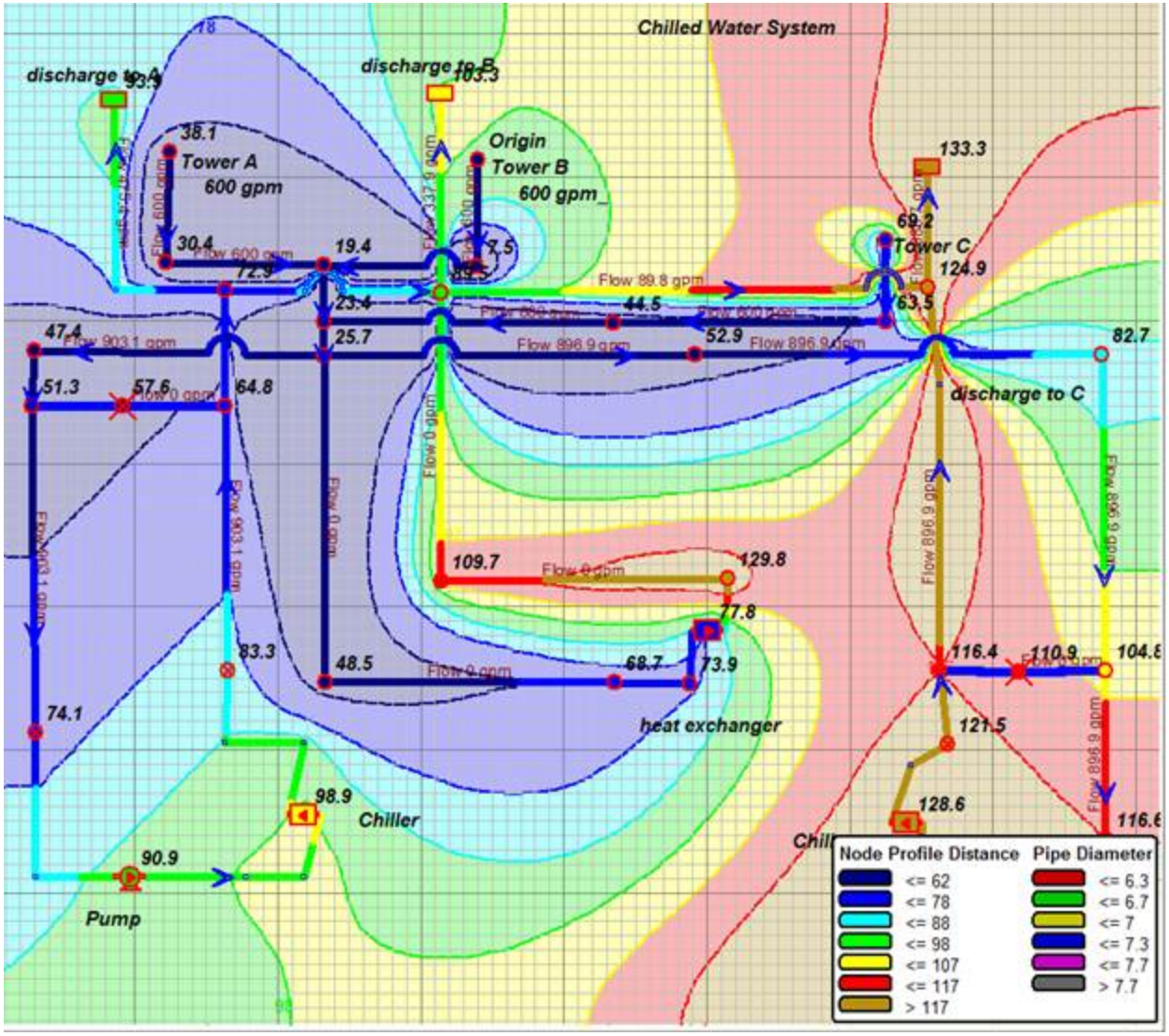
KYnetic interface only.

Select starting node. Go to Facilities Management and select Travel Distance.




The linear pipe distance from the starting node may be shown with both contours and labels.






See also the Contours and Emphasis menus to change settings.

**Travel Path/Time**

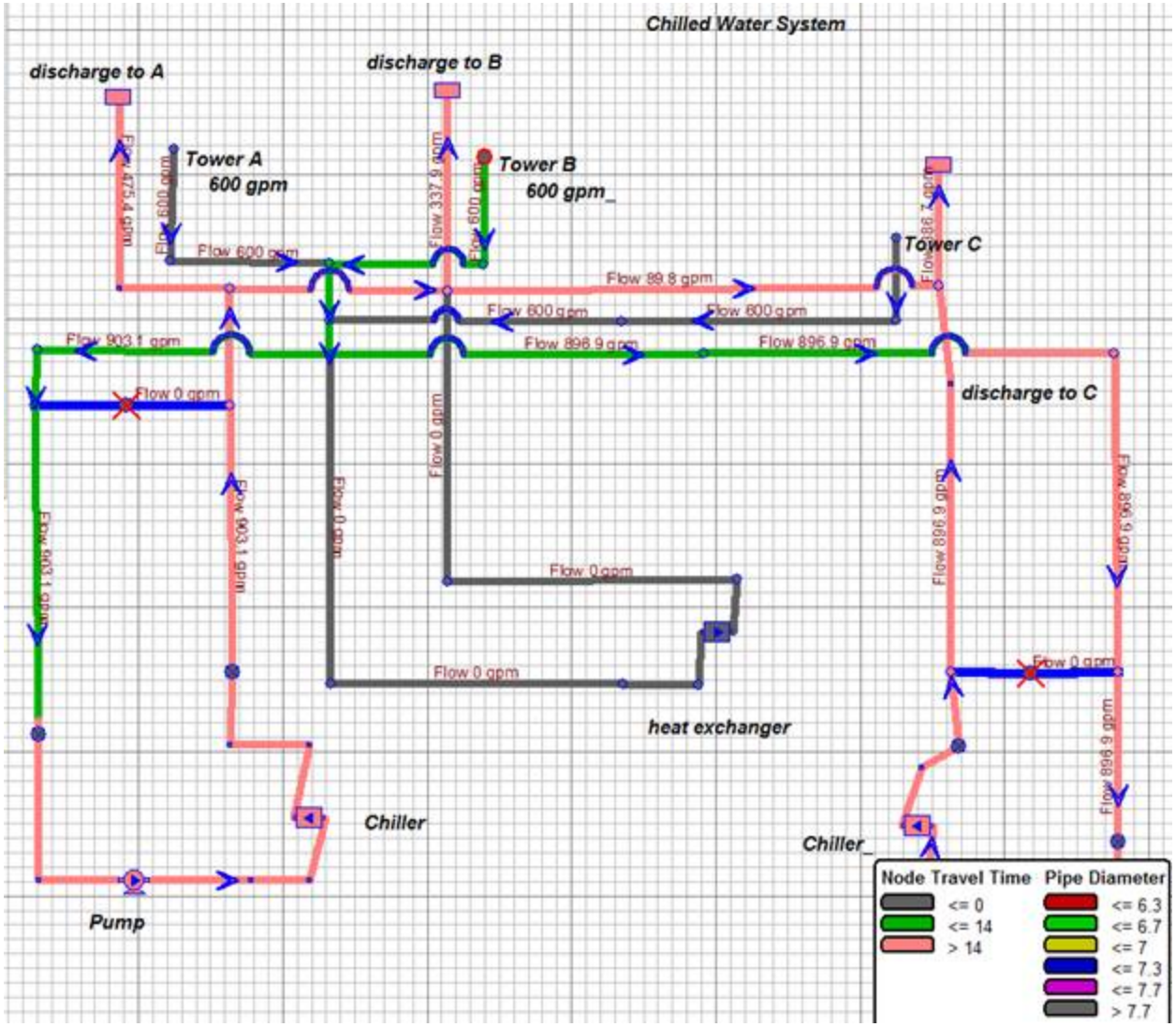
For Surge models, run a Steady State analysis first.  This feature is not for unsteady flow.

Select a starting node.  In the image below, the starting node is labeled Tower B.  Go to Facilities Management and select Travel Path/Time.

The path of the flow emanating from the select node over time is animated.

The time units are seconds.

The color, speed and time range of the animation may be customized.  Animation controls are available: play, pause, and slider bar.




This image shows an animation snapshot of the travel path from Tower B after a segment of time has elapsed. The green color emphasis is expanding to show the progression of flow through the network and the arrows show flow direction.

### 1.3.7.19 Former Analysis Method

The setting for Former versus Current Analysis Method is a check box found in the Analysis Setup menu.

In the former analysis method the results are stored in a file, then read back from the file, and ultimately stored in the main p2k files.

In the current method, the results value are stored in memory and accessed directly by Pipe2018, but are never stored in the p2k file.

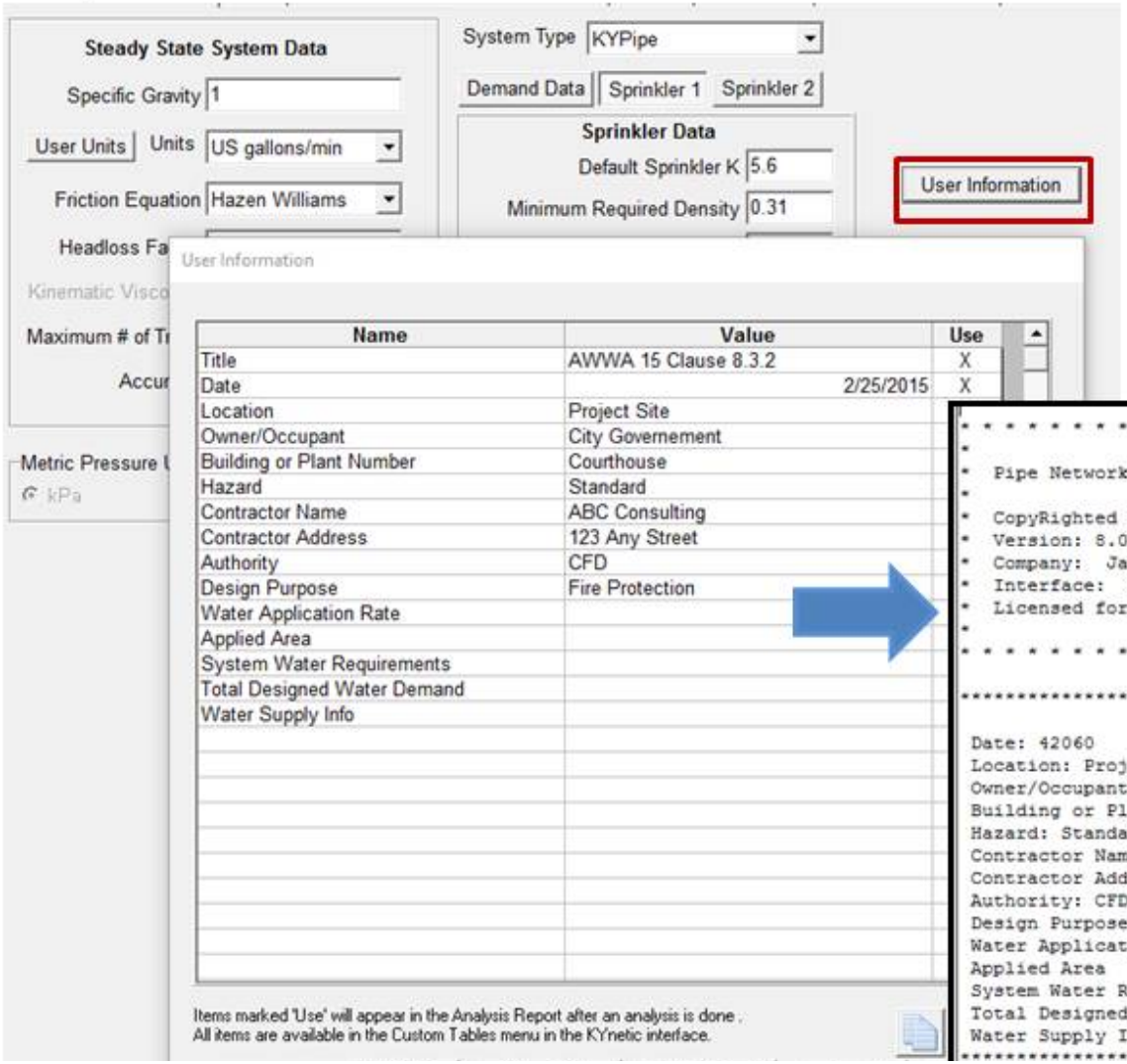
The current method analysis is faster and access to results when making maps and contours is faster, but the nature of complex hydraulic models is such that the former method is sometimes more stable.

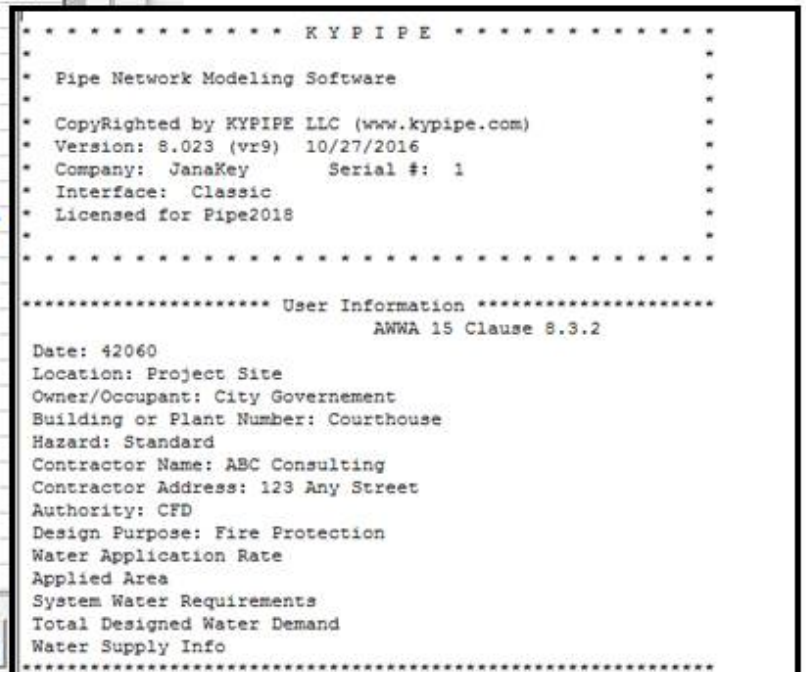
## 1.3.8 Results

### 1.3.8.1 User Information in Report

## NFPA Summary - Additional Information - also Available for **Surge**

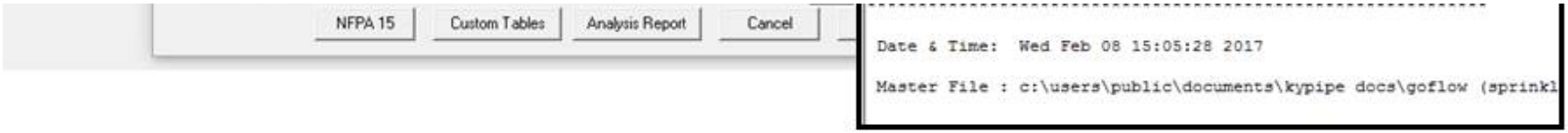
### System Data menu



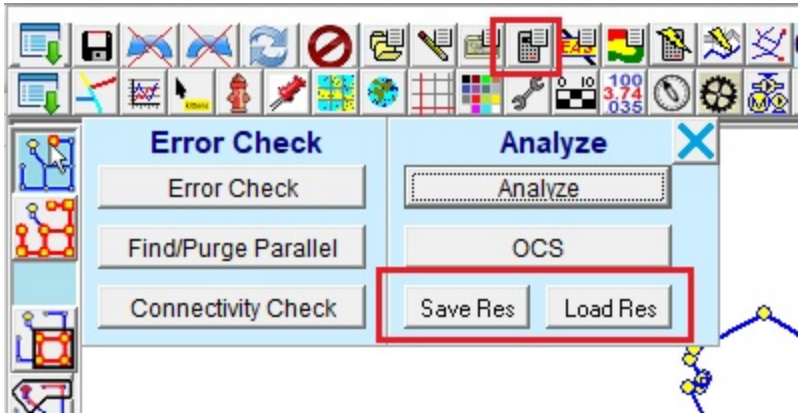




Any data may be added at the beginning of the Report.

Blank rows may be used or an NPFA 15 format form is available.



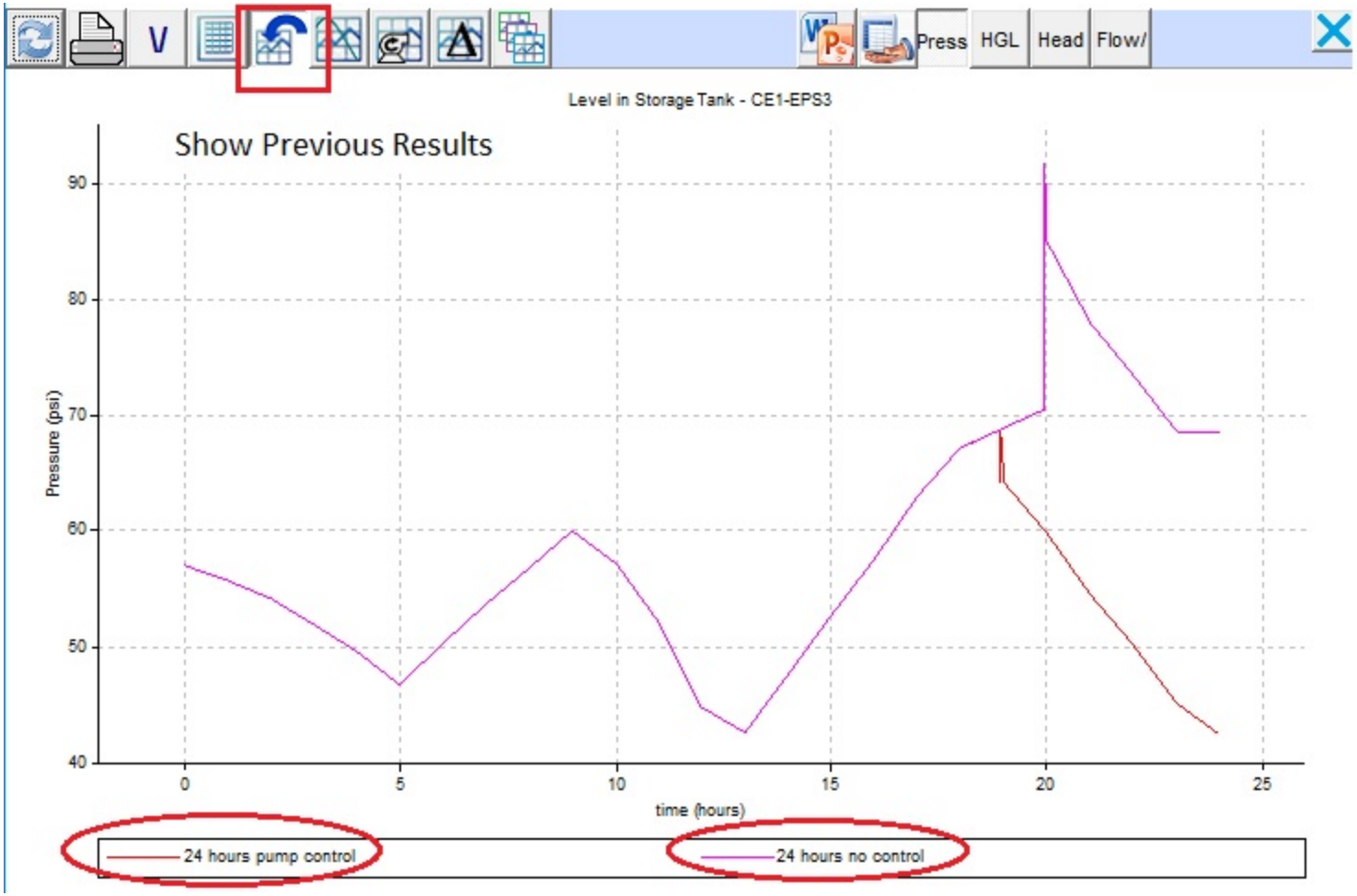

## 1.3.8.2 Saved Results




Save as many sets of results files as needed.

Use Load Res button to load a of results. Two results may be loaded and compared.

To compare two sets of results, load both, then use the Show Previous Results button on Graphs and Tables.




## 1.3.8.3 Customized Reporting

Customized Reporting

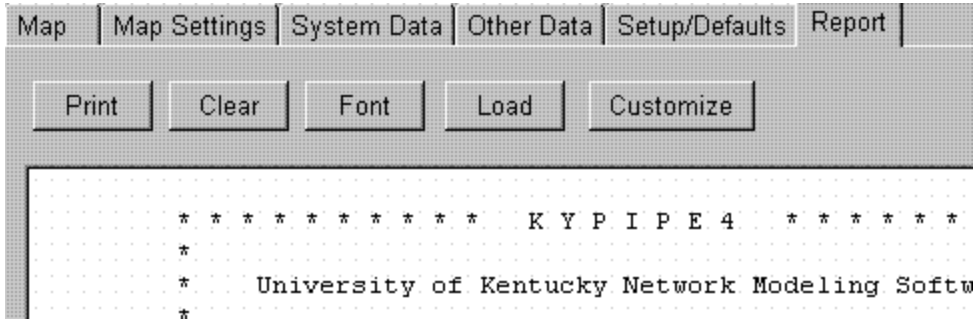
Utility Programs

The Customized reporting utility is used to create a customized printout of data items from the output report.

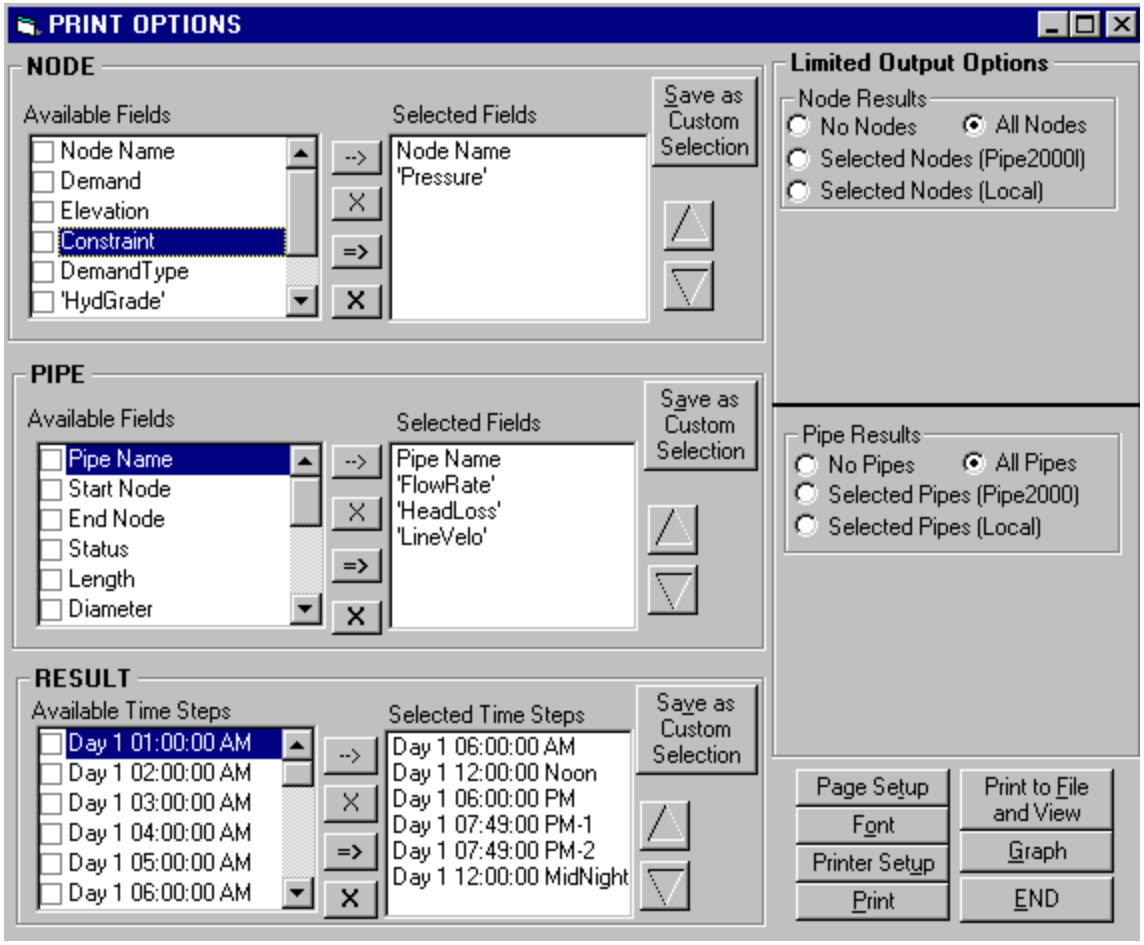
Important Note:  It is necessary to apply the "Use former analysis method" option in Analysis Setup to use this features.

Customized Report is accessed by clicking on the Customize button in the Report screen.

KYnetic Report icon:




The following screen appears:




Selecting the **Node, Pipe and Result** (case number) data to be included in the customized report:

Check (click on) the items to be included and then click the --> to send them to the Selected Fields box.

X will remove the highlighted data item from the Selected Fields box.

=> will include all of the available data items in the customized report.

X will delete all of the data items from the customized report.

Other operations:

Save as Custom Selection This will save the customized reporting settings for the current data file.

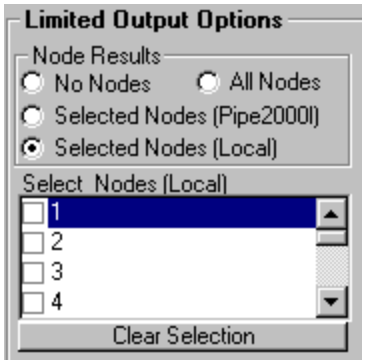
Limited Output Options - There are four Limited Output options for nodes or pipes as follows

No Nodes (Pipes) - Removes all nodes (or pipes) from the customized report

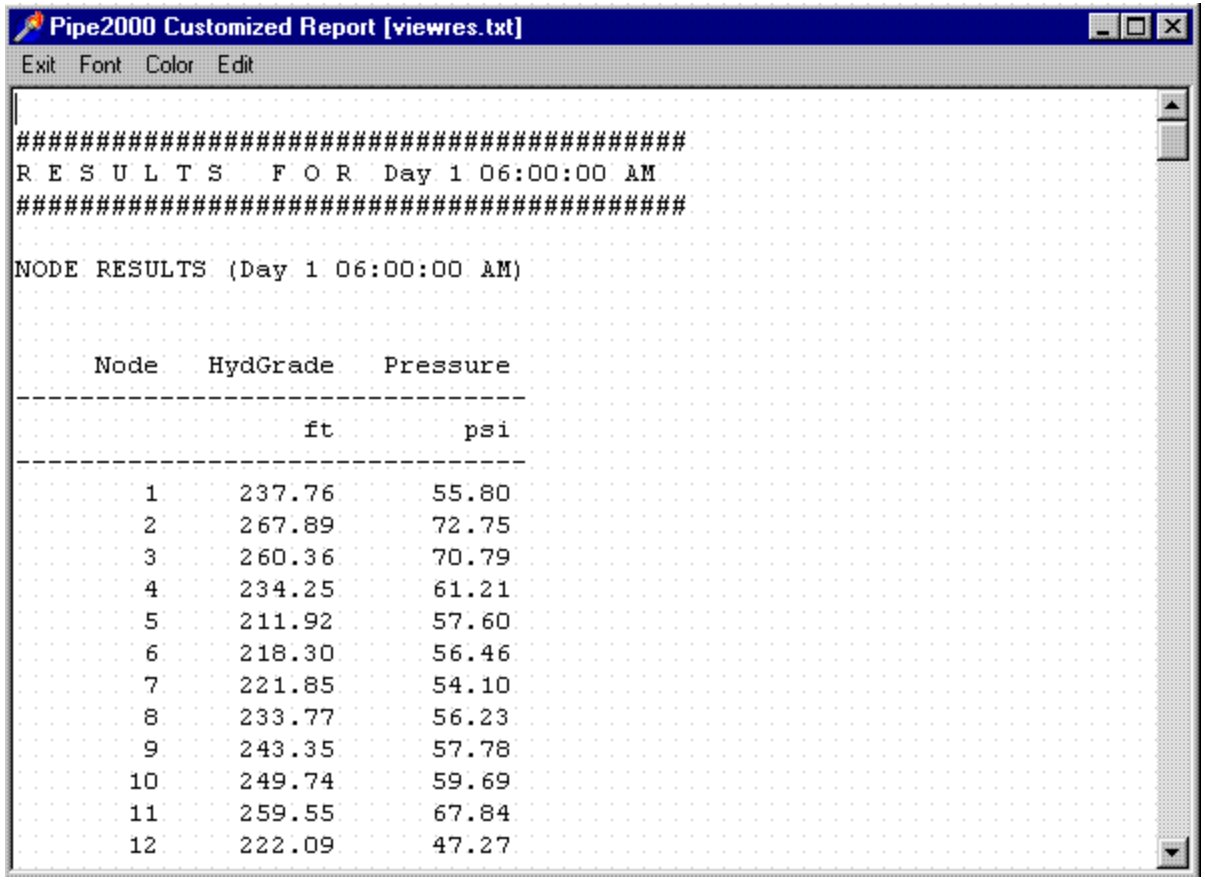
All Nodes (Pipes) - This is the default setting. All nodes (or pipes) are included in the customized report.

Selected Nodes (Pipes) (Pipe2000) - The customized report includes only the nodes (or pipes) as specified for Selected Output. See Selected Output.

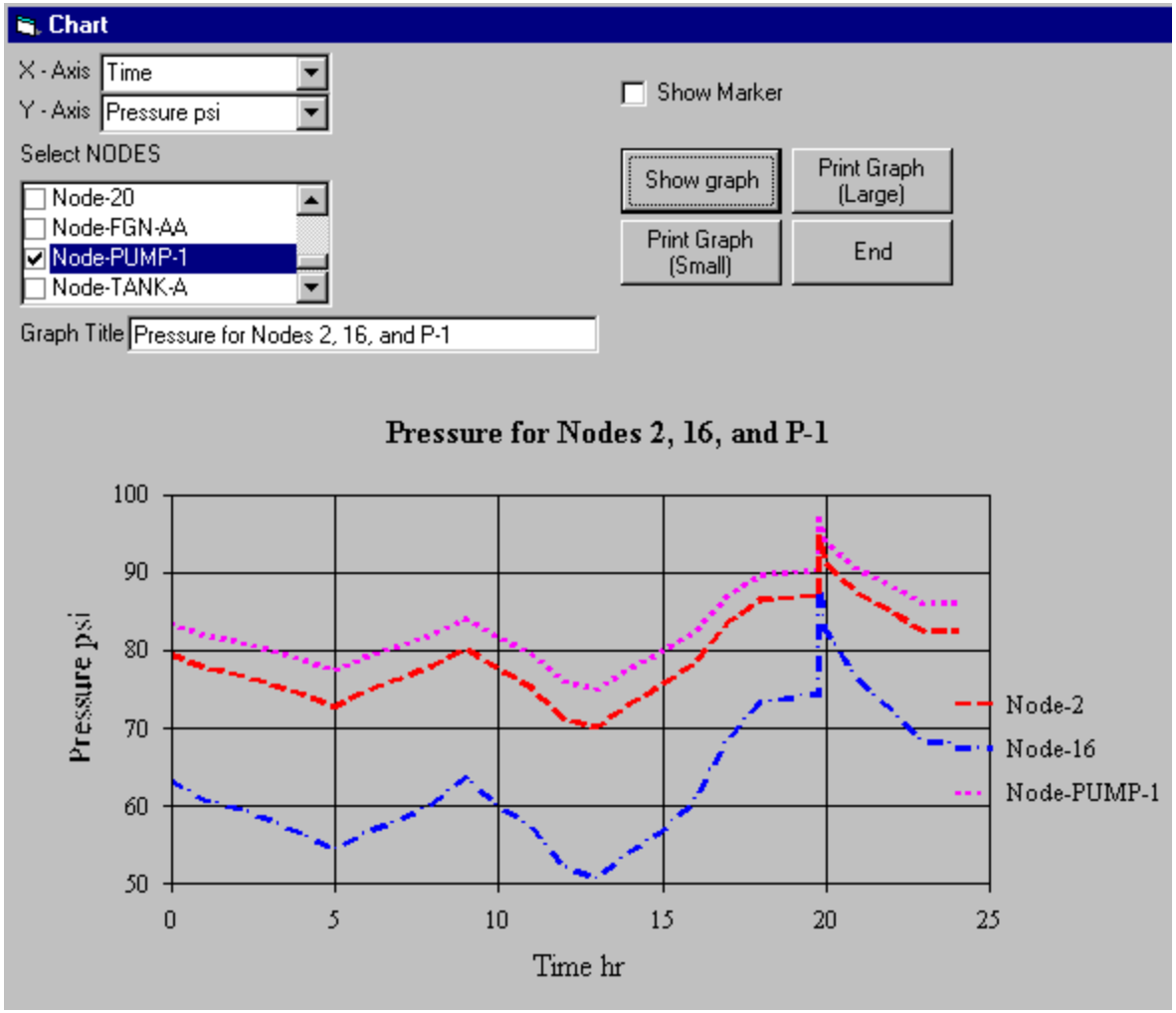
Selected Nodes (Pipes) (Local) - When using this option, a drop-down selector box appears and the individual nodes (or pipes) to be included in the customized report are selected individually by the user as shown:




Print to File and View - Prints the file to a viewing utility for previewing as shown below. The Font and Color settings within this window apply to this viewing module only and not to the report printout.




Graph This button brings up the following window:




X - Axis, Y - Axis, Select NODES - The user selects the x and y-axis parameters and the nodes for which results are to be displayed.

Graph Title - User may enter a title for the graph.

Show Marker - By checking this box, the graph lines are marked with various symbols.

Show Graph - Displays the graph for the chosen selections or updates a graph if selection changes have been made.

End - Exits the window.

### 1.3.8.4 Presentation Generator

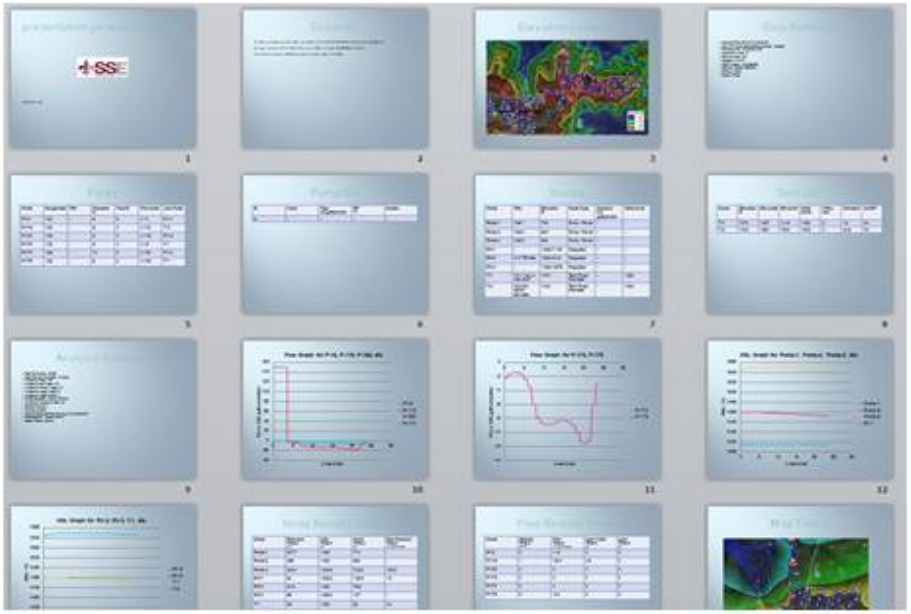
Pipe2014 and later versions.

# Presentation Generator

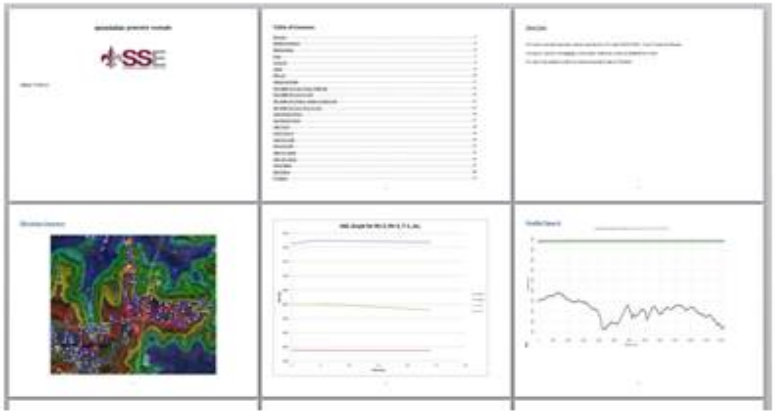
Build automatic or custom reports easily and quickly.
KYnetic interface only.

PDF, HTML, Video

PowerPoint Presentation

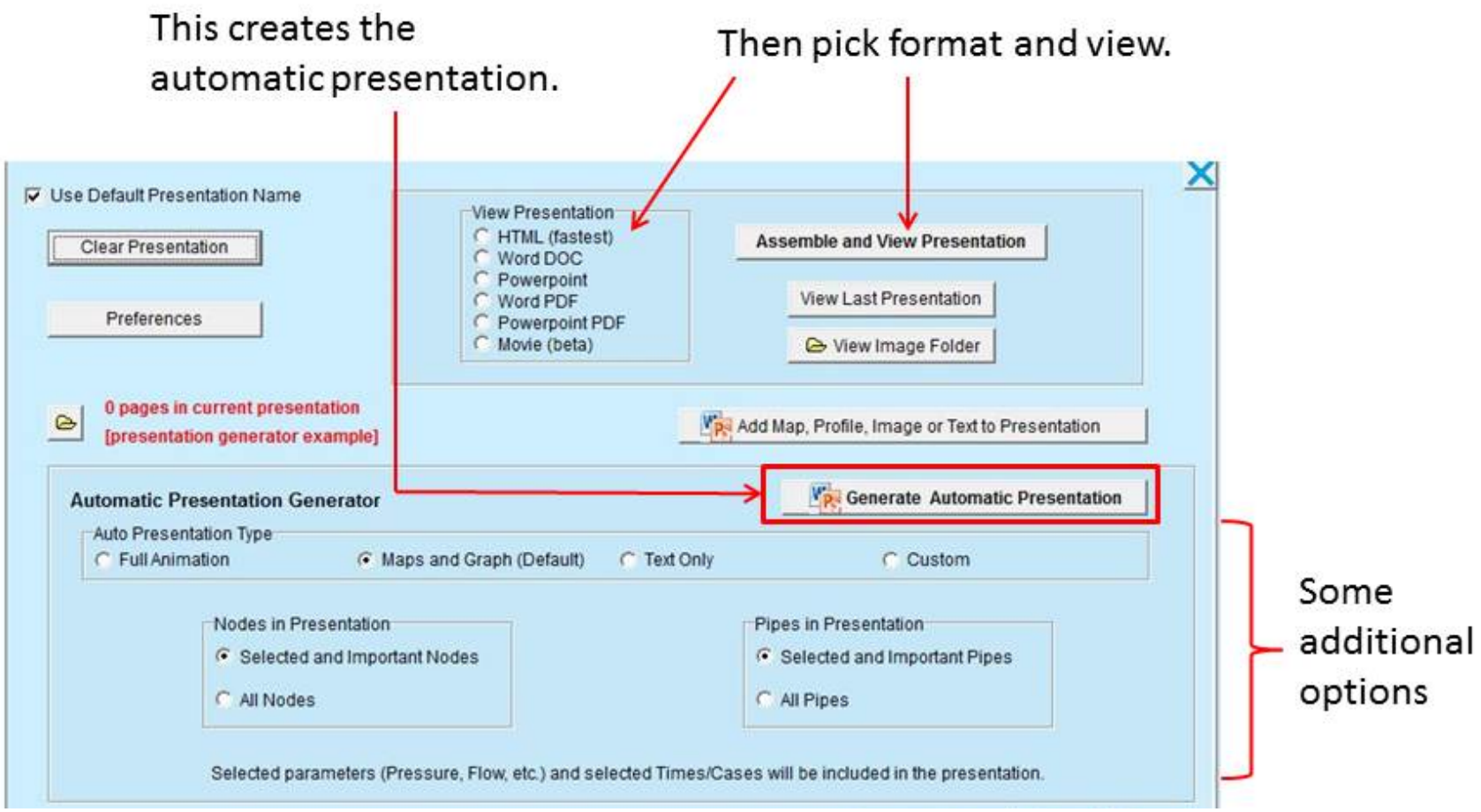


Word Document



# Presentation Generator

## Create an Automatic Presentation



Edit as Custom
Apply
OK

Automatic Presentation is generated and opened in the program for selected format (Word, PowerPoint, HTML viewer, PDF viewer, video player)

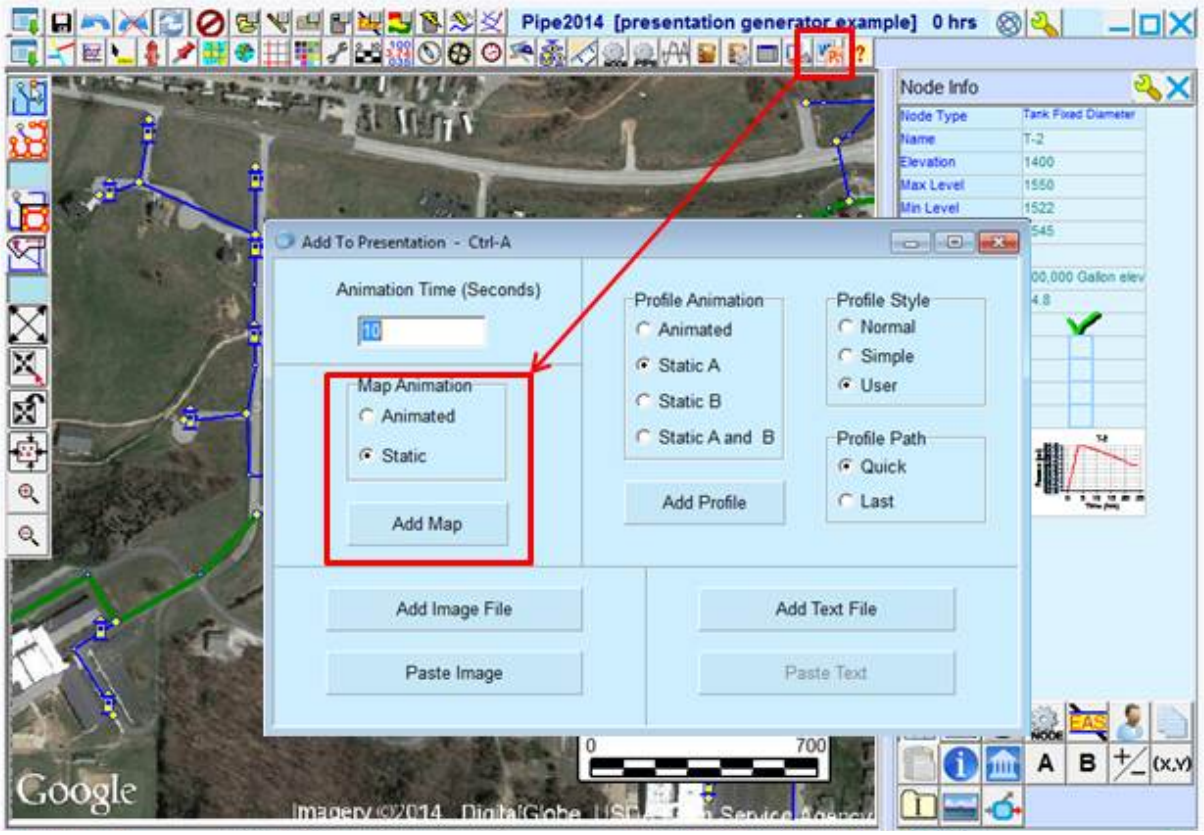
**Table of Contents:**


- 1.....Elevation Contours
- 2.....Data Summary
- 3.....Pipes
- 4.....Pump IDs
- 5.....Nodes
- 6.....Tank List
- 7.....Analysis Summary
- 8.....Flow Graph for P-15, P-119, P-165, etc.
- 9.....Flow Graph for P-174, P-179
- 10.....HGL Graph for Pump-1, Pump-2, Pump-3, etc.
- 11.....HGL Graph for RV-2, RV-3, T-1, etc.
- 12.....Node Results Time 0
- 13.....Pipe Results Time 0
- 14.....Map Time 0
- 15.....Profile Time: 0
- 16.....Node Min/Max
- 17.....Pipe Min/Max
- 18.....Map Min Values
- 19.....Map Max Values
- 20.....Pump Report
- 21.....Tank Report
- 22.....RV Report

# Custom Presentations – Add Map

Presentation icon

In Map view, RIGHT CLICK the icon to add the map (or ctrl-A).

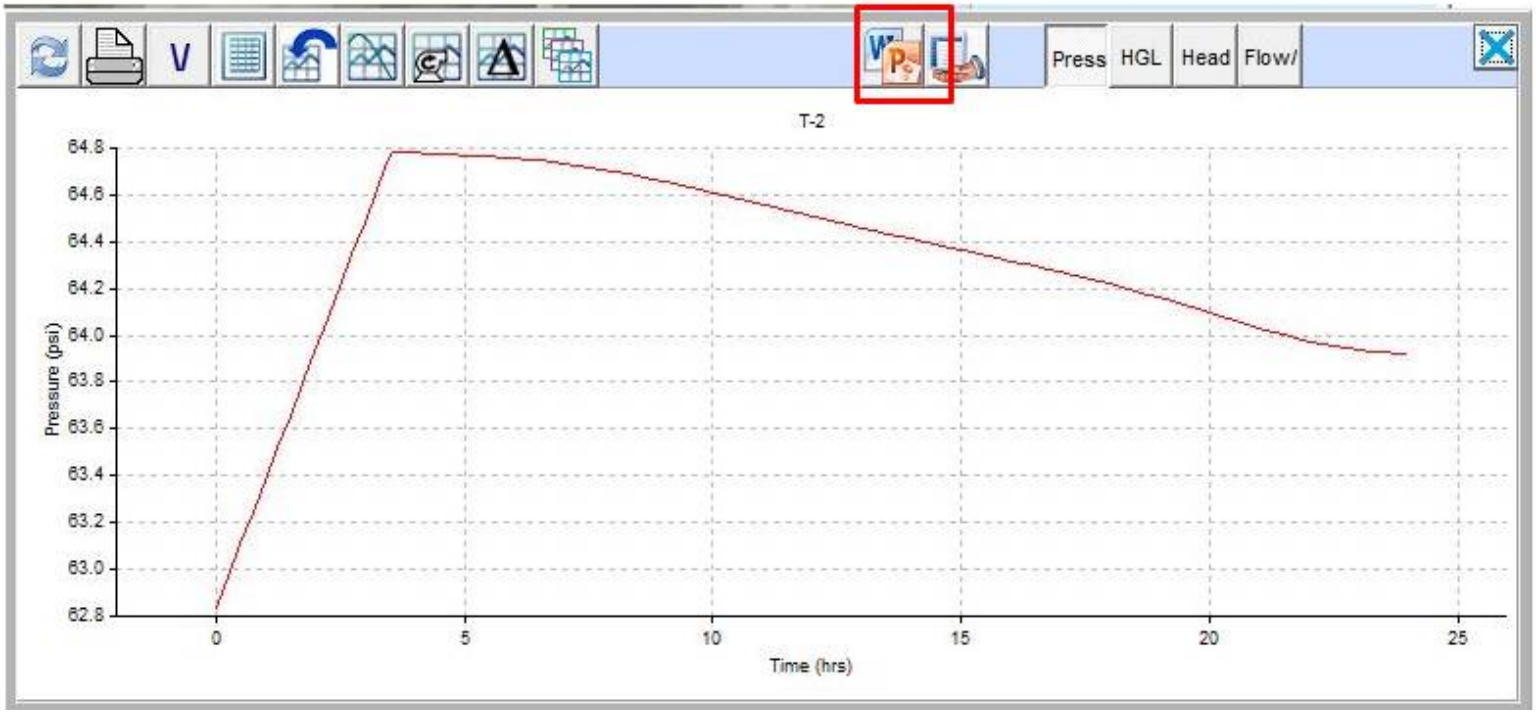


Layout Mode
(1380531.68, 326962.91) D : 993.18
Time 0 - Pressure - Flow

# Custom Presentations – Add Graphs and Tables

Add to Presentation icon

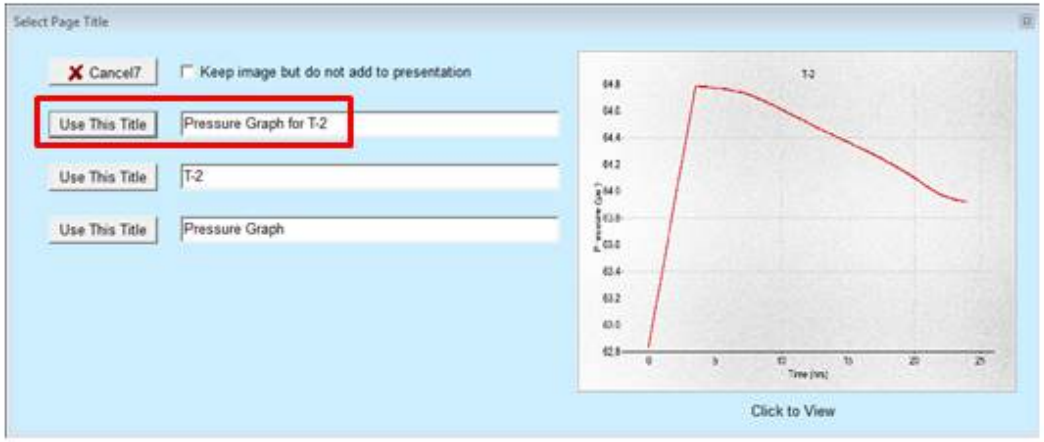
You will see “Add to Presentation” icons in graphs and tables



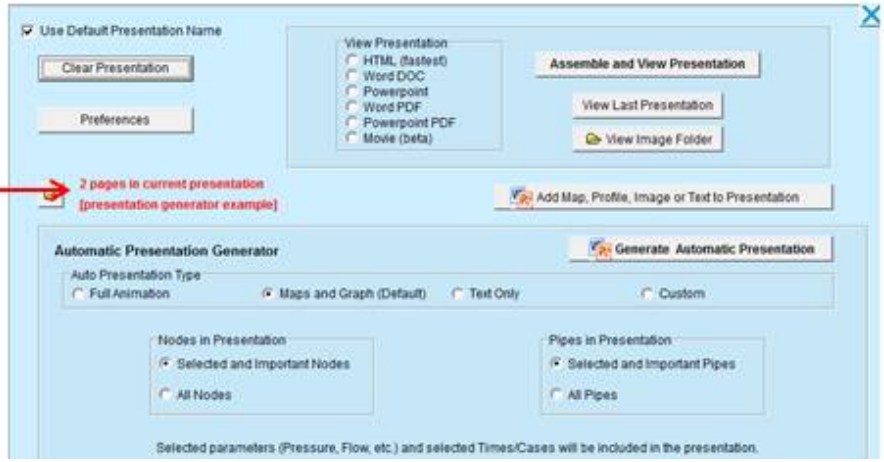
Clicking will add the graph or table.

# Custom Presentations – Add Graphs and Tables

Suggested titles are provided for added item, or you may edit the title, or cancel.

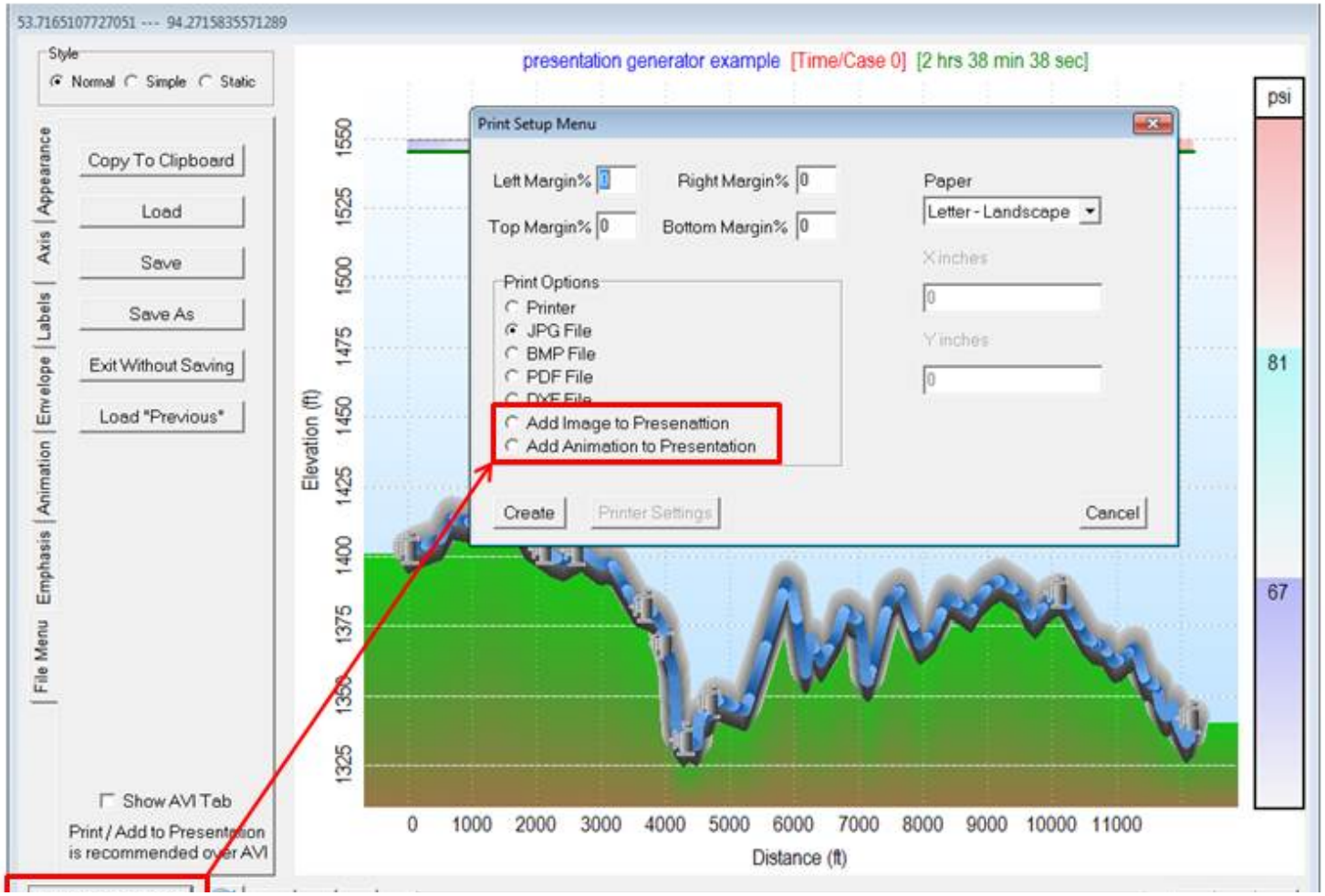



As items are added, the number of items is shown in the Presentation Generator menu



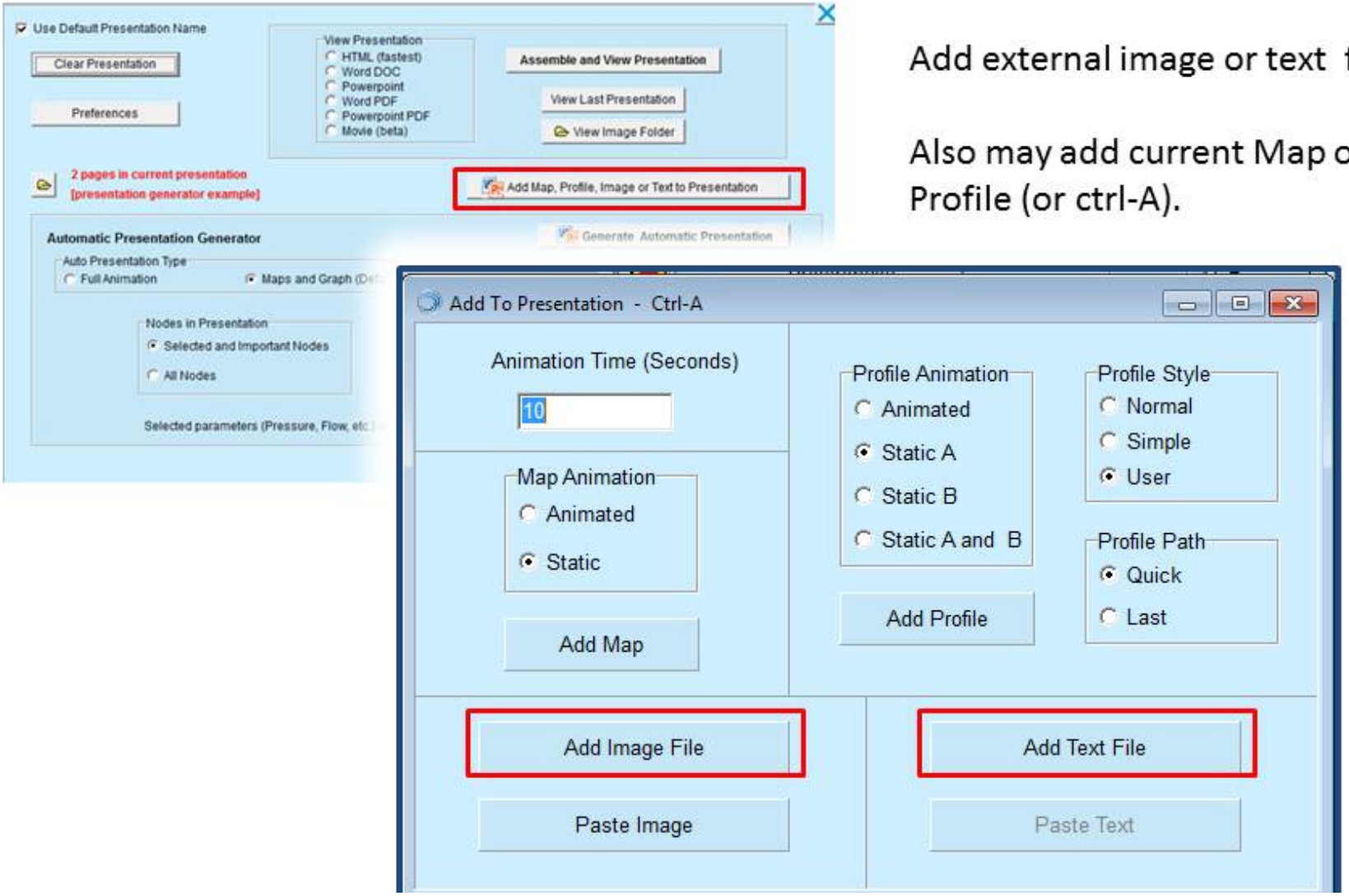
Edit as Custom
Apply
OK

# Custom Presentations – Add Profile



Print/JPG/PDF/DXF
Default
Undo All
Exit
Play
Time/Case 0
© 2014 KYPipe LLC

# Custom Presentations – Add Image or Text Files

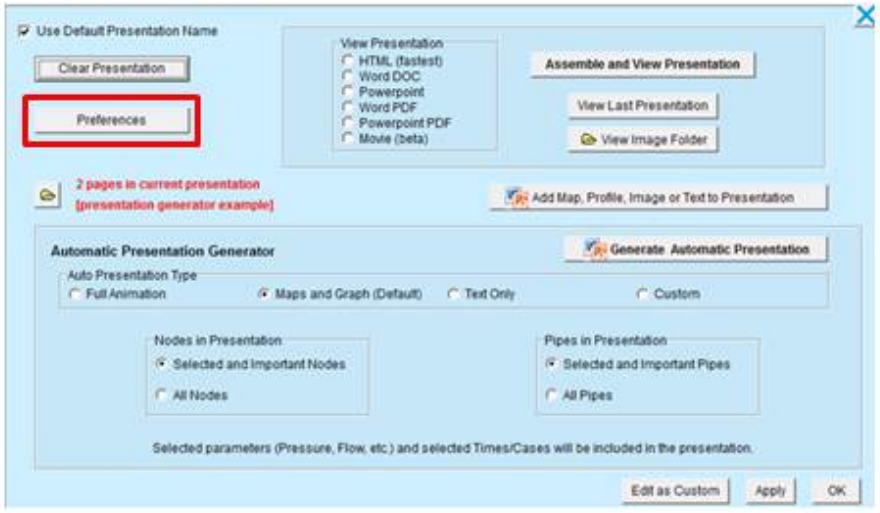



Add external image or text file.

Also may add current Map or Profile (or ctrl-A).

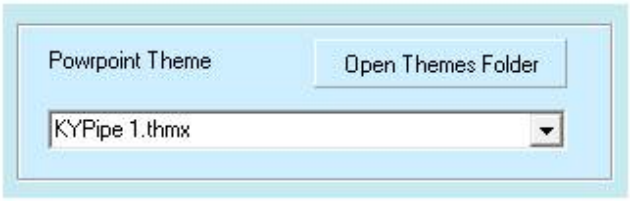
# Preferences

Set Preferences before creating the Presentation.



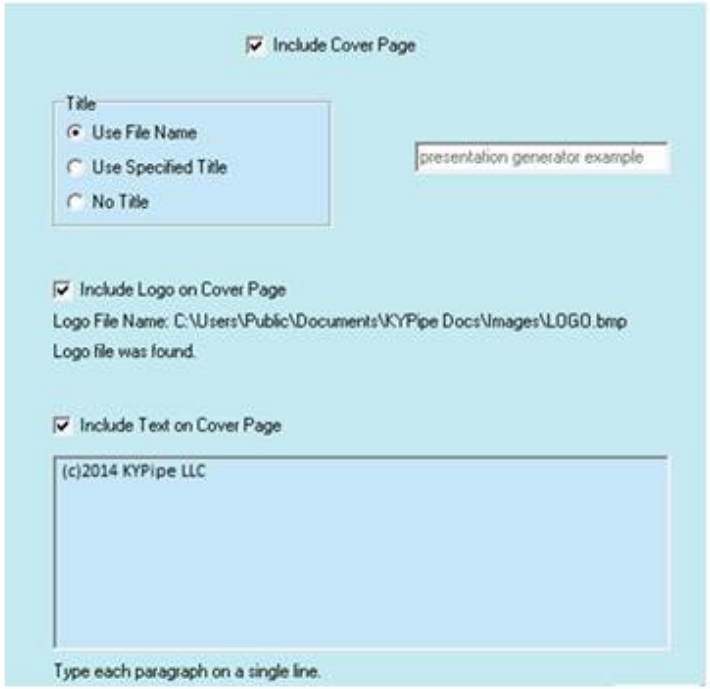

## Video

## PowerPoint



# Preferences

## Cover Page




Edit title and logo.

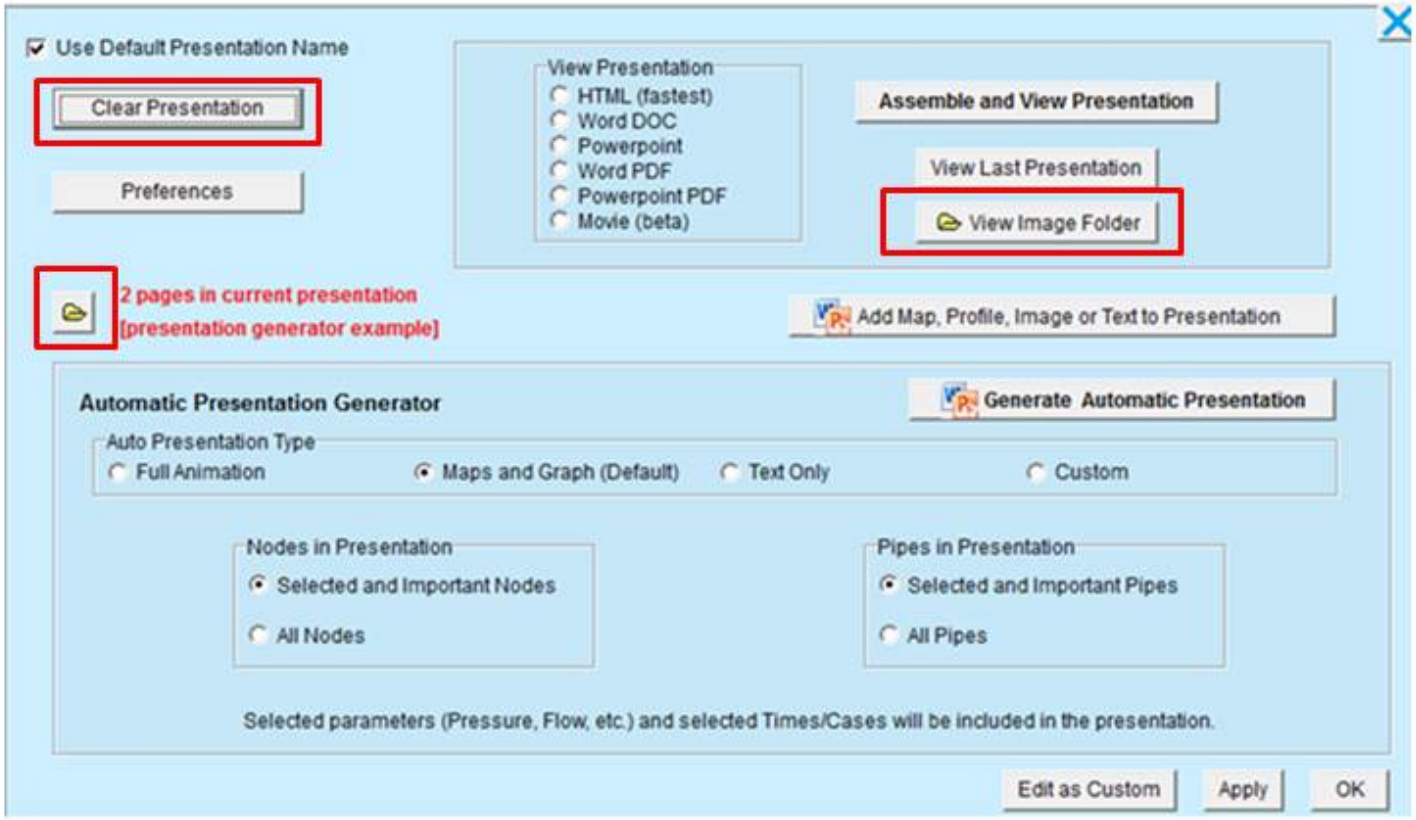
Important! Image file called "logo.bmp" must be placed in KYPipe Docs\Image

## Overview Page

Use default or customize text on Overview page.

must be placed in KTI Pipe Docs\Image
folder to include logo.

# Presentation Files

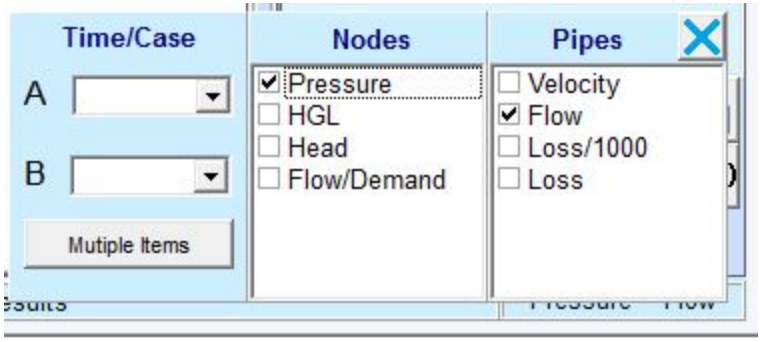



View or delete files associated with Presentation Generator

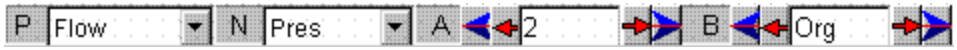
## 1.3.8.5 Result Selector Bar

Result Selector Bar

KYnetic, click bottom right of program screen:




Classic, use the pop-up menus and arrows:




---

The Results Selector Bar is located at the bottom of the display and defines selection for the type of results and time (case) for two sets of results (Result A and Result B)

P / Pipe

Select type of pipe results (Velocity, Flow, Loss, Loss / 1000)

N / Nodes

Select type of node results (Pressure, HGL, Head, Flow/Demand))

A

Select case, time, or the Max, Min, or Avg value for Result A. Small red arrow goes forward or backward one case. Large blue arrow goes forward or backward 5%.

B

Select case, time, or the Max, Min, or Avg value for Result B

Both a Result A and a Result B are provided to allow the user to display two different sets of results on the map alternately or simulaneously. Activation of the display of Result A or Result B or both is done from the Label selection from the main menu or from the Map Settings / Labels Tab.

## 1.3.8.6 Results Presentations

Pipe2018 Presentations

Pipe2018 provides a variety of means of presenting data and simulation results. These are listed below:

Presentation Generator (version Pipe2014 and later)

Tabulated Reports

Network Plots with Labels

Node and Pipe Results Boxes - Quick Tables and Plots

Customized Reports and Plots

Contours

Color Emphasis

Profiles

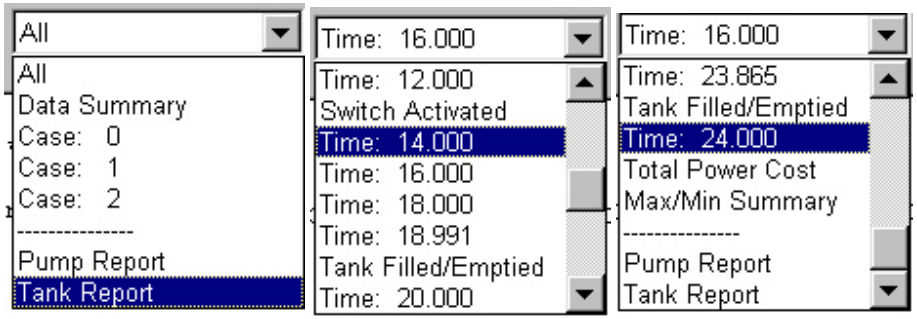
---

Tabulated Report

The analysis produces a tabulated report showing the Input Data Summary and the Results for the Analysis. The report is divided into a number of sections. The whole report or the various sections can be selected and viewed on the screen and printed. By default the results section will report on every pipe and node. However, you can use Selected Output to limit the report to specific pipes and nodes. See also REPORT (Main Menu) and Analysis Report Setup icon or Reports (System Data) (Classic).

The selector can be used to select the data summary, any of the results sections and various other summaries. Examples of the sections which can be selected for

Regular Simulations and EPS are shown below.




Regular Simulations Extended Period Simulations

Some examples of the tabulated reports are shown below.

P I P E L I N E  D A T A

STATUS CODE: XX -CLOSED PIPE CV -CHECK VALVE

| P I P E N A M E | NODE NAMES #1 | #2 | LENGTH (ft) | DIAMETER (in) | ROUGHNESS COEFF. | MINOR LOSS COEFF. |
|---|---|---|---|---|---|---|
| P-1 | J-2 | J-10 | 2859.00 | 6.00 | 130.0000 | 0.00 |
| P-10 | J-10 | J-14 | 1649.00 | 8.00 | 140.0000 | 0.00 |
| P-11 | J-10 | J-11 | 2789.00 | 6.00 | 130.0000 | 0.00 |
| P-12 | J-12 | J-16 | 1464.00 | 8.00 | 140.0000 | 0.00 |
| P-13 | J-12 | T-1 | 2413.00 | 8.00 | 140.0000 | 3.40 |
| P-14 | J-15 | J-1 | 2550.00 | 8.00 | 140.0000 | 0.00 |
| P-15 | J-16 | J-3 | 1735.00 | 10.00 | 120.0000 | 0.00 |
| P-16 | J-4 | J-7 | 1546.00 | 8.00 | 140.0000 | 0.00 |
| P-17 | J-4 | J-14 | 2950.00 | 8.00 | 140.0000 | 0.00 |
| P-18 | J-14 | J-8 | 1942.00 | 8.00 | 140.0000 | 0.00 |
| P-19 | J-7 | J-10 | 3447.00 | 8.00 | 140.0000 | 0.00 |

Data Summary

TIME FROM INITIATION OF EPS = 4.0000 HOURS ( 8.00AM, DAY: 1)

RESULTS OBTAINED AFTER 3 TRIALS: ACCURACY = 0.00118

P I P E L I N E R E S U L T S

STATUS CODE: XX -CLOSED PIPE CV -CHECK VALVE

| PIPE NAME | NODE NUMBERS #1 | #2 | FLOWRATE (gpm) | HEAD LOSS (ft) | MINOR LOSS (ft) | LINE VELO. (ft/s) | HL/ 1000 (ft/ft) |
|---|---|---|---|---|---|---|---|
| P-1 | J-2 | J-10 | -19.50 | 0.14 | 0.00 | 0.22 | 0.05 |
| P-10 | J-10 | J-14 | -26.78 | 0.03 | 0.00 | 0.17 | 0.02 |
| P-11 | J-10 | J-11 | 11.70 | 0.05 | 0.00 | 0.13 | 0.02 |
| P-12 | J-12 | J-16 | -5.48 | 0.00 | 0.00 | 0.03 | 0.00 |
| P-13 | J-12 | T-1 | -34.09 | 0.07 | 0.00 | 0.22 | 0.03 |
| P-14 | J-15 | J-1 | -5.48 | 0.00 | 0.00 | 0.03 | 0.00 |

Set of EPS Results

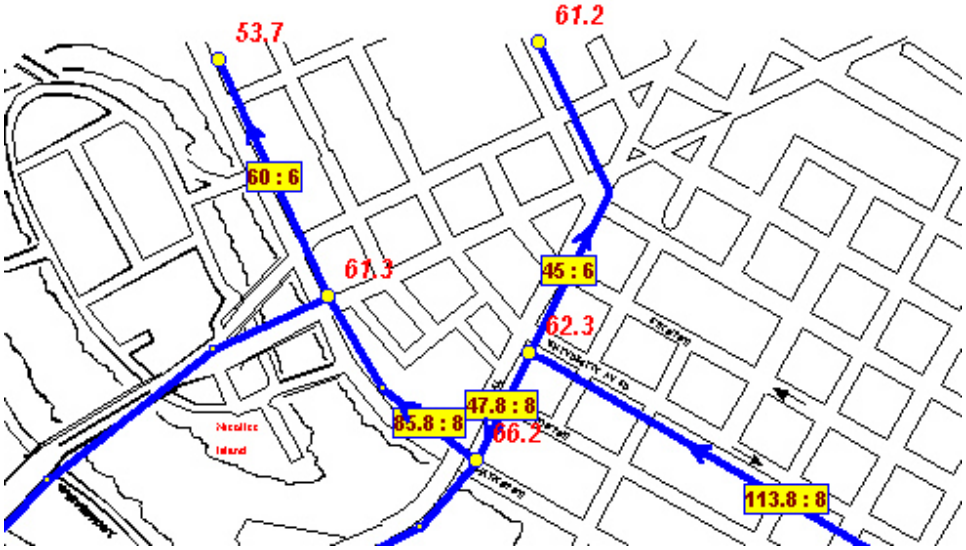
**********
Summary of Max/Min Node Values
**********

| Node | MnPres | MnHead | MnHGL | MnTime | MxPres | MxHead | MxHGL | MxTime | Elevation |
|---|---|---|---|---|---|---|---|---|---|
| J-1 | 55.18 | 127.34 | 734.34 | 16.00 | 66.20 | 152.77 | 759.77 | 22.00 | 607.0 |
| J-10 | 51.96 | 119.90 | 730.90 | 16.00 | 64.36 | 148.53 | 759.53 | 0.00 | 611.0 |
| J-11 | 51.94 | 119.85 | 729.85 | 16.00 | 64.76 | 149.44 | 759.44 | 0.00 | 610.0 |
| J-12 | 53.88 | 124.34 | 734.34 | 16.00 | 65.87 | 152.01 | 762.01 | 22.00 | 610.0 |
| J-13 | 43.44 | 100.26 | 730.26 | 16.00 | 56.10 | 129.46 | 759.46 | 0.00 | 630.0 |
| J-14 | 52.66 | 121.51 | 731.51 | 16.00 | 64.85 | 149.65 | 759.65 | 0.00 | 610.0 |
| J-15 | 56.05 | 129.34 | 734.34 | 16.00 | 71.97 | 166.08 | 771.08 | 22.00 | 605.0 |
| J-16 | 56.05 | 129.34 | 734.34 | 16.00 | 71.76 | 165.60 | 770.60 | 22.00 | 605.0 |
| J-2 | 48.16 | 111.13 | 728.13 | 16.00 | 61.67 | 142.30 | 759.30 | 0.00 | 617.0 |
| J-3 | 56.05 | 129.34 | 734.34 | 16.00 | 73.74 | 170.18 | 775.18 | 22.00 | 605.0 |

EPS Max - Min Summary

Network Plots with Labels

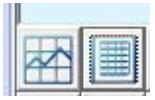
Plots of the entire network or any section of the network can be displayed and printed (with or without a background map). Pipe and node labels can be displayed for a variety of data and results. Labels can be displayed for all or only for selected nodes and pipes. The Label menu (or Map Settings/Labels - Classic) provides the complete range of label selection with control of appearance, font sizes, etc. The Label button on the top menu bar allows a quick selection of various important labels and a selection to turn off pipe, node, or all labels. The Results Selector Bar at the bottom of the display allows users to quickly select the type of results and time (case) to be displayed and choose between displaying one time selection (Result A or Result B) or two selections simultaneously (Result A and Result B). Activation of the display of Result A or Result B or both is done from the LABEL tab on top of the screen. An example of a network plot with labels is shown below.



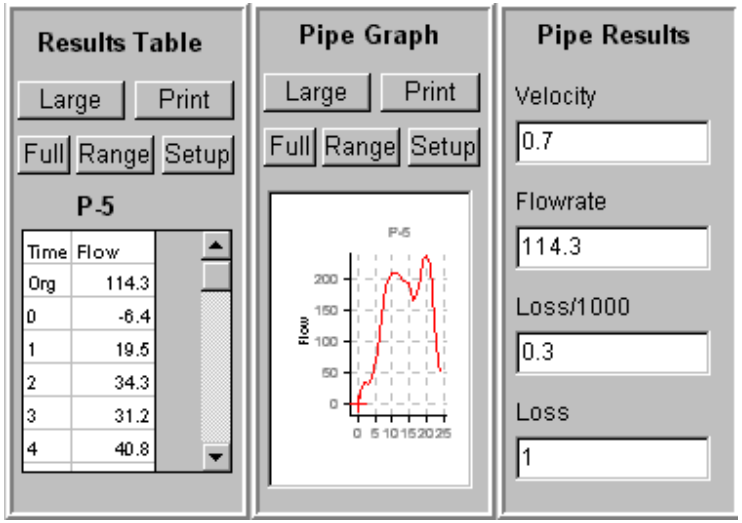

Network Plot showing Pipe Flow and Diameter and Node Pressure

Node and Pipe Results Boxes - Quick Graphs and Tables

These boxes display results for selected nodes and pipes including a Results table and graph as shown below. These provide a very quick and simple method of producing full-sized graphs and tabulated results for nodes and pipes. In group mode, tables and graphs can be generated for a set of nodes or pipes. Some examples are shown below. See also Node Results Boxes and Pipe Results Boxes concerning options for customizing the graphs and tables.

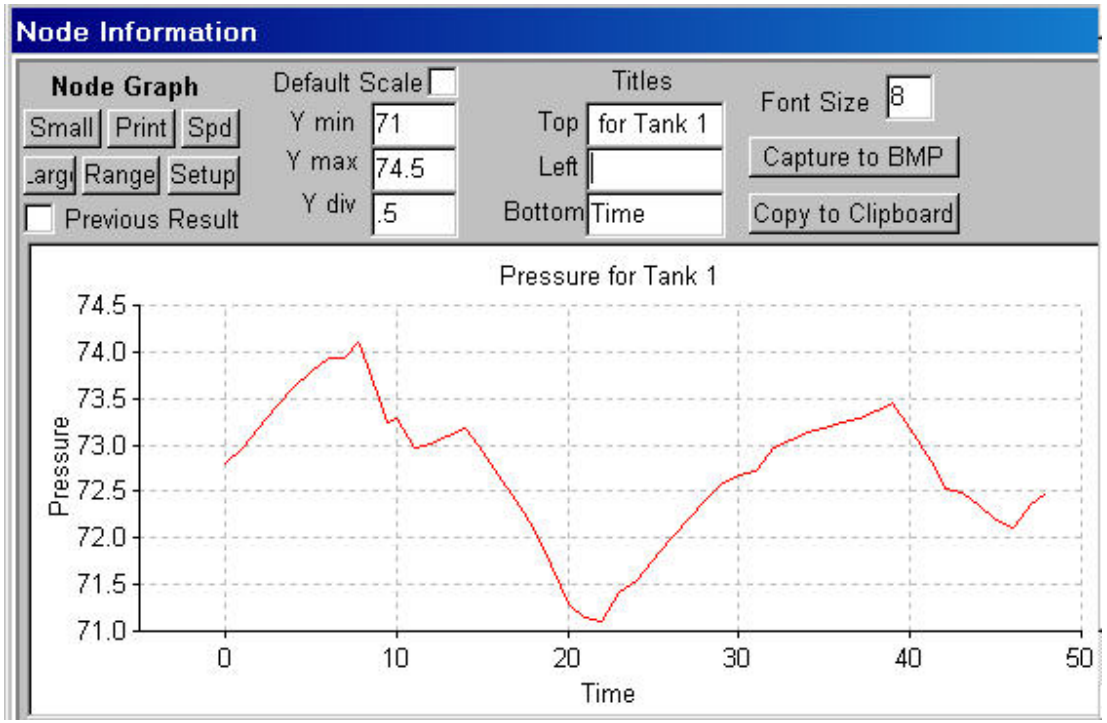
In KYnetic, to access Tables and Graphs: 

Classic:



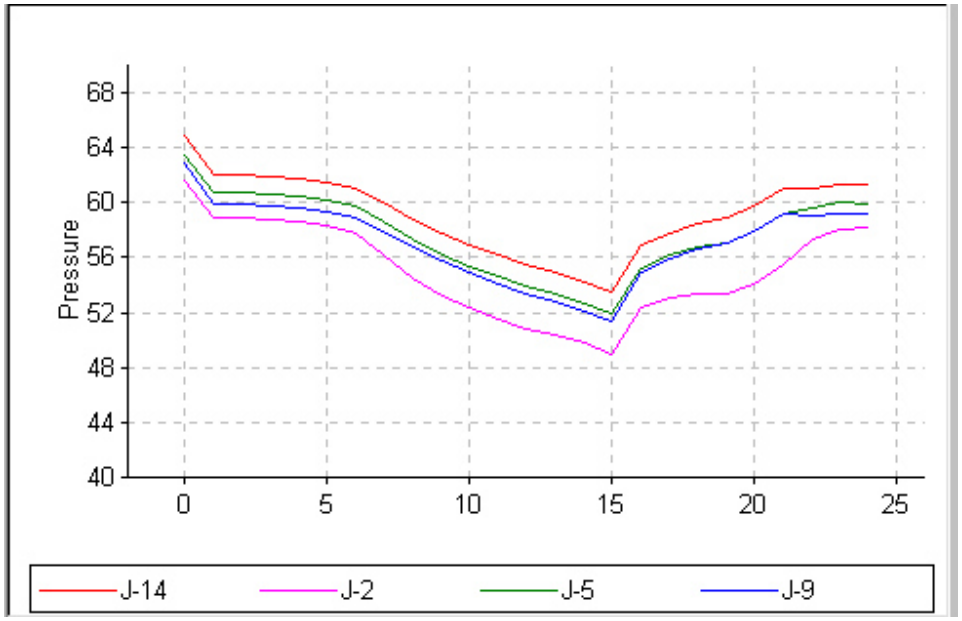

Graphs and Tables (Small View) in Pipe Results Boxes

When the Graphs or Tables are maximized (Full view), more options are available for customizing and using them for other applications, as shown below. Tables also have ASCII and EXCEL export options.


Node Information
Node Graph
Small
Print
Spd
Range
Setup
Previous Result
Default Scale
Y min
71
Y max
74.5
Y div
.5
Titles
Top
for Tank 1
Left
Bottom
Time
Font Size
8
Capture to BMP
Copy to Clipboard
Pressure for Tank 1
Pressure
74.5
74.0
73.5
73.0
72.5
72.0
71.5
71.0
0
10
20
30
40
50
Time

| Time | J-11 | J-13 | J-14 | J-2 | J-5 | J-9 | T-2 |
|---|---|---|---|---|---|---|---|
| Org | 62.3 | 53.7 | 62.8 | 58.8 | 61.2 | 60.8 | 53.7 |
| 0 | 64.7 | 56.1 | 64.8 | 61.7 | 63.4 | 62.8 | 53.7 |
| 1 | 62.0 | 53.3 | 62.1 | 58.9 | 60.7 | 59.9 | 55.6 |
| 2 | 62.0 | 53.3 | 62.0 | 58.9 | 60.7 | 59.8 | 55.6 |
| 3 | 61.9 | 53.2 | 61.9 | 58.8 | 60.6 | 59.7 | 55.5 |
| 4 | 61.7 | 53.0 | 61.8 | 58.6 | 60.4 | 59.6 | 55.4 |
| 5 | 61.4 | 52.8 | 61.5 | 58.3 | 60.2 | 59.3 | 55.2 |
| 6 | 60.9 | 52.3 | 61.1 | 57.8 | 59.8 | 58.9 | 55.0 |
| 7 | 59.7 | 51.1 | 60.1 | 56.2 | 58.6 | 57.9 | 54.7 |
| 8 | 58.2 | 49.7 | 58.8 | 54.5 | 57.3 | 56.7 | 54.2 |
| 9 | 57.1 | 48.6 | 57.8 | 53.3 | 56.2 | 55.7 | 53.5 |

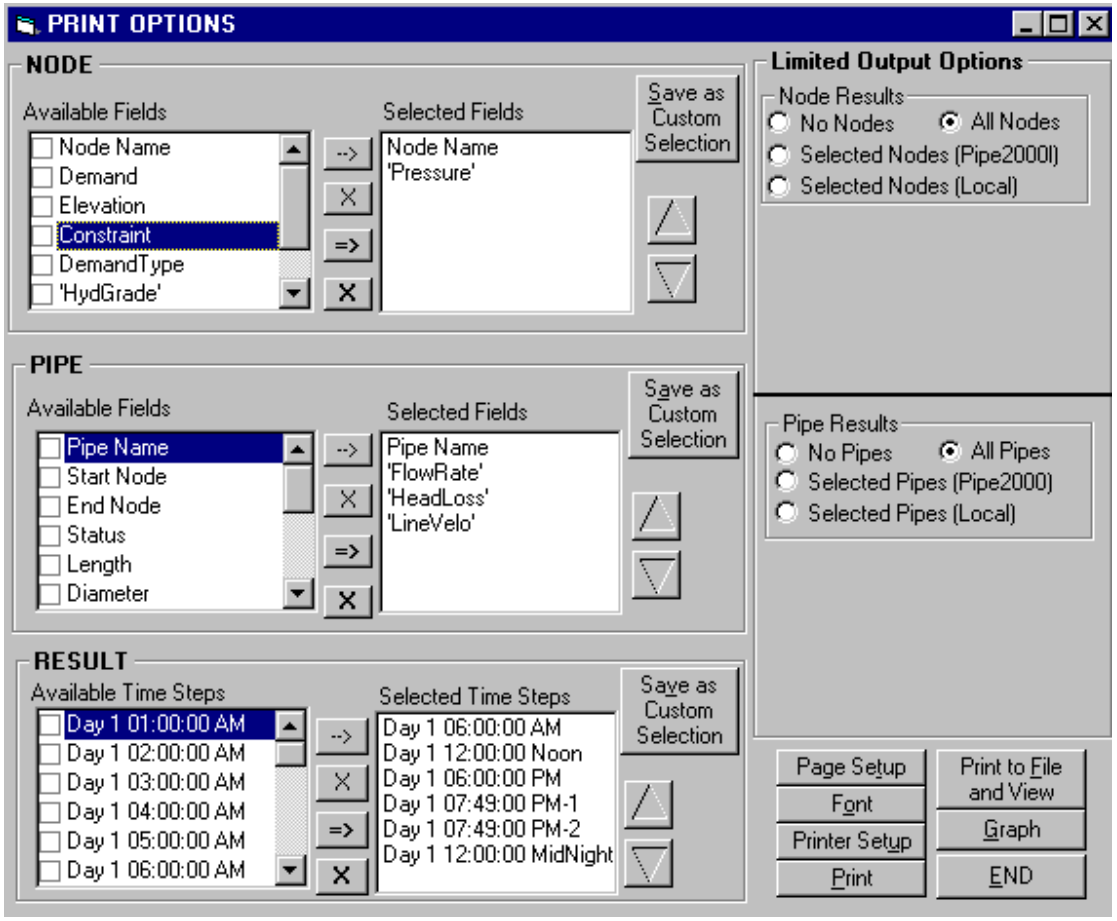
Table of Node Pressures



Plot of Node Pressures

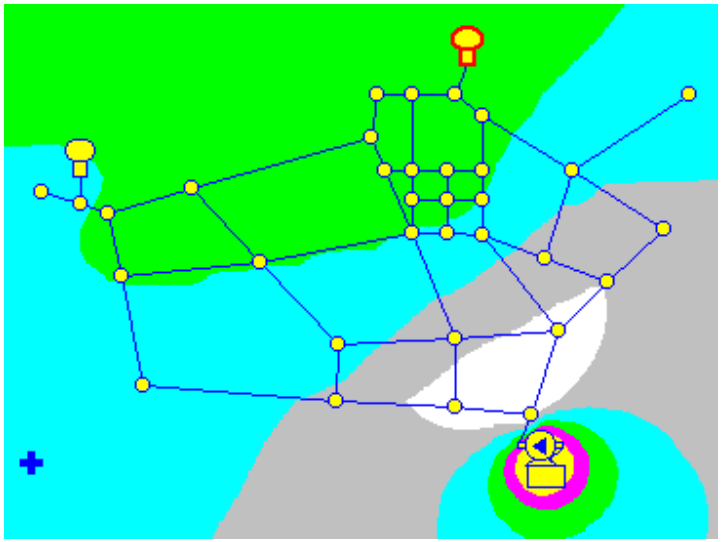
Customized Reports and Plots

**With the customized reporting and graphing module, the user may select the elements and parameter to be included in a report for viewing and printing. A plot may be created and customized by the user as shown below. The axes, the graph title, the plot style (marker), the node or pipe results to be plotted, and the print size may all be selected by the user.**



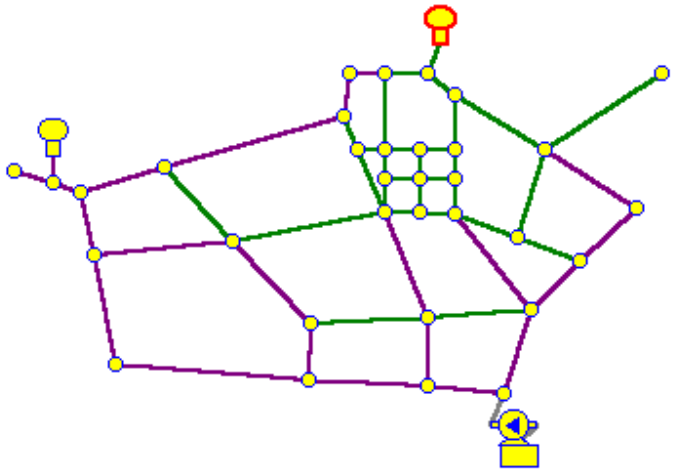

Contours

Contour maps can be generated, displayed, and printed for a variety of node data and results. A variety of contour types can be analyzed including shaded regions and lines. Contours are set up using the Emphasis / Contours tab (Map Settings).



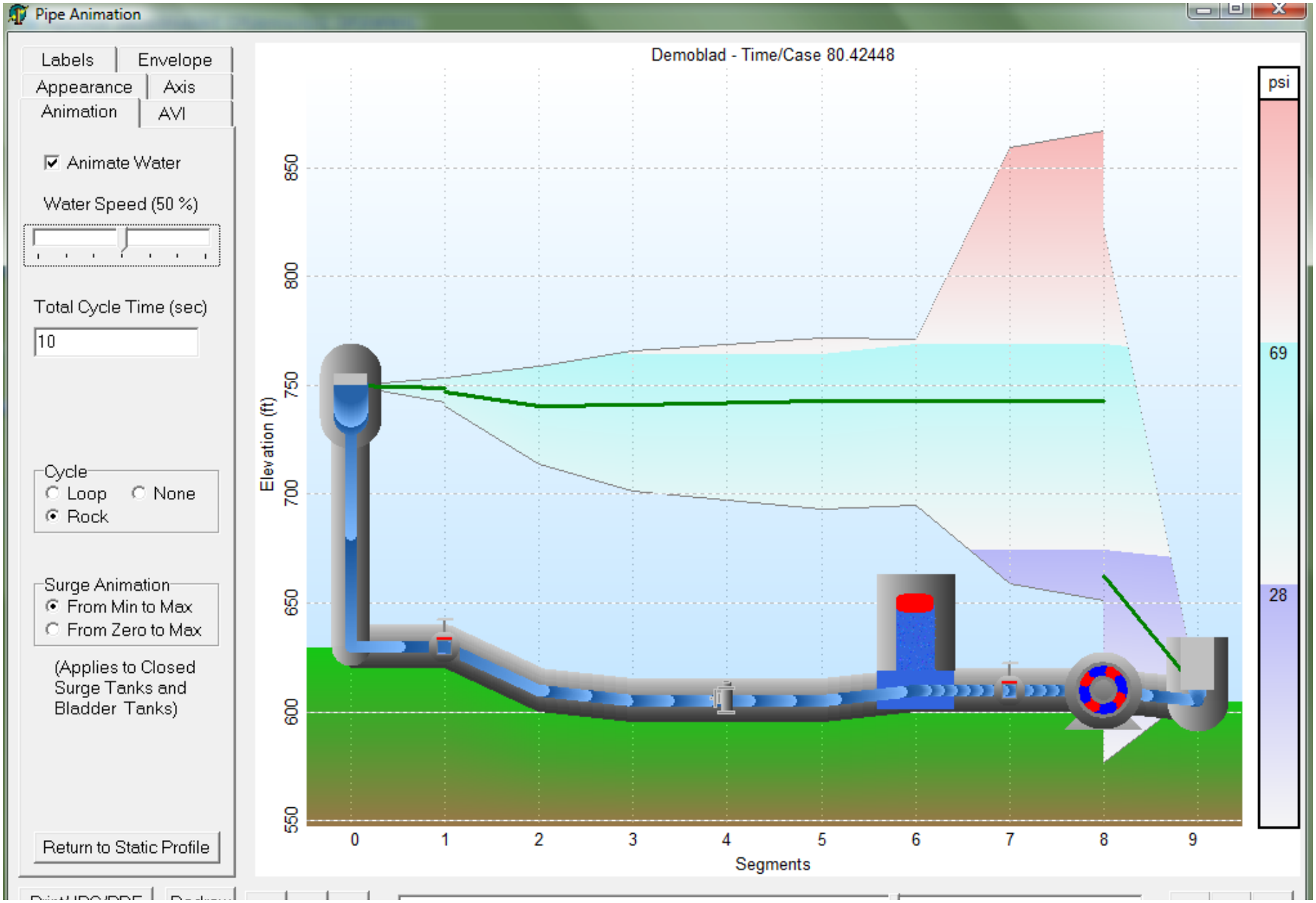
Color Emphasis

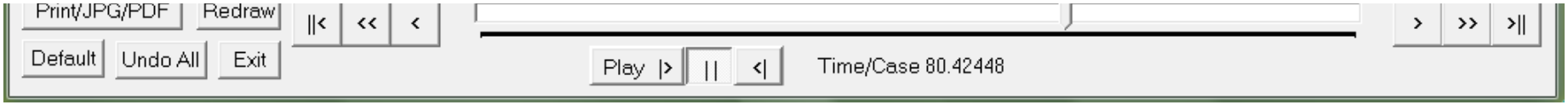
Color emphasis for nodes, pumps, or pipes and sets the color of items based upon data or results values. This is used to show data and visual trends. Color emphasis is set up for nodes and pumps using the Emphasis / Contours - Nodes tab (Map Settings) and for pipes using the Pipe Emphasis tab (Map Settings).



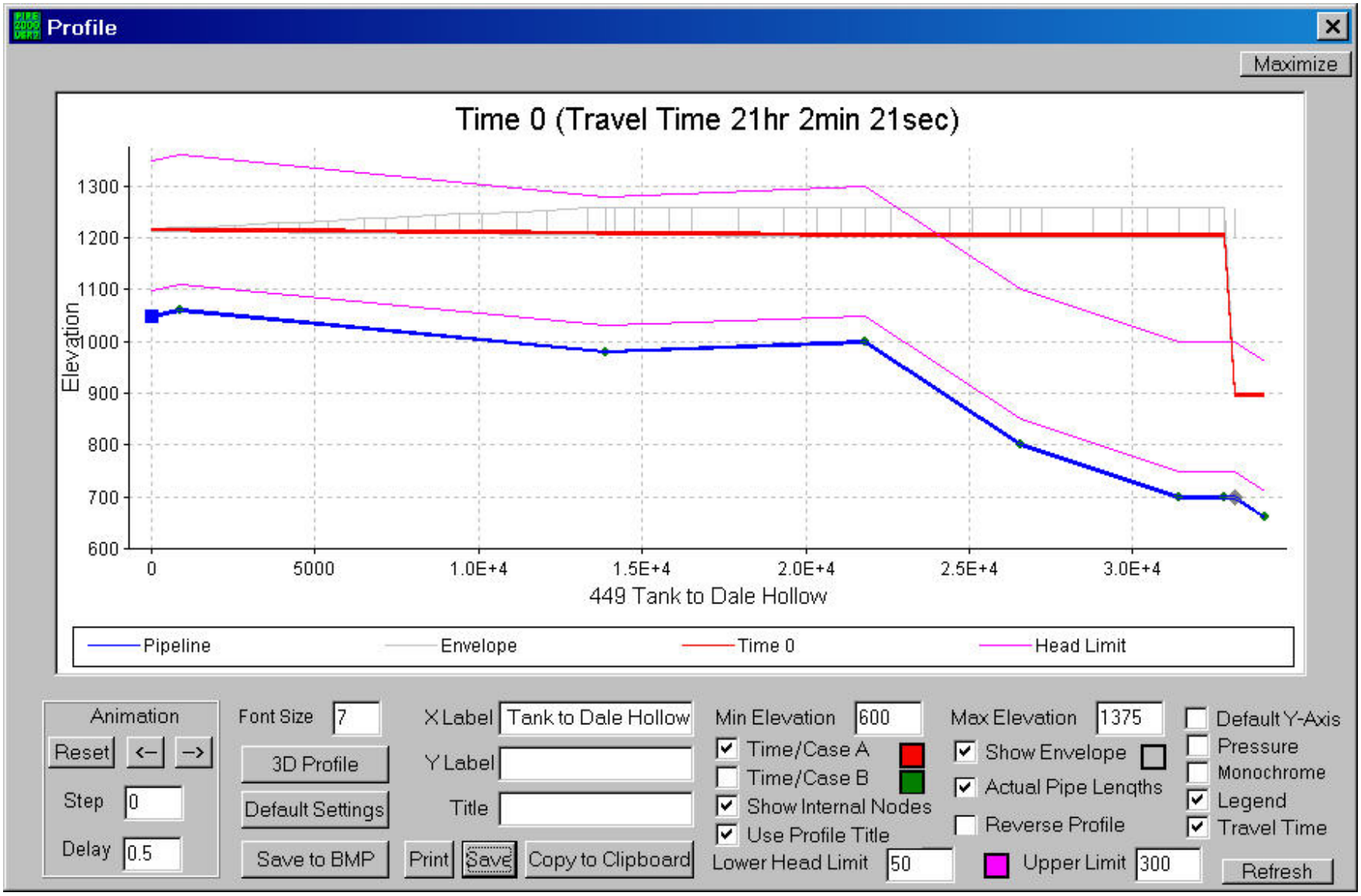
**Profile**

Profile shows a section of pipeline profile with head and maximum/minimum envelopes. Profiles are very useful for the design and operation of pipe systems. See Profile for more information.


Pipe Animation
Labels
Envelope
Appearance
Axis
Animation
AVI
Animate Water
Water Speed (50 %)
Total Cycle Time (sec)
10
Cycle
Loop
None
Rock
Surge Animation
From Min to Max
From Zero to Max
(Applies to Closed Surge Tanks and Bladder Tanks)
Return to Static Profile
Demoblad - Time/Case 80.42448
psi
69
28
Elevation (ft)
850
800
750
700
650
600
550
0
1
2
3
4
5
6
7
8
9
Segments




Previous Profile -


Profile
Maximize
Time 0 (Travel Time 21hr 2min 21sec)
Elevation
1300
1200
1100
1000
900
800
700
600
0
5000
1.0E+4
1.5E+4
2.0E+4
2.5E+4
3.0E+4
449 Tank to Dale Hollow
Pipeline
Envelope
Time 0
Head Limit
Animation
Reset
<--
-->
Step 0
Delay 0.5
Font Size 7
3D Profile
Default Settings
Save to BMP
X Label Tank to Dale Hollow
Y Label
Title
Print
Save
Copy to Clipboard
Min Elevation 600
Time/Case A
Time/Case B
Show Internal Nodes
Use Profile Title
Lower Head Limit 50
Max Elevation 1375
Show Envelope
Actual Pipe Lengths
Reverse Profile
Upper Limit 300
Default Y-Axis
Pressure
Monochrome
Legend
Travel Time
Refresh

## 1.3.8.7 Selected Output

Selected Output

---

Selected Output is a feature which allows the user to specify which pipes and nodes will be included in the Output Report when running a Pipe2018 analysis. This is especially useful for large network models for which the Output Report can be quite large. It may be time-consuming to locate the results of interest to the user. This feature is also used to select nodes for results review for Surge (see Surge Reviewing and Presenting Results)

Below is an example of how to specify and apply a group for Selected Output. For more information about Selected Output features see

Reports (System Data)

Pipe User Box

Node User Box

Sets and Group Mode

Data Tables - Limited Output

**Example:**

Let's say a user would like to see only the output for pipes 5, 6, 7, and 8. In the Map screen, enter Group Mode. Highlight pipes 5, 6, 7, and 8. Displaying User Data, click "Limited Output" row heading (KYnetic) or (Classic)under Edit Pipe Set in the Pipe Information window, click on the Item to Edit drop-down selector box. Select Limited Output (or any other user attribute). Select New Value. In the Value box, assign an integer identifier such as 1 to the Limited Output group and click Proceed.

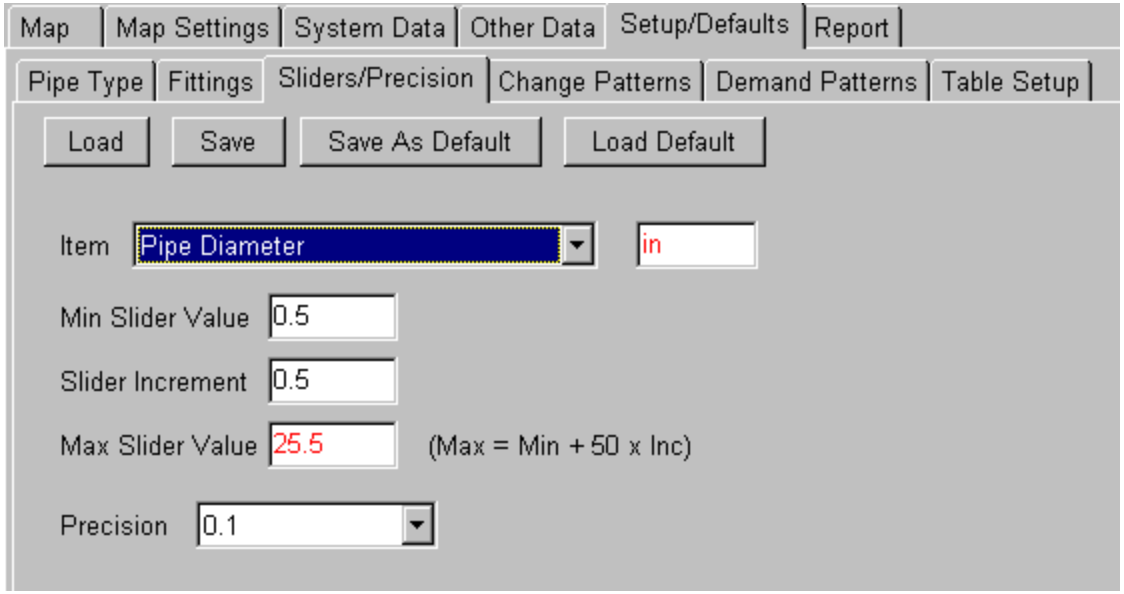
If you would like to verify that there is a Limited Output group called 1 consisting of pipe 5, 6, 7, and 8. Click Clear to unhighlight the pipes. Click the Limited Output Row heading (KYnetic) or (Classis) in the drop down box under Set Selection, click Limited Output. In the Value(s) box, a 1 should appear. Select the 1 and click on New Set. Pipes 5, 6, 7, and 8 will be highlighted.

Now to use this group for Selected Output, go into System Data / Reports. Under Pipe Output, choose Selected. In the drop-down selector under Attribute for Selected Pipe Output, choose Limited Output. Then click on the Value drop-down selector and the integer 1 should be one of the options. Select 1 then analyze the system. When the analysis is finished, view the output Report by selecting the Report tab. Under Pipeline Results, only the output for pipes 5, 6, 7, and 8 will be displayed (although all pipes will have been included in the analysis). See also Selecting Nodes for Limited Ouput.

## 1.3.8.8 Sliders/Precision

Sliders/Precision

In KYnetic, use this icon:




See the Units video on the KYPipe web site.

---

This screen controls the units and characteristics of most of the data and results and associated data sliders for each of the flow units accommodated. Items appearing in red are not directly edited using this screen. They are either set using the System Data or System Data/Simulation Specs (Classic) screen or are calculated values. Although the ability to edit this file provides additional flexibility (such as labels precision), most users will find the defaults acceptable and have no need to modify this data.

Load, Save, Save As Default, Load Default

A Sliders/Precision setting may be saved by the user as a .unt file. These buttons are used to Load an existing .unt file, Save a setting as a .unt file, Save As Default a setting, or Load Default, either the original default .unt file or one specified by the user. Note that if the original default is lost and needs to be recovered, a backup copy is available called unit.bak. Contact Tech Support.

Item

Two data fields are located by this heading. The first is a drop-down box where the user may select the data item for which Slider values and Precision may be set. In the next field, the units appear in red, meaning the units are not entered directly into this field. To set the units for a data item, go the the System Data or System Data/Simulation Specs (Classic) screen.

Min Slider Value

minimum value for data slider

Slider Increment

increment for data slider

Max Slider Value

maximum calculated value for data slider (appears in red, may not be directly edited)

**Precision**
a drop-down box provides a selection of precision choices

## 1.3.9 Import/Export

### 1.3.9.1 ArcView Import Utility

ArcView Import Utility

Utilities / Data Exchange

See also ArcView Export

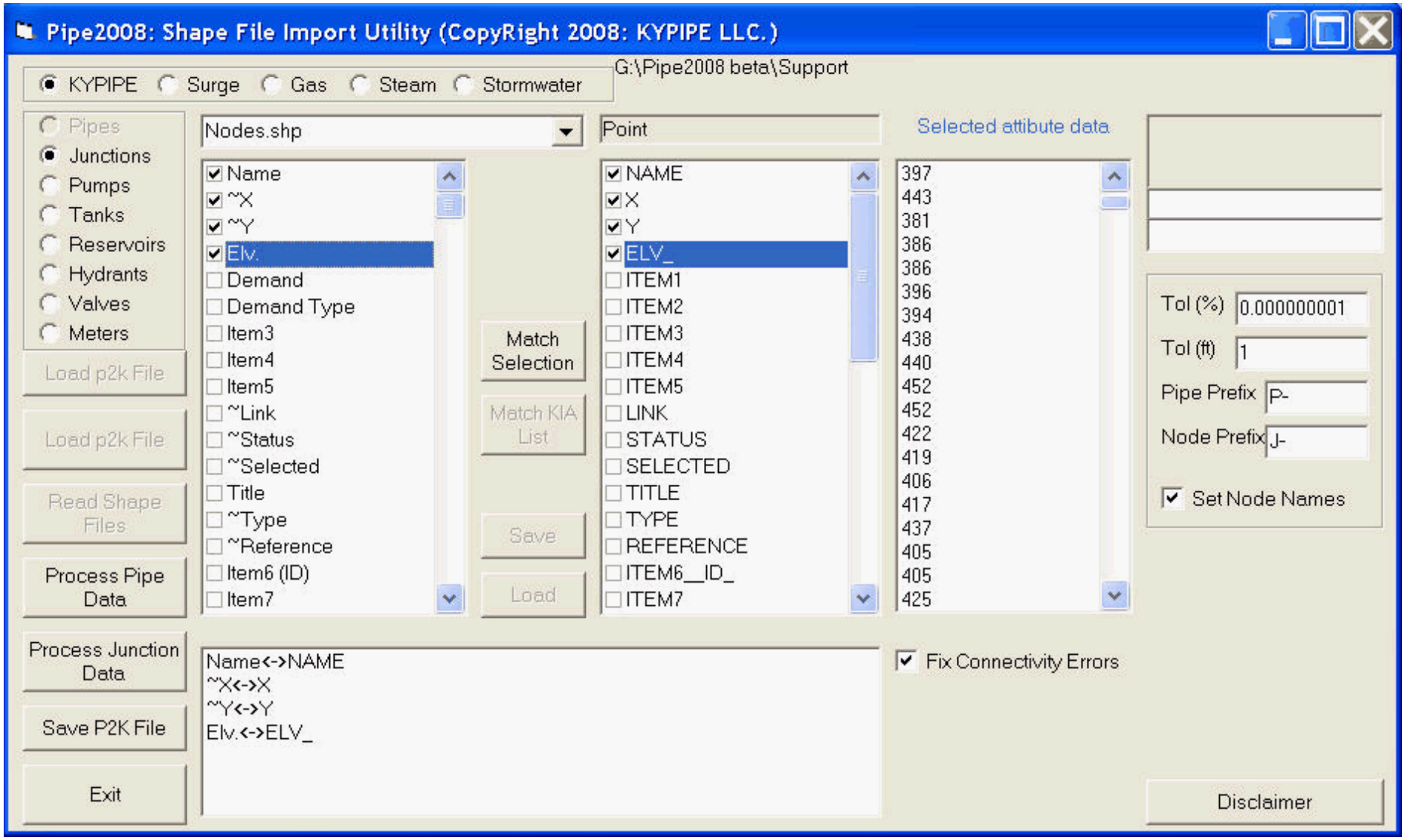
Utility Programs

# ArcView Import

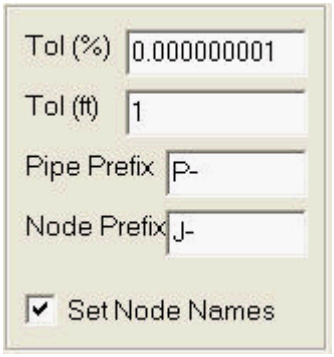
Notes about the Arcview import module. This utility has the ability to associate data with different elements (junctions, pumps, tanks etc), and save the matched items settings for future usage. There is an option to check for overlapping pipes but using this option increases conversion time. The Pipe2018 program limits the number of intermediate nodes in a single pipeline to 950. Import module allows for processing multiple line and point shape files. All line shape files must be read before processing the pipe data. The number of shapes in a shape file are displayed.

From within Pipe2018, under File the import utility may be found.

When the utility starts you will see the following window:




Before proceeding, you may set the following options:




Tol % - See below

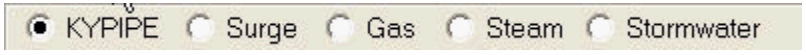
Tol (ft) - See below

Pipe Prefix - allows the user to choose a prefix for assigning pipe names.

Node Prefix - allows the user to choose a prefix for assigning node names.

Set Node Names -

To import shp files:

1. 

- Choose the file type you will be creating from these choices.

2.

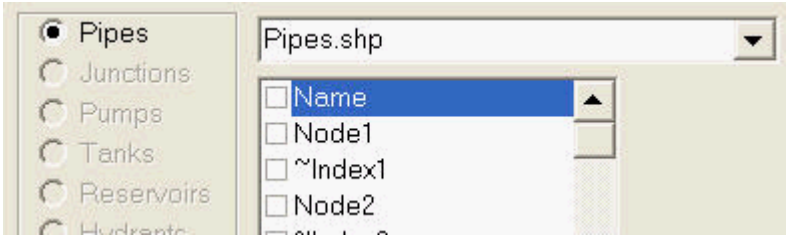
- If you wish to add GIS data to an existing .p2k file, click on this button and browse for the p2k file you wish to append.

3.

- Click on this button and browse to select one of the .shp files in the target folder.

4.

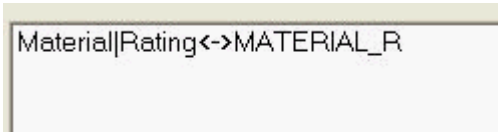
- Click to load the data in all the .shp files.

5. 

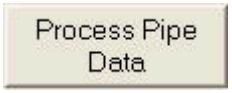
- In the drop-down selector, choose the name of the pipes .shp file.

6.

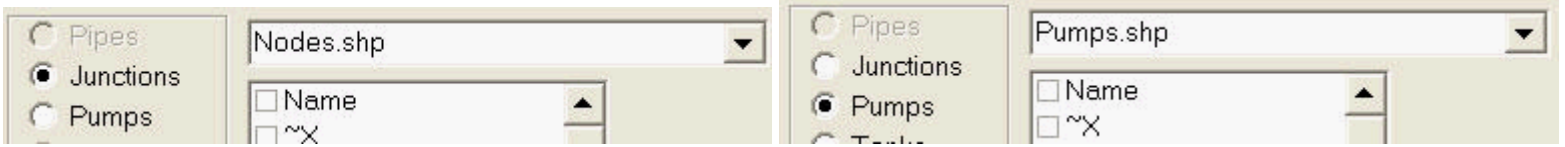
- Highlight, check, and use the Match Selection button to associate attributes in the two columns. Matched selections will appear in the bottom field as follows:



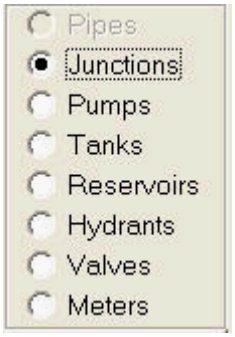
. See "Matching Nodes" below for data type information for various nodes.

7. 

- Click this button to process the pipe data.

8. 

- Next, use the drop-down selector to choose available node .shp files.



- **IMPORTANT**: select the type of node represented by the shp file in the top left side of the window.

9. Match attributes as described in step 6.

10.

- Click the Process Data button for each node type.

11.

- When all the node .shp files have been processed, save the .p2k file.

12. In Pipe2018, open the newly created .p2k file. Use the Zoom All (Z All) button to view the system.

### Matching Nodes

With node items usually very few matches are made automatically because of the diverse names (like Pump Power for pumps and Tank Inlet Height for tanks). You will need to look at the table below to determine where each item needs to be matched for a specific node type.

| (~type) Type | ITEM1 | ITEM2 | ITEM3 | ITEM4 | ITEM5 | ITEM6 | ITEM7 | ITEM8 | ITEM9 | ITEM10 |
|---|---|---|---|---|---|---|---|---|---|---|

| | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|
| (1) Junction | Dmd 1 | Dmd 1 Type | Dmd 2 | Dmd 2 Type | Dmd 3 | Dmd 3 Type | Dmd 4 | Dmd 4 Type | Dmd 5 | Dmd 5 Type |
| (2) Tank | Max Level | Min Level | Init. Level | Inflow | Vol | Shape ID | | | | |
| (2) Tank (Fixed Diameter) | Max Level | Min Level | Init. Level | Inflow | Diam * -1 | | | | | |
| (3) Reservoir | Grade | | | | | | | | | |
| (4) Pump (Table)<a> | Speed | Grade | | | 0 | Pump ID | | | | |
| (4) Pump (Power)<a> | Power | Efficiency | Grade | | 1 | | | | | |
| (4) Pump (Rated)<a> | Pressure | Flow | Grade | | 3 | | | | | |
| (5) Check Valve <b> | | | | | | | | | | |
| (6) Hydrant | Static Pressure <c> | Residual Pressure <c> | Residual Flow <c> | | | | | | | |
| (7) Valve | | | | | | | | | | |
| (8) Sprinkler | K Factor | Riser Length | Riser Diameter | Riser Elevation Change | Elbows | | | | | |
| (9) Regulator <a> | Setting | Type [0-4]<d> | | | | | | | | |
| (10) Metered Connection | | STATUS = 0 | | | | | | | | |
| (10) In-Line Meter | | STATUS = 1 | | | | | | | | |
| (11) Loss Element<a> | Grade | | | | | Loss Element ID | | | | |
| (12) Active Valve | Resistance WO | Initial ratio | Grade | Type [0-5] <e> | | | | | | |
| (13) SDO | Inflow R | Outflow R | | | | | | | | |

| | | | | | | |
|---|---|---|---|---|---|---|
| (14) Pressure Supply | Gauge Elevation | | | | | Pressure Supply ID |
| (15) Intermediate Node | | | | | | |
| (16) BFP <a> | Unique ID # | | | | | |
| (17) Rack Sprinkler | K Factor | Riser Length | Riser Diameter | Elevation Change | Elbows | |

<a> = direction (inlet to outlet) determined by status [0,1] 0=right to left, if pipe connects to left use negative index #

<b> = always in direction specified by pipe N1 to N2

<c> =measured values

<d> = [PRV1, PRV2, PSV, FCV1, FCV2]

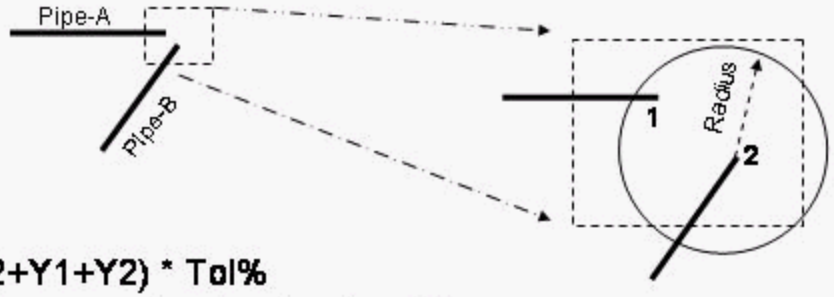
<e> = [Ball, Butterfly, Gate, Globe, Needle, User-defined]

Tolerances

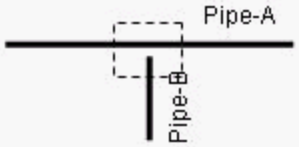
Note if there are nodes in a model that are very close to each other - e.g. within 0.5ft or so, setting the length tolerance to a number such as 0.1 may be helpful.

**Tolerances (Tol% and Tol_Len) - used for checking the proximity of nodes.**

1. **While processing pipe information read from line shape file:**
   - **The proximity of end nodes of different pipe segments is checked.**



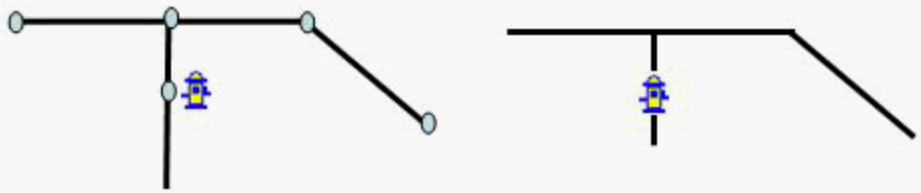

- **Radius = 0.25*(X1+X2+Y1+Y2) * Tol%**
  - **X1,Y1 and X2,Y2 are coords of nodes 1 and 2**
- **if Radius > Tol_Len then Radius = Tol_Len**
- **If distance between nodes 1 and 2 is less than Radius then pipes A and B are joined (Node 2 is moved to Node1 location)**




**Pipe-B will be joined to Pipe-A if the gap between the two Pipes is less than the "Radius"**

**Tolerances (Tol% and Tol_Len) - Used for checking the proximity of nodes.**

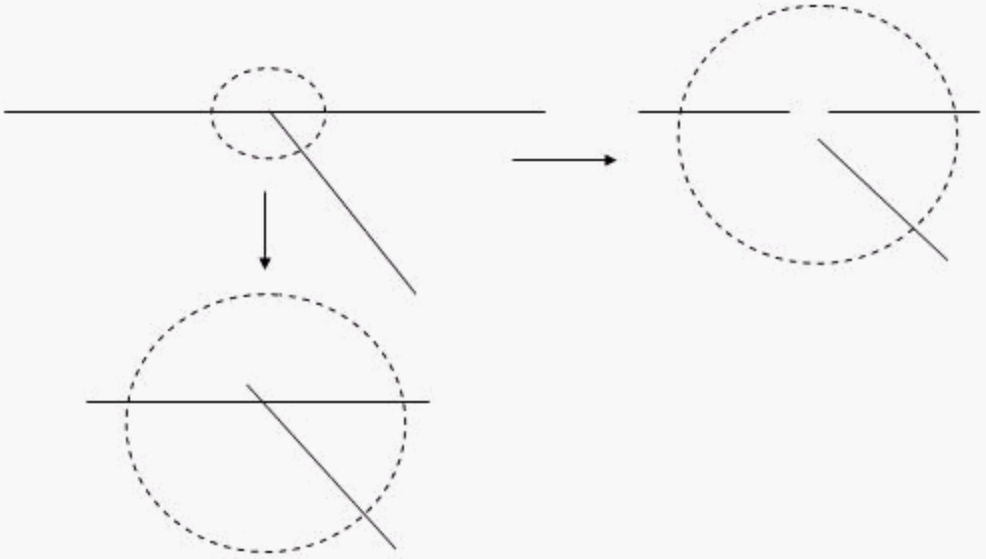
2. While merging data from point shape files with the existing pipe network model



Proximity of nodes to a hydrant will be tested first and if an intermediate or a junction node exists within the "Radius" then that node will be replaced with the hydrant node. If not, the hydrant will be moved to the nearest pipeline.

# Helpful hints in generating quality shape files

- Make sure the end points of pipe segments are connected correctly.
- At certain zoom levels, the pipe segments might look connected but they may not be.



## 1.3.9.2 ArcView Export Utility

ArcView Export Utility

Utilities / Data Exchange

See also ArcView Import Utility

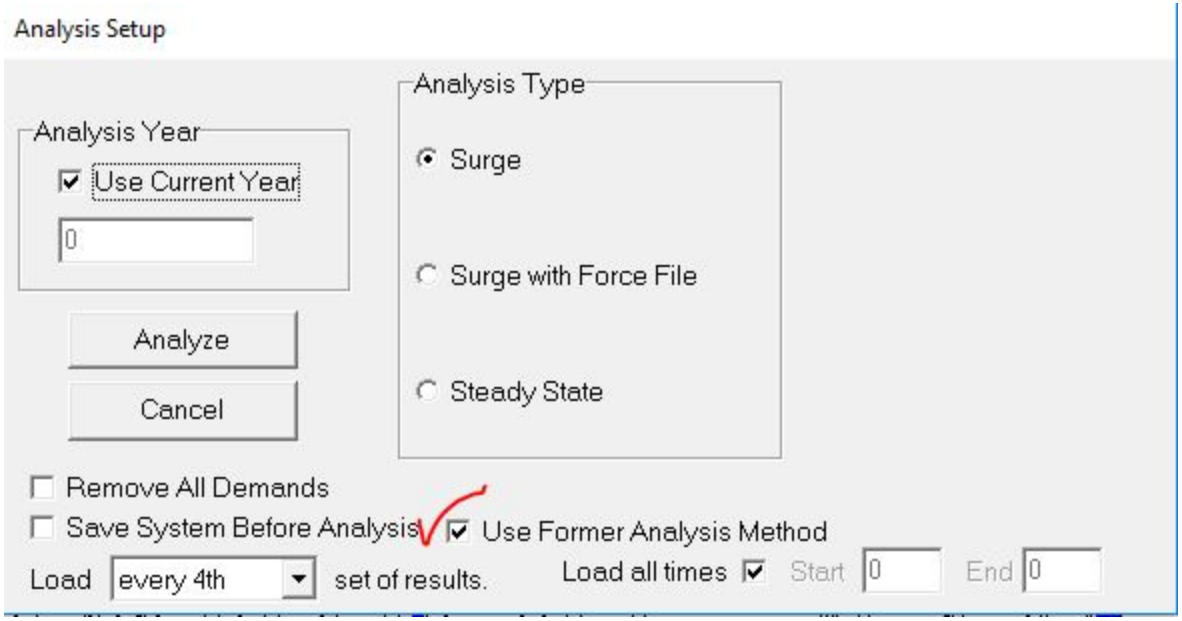
Utility Programs

---

The ArcView Export Utility is a program that exports your Pipe2018 piping system along with selected data items to an ArcView shape file.

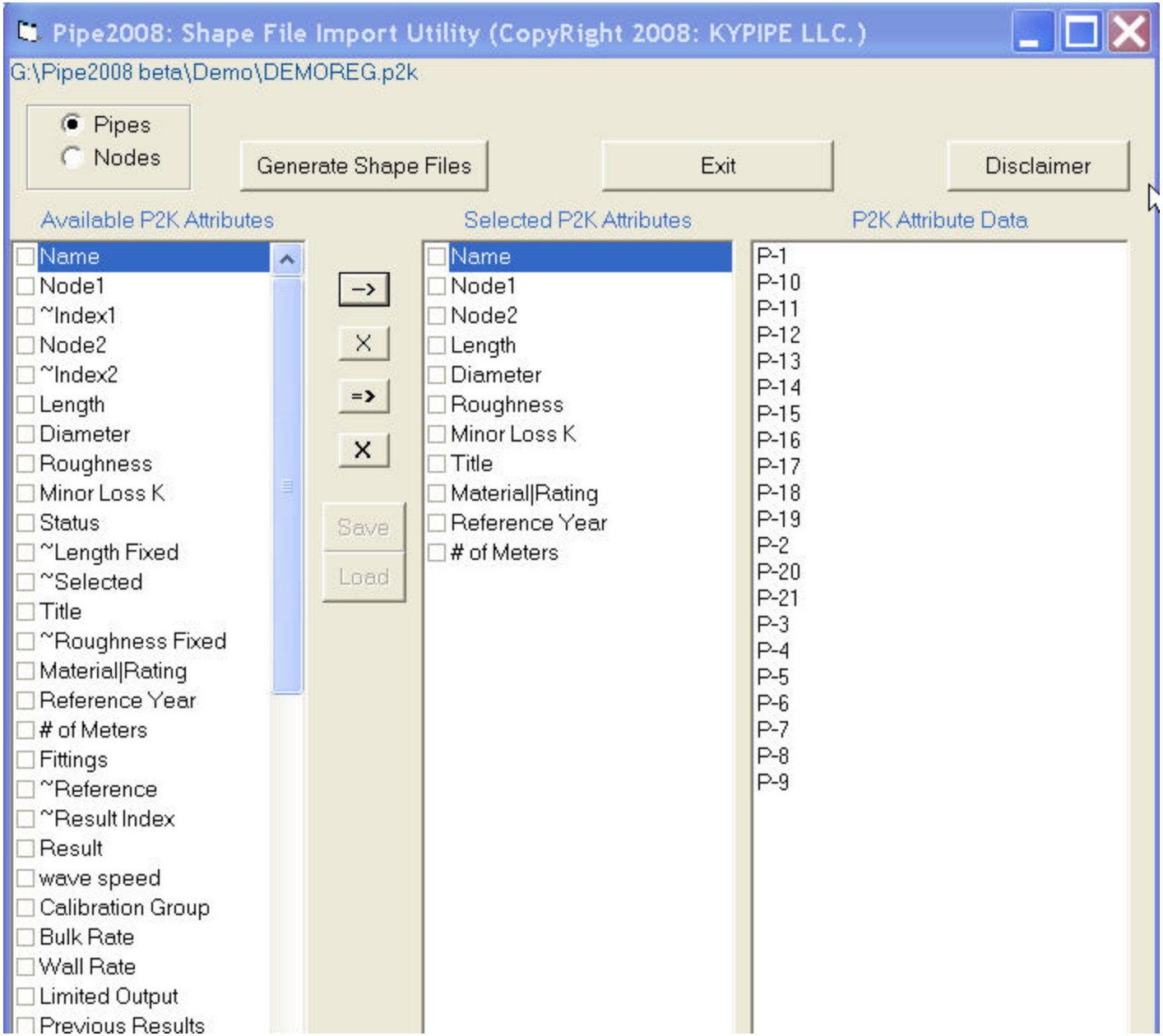
Below are the steps to export your information to an ArcView shape file. The Export Arcview utility may be found under File in Pipe2018.

An analysis must first be run in order to export the model. Open the Analysis Setup window and make sure "Use Former Analysis Method" is checked.




The results do not necessarily have to be valid, but the analysis process will produce necessary internal data files used by the ArcView Export utility.

The following window appears:


Pipe2008: Shape File Import Utility (CopyRight 2008: KYPIPE LLC.)
G:\Pipe2008 beta\Demo\DEMOREG.p2k
Pipes
Nodes
Generate Shape Files
Exit
Disclaimer
Available P2K Attributes
Selected P2K Attributes
P2K Attribute Data
Name
Node1
~Index1
Node2
~Index2
Length
Diameter
Roughness
Minor Loss K
Status
~Length Fixed
~Selected
Title
~Roughness Fixed
Material|Rating
Reference Year
# of Meters
Fittings
~Reference
~Result Index
Result
wave speed
Calibration Group
Bulk Rate
Wall Rate
Limited Output
Previous Results
->
X
=>
X
Save
Load
Name
Node1
Node2
Length
Diameter
Roughness
Minor Loss K
Title
Material|Rating
Reference Year
# of Meters
P-1
P-10
P-11
P-12
P-13
P-14
P-15
P-16
P-17
P-18
P-19
P-2
P-20
P-21
P-3
P-4
P-5
P-6
P-7
P-8
P-9

Use the buttons to select which attributes will be exported by populating the 'Selected P2K Attributes' column.

- causes the attributes checked in the 'Available' column to be copied to the 'Selected' column

- causes the attributes checked in the 'Selected' column to be deleted.

- causes ALL of the attributes in the 'Available' column to be copied to the 'Selected' column

- causes ALL of the attributes in the 'Selected' column to be deleted.

Do this for both Pipes and Nodes.

Once the 'Selected' column is finished, click Generate Shape Files. Shape files called 'Pipes' and 'Nodes' will be created in the same file folder as the original p2k file.

## 1.3.9.3 AutoCad Exchange

AutoCad Exchange

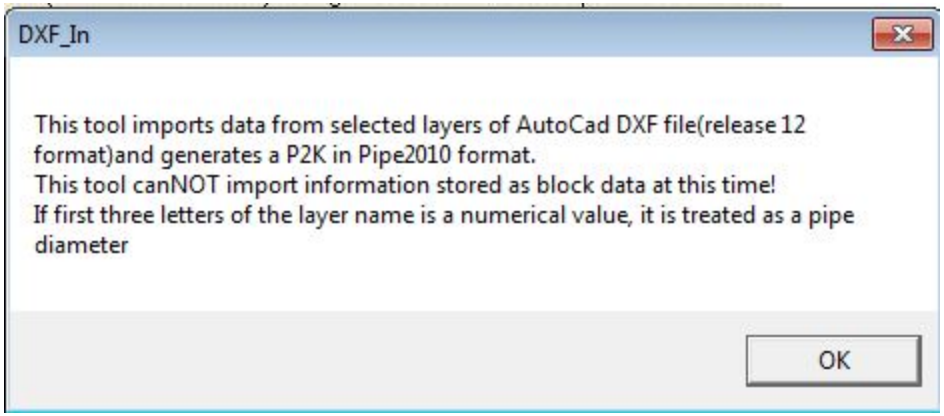
To make your Pipe2018 data file appear in AutoCad.

Utility Programs

---

**AutoCad Import**

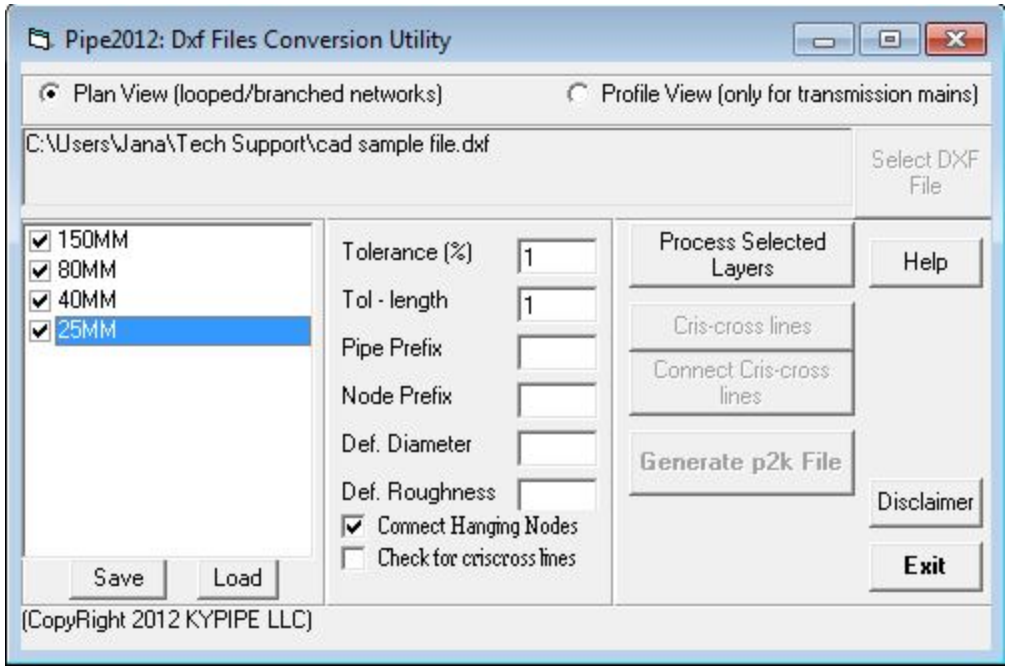
Here are some **important notes** before proceeding with the import:




Note that AutoCAD R12 format is required. Save to this format before before proceeding with import.

From within Pipe2018 the Import DXF is found under File - Import Export (or Pipe2000 Utilities in Classic)

You will be prompted to save your current p2k file. Then the following window appears:



Select Plan View or Profile View.

Note: Some elevation data in DXF may be successfully imported. Be sure to use correct configuration: Plan View or Profile View. Profile View is only for transmission mains. Coordinate X, Y and Z data have different interpretations in these two layouts.

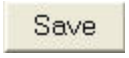
Click

and browse to the DXF release 12 file you have prepared.

Once the file name appears at the top, click

.

Layer names will appear on the left. Check the layers to be imported.

At this point, if you wish for the selected layers to be available for future imports, you may click 

. Then next time you import this same dxf, click

to load the layer selection.

Tolerance - This refers to the distance between lines in AutoCAD. Distances smaller than this number will be considered to be connected.

Tolerance Length - Pipes shorter than this length will not be imported as an individual pipe. This will reduce the overall number of pipes in the resulting p2k model.

Default Diameter and Default Roughness - these values will be applied to the imported pipes, unless a layer named with three numerical characters is defined for pipe diameters.

Click

.

Then you may either click

or

. A file will be generated called filename.p2k or filename.DT2 in the same folder as the original dxf file.

Click

. You may now open the newly created p2k file.

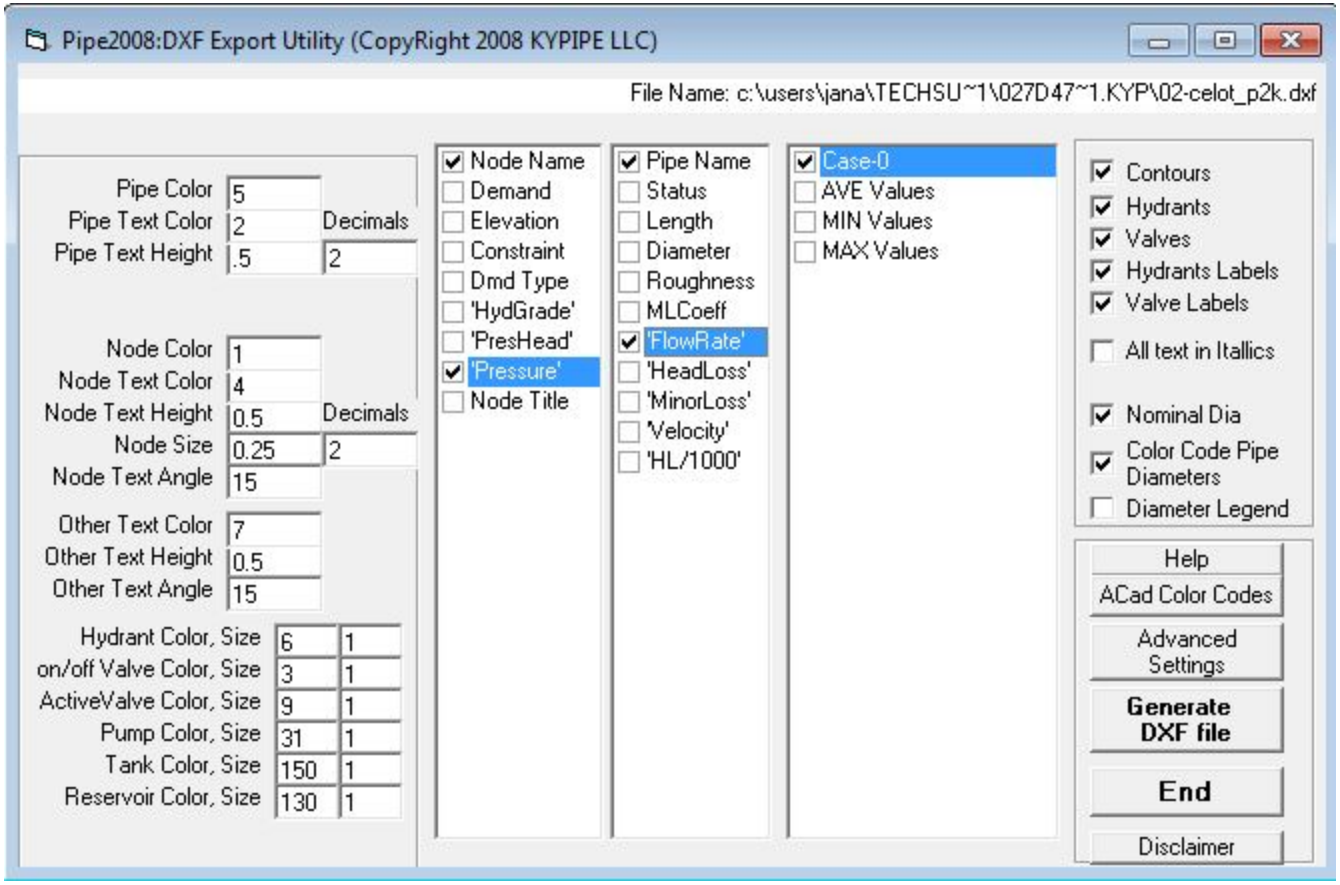
**AutoCAD export:**

To make your Pipe2018 data file appear in AutoCad, do the following:

1. Run Pipe2018.
2. Load your data file.
3. Select Analysis.  In the Analysis Setup box, check the "Use Former Analysis Method" box.
4. Run the Analysis.
5. Under File select Export DXF.

Export Contours - Important Note - To export contours, the **contour style must be set to Line style** before opening the DXF Export utility.

You will be prompted to save your current p2k file. Then the following window will appear:




Once the text, node, and pipe sizes and colors have been chosen, the settings may be saved (click Save). Next time the DXF Export is opened, click Load to load the saved settings.

Use the ACAD Color Codes button to reference color codes and choose colors for the dxf file.

Choose one node and one pipe label and one Case/Time from each list.

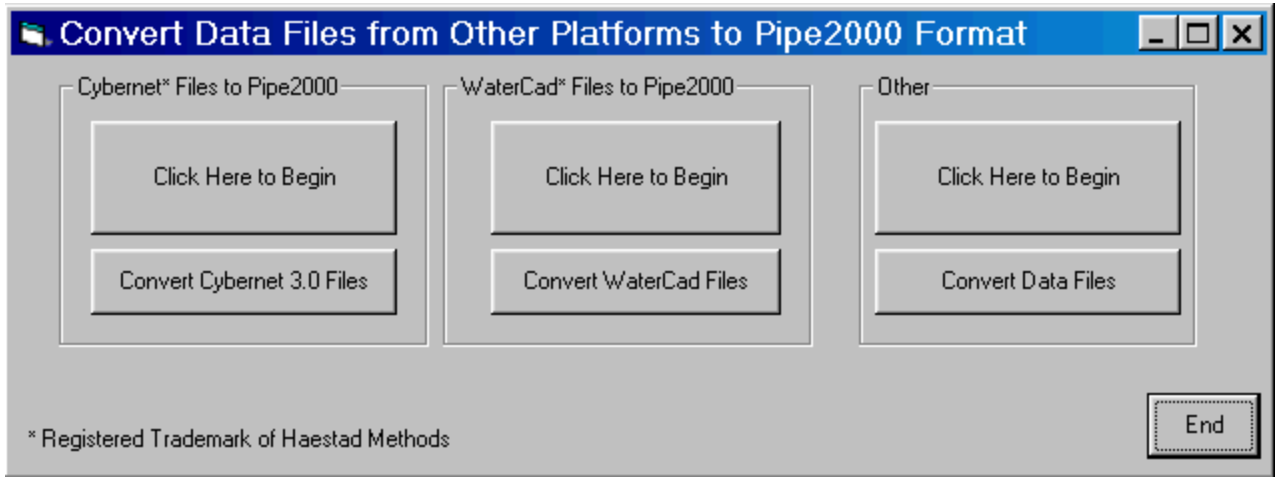
When the settings have been chosen, select Generate DXF file. A file called "filename"_p2k.dxf will be created in the same folder as the original .p2k file.

## 1.3.9.4 Convert WaterCAD

Convert WaterCAD

---

Found under File. Each conversion utility contains detailed step-by-step instructions.




## 1.3.9.5 Copy and Paste

Copy and Paste

---

### Copy and Paste Pipes

You may select a group of pipes and nodes and copy them with all assigned attributes to another system (another .p2k file). Using the GBox or individually selecting nodes and pipes in Group Mode, select the portion of the system which you want to copy. Under Edit in the Main menu, select copy. Close the file and open the file to which you want to paste. Under Edit select paste. This will paste the nodes and pipes at the same coordinates at which there were located in the original file. However, if you enter Layout mode and select a node and then paste, the paste will occur at that node. The pasted system will not be connected to or overwrite any part of the system to which it was pasted, even if two nodes are at the same coordinate location. After pasting the system you can click near the

pasted pipes and nodes and drag them to the desired position. It may be necessary to de-select and re-select one of the nodes if the click and drag doesn't work initially.

## Excel – Merging Pipe2018 Data Files using Excel (For Version 1 Users)

**The main purpose of merging the Pipe2018 datafiles is so the user can work on a simple subsystem and later merge the subsystem datafile into an existing main system datafile. There are a few restrictions that must be adhered to concerning the subsystem file before the merger can be successful.**

1. No Duplicate Names. The Subsystem Datafile cannot have any pipe, node or component names identical within the Subsystem file itself or identical to an existing name within the Main System Datafile.

2. No Change Data or EPS Data can be merged from the Subsystem Datafile into the Main System Datafile. Any of this data for the Subsystem file will need to be added through the final merged datafile.

3. The Subsystem Datafile cannot have any Intermediate nodes or components in a pipe between end nodes such as check valves, hydrants, control valves, etc. Only those components that can be deemed as end nodes may be included in the Subsystem Datafile such as tanks and pumps. However, using the Edit Group feature of Pipe2018 the user can change the intermediate data into end node data, rather than lose it, and change it back to intermediate data in the merged file.

### MERGING DATAFILES USING EXCEL

### IN Pipe2018:

**1. LOAD Pipe2018.**

**2. OPEN the Main System File** and make any necessary changes before opening EXCEL.

**3. Click FILE | Import Export (Pipe2000 Utilities in Classic interface) | EXPORT EXCEL FILE.**

**4.** On the Save the file? Information box, **Click Yes**.

**5.** On the Saving C:\pathname\systemfilename.XLS Pipe2018 box, **Click OK**.

**6. OPEN the Subsystem File** and make any necessary changes to comply with the rules defined above before opening EXCEL.

**7. Click FILE | Import Export (Pipe2000 Utilities in Classic interface) | EXPORT EXCEL FILE.**.

**8.** On the Save the file? Information box, **Click Yes**.

**9.** On the Saving C:\pathname\subsystemfilename.XLS Pipe2018 box, **Click OK.**

## IN MICROSOFT EXCEL:

**1. LOAD Microsoft Excel.**

**2. OPEN the System Excel File.**

**3. Align the System Excel File Window** so the top of the System file is at the top of the Excel file window and the bottom of the System file is below the halfway point of the Excel file window. See figure below.

**4. Open the Subsystem Excel File. Do Not** close the System file opened in step 2.

**5. Align the Subsystem Excel File Window** so the bottom of the Subsystem file is at the bottom of the Excel file window and the top of the Subsystem file overlaps the bottom of the System file. See figure below.

**MERGE NODES:**

**1. Click on the NODES tab** at the bottom of the Subsystem file.

2. Starting at Column A, Row 2 **hold left click down and drag mouse down and to the right** to highlight all rows of nodes and all columns from Column A to Column AA.

**3. Click Edit | Copy.**

**4. Click anywhere on System file.**

**5. Click on the NODES tab** at the bottom of the System file.

**6. Click on Column A on the first row after all node data.** The cell becomes highlighted.

**7. Right click highlighted cell and Paste.** Node data is now merged.

**MERGE PIPES:**

**1. Click anywhere on Subsystem file.**

**2. Click on the PIPES tab** at the bottom of the Subsystem file.

**3.** Starting at Column A, Row 2 **hold left click down and drag mouse down and to the right** to highlight all rows of pipes and all columns from Column A to Column Z.

**4. Click Edit | Copy.**

**5. Click anywhere on System file.**

**6. Click on the PIPES tab** at the bottom of the System file.

**7. Click on Column A on the first row after all pipe data.** The cell becomes highlighted.

**8. Right click highlighted cell and Paste.** Pipe data is now merged.

**MERGE PUMP IDs:**

**1. Click anywhere on Subsystem file.**

**2. Click on the Pump ID Data tab** at the bottom of the Subsystem file.

3. Starting at Column A, Row 2 **hold left click down and drag mouse down and to the right** to highlight all rows of pump ID s and all columns from Column A to Column AM.

**4. Click Edit | Copy.**

**5. Click anywhere on System file.**

**6. Click on Column A on the first row after all pump ID data.** The cell becomes highlighted.

**7. Right click highlighted cell and Paste.** Pump ID data now merged.

**SAVING MERGED FILE:**

**1. Click anywhere on Subsystem file.**

**2. Click File | Close.**

**3. Click File | Save As.**

**4.** Give the merged file a new filename and **click Save**.

**5. Click File | Exit.**

## IN Pipe2018:

**1. Reopen the Pipe2018 window.** If Pipe2018 is not still loaded then **RELOAD Pipe2018**.

**2. Click FILE | Import Export (Pipe2000 Utilities in Classic interface) | IMPORT EXCEL FILE.**

**3.** On the Save the File? Information box, **Click No.**

**4.** On the Opening File box **select the merged file and click OK.**

**5. Correct the Pump ID s for Subsystem pumps.** The pumps from the Subsystem file will have new ID numbers assigned in the new merged file. The correct pump curve data is included in the merged file but the assigned ID numbers are now different and needs to be corrected.

For copy and paste of node data, see Node Information Window

## 1.3.9.6 Cybernet Import

Cybernet Import

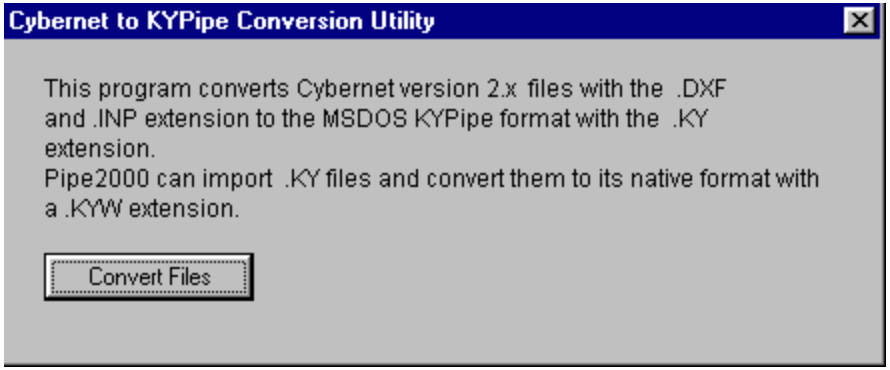
Pipe2000 Utilities / Data Exchange

Utility Programs

---

This utility converts Cybernet 2.x files with the .DXF and .INP extension to the KYPipe format .KY used by later version of KYPipe. Once in the .KY format, the user may enter Pipe2018 and use the Import KY command (Main Menu/File) to import the file and create a new .P2K file.

To enter this utility, click the Cybernet Import icon in the Pipe2018 directory. The screen below will appear. Click Convert Files. This prompts the user for the .DXF and .INP files to be converted. Once selected the conversion is done automatically and the new .KY file will be found in the same directory as the original .DXF file.




For later version of WaterCAD, see Convert WaterCAD

## 1.3.9.7 DAT Import

DAT Import

Pipe2000 Utilities / Data Exchange

Utility Programs

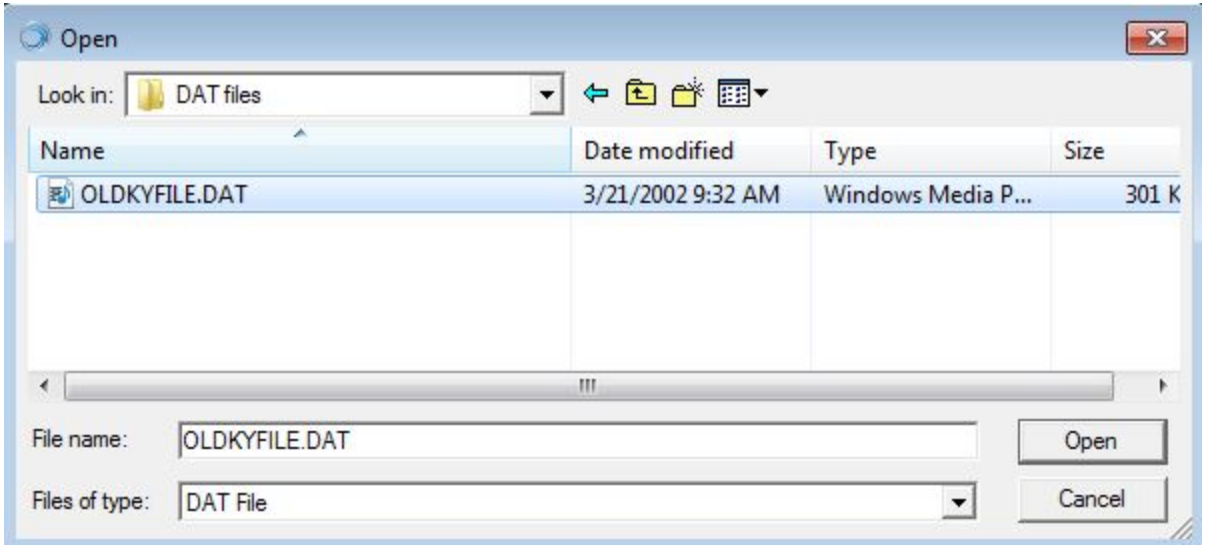
This utility is used specifically to convert KYPipe files from KYPipe versions prior to KYPipe2 Plus. The utility converts these MSDOS files from the .DAT format to .P2K format used by the current Pipe2018 program.

You may append existing p2k files or create a new file. If you are appending and existing file, load it first. Then launch the DAT Import utility.

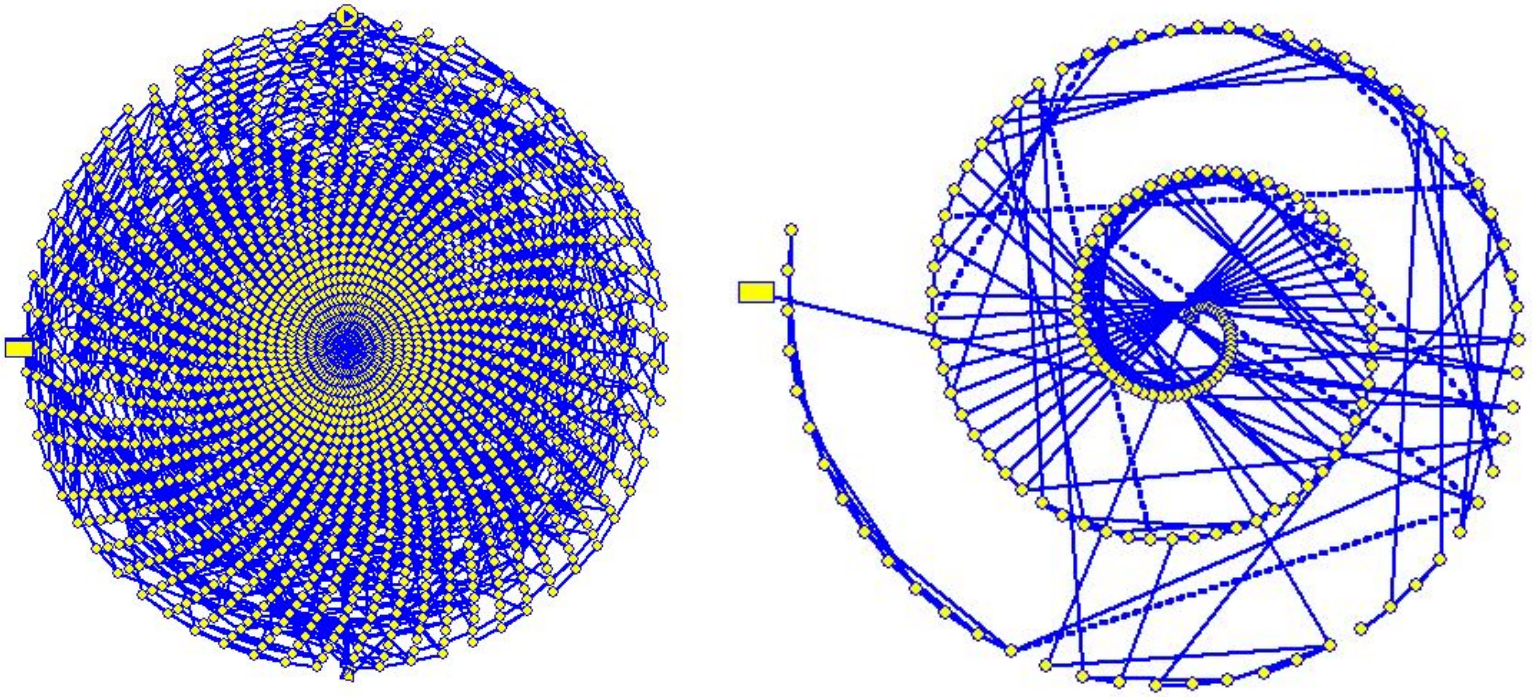
If you are creating a new file, go to File - New and then open the DAT Import utility.

To launch, go to File - Import/Export (KYnetic) or File - Pipe2000 Utilities (classic). Click this button:

Browse to the DAT file and click Open.




The file is imported and nodes are spread out in a spiral shape. Some examples are shown below.



The reason for this shape is that old KYPIPE 2 and earlier models lack coordinate data. In order to be able to manage data graphically, the nodes need to be visible in the Map window. Lacking coordinate data, all nodes imported from the DAT file would appear at coordinate (0,0) and being in the same location, they are stacked on top of one another and cannot be selected with the mouse pointer. The nodes being spread out in a spiral allows the modeler to find and select specific nodes.

Tip: use View - Find Node and type in the node name to find a specific node on the map.

Notes:

Disconnected reservoirs - Original DAT file format allowed for the input of reservoir grade data without an associated node or pipe. Thus disconnected reservoirs may exist in the converted p2k file.

Pump elevation - Original DAT file format did not require an elevation for pumps. Pump elevation data will need to be added.

EPS and Change Data - This associated information is NOT imported in the DAT Import process. The .p2k file represents a baseline steady state model only.

System Data - Check the System Data before proceeding. Some of this is imported from the original DAT, but Darcy Weisbach equation selection and Specific Gravity are some examples of data that may not be set in the .p2k file.

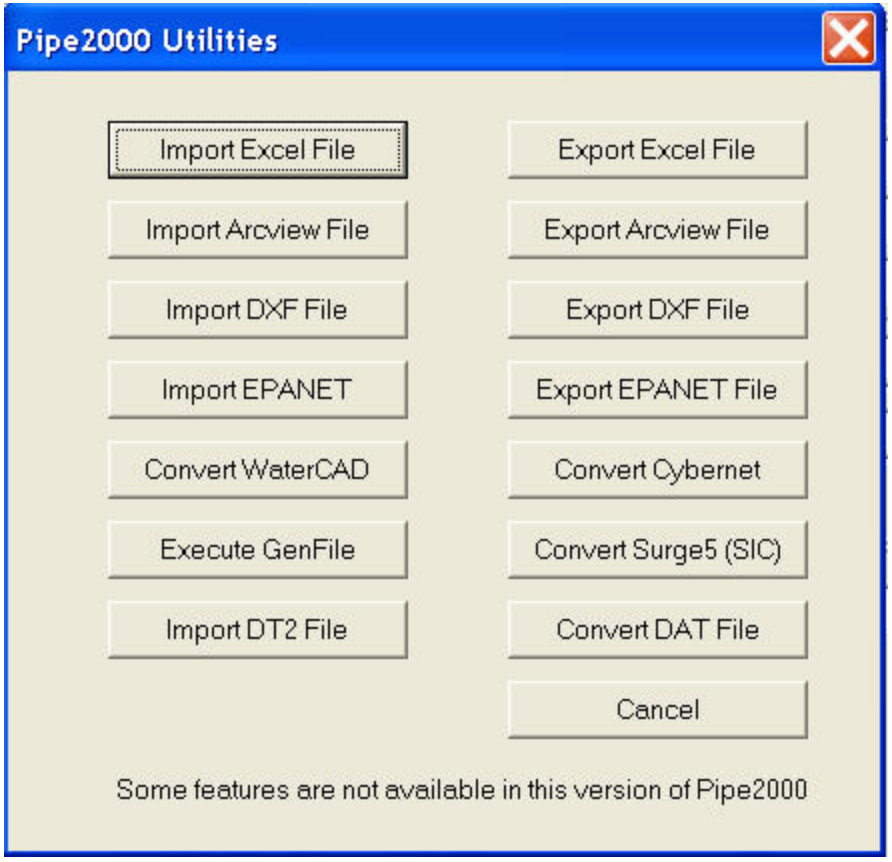
Regulators / Pressure Reducing Valves - these nodes are not converted.

## 1.3.9.8 Data Exchange

Pipe2000 Utilities / Data Exchange

---

See also Utility Programs. Click on the buttons for more information.




## 1.3.9.9 DEM Elevations Import Tool

DEM Elevations Import

---

See also Internet Maps and Elevations.

Located under Tools | Extract Elevation from DEM files (main menu).

This is a series of utilities for importing elevation information from a DEM file.

1 - How to Extract Elevations from SDTS/DEM file
2 - Download STDS File
3 - Convert SDTS to DEM
4 - Create XY File
5 - Convert DEM and XY files to XYZ file
6 - Import Elevations from XYZ file

When a model is created from an AutoCAD file, elevation data may not be imported. If the AutoCAD model uses state plane coordinates, elevation data may be imported from a DEM file.

Download SDTS File

This takes you to the GeoCommunity[tm] web site (http://data.geocomm.com/dem/demdownload.html). USGS DEM files may be downloaded for free. The download is a .tar.gz data set which is a packed (zipped) file.

Convert SDTS to DEM

This opens the **sdts2dem** conversion utility in a DOS window. The guidelines for using sdts2dem successfully on a Windows system are as follows:

- Do we need to copy sdts2dem into the folder where we upzipped????
- Each .tar.gz dataset downloaded collects several *.DDF files representing just one "SDTS transfer" -- a single elevation grid. Unpack each transfer in its own directory, for the file names in different transfers may not be unique.
- If you unpack using WinZip, be sure that the "TAR file smart CR/LF conversion" box is NOT checked on the Options/Configuration/Miscellaneous screen.
- To run sdts2dem by clicking its icon, put it in the same directory as the .DDF files.
- If sdts2dem exits unexpectedly, open a command window and run it there so that you can see any error messages. (Make the directory containing the DDF files your current directory.)
- The program can accept command-line options, but if you just run it with none it will print a usage message and prompt you for what it needs.

**Create XY File**

This takes the currently loaded Pipe2018 model file and creates an XY coordinate file.

Convert DEM and XY files to XYZ file

This takes the elevations (or Z coordinates) from the DEM file and adds them to the XY file.

Import Elevations from XYZ file

This brings the XYZ data back into the Pipe2018 model file.

## 1.3.9.10 EPANET Conversion

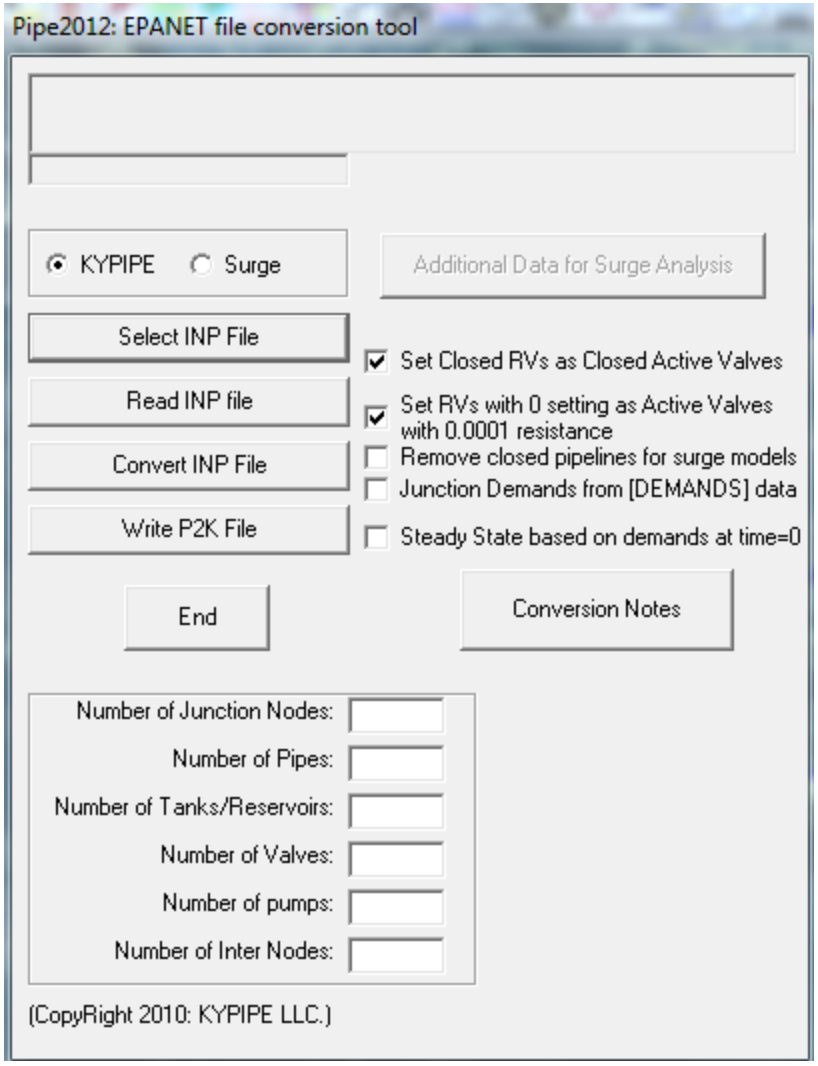
EPANET Conversion

Pipe2000 Utilities / Data Exchange

---

Pipe2000 Utilities / Data Exchange

The EPANET2 import provides the capability to utilize most of the other widely used pipe system software. WaterCad ® (version 5) and H2ONET can both export to an EPANET2 file. This provides a very convenient means of using Pipe2018 KYPipe or Surge features with these models.




1) select KYPipe or Surge as the target model type.

2) Click "Select INP File", then browse for the .inp

3) Click "Read INP File"

4) For Surge modes, click the "Additional Data for Surge Analysis" button and fill in desired additional data.

5) Click "Convert INP File"

6) Click "Write P2K File"

The new .p2k file will be named the same as the .inp with "(new)" added and located in the same file folder as the .inp file from step 2 above.

NOTE: the imported file is NOT automatically loaded into Pipe2018.

Other Settings:

Set Closed RVs as Closed Active Valves - In Pipe2018, regulating valves cannot be closed, so this is the default setting. Otherwise, RVs will be imported as opened even if they are closed in EPANET.

Set RVs with 0 setting as Active Valves with 0.0001 Resistance - regulators in Pipe2018 cannot have a setting = 0. An active valve with a resistance of 0.0001 has essentially no loss across it.

Remove closed pipelines for surge model - In Surge, pipelines must be closed using a closed valve rather than closing the pipeline directly.

Junction Demands from [DEMANDS] data - If demands for junction nodes are specified more than once under [DEMANDS] data, only the last occurrence of the demand specified at a junction node will be processed and all earlier occurrences ignored.

Steady State based on Demands at time=0

"Number of [elements]" - these are filled in after the conversion is complete to indicate the final number for each element in the imported model.

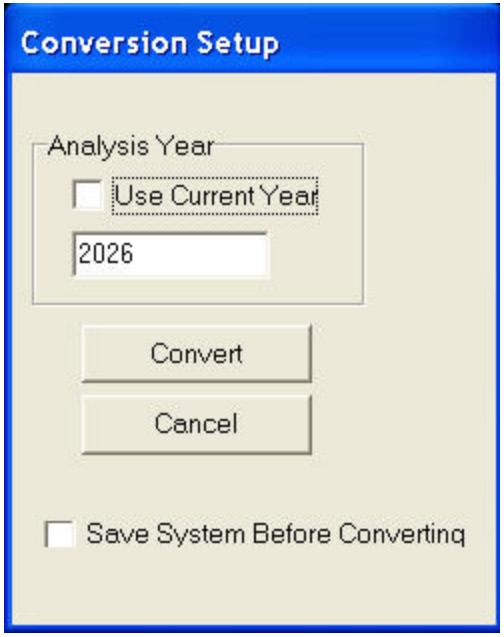
Demand Pattern, Change Pattern and Control Switch data may also need to be manually added to converted models (import and export) using the EPANET conversion tool.

EPANET General Purpose Valves (GPVs) are not supported in import.

## 1.3.9.11 EPANET Export

EPANET Export

Pipe2000 Utilities / Data Exchange




Export to EPANET without running a Pipe2018 water quality analysis.

## 1.3.9.12 Excel Import and Export

Excel Import

Pipe2000 Utilities / Data Exchange

Utility Programs

---

### Export

Select File | Import Export (or Pipe2000 Utilities in Classic) | Excel Export. This will create an xls file with the same name as the p2k file. For an explanation of data column headings, see below or see Data Tables.

### Import

The conversion process involves copying columns of information from a source spreadsheet to a destination spreadsheet. You can work within Pipe2018 or Excel to access the two spreadsheets. Columns of data may also be copied and pasted from Excel directly into the Pipe2018 Advanced Editor (in Data Tables, click "All" button to access editor)

1. The easiest way to start creating your destination spreadsheet is to enter Pipe2018 and create a New File or load an existing file (in which you will overwrite the data). If you want to work in Excel, then use the Export to Excel option under File/Data Exchange on the Pipe2018 main menu, then load the file you created in Excel. If you want to work within the Pipe program then choose the Edit/Data Tables option from the main menu. When the data table appears, click the All button to enter the advanced user spreadsheet editor.

You should now be in a spreadsheet editor with a Pipe2018 P2K file loaded and ready to edit. Note that you now have access to ALL of the information in the P2K file. You can edit the file manually or you can copy data into it from another spreadsheet.

2. If you are working in Excel, then load your source spreadsheet so that you have access to both the source and destination files. If you are working in Pipe2018, then start another instance of Pipe2018 and go to the advanced user spreadsheet as before. Now use the File/Read option from the main menu and load the Excel file (remember to set the File Type to Excel). Note that you have now loaded into the instance of Pipe2018 a file which is not in a meaningful P2K format. Note that when you quit the advanced user spreadsheet editor and go back to this instance of Pipe2018, it will contain meaningless data so you should quit this instance of Pipe2018 without saving any changes to your file.

3. In your destination spreadsheet editor, the first three sheets will be named REFERENCE, NODES, and PIPES. Select the NODES sheet. The column headings on this sheet should be labeled NAME, ~X, ~Y, ELV, ITEMS1, etc. Copy columns of node information from the source spreadsheet and place them in the appropriate column in the destination spreadsheet. The first five columns of the destination spreadsheet should now contain Node Name, X coordinate, Y coordinate, Elevation, and Demands.

If you have additional information pertaining to the nodes it can be placed in columns 26 or greater. This information will then be available in Pipe2018 as User Data which can be edited, or used for labeling maps or making contours, and can be exported to DXF (AutoCAD) files or ArcView files.

VERY IMPORTANT NOTE for Node Page: The 14th column of the destination spreadsheet (which should be labeled TYPE) should contain a 1 for every junction, 2 for pumps, 3 for reservoirs, or a 4 for tanks. If this column is empty Pipe2018 will delete the entire row of data. You may choose to enter a 1 for each node and then when you return to the Pipe2018 graphical editor change the nodes which are not junctions into the appropriate type of node.

4. Now select the PIPES sheet. The column headings on this sheet should be labeled NAME, NODE1, ~INDEX1, NODE2, ~INDEX2, etc. Copy columns of pipe information from the source spreadsheet and place them in the appropriate column in the destination spreadsheet. The first columns of the destination spreadsheet should now contain the Pipe Name, with Node 1 in the second column, Node 2 in the fourth column, and Length, Diameter, and Roughness in the sixth, seventh and eighth columns. Minor loss can be placed in the ninth column. Pipe material can be placed in the 15th column.

If you have additional information pertaining to the pipes, it can be placed in columns 26 or greater. This information will then be available in Pipe2018 as User Data which can be edited, or used for labeling maps, and can be exported to DXF (AutoCAD) files or ArcView files.

VERY IMPORTANT NOTE for Pipe Page: The 11th and 14th columns of the spreadsheet (which should be labeled LENGTH FIXED and ROUGHNESS FIXED) should contain a 1 for every row. This will tell Pipe2018 to use the pipe lengths and roughnesses form columns 6 and 8. If there is no value in column 11, then Pipe2018 will calculate pipe lengths based on node coordinates.

5. If you have any rows of node data or pipe data that are blank or contain other information then delete these rows now.

6. If you are working in Excel, save your work and quit now. Now start Pipe2018 and select File | Import Export (or Pipe2000 Utilities in Classic)| Excel Import to import the Excel file into Pipe2018.

If you are working in Pipe2018, do a File/Write command to save your work, using the Excel format. Now close the advanced spreadsheet editor. This should take you back to Pipe2018 in the data table mode. Click the Map button to go back to the Pipe2018 graphical editor environment.

7. Do a Zoom All by clicking the Z all button. If you don't see a map of your system with pipes and nodes visible, then return to step 6 and check your work.

8. If everything is okay, then add tanks, reservoirs, pumps and RVs or click on a junction and change it to these elements. Then edit the data for these new elements.

9. Add system data, change patterns, and demand patterns of desired. Save the new P2K file.

## 1.3.9.13 Execute GenFile

Execute GenFile

Pipe2000 Utilities / Data Exchange

---

This is an interface program used to import ASCII files in system generation for GoFlow. A file is saved as a GEN file. When executed, a system with pipes data will be created. This is a specialized application. Contact tech support to determine the ASCII, comma-delimited format.

### 1.3.9.14 Google Earth Import and Export

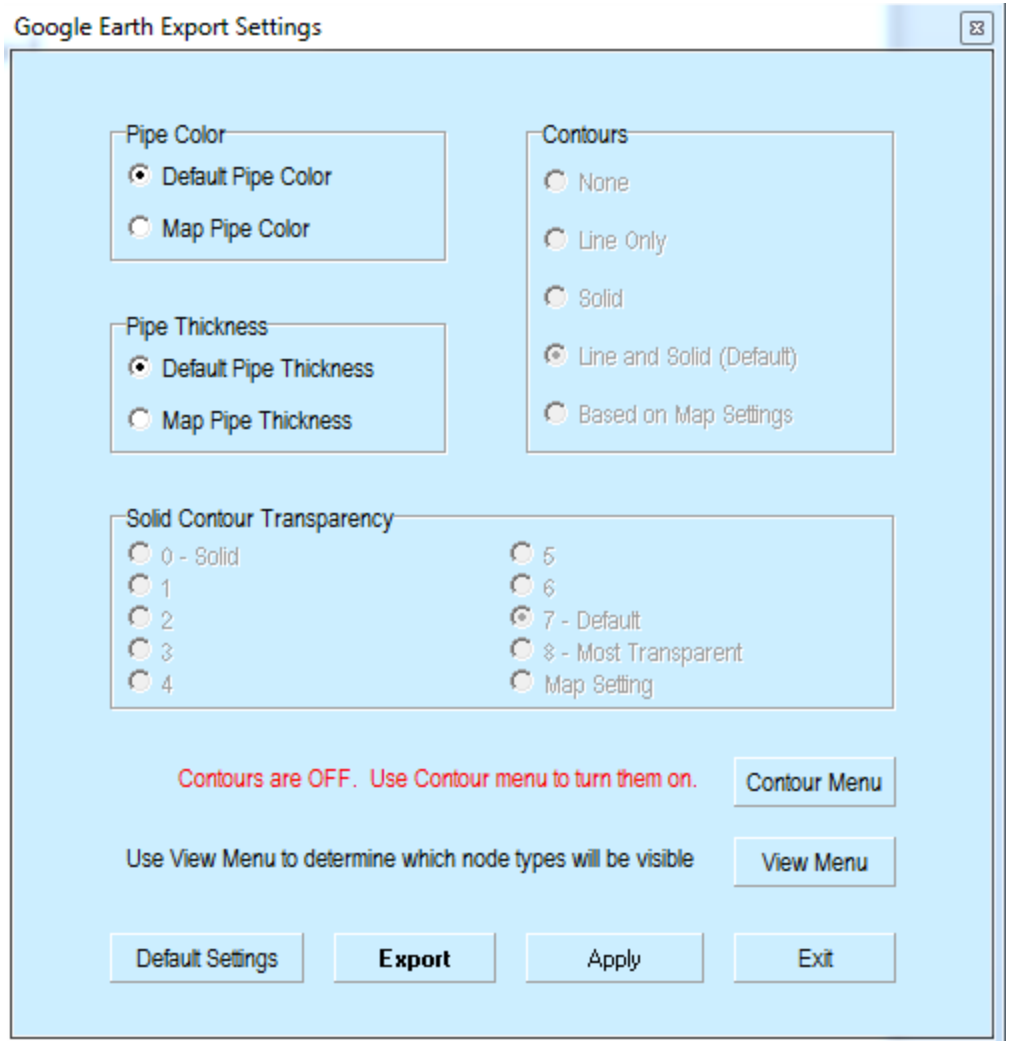
# Google Earth Import and Export

Available in both KYnetic (File – Import/Export) and Classic (File – Pipe2000 Utilties) interfaces.

This is a Professional package feature.

# Google Earth Export

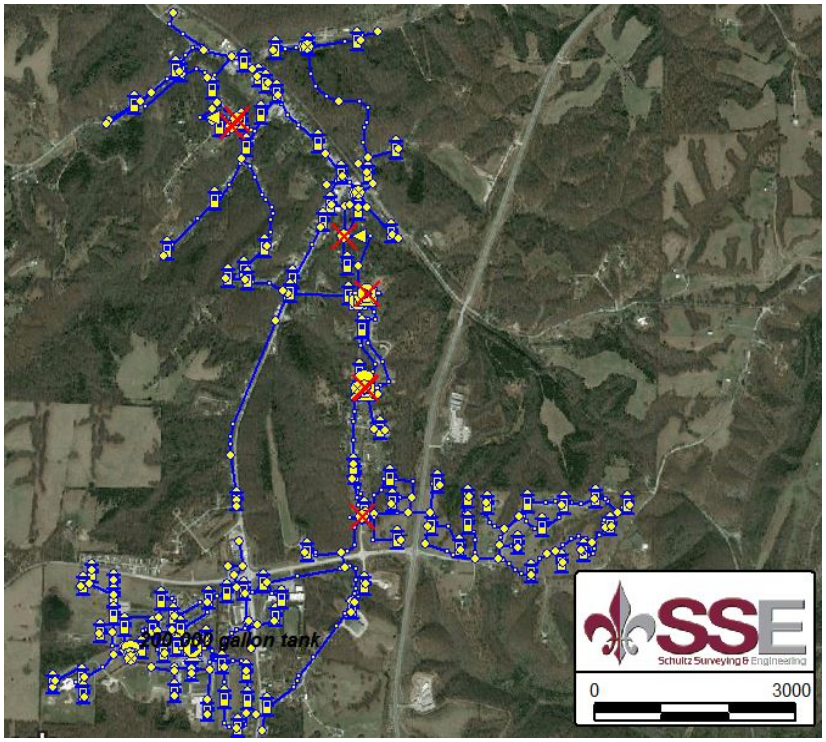
Must be used in conjunction with Internet Maps.



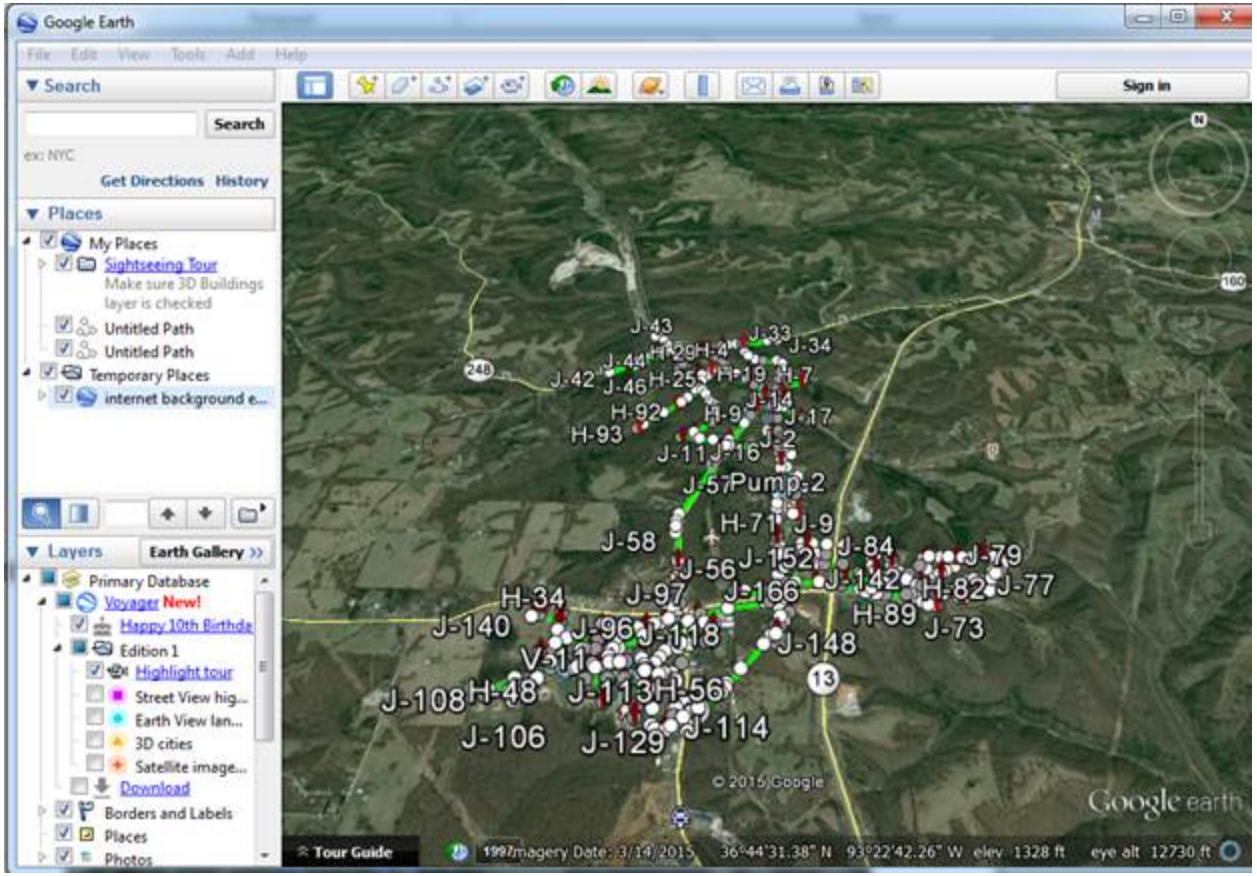

**Pipe Color and Thickness** – the options are to use the color which is the default for Google Earth, or to export colors which are currently being used in the Pipe program.

**Contours** –It is important to Zoom All and refresh the contours prior to Export. In addition to the settings in the Google Earth Export menu, the Contour and View menus may be launched from within the utility.

Original model




Export to Google Earth:




Detail of node symbols.

# Google Earth Import

KMZ File with embedded KML may be imported to build a P2K model file.

Attributes from the KMZ and KML files may be matched with data attributes in the model. For example, if pipe diameter is associated with the Google Earth file, it may be matched with Diameter data in the Pipe program.

Important Note: At this time, pipes must be defined as polylines.  Pipes drawn as polygons will not be imported!

## 1.3.9.15 Import DT2 File

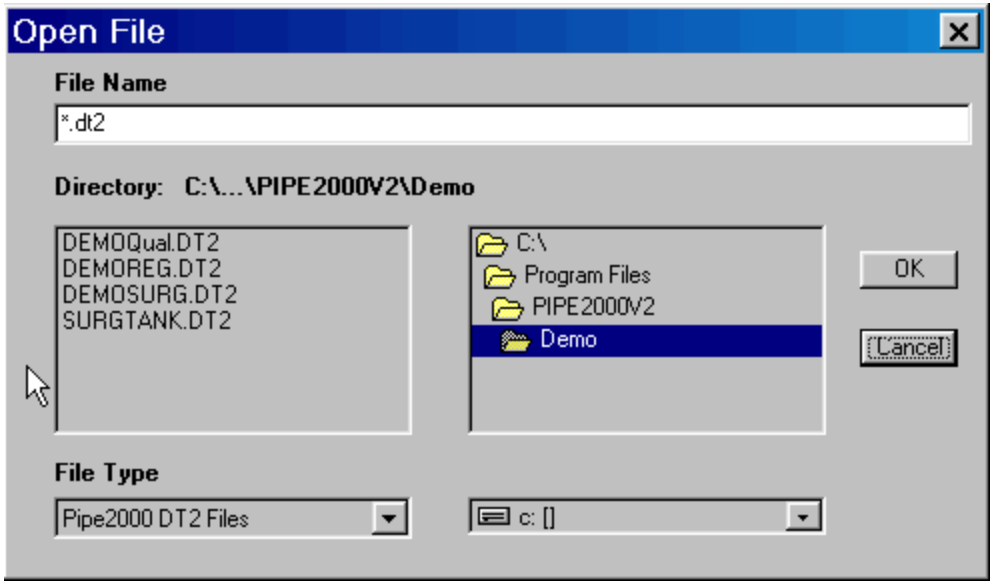
Import DT2 File

Pipe2000 Utilities / Data Exchange

---

If your P2k and backup files become lost, the Import DT2 Utility may be used to build a new P2k file if a DT2 file is available. Some data will not be recovered, such as intermediate nodes.

All of the following import utilities create a DT2 file: EPANET, Surge5 (Version 2), and WaterCAD (Version 2). The DT2 Import is then used to create a P2K file.




## 1.3.9.16 Import KY

Import KY

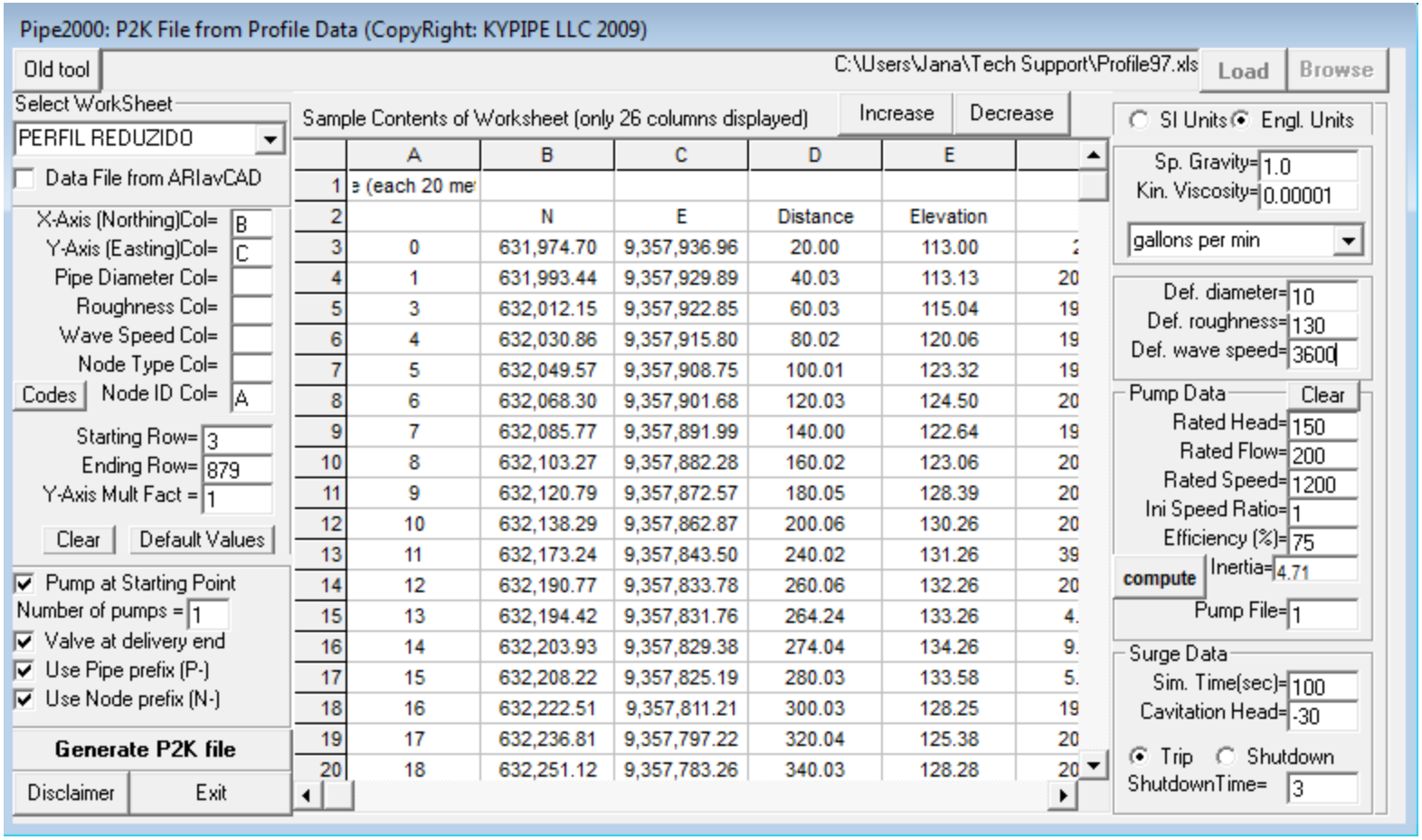
Utility Programs

---

To import a KY data file created in a previous version of KYPIPE simply click on File | Import KY, select the KY application, then browse for the file to be converted. A new p2k file is created and opened in Pipe2018. If a PCX background was included with the KY file, Pipe2018 will attempt to convert that file and place use it as a background as well.

For earlier versions of KYPIPE with DAT files, use the DAT Import to create a KY file and then import that file as described above. This is found under File | Pipe2000 Utilities.

## 1.3.9.17 Profile Data Import




Browse - To locate xls or csv file to import, click the Browse button.

Load - This button loads the file that appears in the pathname displayed to the left of the button.

Old Tool - This brings up the previous profile importer. The data read by this earlier tool is more limited.

Select Worksheet - If the profile data loaded (using the Load button) is located on a sheet other than first one, you can use this drop-down selector to choose the correct sheet

Data File from ARIavCAD - if the profile data was generated in ARIavCAD, check this box.

Increase/Decrease - Increases or decreases the width of the columns displayed

**Matching Data with spreadsheet column**

Enter the column letter next to each data itemFor any data that exists in the spreadsheet

As shown in the example above, the Northing values are in column B. so "B" is entered next to "X-Axis (Northing) Col =".

If nodes besided junctions existing within the pipeline, these may be specified with a Node Type Code. Click the **Codes** button to see the node type codes.

When column headings are present, specify the **Starting Row**. In the example above, the data begins in row 3. The **Ending Row** will be automatically entered, unless the user wishes to overwrite this value.

Pump At Starting Point - a single or multiple pumps may be placed at the beginning of the pipeline. The data describing the pump may be specified in the Pump Data area on the right side of the screen. For a pump trip, the pump must be described with a Suter file. In this case, in addition to the Rated Head and Rate Flow, the Rated Speed, Initial Speed Ratio (usually 1), and Efficiency must be entered. Click the **compute** button and the Intertia and Pump File number will be calculated

Valve at delivery end - when this is checked a valve will be added to the pipeline at the opposite end which is discharging to atmosphere.

Units - Select SI or English, specific gravity of fluid, kinetmatic viscosity of the fluid, flow units, default values for diameter, roughness and wave speed. See "Pump at Starting Point" for more infomation on Pump Data

Surge Data - To set up a Surge simulation, enter a total Sim(ulation) Time and specify the event type, either a Pump Trip or Pump Shutdown. For Shutdown, specify the Shutdown time. You may leave the default value for Caviation Head.

tips:

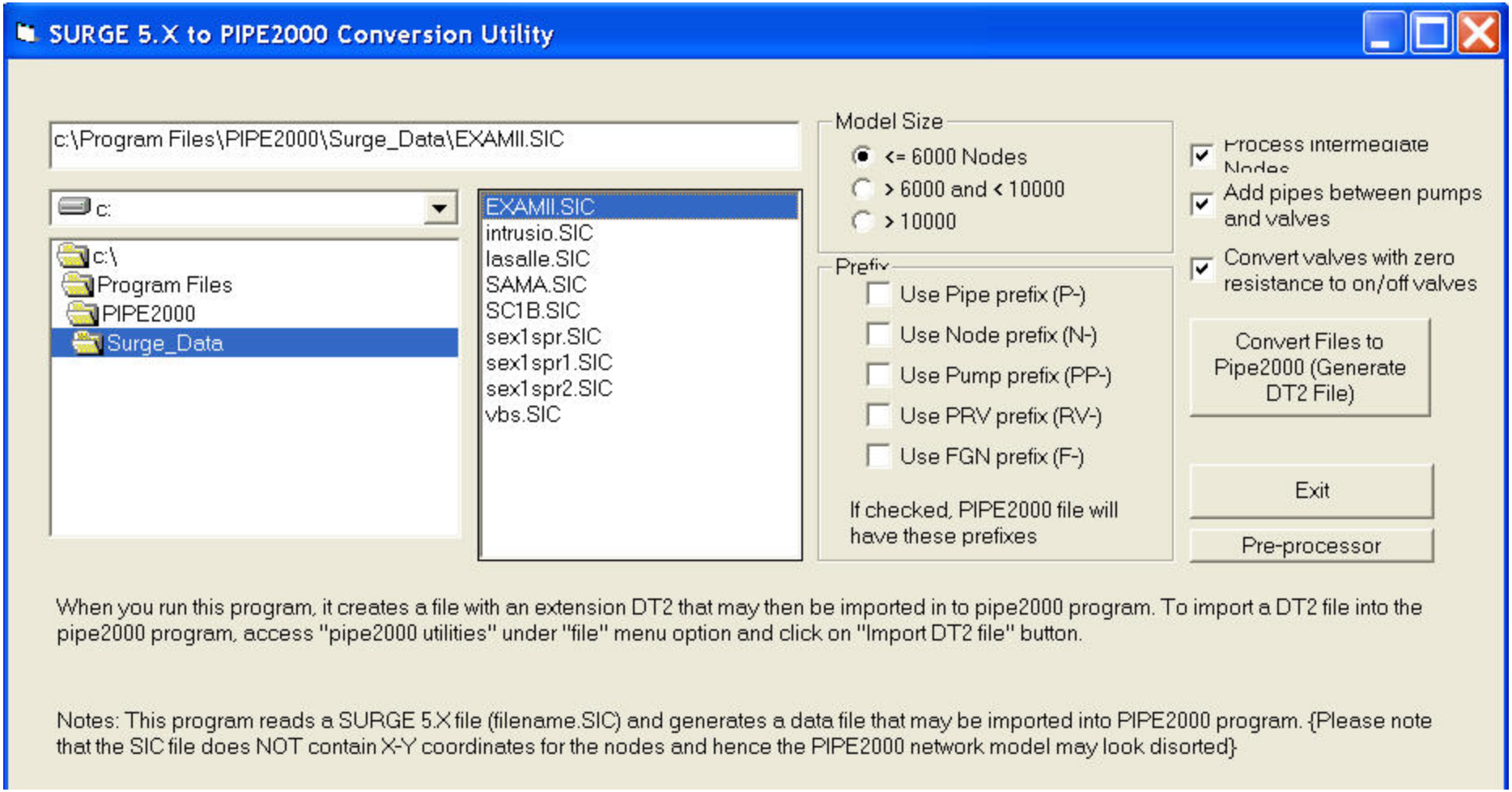
If you load the wrong profile data file or want to load a different one, completely exit the Profile Data Import Tool and open it again. Then browse for the correct file.

## 1.3.9.18 Surge5 Conversion

Surge5 Conversion

Pipe2000 Utilities / Data Exchange

Click File | Pipe2000 Utilities or Import Export (KYnetic) to conduct a Surge 5 conversion. Follow the instructions in the window.




## 1.3.9.19 To TIFF

To TIFF

Utility Programs

To TIFF is a Pipe2018 utility which converts .pcx files into .tif files. When a user converts a KYPIPE3 or KYTMP file using the KY Import function (File - Main Menu), this step is carried out automatically. A reference file will be created and the map will appear in Pipe2018 in the appropriate scale and location. However, if a user needs to add a PCX map to a Pipe2018 file, the To TIFF utility is used.

To use To TIFF, browse to the click on the Pipe2018 directory and click on the To TIFF application. You will be prompted for the file which you would like to convert. Upon selecting the file, the conversion is automatically carried out and the converted file may be found in the same directory as the original PCX file. You may now create a reference file for the new background in the Map Link utility, and proceed to add the map in Pipe2018, then Scale the Background Map.

## 1.3.9.20 Utility Programs

Pipe2000 Utilities / Data Exchange

---

There are several programs that are bundled with Pipe2018 that assist you in various tasks.

### ArcView Export Utility

Arcview data exchange utility.

### ArcView Import Utility

Arcview data exchange utility.

### AutoCad DXF Import and Export

AutoCad exchange utility.

### Customized Reporting

Allows the user to create a customized printout of analysis results.

### Cybernet Import

Convert Cybernet 2.x files to a format which may be imported by Pipe2018

### DAT Import

Convert pre-KYPipe2 Plus data files to a format which may be imported by Pipe2018.

### DT2 Import

Import DT2 files. The main vehicle for importing EPANET, WaterCAD, Surge5 and other data files.

EPANET import and export
Enter data for water quality simulation using EPANET.

Excel Import and Export
Import Excel-format data into Pipe2018. For merging two Pipe2018 files see Copy and Paste.

Execute GenFile
For GoFlow applications

International Decimal Settings
Easy way to check decimal settings and change to compatible format. Also use to change decimal back to comma when Pipe program is not in use

KY Import
Import KY data file from a previous version of KYPIPE into Pipe2018.

MapLink
Create a reference file for a background map.

Pipe2018 Help
Complete online help for Pipe2018

RuralVB
The Peak Demand Allocation utility for Rural Water calculations.

To Tiff
Convert a .PCX background file to a TIFF format readable by Pipe2018.

## 1.3.9.21 WaterCad Conversion

WaterCad Conversion

Pipe2000 Utilities / Data Exchange

---

WaterCAD, WaterGEMS, Hammer and other modeling software that uses EPANET for hydraulic calculations may be imported to Pipe2018 via the EPANET file format with file extension ".inp".

The conversion pathway may be easily followed a different ways. Not all model create the .inp file directly. In Bentley programs, most have an Export to EPANET option. When this is used, the option to use "ID Numbers" should be selected. This generates an .inp file. To import the .inp, go to File | Pipe2000 Utilities and select EPANET import.

Alternatively, if EPANET .net and .map files are available, these may be saved as .inp format within the EPANET program. EPANET 2.0 may be downloaded for free. Upon loading the .net file into the EPANET program, go to File | Export and select Network. This generates the .inp which may then be imported to Pipe2018 by going to File | Pipe2000 Utilities and selecting EPANET import.

There is a separate utility for converting Cybernet 2.0 file.

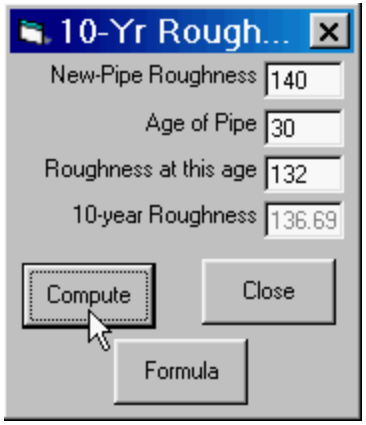
See also EPANET import.

# 1.3.10 Tools - Calculators

## 1.3.10.1 10 Year HW Coefficient

**10 Year HW Coefficient**

This tool calculates the value for the 10-year Hazen Williams coefficient required to utilize the Pipe2018 aging capability. The New Pipe roughness and a value for the roughness at a known age are required.




See also Pipe Type and Age-Based Roughness.

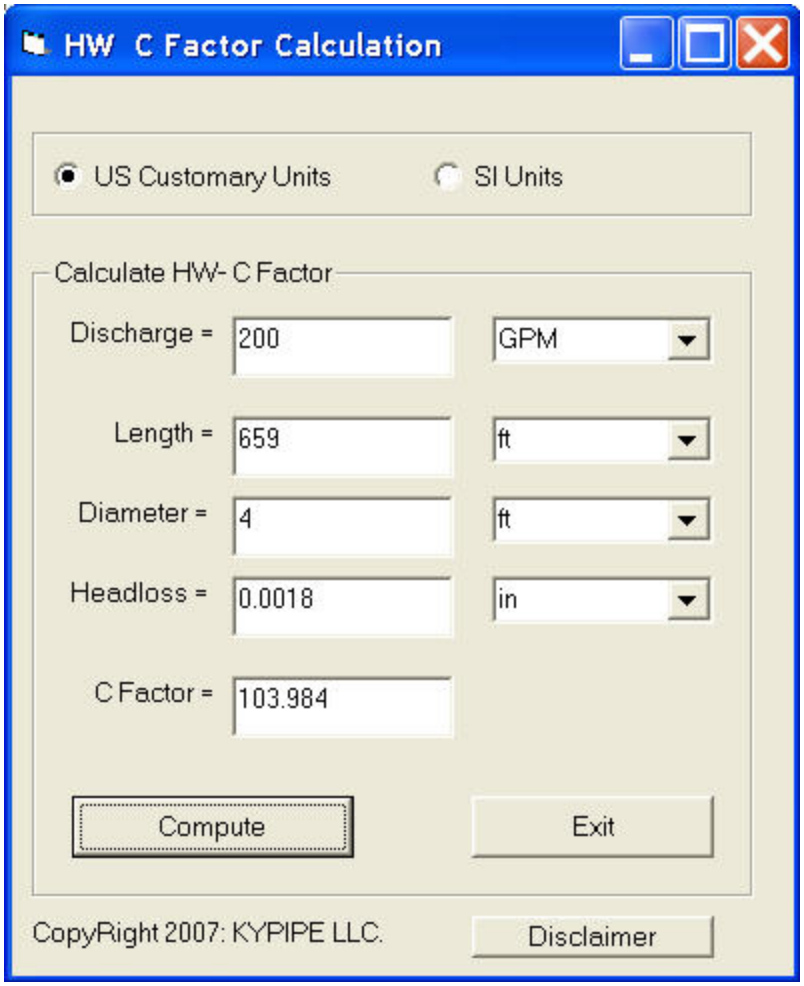
## 1.3.10.2 Calculator Tool

**Calculator**

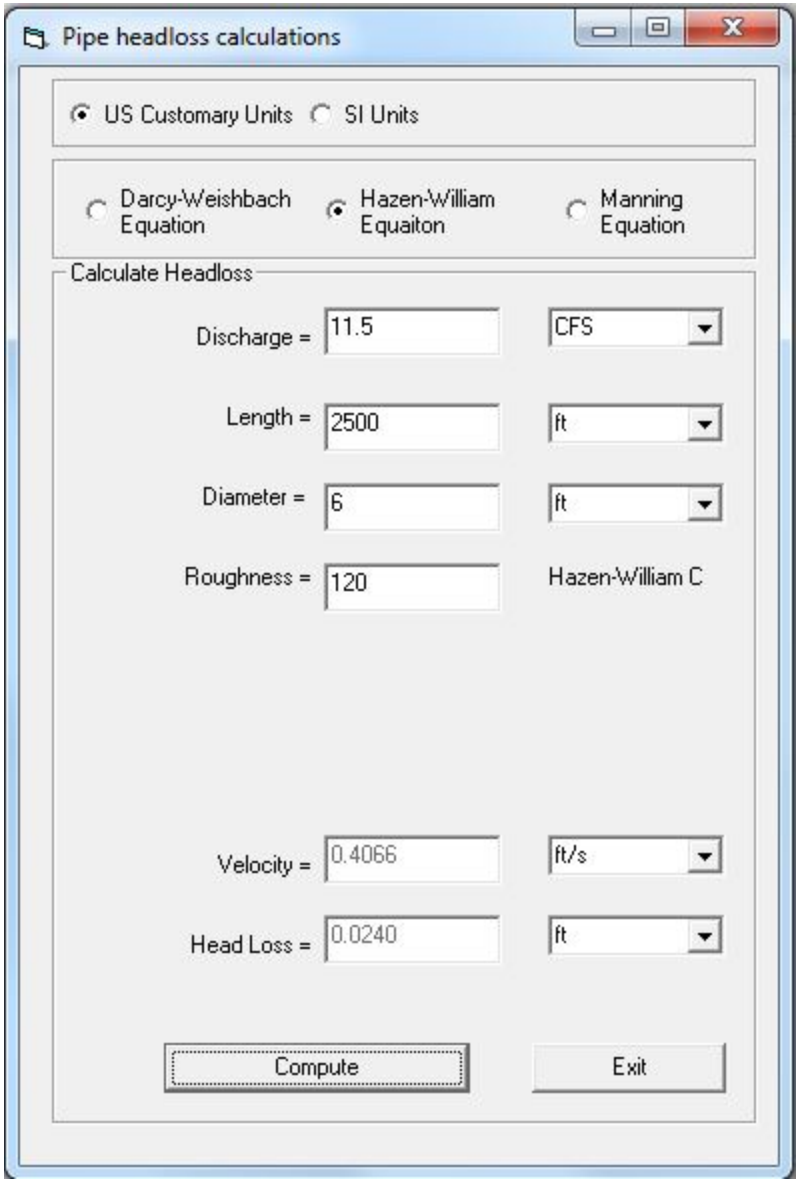
This tool is for simple mathematical operations.

## 1.3.10.3 C Factor Calculator Tool

**C Factor Calculator**



#### 1.3.10.4 Headloss Calculator Tool




Select Units, Equation and enter the flow and pipe characteristics to calculate the head loss and velocity in a particular section of pipe.

## 1.3.10.5 Hose and Nozzle Constants Tool

### Hose and Nozzle Constants

Combining Hydrant and Hose Constants. This is done automatically when the Fireflow and Hydrant Analysis option for Hydrant Flow - Hose Attached is selected. But the following formula shows how this calculation is done:

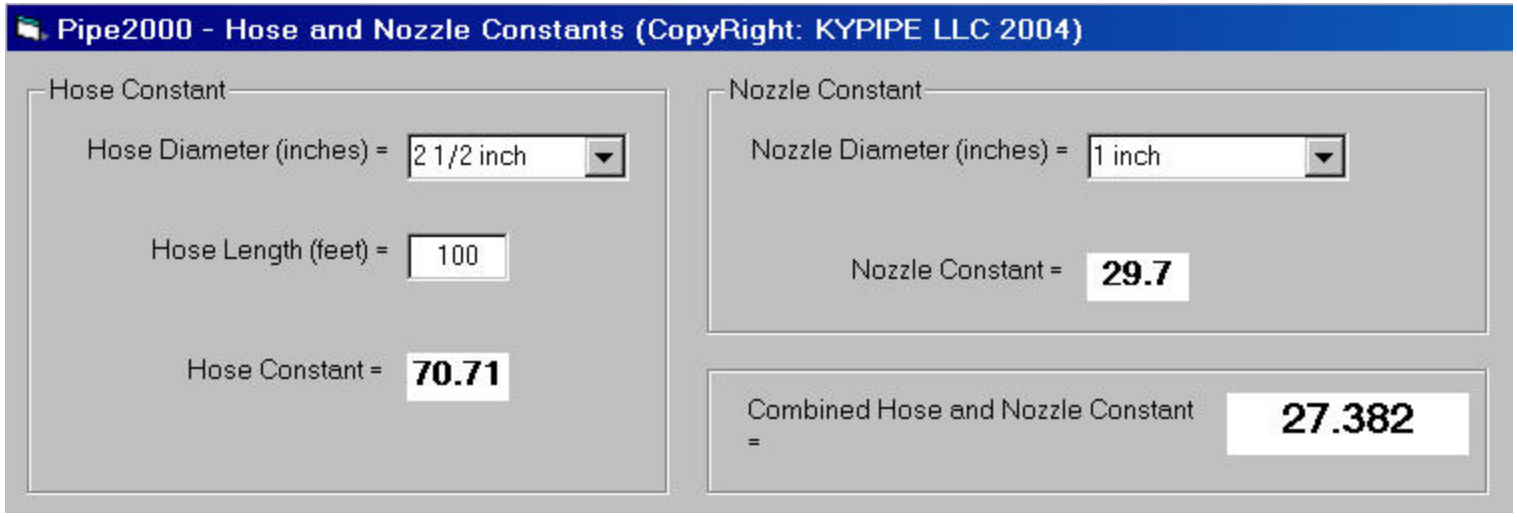
C1 = constant 1 (hydrant nozzle)

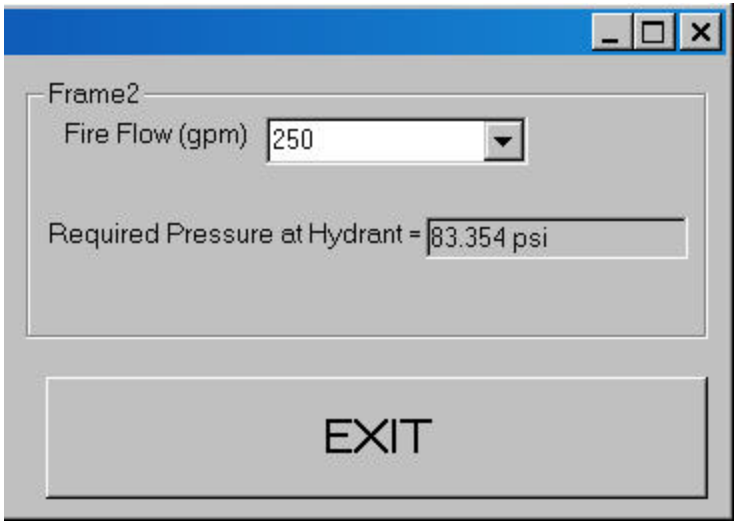
C2 = constant 2 (hose constant)

Cc = combined constant

Cc = [(C1*C2)^2/ (C1^2 + C2^2)] ^0.5

Example C1=100, C2 = 200, Q = 500 gpm, Cc = 89.44

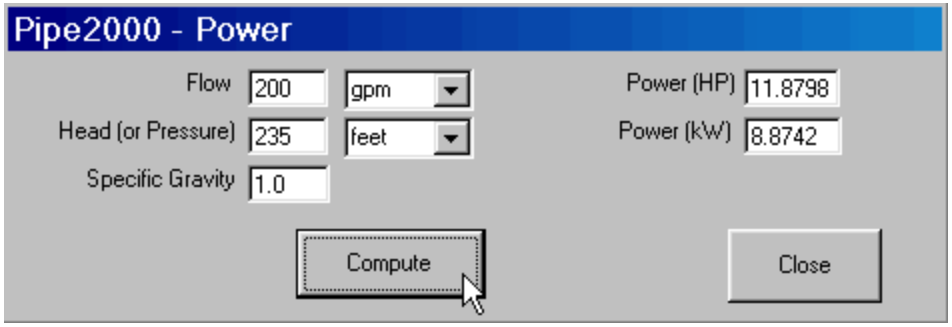





## 1.3.10.6 Power (HP or KW) Calculations Tool

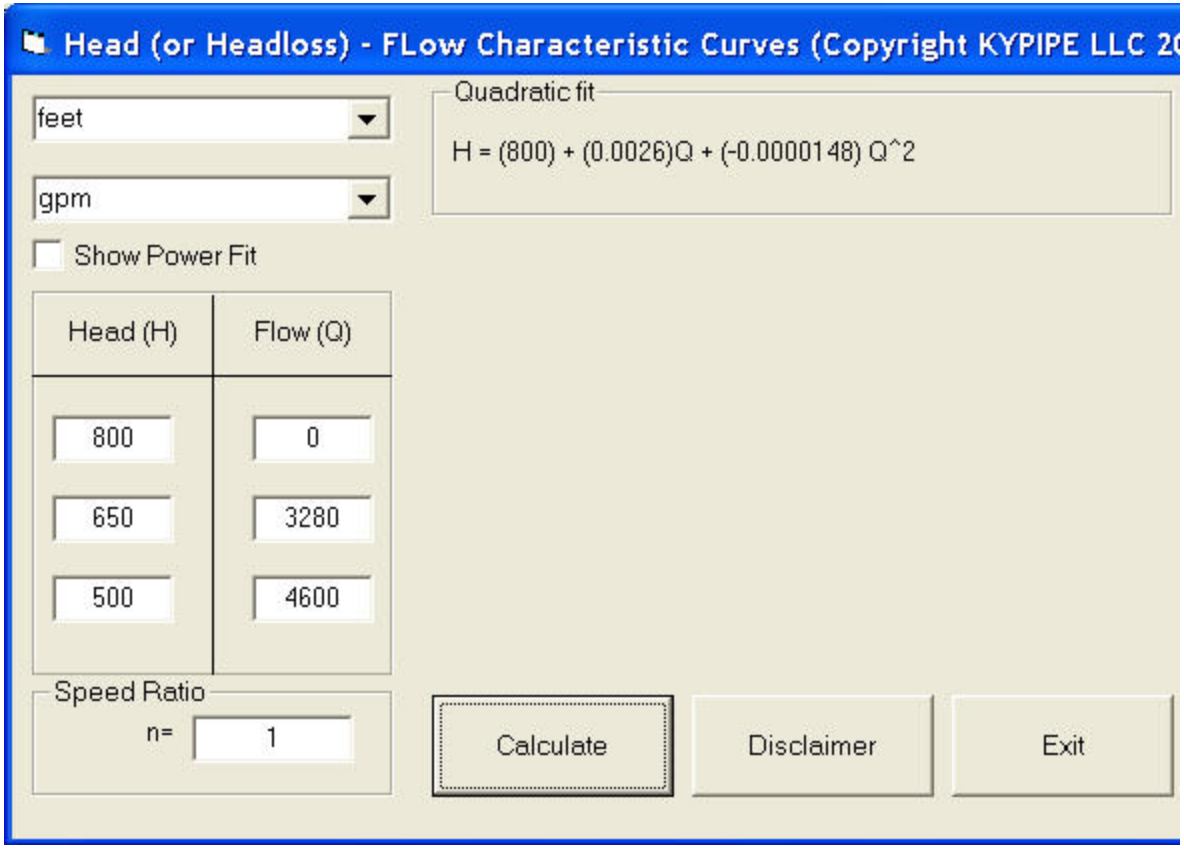
**Power (HP or KW) Calculations**

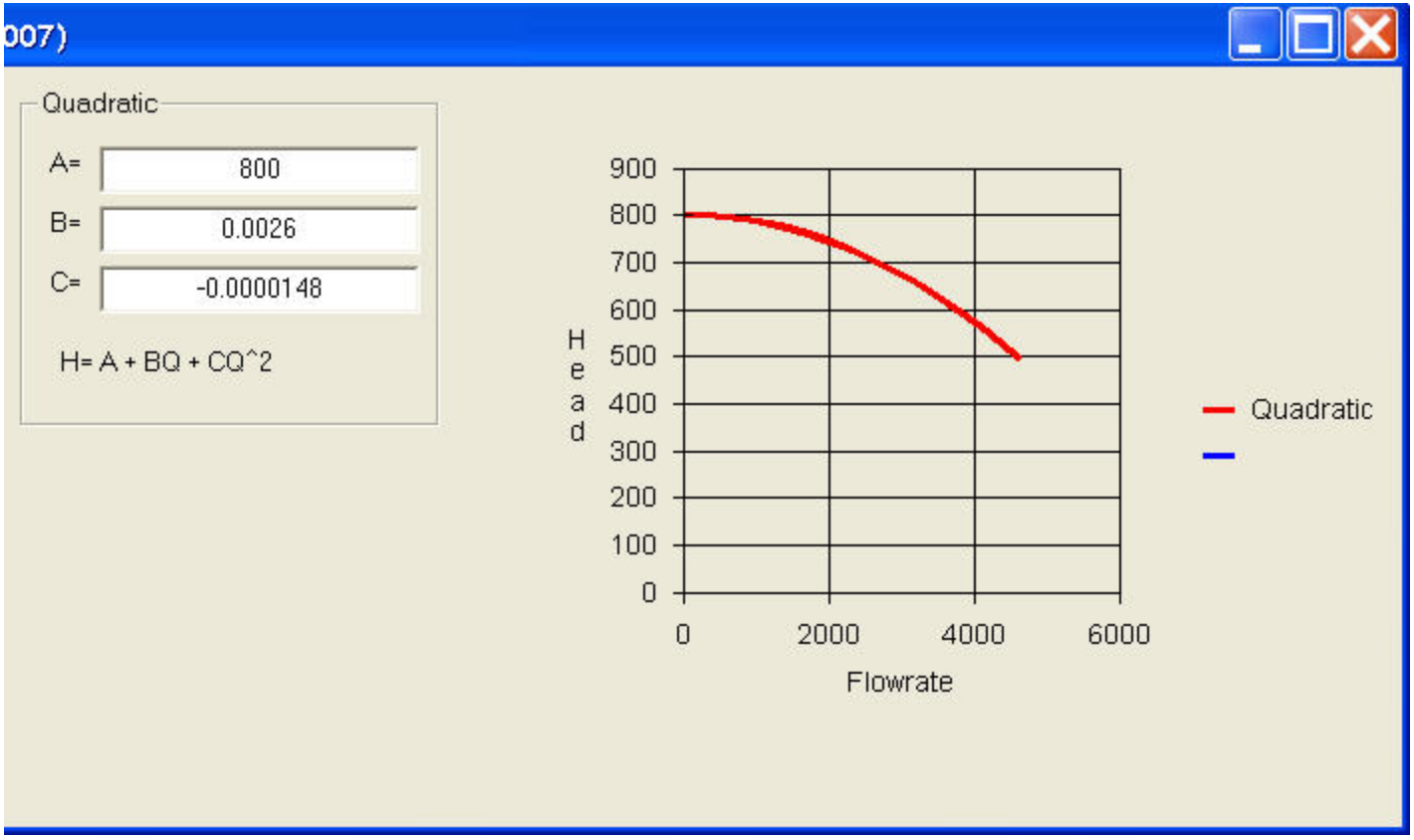
This tool calculates the useful power used by a pump for a specified operating condition. See Constant Power Pump.




## 1.3.10.7 Pump Characteristic Curves Tool

**Pump Characteristics Curve**


Head (or Headloss) - FLow Characteristic Curves (Copyright KYPIPE LLC 2
feet
gpm
Show Power Fit
Head (H)
Flow (Q)
800
0
650
3280
500
4600
Speed Ratio
n=
1
Quadratic fit
H = (800) + (0.0026)Q + (-0.0000148) Q^2
Calculate
Disclaimer
Exit




## 1.3.10.8 Pump Selection Tool

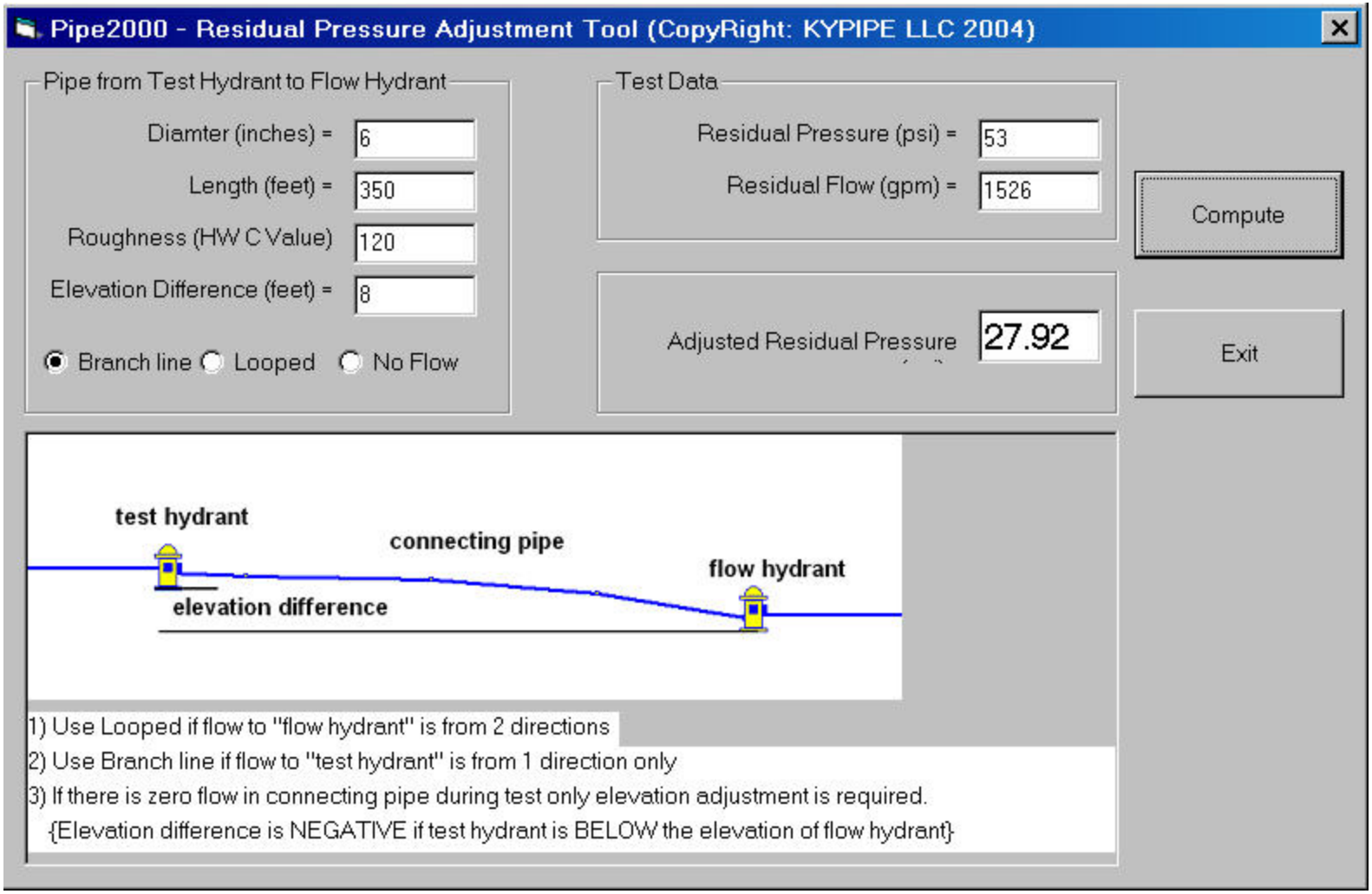
**Pump Selection**

User defines a pump head and flow. The tool then searches all existing pump curves entered by the user or in default files and finds the closest match. The curve is then displayed on the pump curve graph.

## 1.3.10.9 Residual Pressure Adjustment Tool

**Residual Pressure Adjustment**

May be used when a fire flow test point is located apart from the location where a residual pressure value is desired.




## 1.3.10.10 Resistance Calculations Tool

**Resistance Calculations**

The resistance, R, is the head drop (in ft. or m) over the flow squared (in cfs or cms).

This tool does some very useful resistance calculations based on a variety of information. The calculations include the resistance based on

a) minor loss coefficient

b) valve flow coefficient

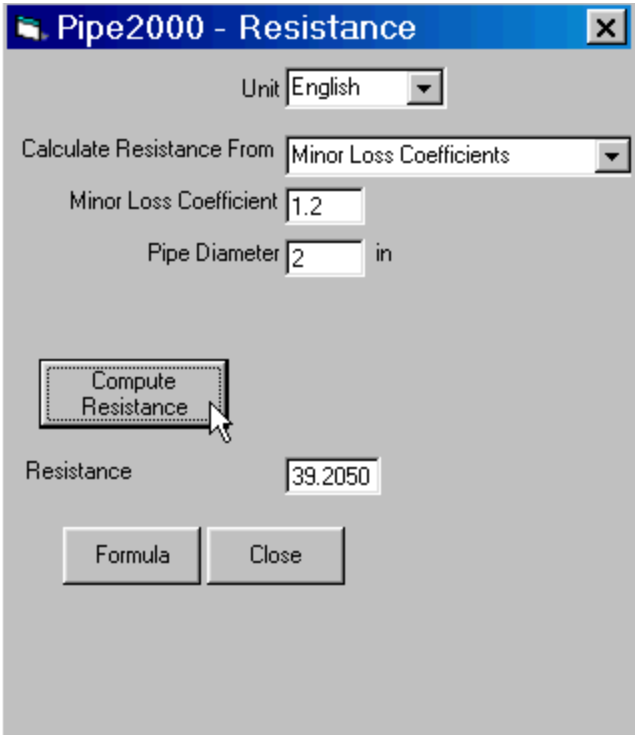
c) orifice data

d) connection to a tank

e) head-flow data

f) piping section (parallel or series pipes)

May be used with active valves, surge tanks, SDOs, pressure relief valves, parallel or series pumps.




For Active Valves: R 100% (also Cv 100%) - This is the resistance of the valve while it is 100% open. For example, a wide open valve which has a head drop of

1.2 ft at 500 gpm (1.114 cfs) has a wide open resistance of (1.2 / (1.114^2) ) = 0.97.

Under System Data/Preferences, a check box is provided to allow the use of a flow coefficient, Cv, instead of R. The Cv is usually provided by the manufacturer, in lieu of the calculated resistance. Cv 100% must be a non-zero number. Cv is the flowrate in gpm required to cause a 1 psi drop in pressure (units gpm/psi^0.5). In SI units, Kv is used (m3/h at sqr1bar). The conversion is made internally (Cv= Kv*0.856).

Combining resistances for pipe sections. To obtain a combined R for two parallel pipe sections, for example, two 30" sections use this relationship:

1/square root (total Resistance for 30" pipes)= 1 /square root (R 30") + 1/square root (R 30")

$R_t$ = total resistance combined

$$\frac{1}{\sqrt{R_t}} = \frac{1}{\sqrt{R_1}} + \frac{1}{\sqrt{R_2}}.$$

## 1.3.10.11 Sprinkler or Blowoff Constant Tool

### Sprinkler/Blowoff Constant

This tool provides several options for calculating the constant to be used for a sprinkler/blowoff element. The sprinkler/blowoff constant is defined as the flow (in gpm or l/s) discharged through the device at a pressure drop of 1 psi or 1 meter. See also blowoff/hydrant.

Combining Hydrant and Hose Constants. This is done automatically when the Fireflow and Hydrant Analysis option for Hydrant Flow - Hose Attached is selected. But the following formula shows how this calculation is done:

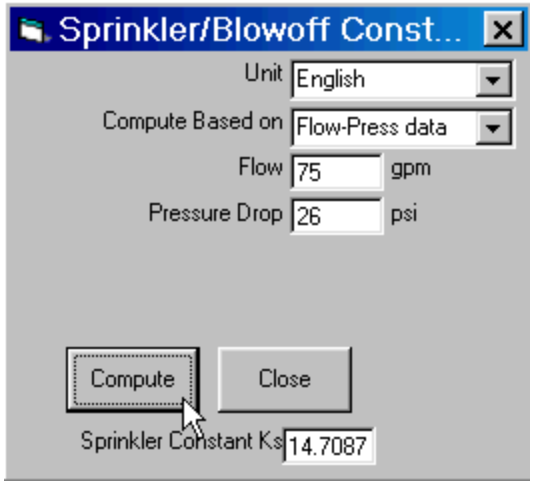
C1 = constant 1 (hydrant nozzle)

C2 = constant 2 (hose constant)

Cc = combined constant

Cc = [(C1*C2)^2/ (C1^2 + C2^2)] ^0.5

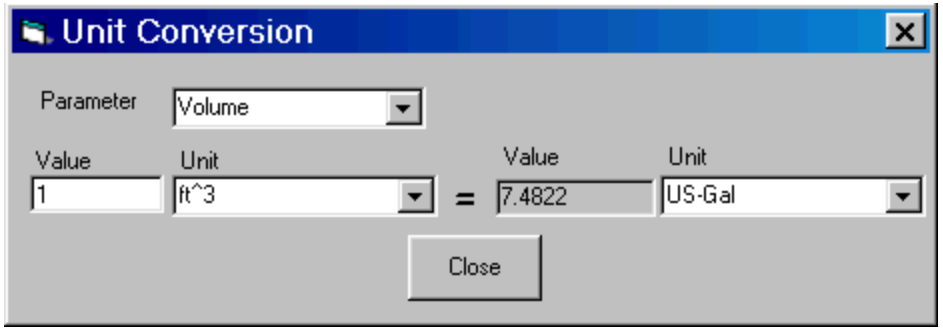
Example C1=100, C2 = 200, Q = 500 gpm, Cc = 89.44




## 1.3.10.12 Units Converter Tool

**Units Convertor**

This tool provides a conversion factor for a variety of parameters and units.




## 1.5 KYPipe - Water

## 1.4.1 Water Quality/EPANET

### 1.4.1.1 EPA Stage 2 DBP Rule

EPA Stage 2 DBP Rule

---

## Summary of Stage 2 Disinfectants and Disinfection Byproducts Rule:

When a water distribution system uses a form of disinfection other than UV light , principally in the form of a chemical additive, there are disinfectant byproducts (DBP's) created. Levels of the DBPs will vary throughout the distribution system but they can and do reach consumers. Studies have linked DBP exposure to a number of health problems including increased risk of certain cancers (bladder, colon, and rectal). Additional studies suggest that exposure to DBPs is a potential reproductive and developmental health hazard. Understanding that DBP's are found in nearly all distribution systems and the risks of exposure to high levels, the EPA proposal is to require more stringent monitoring standards to ensure the safety of the public. This final rule is effective on March 6, 2006.

### What are the requirements of the final rule?

**• Conduct Initial Distribution System Evaluations (IDSEs) on a required schedule. Systems may comply by using any of four approaches for which they qualify (standard monitoring, system specific study, 40/ 30 certification, or very small system waiver). IDSE's are due between October 2006 and April 2008 depending on the size of the system.**

**• Determine Stage 2 monitoring locations based on the IDSE.**

**• Comply with Stage 2 MCLs on a required schedule.**

## Initial Distribution System Evaluation.

Under the Stage 2 DBP rule, systems will conduct an evaluation of their distribution systems, known as an **Initial Distribution System Evaluation (IDSE)**, to identify the locations with high disinfection byproduct concentrations. These locations will then be used by the systems as the sampling sites for Stage 2 DBP rule compliance monitoring.

**What is an IDSE?** The first provision of the Stage 2 DBP Rule, designed to identify higher risk systems, is the Initial Distribution System Evaluation (IDSE). The purpose of the IDSE is to identify Stage 2 DBPR compliance monitoring sites that represent each system's highest levels of DBPs. Because Stage 2 DBPR compliance will be determined at these new monitoring sites, only those systems that identify elevated concentrations of TTHM and HAA5 will need to make treatment or process changes to bring the system into compliance with the Stage 2 DBPR. By identifying compliance monitoring sites with the highest concentrations of TTHM and HAA5 in each system's distribution system, the IDSE will offer increased assurance that MCLs are being met across the distribution system and that customers are receiving more equitable public health protection. Both treatment changes and awareness of TTHM and HAA5 levels resulting from the IDSE will allow systems to better control for distribution system peaks.

**Who needs an IDSE?** All community water systems (CWSs) and nontransient noncommunity water systems (NTNCWSs) that add a primary or residual

disinfectant or deliver water that has been disinfected by a primary or residual disinfectant other than UV light. PWSs with fewer than 10,000 customers are exempt.

**IDSE Options**

The IDSE is designed to offer flexibility to public water systems. The IDSE requires TTHM and HAA5 monitoring for one year on a regular schedule that is determined by source water type and system size. Alternatively, systems have the option of performing a site-specific study based on historical data, water distribution system models, or other data; and waivers are available under certain circumstances.

For those systems not receiving a very small system waiver, there are three possible approaches by which a system can meet the IDSE requirement. They are:

1. **Standard monitoring**. Standard monitoring requires one year of DBP monitoring throughout the distribution system on a specified schedule. Prior to commencing standard monitoring, systems must prepare a monitoring plan and submit it to the primacy agency for review. The frequency and number of samples required under standard monitoring is determined by source water type and system size.

2. **System specific study**. Under this approach, systems may choose to perform a system specific study based on earlier monitoring studies or distribution system hydraulic models in lieu of standard monitoring. Prior to commencing a system specific study, systems must prepare a study plan and submit it to the primacy agency for approval. The two options for system specific studies are: (1) TTHM and HAA5 monitoring data that encompass a wide range of sample sites representative of the entire distribution system, including those judged to represent high TTHM and HAA5 concentrations, and (2) **Extended Period Simulation Hydraulic Models** that simulate water age in the distribution system, in conjunction with one round of TTHM and HAA5 sampling. This option may be accomplished with **Pipe2018 : KYpipe software**. Under our Utility Partner Plan, there is even an option for KYPipe to do this modeling for you. This includes maintaining the model after completion.

3. **40/30 certification**. Under this approach, systems must certify to their State or primacy agency that every individual compliance sample taken under subpart L during the period specified in Table IV.F- 2 were less than or equal to 0.040 mg/L for TTHM and less than or equal to 0.030 mg/L for HAA5, and that there were no TTHM or HAA5 monitoring violations during the same period. The State or primacy agency may require systems to submit compliance monitoring results, distribution system schematics, or recommend subpart V compliance monitoring locations as part of the certification.

**What are the requirements if I choose System Specific Study - Hydraulic Model Option?**

**System Specific Study - Hydraulic Model**

Hydraulic models must meet the following criteria:

- Extended period simulation hydraulic model.
- Simulate 24 hour variation in demand and show a consistently repeating 24 hour pattern of residence time.
- Represent 75% of pipe volume; 50% of pipe length; all pressure zones; all 12-inch diameter and larger pipes; all 8- inch and larger pipes that connect pressure zones, influence zones from different sources, storage facilities, major demand areas, pumps, and control valves, or are known or expected to be significant conveyors of water; all pipes 6 inches and larger that connect remote areas of a distribution system to the main portion of the system; all storage facilities with standard operations represented in the model; all active pump stations with controls represented in the model; and all active control valves. [[Page 421&cd;
- The model must be calibrated, or have calibration plans, for the current configuration of the distribution system during the period of high TTHM formation potential. All storage facilities must be evaluated as part of the calibration process.
- All required calibration must be completed no later than 12 months after plan submission. Submission must include:
- Tabular or spreadsheet data demonstrating percent of total pipe volume and pipe length represented in the model, broken out by pipe diameter, and all

required model elements.
• A description of all calibration activities undertaken, and if calibration is complete, a graph of predicted tank levels versus measured tank levels for the storage facility with the highest residence time in each pressure zone, and a time series graph of the residence time at the longest residence time storage facility in the distribution system showing the predictions for the entire simulation period (i.e., from time zero until the time it takes for the model to reach a consistently repeating pattern of residence time).
• Model output showing preliminary 24 hour average residence time predictions throughout the distribution system.
• Timing and number of samples planned for at least one round of TTHM and HAA5 monitoring at a number of locations no less than would be required for the system under standard monitoring in Sec. 141.601 during the historical month of high TTHM. These samples must be taken at locations other than existing subpart L compliance monitoring locations.
• Description of how all requirements will be completed no later than 12 months after submission of the system specific study plan.
• Schematic of the distribution system (including distribution system entry points and their sources, and storage facilities), with notes indicating the locations and dates of all completed system specific study monitoring (if calibration is complete) and all subpart L compliance monitoring.
• Population served and system type (subpart H or ground water).
• If the model submitted does not fully meet the requirements, the system must correct the deficiencies and respond to State inquiries on a schedule the State approves, or conduct standard monitoring.

## 1.4.1.2 EPANET

EPANET

Trace Analysis

Water Age Analysis

Example 2

See also KYPipe - Water Quality Analysis Demo

Utility Programs

---

### Water Quality Modeling

Water quality modeling in a distribution system is becoming increasingly important after the reauthorization of the Safe Drinking Water Act. Pipe2018 provides a powerful interface to the EPANET program to perform water quality simulations on an already-created hydraulic network model. The following example illustrates the necessary steps in developing a water quality model. It should be noted that EPANET does not allow for steady state water quality simulations and hence the example deals with only extended period water quality simulation.

### Example

EXQUAL, an example 24-hour extended period simulation hydraulic network model, will be used to illustrate water quality modeling using the Pipe2018-EPANET interface. Different features of the Pipe2018-EPANET interface are demonstrated with EXQUAL in two parts.

Part 1: Provide the following water quality data to EXQUAL in Pipe2018 environment and run an EPANET water quality analysis in the EPANET environment.

* Constituent to be modeled is Chlorine

* Bulk decay rate for all pipes in the distribution system is -0.15/day

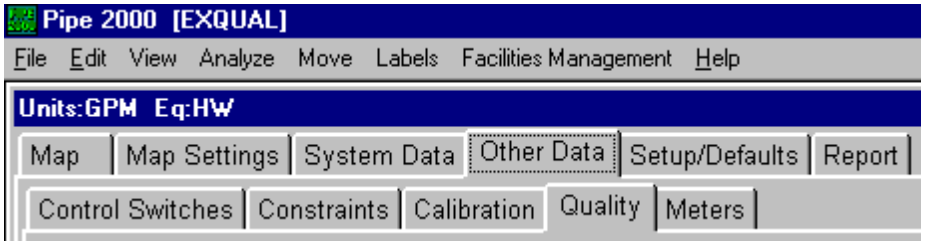
* Wall decay rate for all pipes in the distribution system is -0.5 ft/day

* Chlorine concentration in the supply reservoir is 3 ppm (or mg/l) and in the tank is 1.5 ppm
(if the bulk/wall decay rates are zero the program assigns a default value)

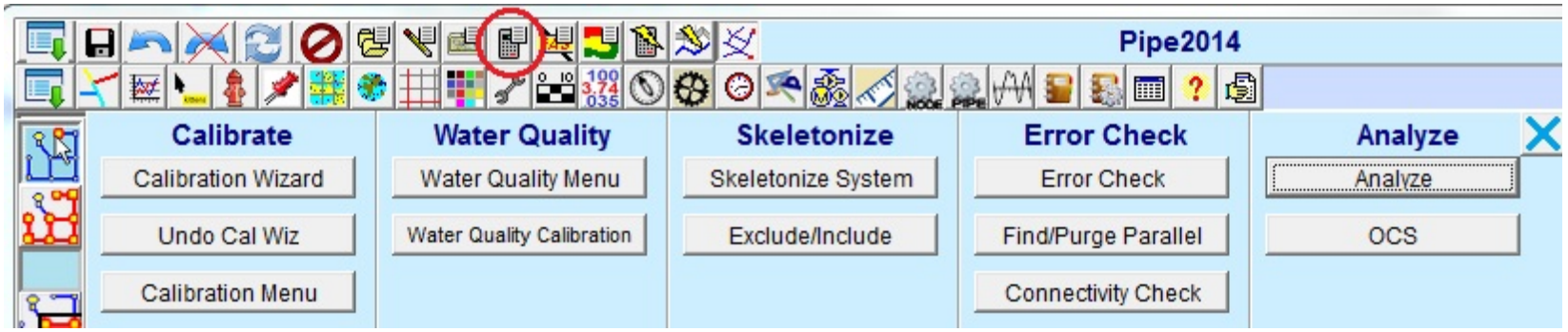
Solution to Part 1:

* Start Pipe2018 and open the data file called EXQUAL in the DataFiles folder

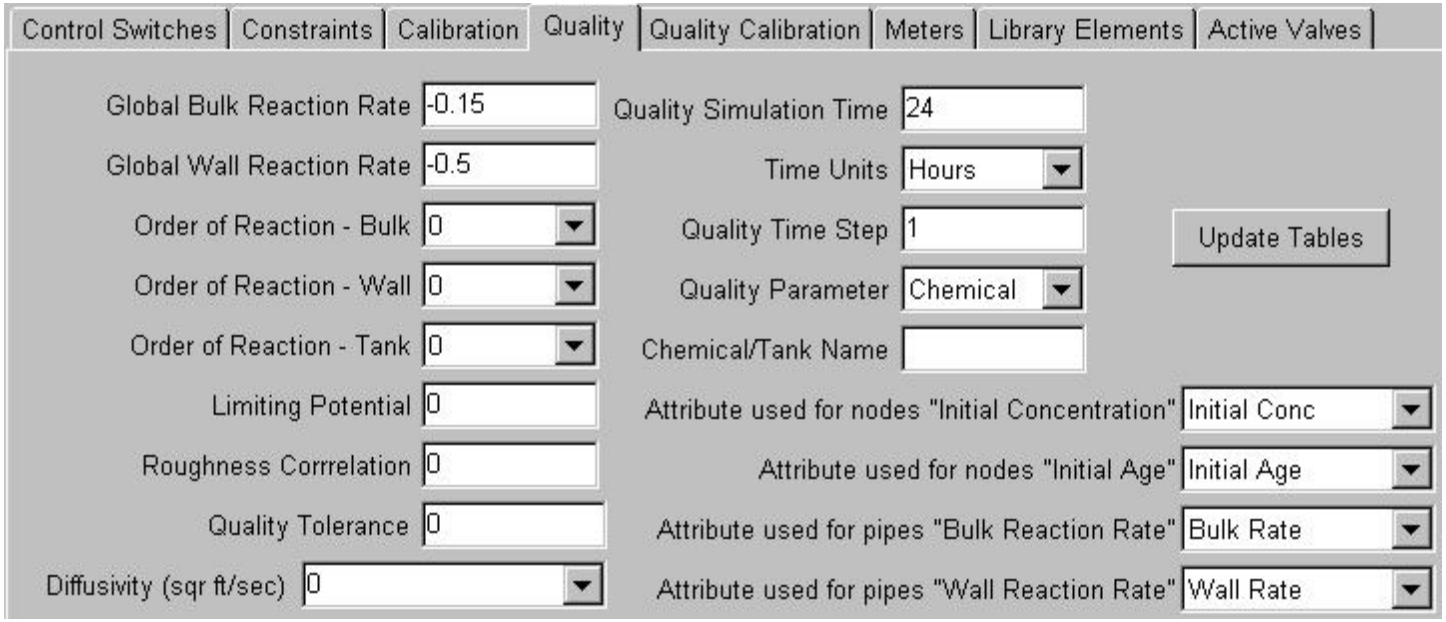
* Classic: Access the water quality data screen by clicking the "Other Data" and "Quality" tabs

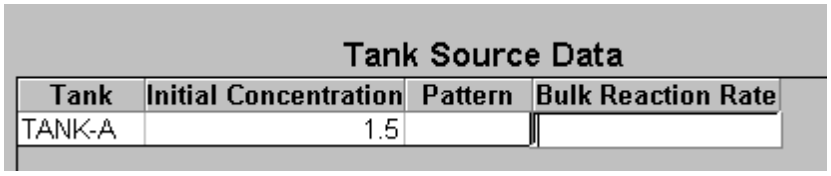


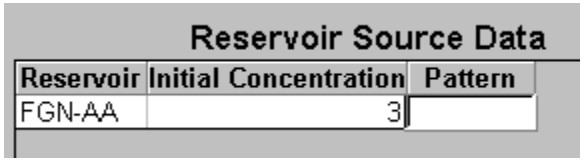

* KYnetic: Access under Analyze | Water Quality Menu



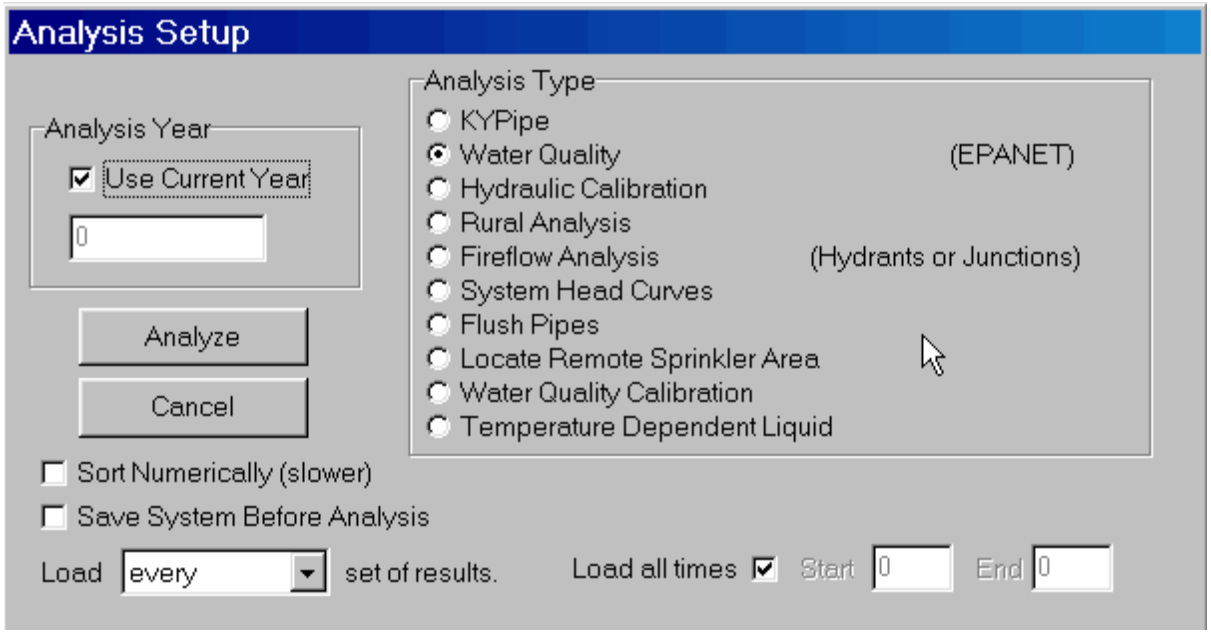
* Provide specified water quality data as shown in the following screens. Please refer to the EPANET user's manual for detailed explanation on decay rates and other related data.



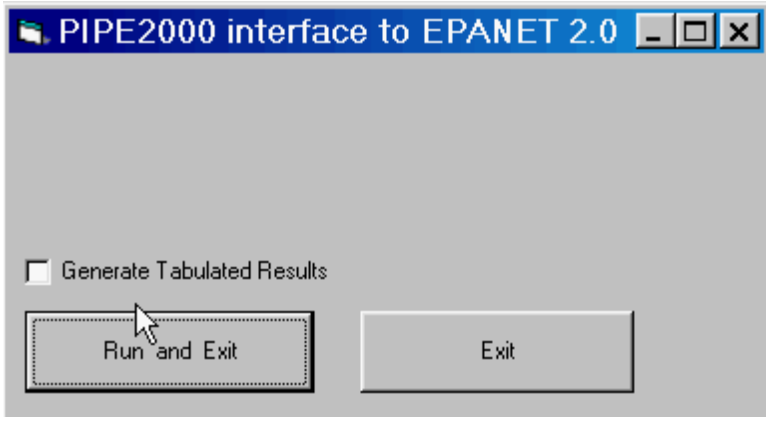




* Access the EPANET environment using the following steps:
    * Click on "Analyze" and then on "Analyze"
    * The following menu appears. Select "Water Quality" and click "Analyze"

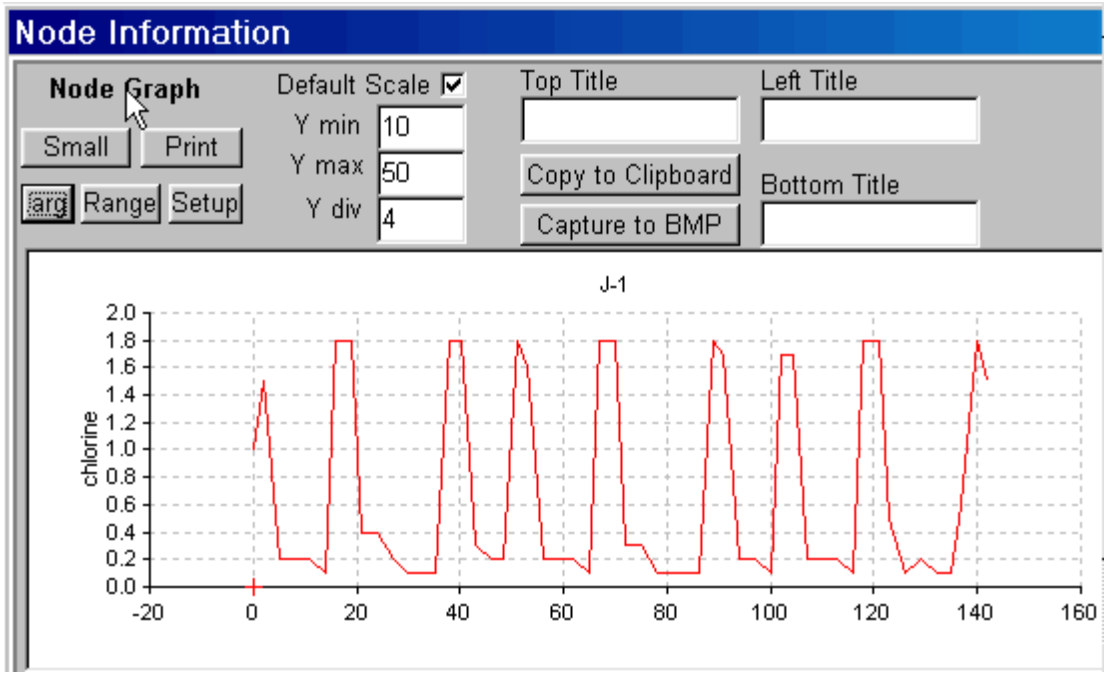



* The following menu appears. At this stage, the data file in the required EPANET is created.




* There are different ways to review the results - graphical, tabulated, and map labels. If the above box, Generate Tabulated Results, is checked, EPANET output is included in the Report. Node tables and graphs can be used to view chlorine levels as well as labels on the

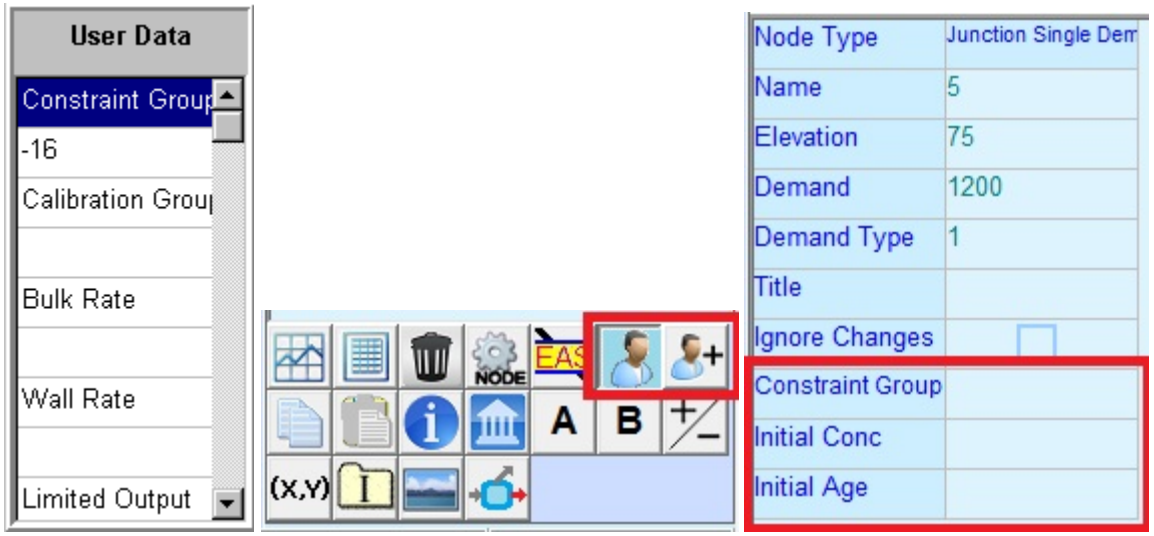
Map.




Part 2: Assume that all pipes 6" and larger have a bulk decay rate of -0.10/day and a wall decay rate of -0.75 ft/day and rerun an EPANET water quality analysis without leaving the Pipe2018 environment. Note: Other pipes in the system will have same bulk and wall decay rates as in the previous part.

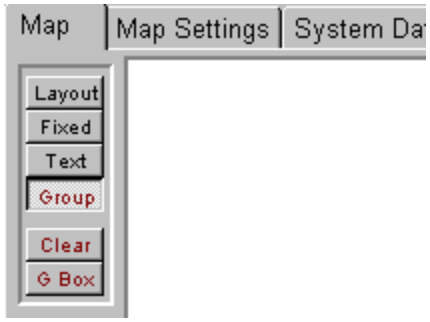
Solution to Part 2:

* To override global bulk and wall decay rates specified in "quality" data screens, provide specified decay rates for all 6" diameter pipes by selecting "User Data" option in "pipe information window." Click on any 6" diameter pipe, and then click on "User" icon to access the following "User Data" menu.

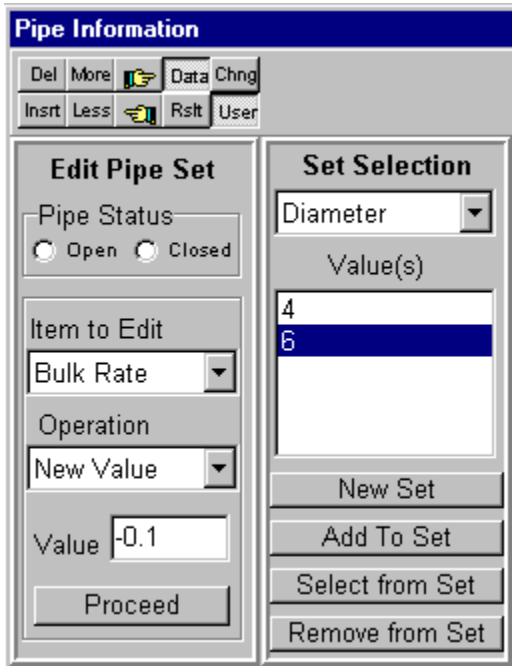


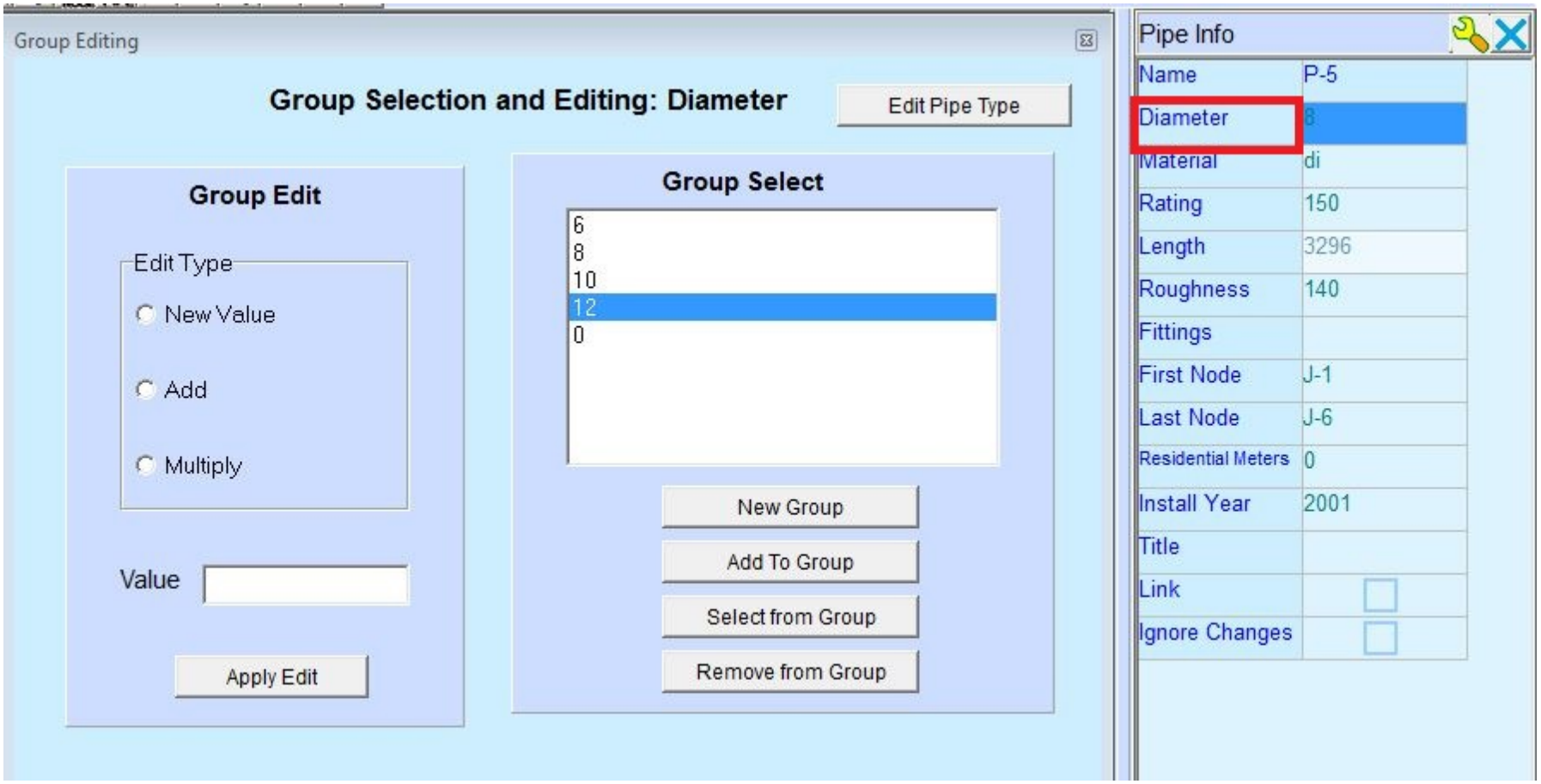

* Provide appropriate bulk and wall decay rates. Repeat this process for all other 6" diameter pipes. Alternatively, one could use "Group" operations feature to set bulk and wall decay rates for all 6" diameter pipes. See also Pipe User Box and Sets and Group Mode. This procedure is explained in the following.

* First click on any pipe and then click "Group" button as shown below. This operation displays "Group Operation" and "Set Selection" windows under "Pipe information windows".

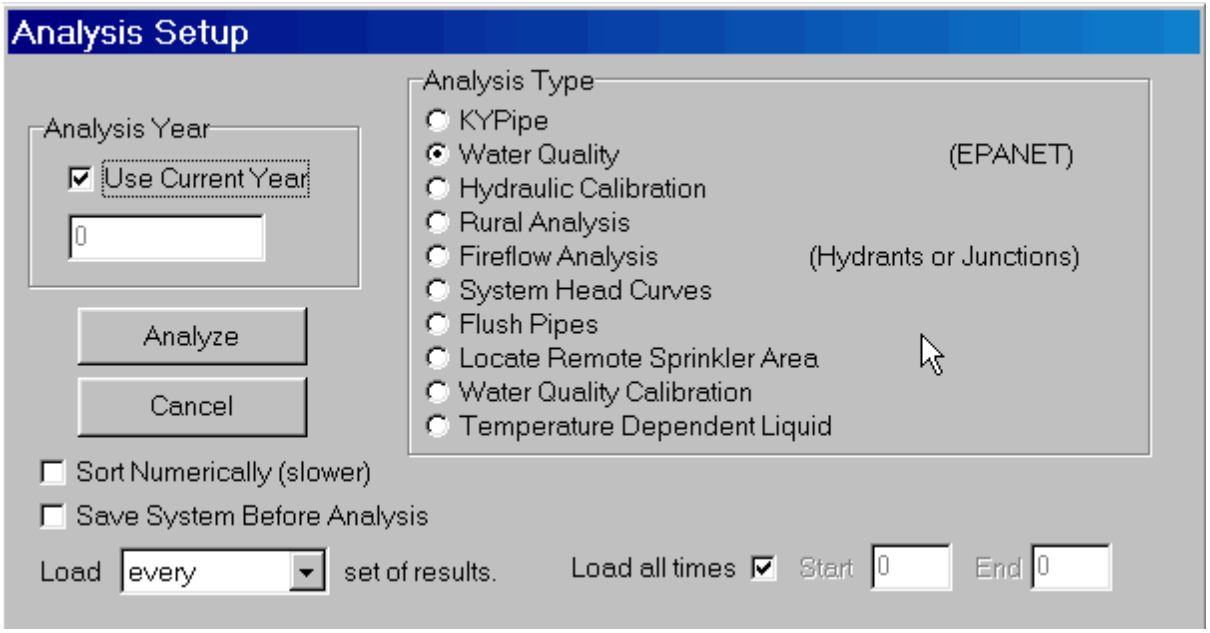



* Select "Diameter" option under "Set Selection" menu, click on 6 and then click on the "New Set" button. This process highlights all 6" (or 12" in the KYnetic example) diameter pipes. With all 6" pipes now highlighted, select "Bulk Rate" option under "Item to Edit" menu and provide a "New Value" of -0.10 and click "Proceed". This process sets the bulk decay rate for all 6" diameter pipes to -0.10/day. Repeat this process to set wall decay rate.


Pipe Information
Del
More
Data
Chng
Insrt
Less
Rslt
User
Edit Pipe Set
Pipe Status
Open
Closed
Item to Edit
Bulk Rate
Operation
New Value
Value
-0.1
Proceed
Set Selection
Diameter
Value(s)
4
6
New Set
Add To Set
Select from Set
Remove from Set




* Click "Analyze" on the top menu bar and select "Analyze" option to access the following analysis options window.

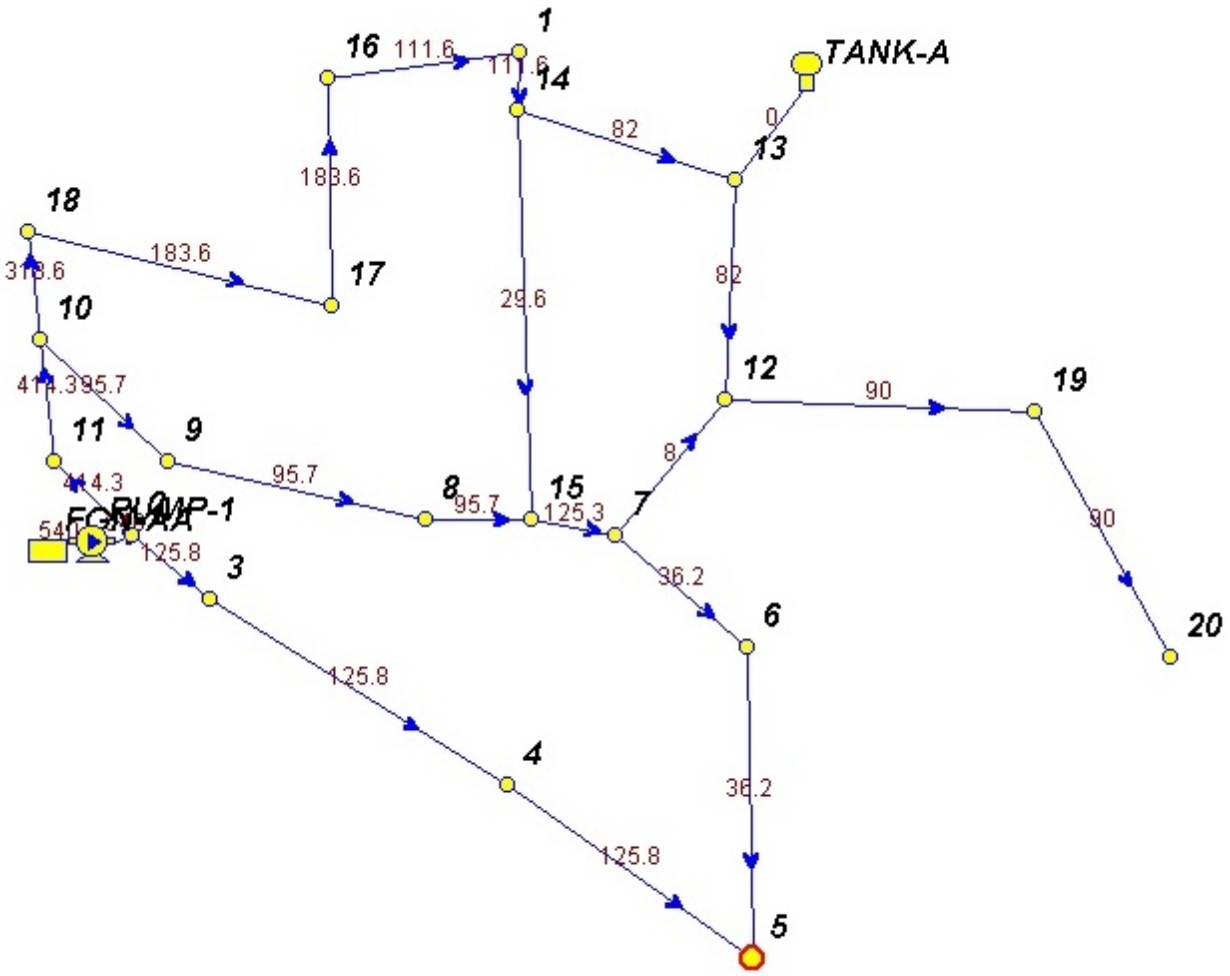



* Select "Water Qualtiy" option, click "Analyze" and follow the on-screen instructions. This process generates an EPANET water quality data file, performs an EPANET water quality analysis and loads the EPANET-generated, tabulated output into the Pipe2018 Report (if the Generate Tabulated Report box is checked). In addition, this process also converts EPANET hydraulic results into a Pipe2018 format for graphical review. Click on the "Report" tab to view the EPANET-generated, tabulated output. A sample output showing the chlorine concentrations at 24 hrs is shown in the following.

Node Results at 24:00 hrs:

| Node | Elev. ft | Demand gpm | Grade ft | Pressure psi | Chemical mg/L | |
|---|---|---|---|---|---|---|
| 1 | 109.00 | 0.00 | 257.62 | 64.40 | 1.89 | |
| 2 | 100.00 | 0.00 | 283.26 | 79.41 | 2.63 | |
| 3 | 97.00 | 0.00 | 278.50 | 78.64 | 1.99 | |
| 4 | 93.00 | 0.00 | 262.01 | 73.23 | 1.60 | |
| 5 | 79.00 | 180.00 | 247.90 | 73.18 | 1.08 | |
| 6 | 88.00 | 0.00 | 249.35 | 69.91 | 0.73 | |
| 7 | 97.00 | 90.00 | 250.17 | 66.37 | 0.95 | |
| 8 | 104.00 | 0.00 | 257.19 | 66.38 | 1.42 | |
| 9 | 110.00 | 0.00 | 264.66 | 67.01 | 1.87 | |
| 10 | 112.00 | 0.00 | 269.65 | 68.31 | 2.44 | |
| 11 | 103.00 | 0.00 | 277.00 | 75.39 | 2.46 | |
| 12 | 113.00 | 0.00 | 250.12 | 59.41 | 0.96 | |
| 13 | 119.00 | 0.00 | 254.81 | 58.85 | 1.30 | |
| 14 | 113.00 | 0.00 | 255.49 | 61.74 | 1.36 | |
| 15 | 100.00 | 0.00 | 254.16 | 66.80 | 1.09 | |
| 16 | 113.00 | 80.00 | 258.63 | 63.10 | 2.02 | |
| 17 | 110.00 | 0.00 | 261.60 | 65.69 | 2.15 | |
| 18 | 107.00 | 150.00 | 265.69 | 68.76 | 2.30 | |
| 19 | 109.00 | 0.00 | 242.29 | 57.75 | 0.91 | |
| 20 | 105.00 | 100.00 | 235.16 | 56.40 | 0.52 | |
| 22 | 95.00 | 0.00 | 287.36 | 83.35 | 2.81 | |
| 25 | 95.00 | 0.00 | 115.90 | 9.05 | 2.81 | |
| 21 | 120.00 | -600.00 | 120.00 | 0.00 | 3.00 | Reservoir |
| 23 | 120.00 | 0.00 | 250.00 | 56.33 | 0.09 | Tank |

* The following figure depicts flowrates at 23 hours, as computed by the EPANET hydraulic simulation model, overlaid on Pipe2018 network schematic. These results are displayed using the options under Labels in the Main Menu.




Trace Analysis

To conduct a trace analysis, specify "Trace" in the Quality Parameter field. Then enter the exact name of the tank from which the trace is done in the "Chemical/Tank Name" field. Run the analysis. To view the results, display Node Results A as a map label. At the bottom of the map screen, next to "N" choose "Trace". Now you may use the red selector arrows to scroll through the trace results or select a node and view the table or graph for that node.

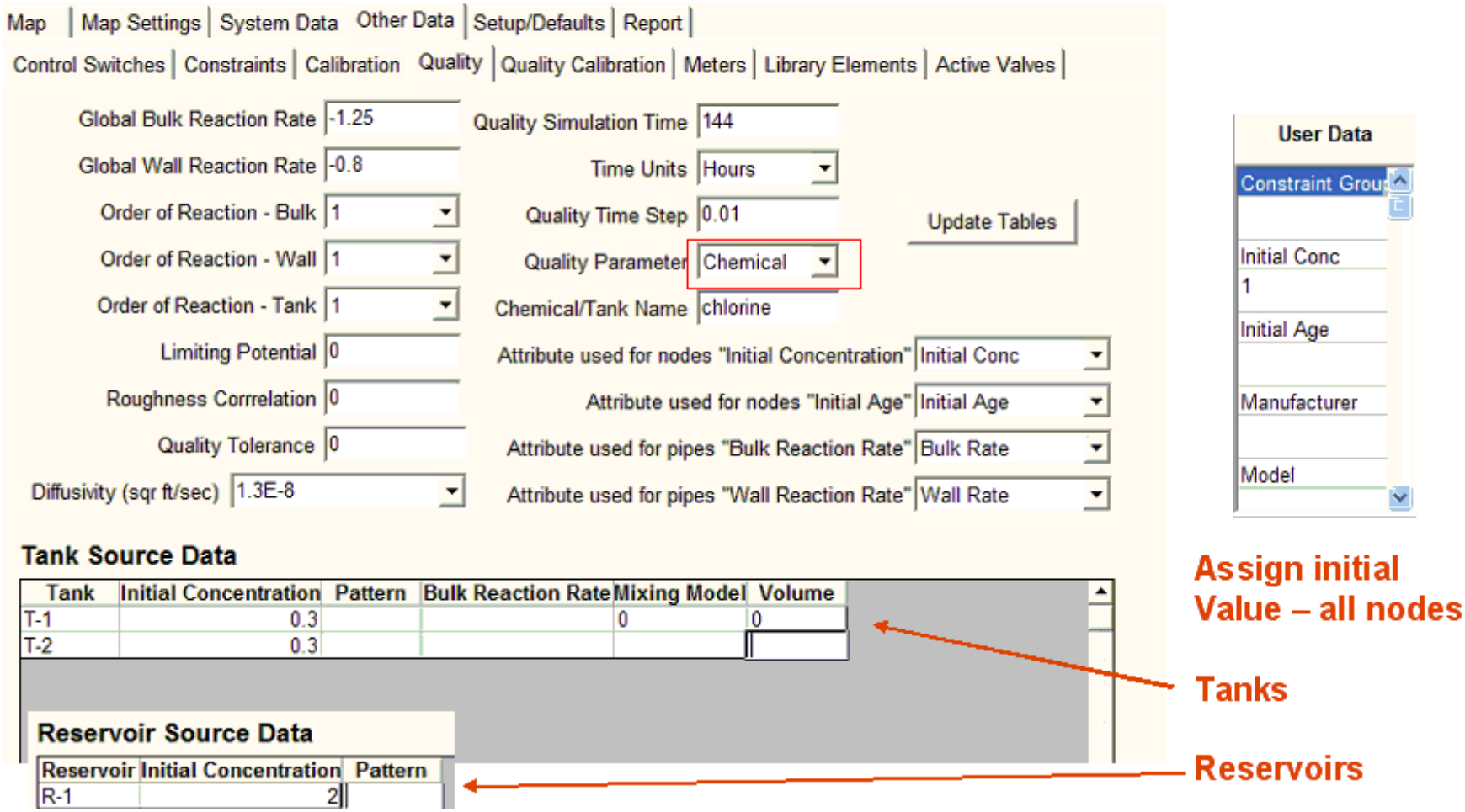
### Water Age

To conduct a water age analysis, specify "Age" in the Quality Parameter field. You may specify an initial age for the various sources in the model or leave the default as zero (note that the "Age" data field is shared by the Quality analysis setup as "Initial Concentration" field, entered data may be carried over as "Age"). Run the analysis. To view the results, display Node Results A as a map label. At the bottom of the map screen, next to "N" choose "Age". Now you may use the red selector arrows to scroll through the age results or select a node and view the table or graph for that node.

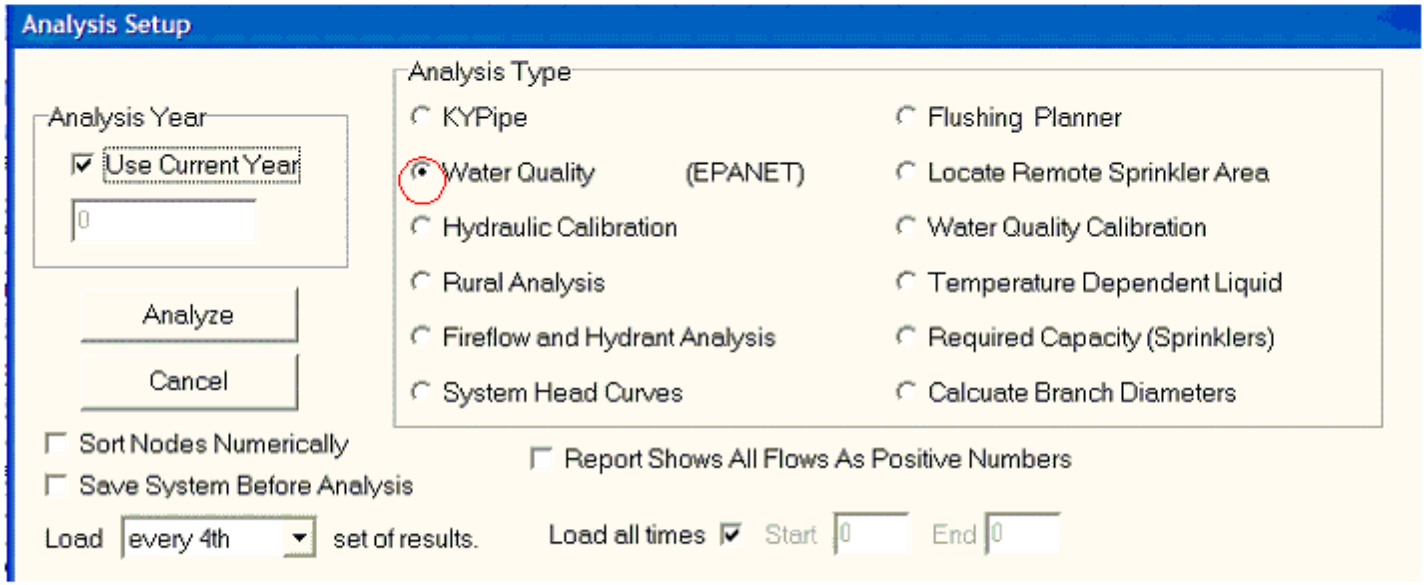
## Example 2

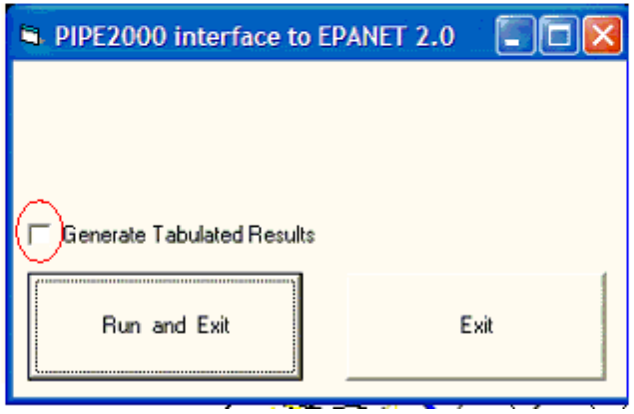
# Water Quality Data – Chlorine Residuals

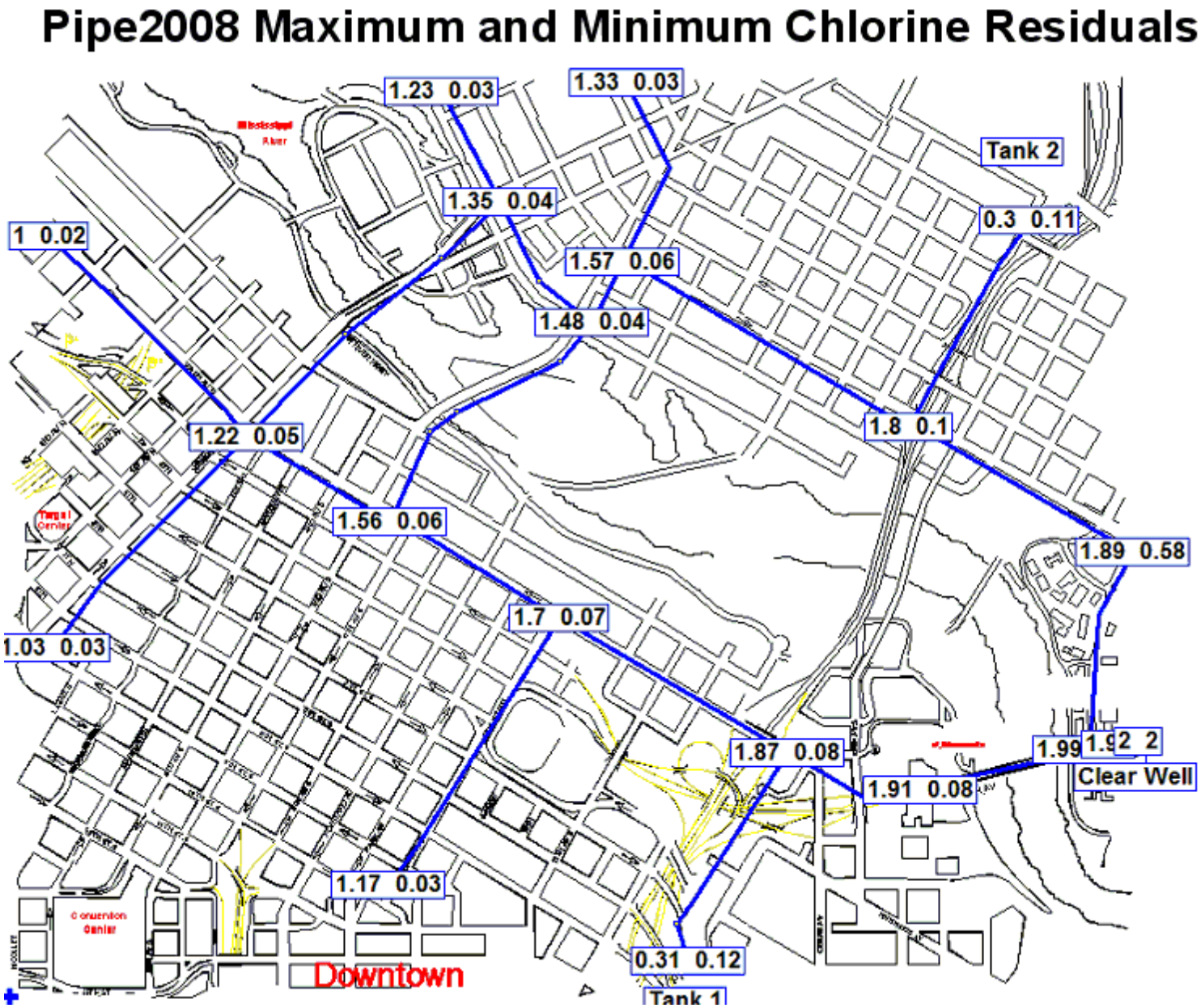
## Examples Manual – Demo 1-6 (demoqual.p2k)



# Running the WQ Analysis




Pipe2008 Maximum and Minimum Chlorine Residuals
1.23 0.03
1.33 0.03
Tank 2
1.35 0.04
1 0.02
0.3 0.11
1.57 0.06
1.48 0.04
1.8 0.1
1.22 0.05
1.56 0.06
1.89 0.58
1.7 0.07
1.03 0.03
1.87 0.08
1.99
Clear Well
1.91 0.08
1.17 0.03
0.31 0.12
Downtown
Tank 1

# EPANET Tabulated Results

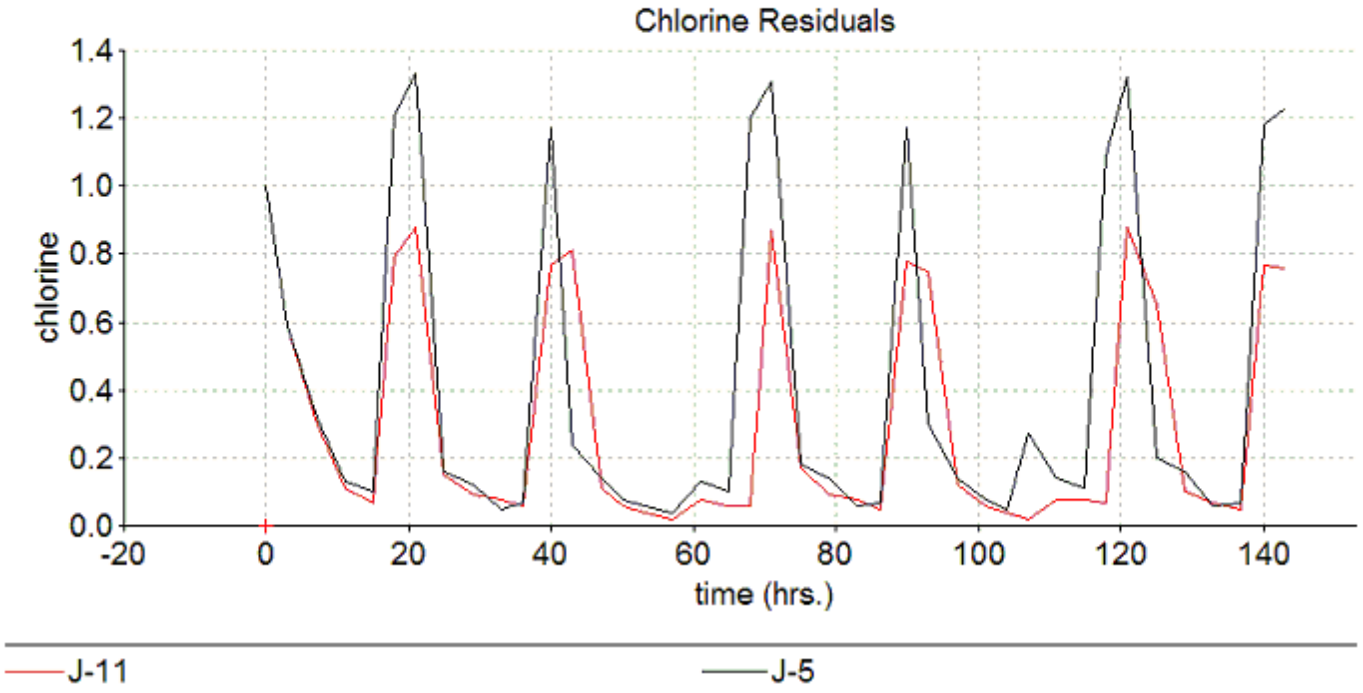
```
Page 1                                        Sun Sep 24 12:47:39 2006

******************************************************************
*                          E P A N E T                           *
*                  Hydraulic and Water Quality                   *
*                  Analysis for Pipe Networks                    *
*                          Version 2.0                           *
******************************************************************


    Input Data File ..................... C:\PROGRA~1\PIPE2006\EXAMPL~1\demorest\demoWQ.inp
    Number of Junctions.................. 18
    Number of Reservoirs................. 1
    Number of Tanks ..................... 2
    Number of Pipes ..................... 21
    Number of Pumps ..................... 1
    Number of Valves .................... 0
    Headloss Formula .................... Hazen-Williams
    Hydraulic Timestep .................. 1.00 hrs
    Hydraulic Accuracy .................. 0.005000
    Maximum Trials ...................... 20
    Quality Analysis .................... CHLORINE
    Water Quality Time Step ............. 0.58 min
    Water Quality Tolerance ............. 0.01 mg/L
    Specific Gravity .................... 1.00
    Relative Kinematic Viscosity ........ 1.00
    Relative Chemical Diffusivity ....... 1.00
    Demand Multiplier ................... 1.00
    Total Duration ...................... 144.00 hrs
    Reporting Criteria:
       All Nodes
       All Links

Analysis begun Sun Sep 24 12:47:39 2006
```

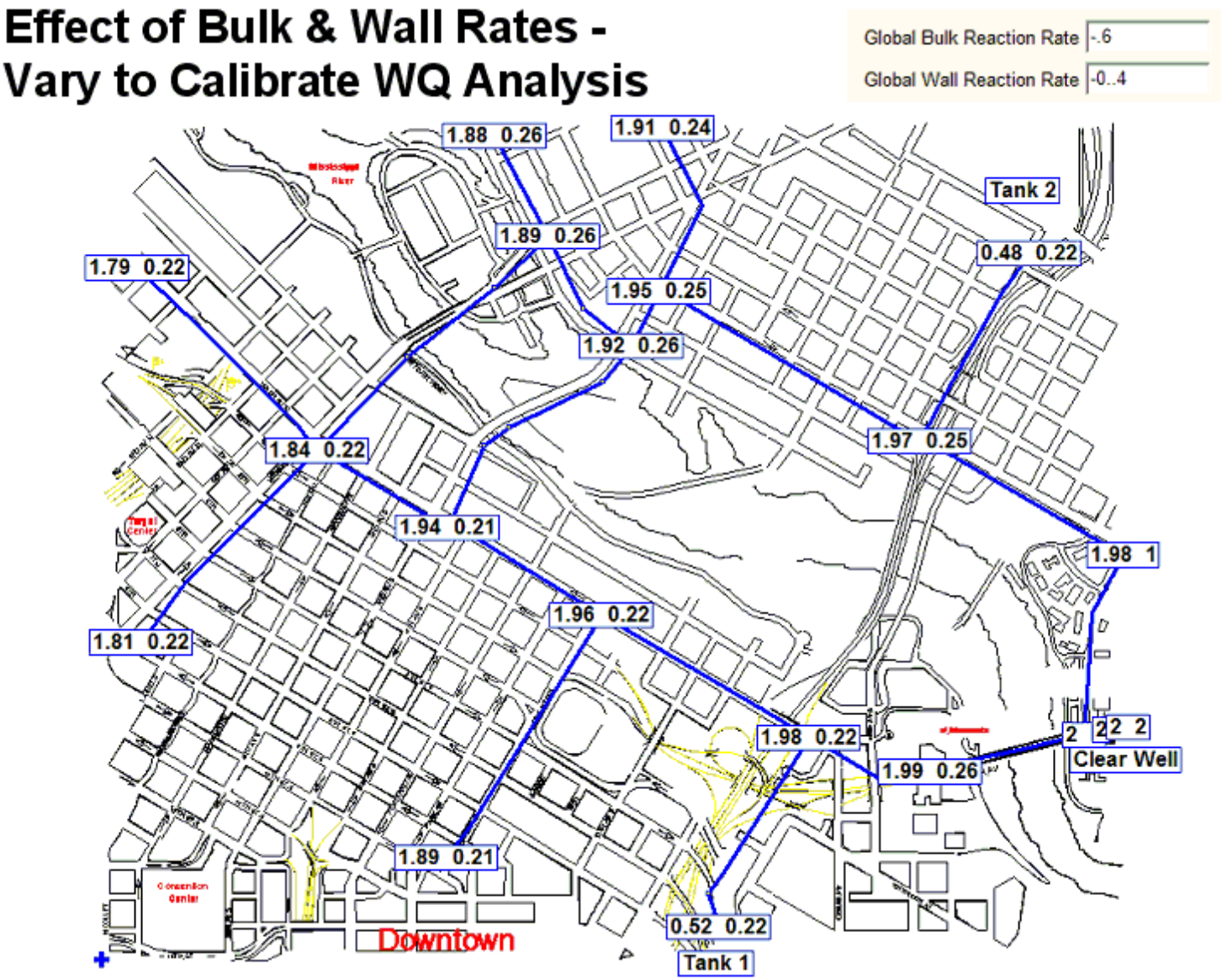
# EPANET Tabulated Results

Node Results at 100:00:00 hrs:

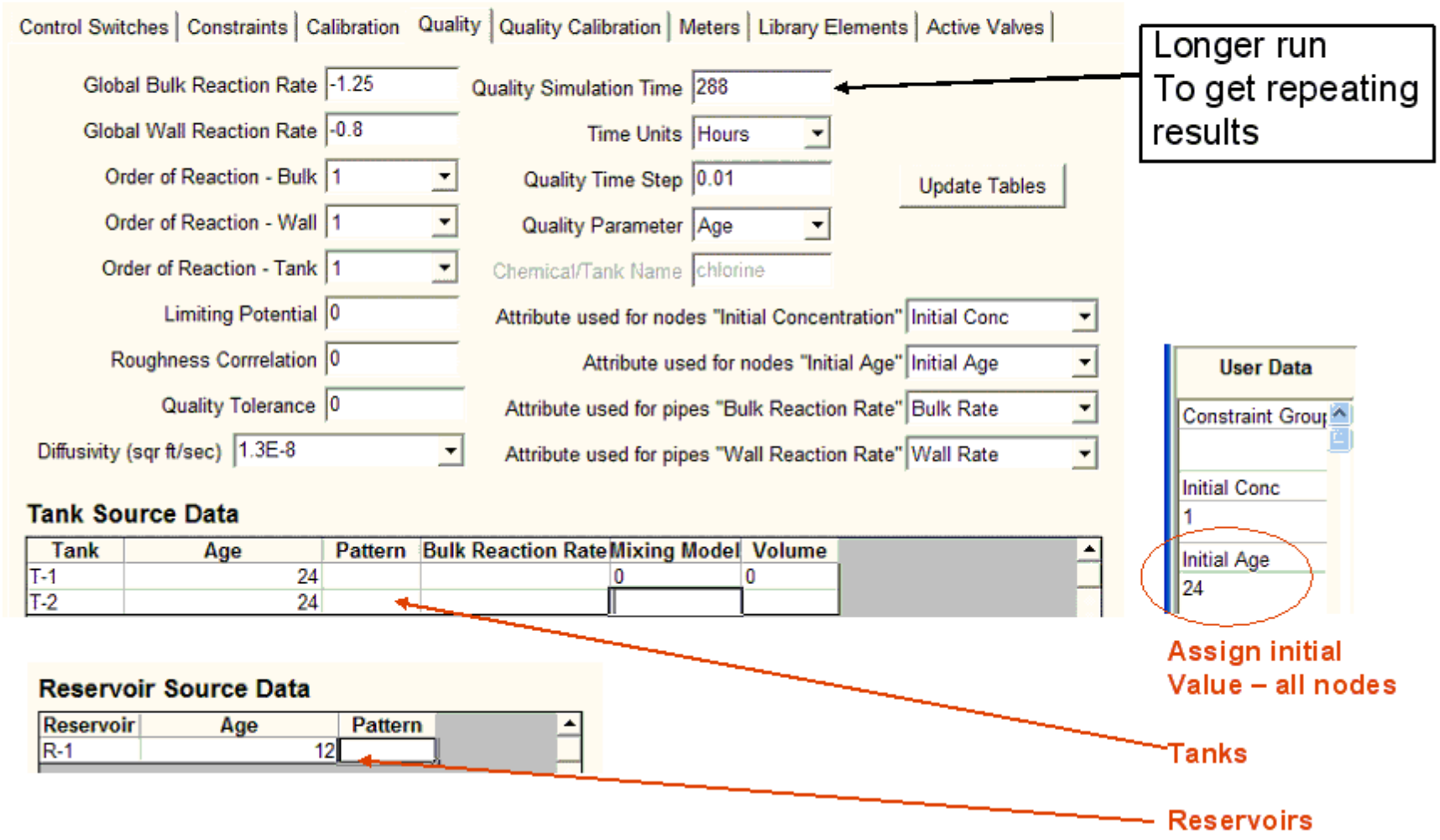
| Node | Elevation ft | Demand gpm | Head ft | Pressure psi | CHLORINE mg/L | |
|---|---|---|---|---|---|---|
| J-1 | 607.00 | 2.97 | 737.22 | 56.43 | 0.13 | |
| J-10 | 611.00 | 4.95 | 736.94 | 54.57 | 0.09 | |
| J-11 | 610.00 | 14.85 | 736.86 | 54.97 | 0.07 | |
| J-12 | 610.00 | 0.00 | 737.22 | 55.13 | 0.13 | |
| J-13 | 630.00 | 19.80 | 736.89 | 46.32 | 0.10 | |
| J-14 | 610.00 | 0.00 | 736.99 | 55.03 | 0.10 | |
| J-15 | 605.00 | 0.00 | 737.22 | 57.29 | 0.83 | |
| J-16 | 605.00 | 0.00 | 737.22 | 57.29 | 0.11 | |
| J-2 | 617.00 | 24.75 | 736.72 | 51.87 | 0.07 | |
| J-3 | 605.00 | 0.00 | 737.22 | 57.29 | 0.77 | |
| J-4 | 602.00 | 0.00 | 736.99 | 58.49 | 0.13 | |
| J-5 | 613.00 | 14.85 | 736.95 | 53.71 | 0.08 | |
| J-6 | 611.00 | 6.93 | 737.01 | 54.60 | 0.12 | |
| J-7 | 613.00 | 0.00 | 736.96 | 53.71 | 0.10 | |
| J-8 | 611.00 | 5.94 | 737.06 | 54.62 | 0.10 | |
| J-9 | 615.00 | 9.90 | 737.01 | 52.87 | 0.08 | |
| I-Pump-1 | 610.00 | 0.00 | 610.00 | 0.00 | 1.99 | |
| O-Pump-1 | 610.00 | 0.00 | 737.22 | 55.13 | 1.23 | |
| T-1 | 630.00 | -52.10 | 737.39 | 46.53 | 0.17 | Tank |
| T-2 | 625.00 | -52.84 | 737.39 | 48.70 | 0.14 | Tank |
| R-1 | 610.00 | 0.00 | 610.00 | 0.00 | 2.00 | Reservoir |

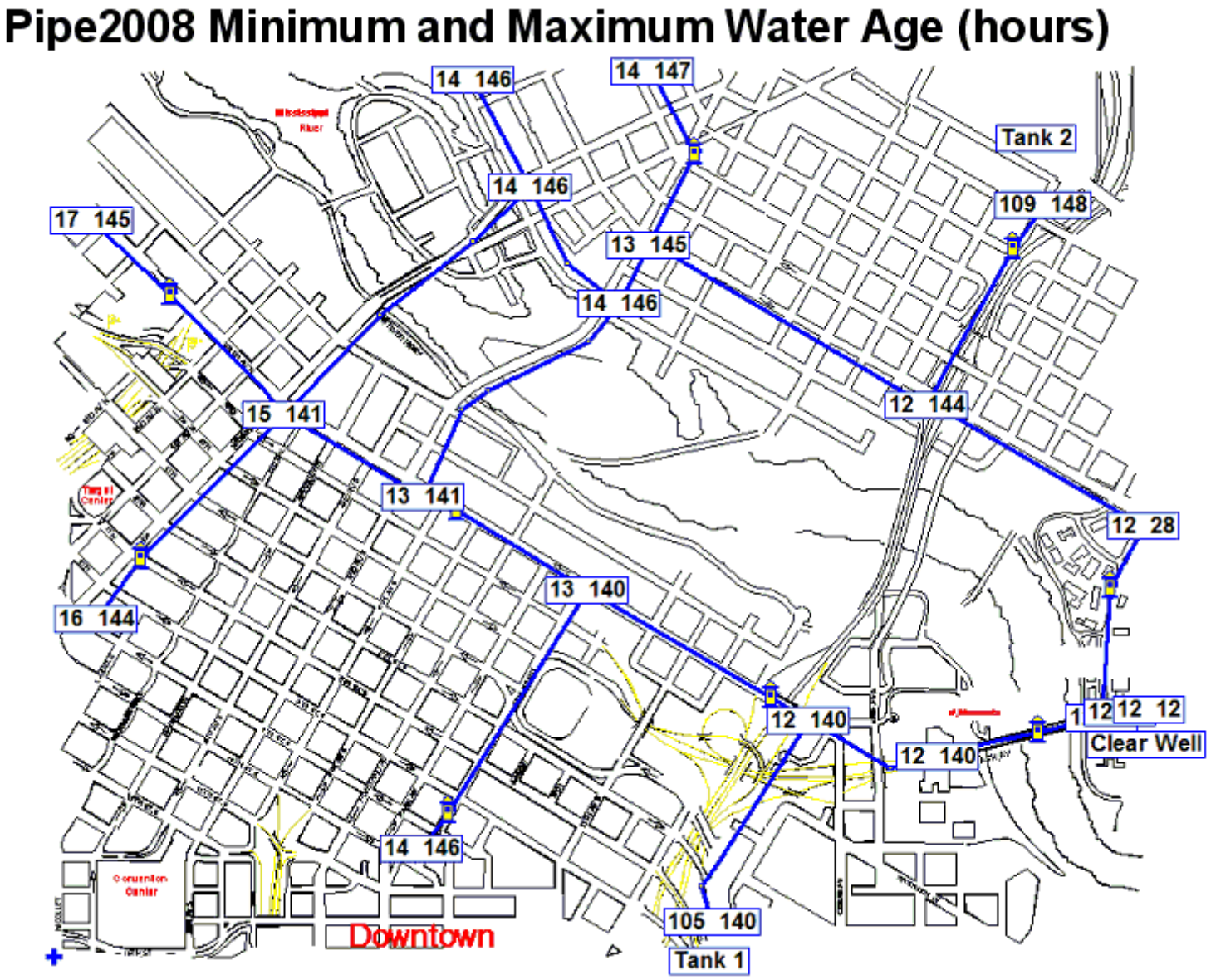
# Pipe2008 Plotted Results




Effect of Bulk & Wall Rates -
Vary to Calibrate WQ Analysis
Global Bulk Reaction Rate -.6
Global Wall Reaction Rate -0..4
1.88 0.26
1.91 0.24
Tank 2
1.89 0.26
0.48 0.22
1.79 0.22
1.95 0.25
1.92 0.26
1.97 0.25
1.84 0.22
1.94 0.21
1.98 1
1.96 0.22
1.81 0.22
1.98 0.22
1.99 0.26
Clear Well
1.89 0.21
0.52 0.22
Tank 1
Downtown

# Water Age Calculations




Pipe2008 Minimum and Maximum Water Age (hours)
14 146
14 147
Tank 2
14 146
109 148
17 145
13 145
14 146
12 144
15 141
13 141
12 28
13 140
16 144
12 140
12 12 12
Clear Well
12 140
14 146
Downtown
105 140
Tank 1

## 1.4.1.3 Water Quality Calibration

Water Quality Calibration

---

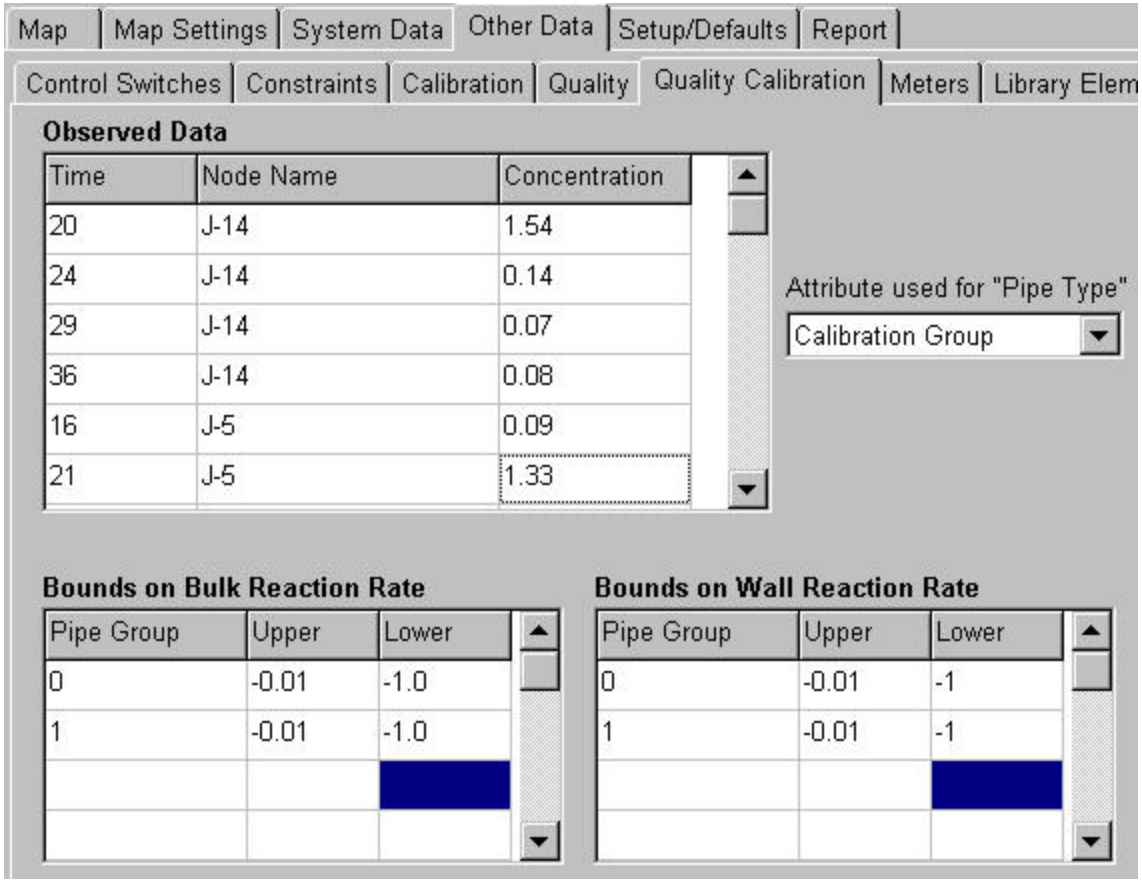
This feature is used to calibrate the Bulk and Wall reaction rates for model based on concentration data collected in the field.

Go to Other Data | Quality Calibration. This calibration module may be used with a system for which water quality data has been defined. Use observed concentration data collected in the field at various times as shown in the example below. Create two or more groups of pipes using Group mode using the Calibration or other User-defined attribute. Be sure the name you have given for this pipe attribute is selected under "Attribute used for "Pipe Type"" Assign bounds for the Bulk and Wall Reaction Rates.

Input times as decimal, for example 10.25 instead of 10:15.

If times for observed data are in decimal, it must match the time increments specified in the Water Quality menu.

Do not use symbols in node or pipe names.




Then run the Water Quality Calibration analysis. A Water Quality Calibration report is generated. This report includes calculated values of the Bulk and Wall Reaction rates as shown. These are not automatically applied to the system. If the user wishes to use these values they may be applied using Group mode.

```
Optimal values for the Decision variables :

      For Group# 0
            Bulk reaction rate  =  -0.728    [ -0.01  < > -1     ]
            Wall reaction rate  =  -0.791    [ -0.01  < > -1     ]

      For Group# 1
            Bulk reaction rate  =  -0.446    [ -0.01  < > -1     ]
            Wall reaction rate  =  -0.945    [ -0.01  < > -1     ]
```

## 1.4.1.4 Water Quality Sensor Placement Tool

For Pipe2012 and later versions.

Drinking water utilities have recognized the importance of online sensors in minimizing distribution system vulnerability to accidental or intentional contamination. Online sensors with real-time data acquisition and transmission capability can be quite expensive forcing small and medium water utilities to limit the number of sensors to just one or a few. Placement of sensors is not a trivial task, considering the limitations on number of affordable sensors and the need for limiting the spread of contamination within the distribution system. Though TEVA-SPOT (Threat Ensemble Vulnerability Assessment and Sensor Placement Optimization Tool) can provide guidance on placement of water quality sensors for water distribution networks, it requires the use of complex water quality models of distribution networks coupled with sophisticated optimization methods for efficient sensor placement guidance. Most small and medium water utilities may not have technical or financial resources required for meaningful use of TEVA-SPOT and may not be able take full advantage of water quality sensor technology and may continue to face higher vulnerability risk.

The Optimal Water Quality Sensor Placement Tool (OWQSPT) is a simplified guidance tool for optimal placement of sensors in small and medium water distribution networks. The input for this tool is a good hydraulic model setup for extended period simulation (EPS). Optimal water quality sensor placement analysis (to minimize average detection time) can be performed with little or no additional input from water utilities. Using the default data built into the KYPipe Water Quality Sensor Tool, the KYPipe users can determine optimal locations for water quality sensors in their extended period simulation model with just three simple key strokes! In addition to generating a detailed report for a sensor placement study, the optimal locations for water quality sensors will be automatically highlighted on the network model.

The Optimal Water Quality Sensor Placement Tool is an ideal, affordable tool for small and medium water utilities that will help circumvent water distribution

system contamination. This tool is only available in the KYnetic interface and the Professional and Premium packages.

.

KYPIPE-WQSensor
Pipe2012 Tool for
Optimal Placement of Water Quality Sensors
for Small and Medium Water utilities

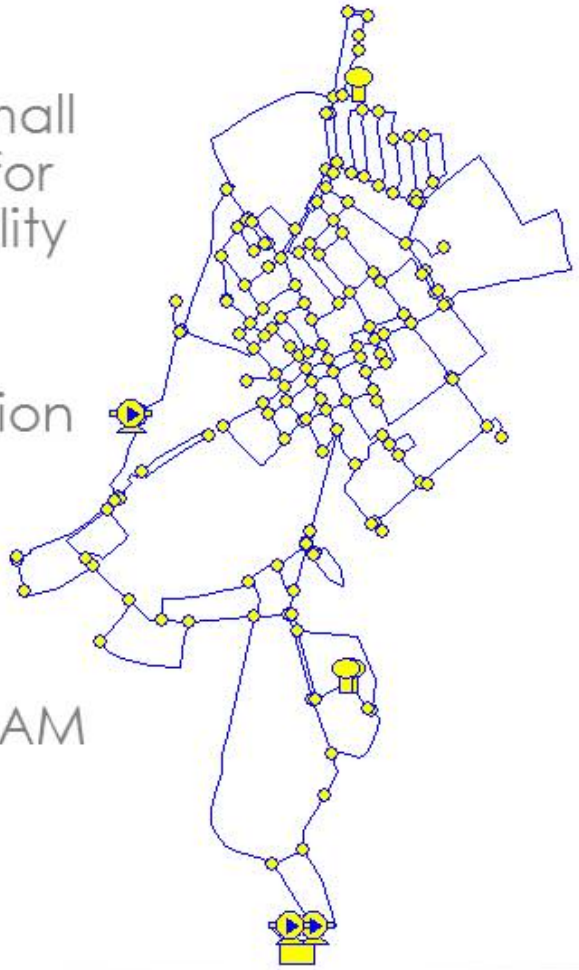
.

_____

.

# Optimal Water Quality Sensor Placement Tool

- Use of optimal water quality sensor placement tool is demonstrated on a small water distribution network model setup for extended period simulation (Water Quality Sensor Placement.P2K under KYPIPE example files folder).
- Network comprises one main pump station and a booster pump station and three elevated storage tanks.
- Mass injection rate of contaminant = 1000mg/min
- Duration of injection: 1 hour starting at 4AM
- Number of water quality sensors: 2



www.kypipe.com

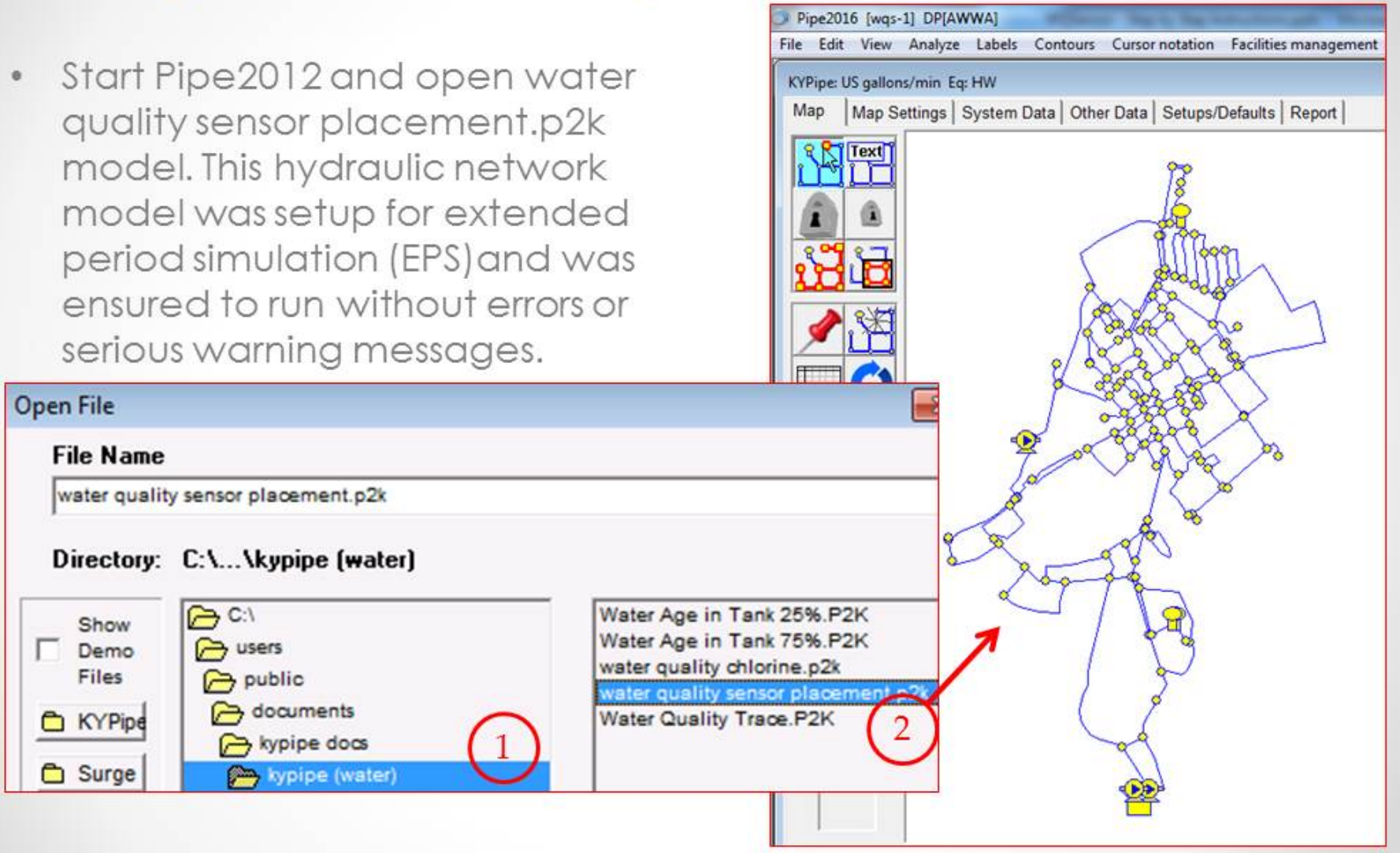
.

_____

.

# Optimal Water Quality Sensor Placement Tool

- Start Pipe2012 and open water quality sensor placement.p2k model. This hydraulic network model was setup for extended period simulation (EPS) and was ensured to run without errors or serious warning messages.



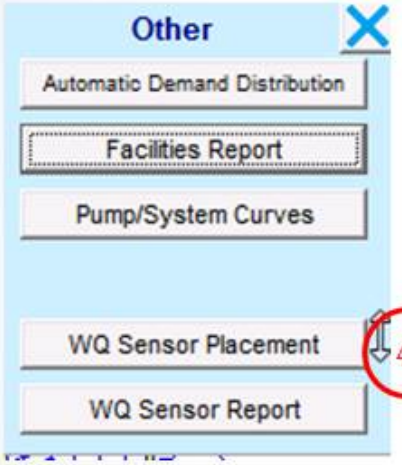
.

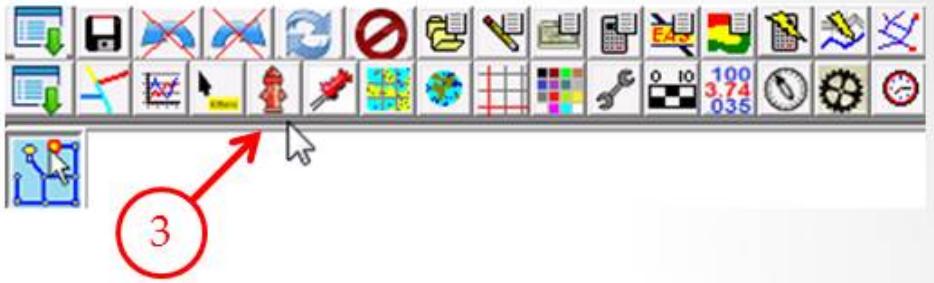
_____

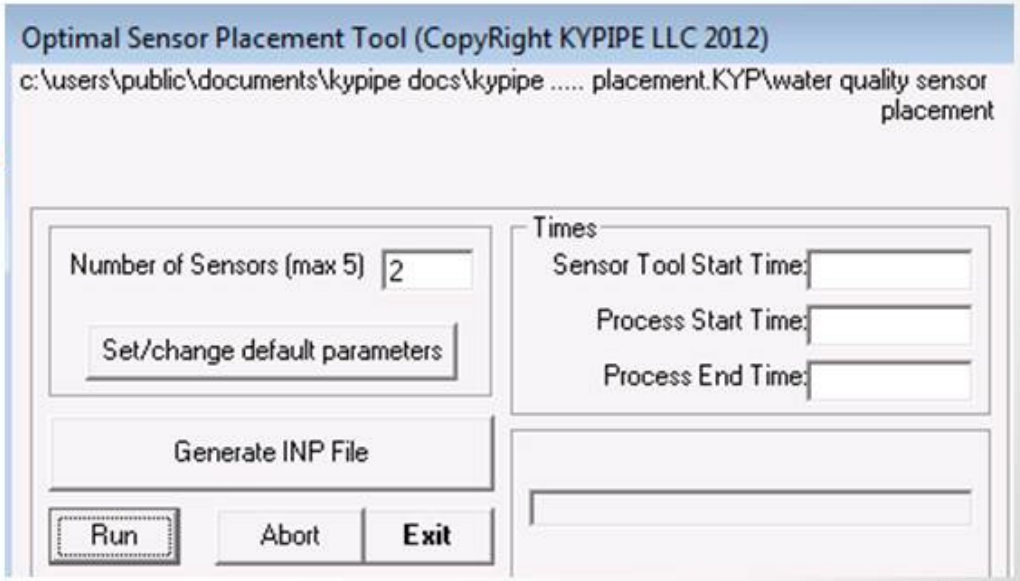
.

# Optimal Water Quality Sensor Placement Tool

- Access WQ Sensor Placement Tool from Facilities Management Window




www.kypipe.com

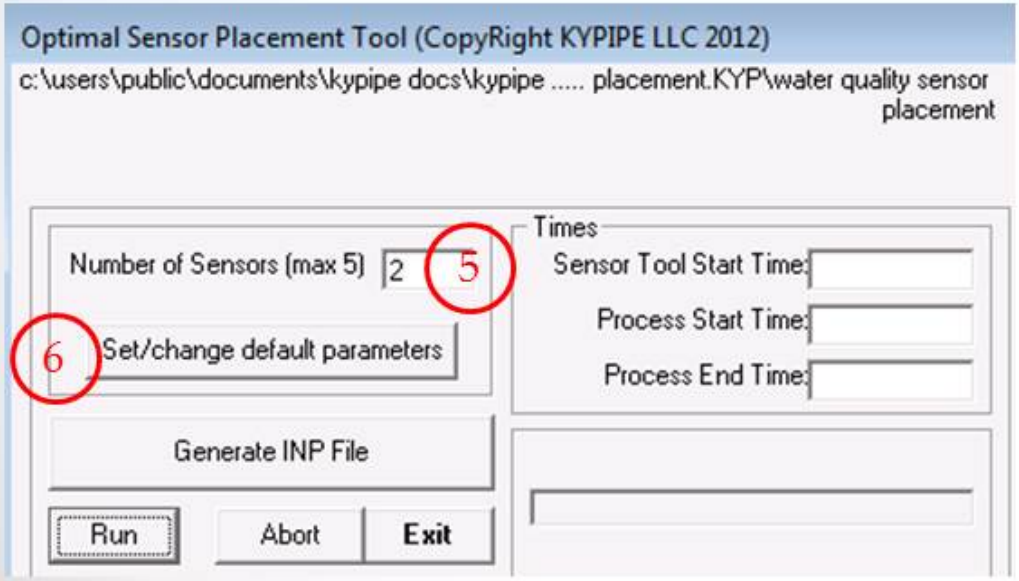
.

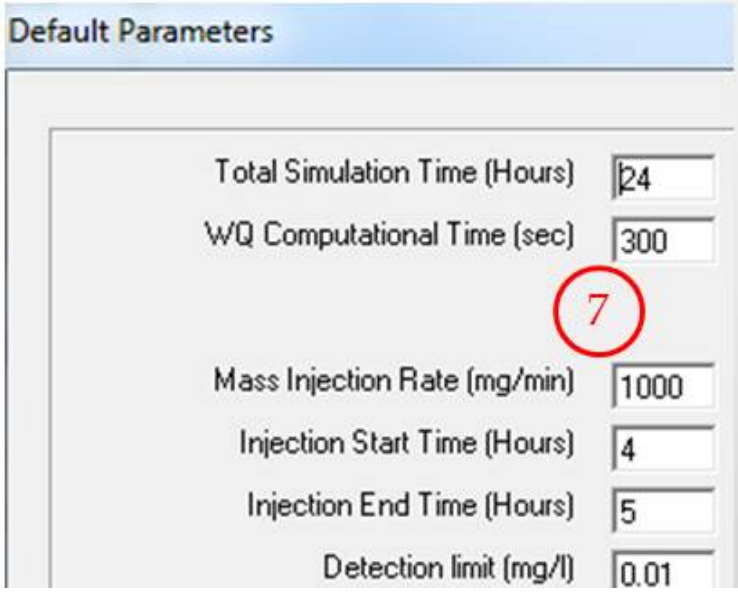
_____

.

# Optimal Water Quality Sensor Placement Tool

- Enter 2 under "Number of Sensors" box
- Click on "Set/change default parameters" button and enter data or make changes as needed in the default parameter window.



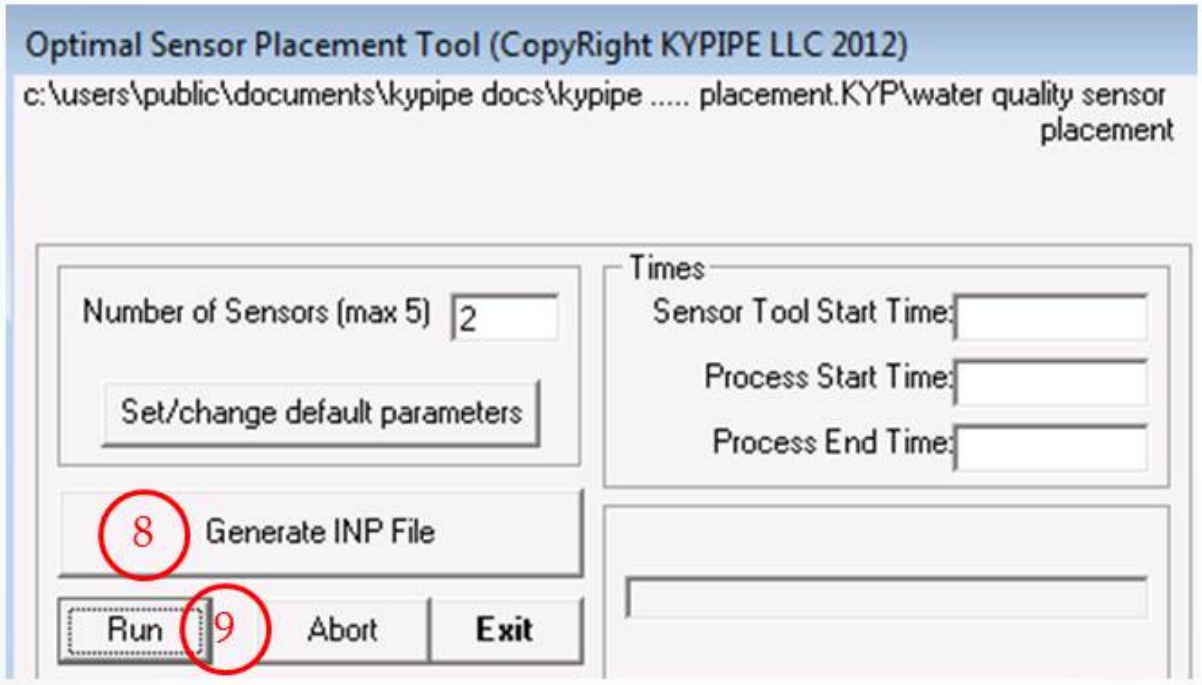
.

_____

.

# Optimal Water Quality Sensor Placement Tool

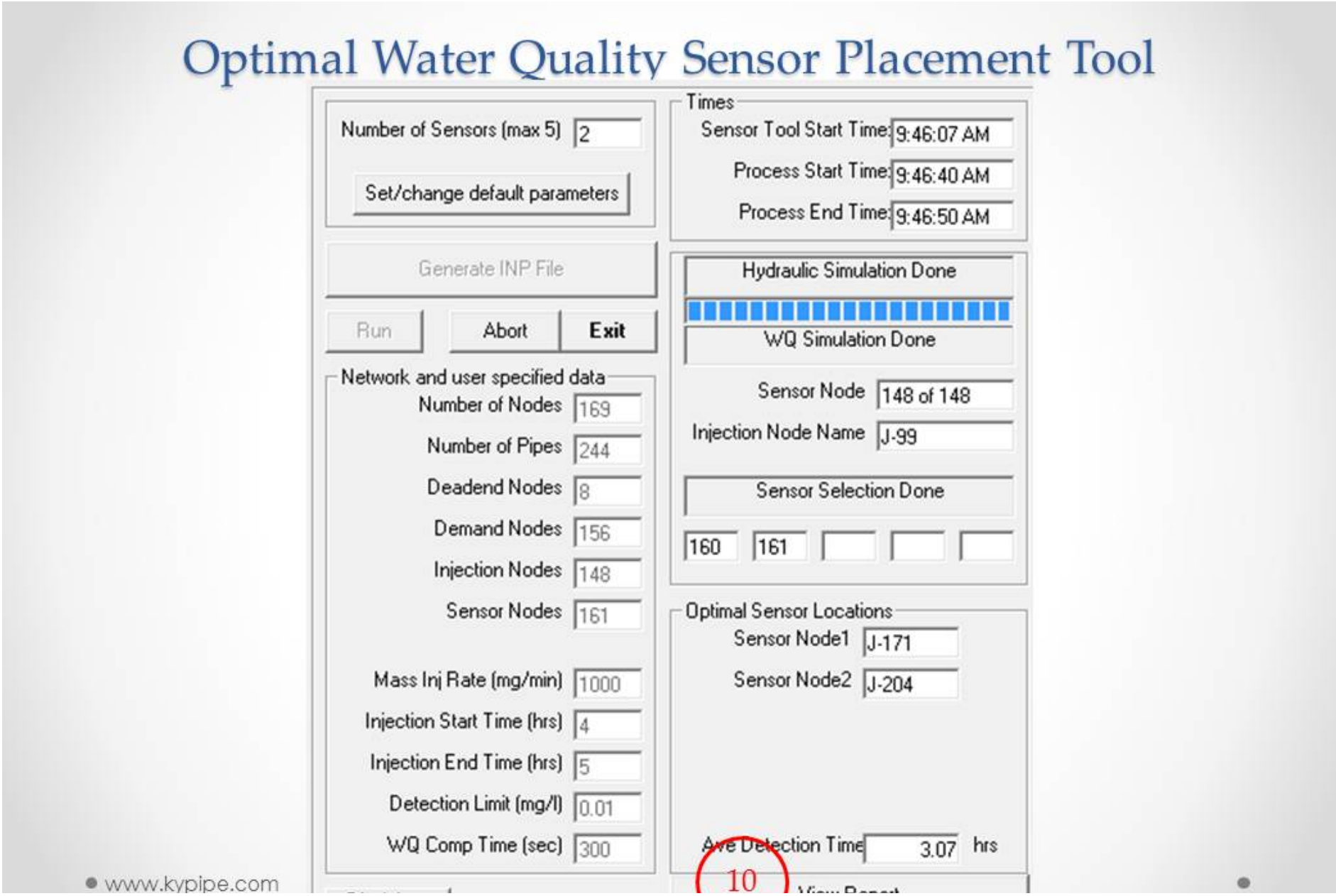
- Go back to the main menu and click on "Generate INP file" button. Then click on "Run" button.



.

_____

.


Optimal Water Quality Sensor Placement Tool
Number of Sensors (max 5) 2
Set/change default parameters
Generate INP File
Run
Abort
Exit
Network and user specified data
Number of Nodes 169
Number of Pipes 244
Deadend Nodes 8
Demand Nodes 156
Injection Nodes 148
Sensor Nodes 161
Mass Inj Rate (mg/min) 1000
Injection Start Time (hrs) 4
Injection End Time (hrs) 5
Detection Limit (mg/l) 0.01
WQ Comp Time (sec) 300
Times
Sensor Tool Start Time: 9:46:07 AM
Process Start Time: 9:46:40 AM
Process End Time: 9:46:50 AM
Hydraulic Simulation Done
WQ Simulation Done
Sensor Node 148 of 148
Injection Node Name J-99
Sensor Selection Done
160
161
Optimal Sensor Locations
Sensor Node1 J-171
Sensor Node2 J-204
Ave Detection Time 3.07 hrs
10
www.kypipe.com

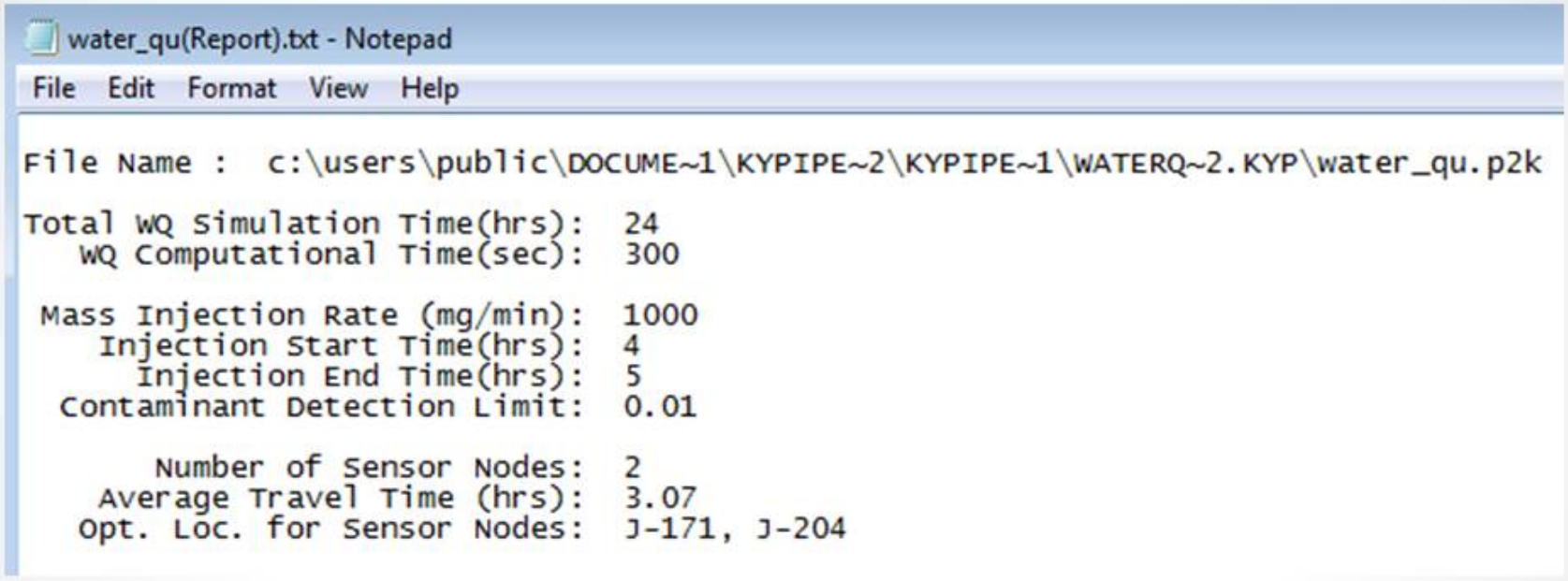
Disclaimer
View Report

.

_____

.

# Optimal Water Quality Sensor Placement Tool

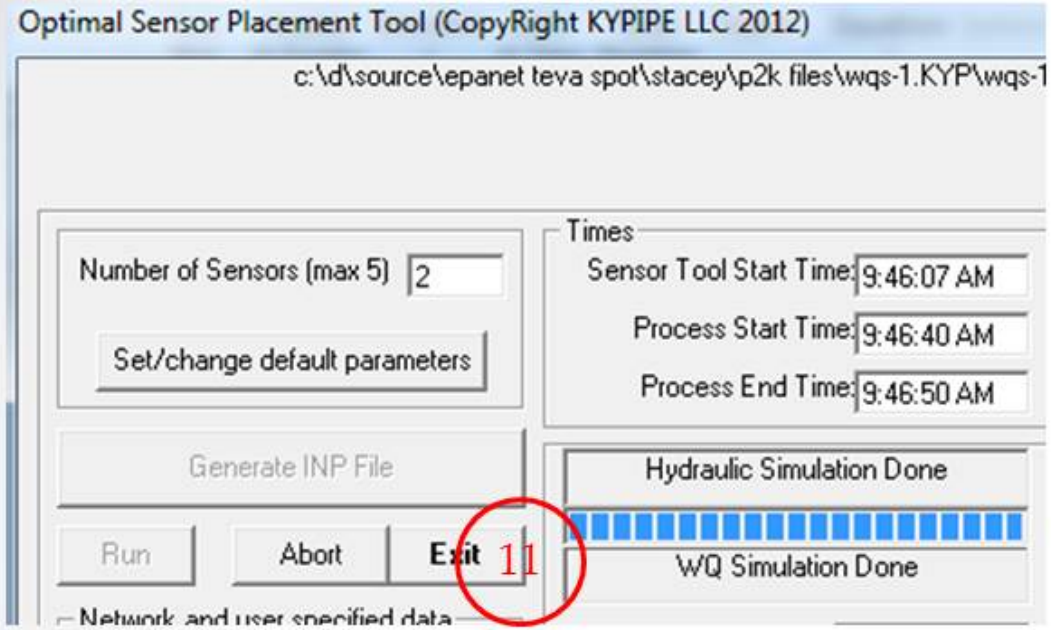


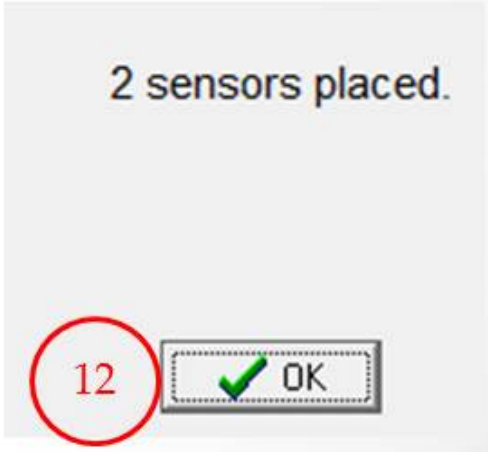
.

_____

.

# Optimal Water Quality Sensor Placement Tool



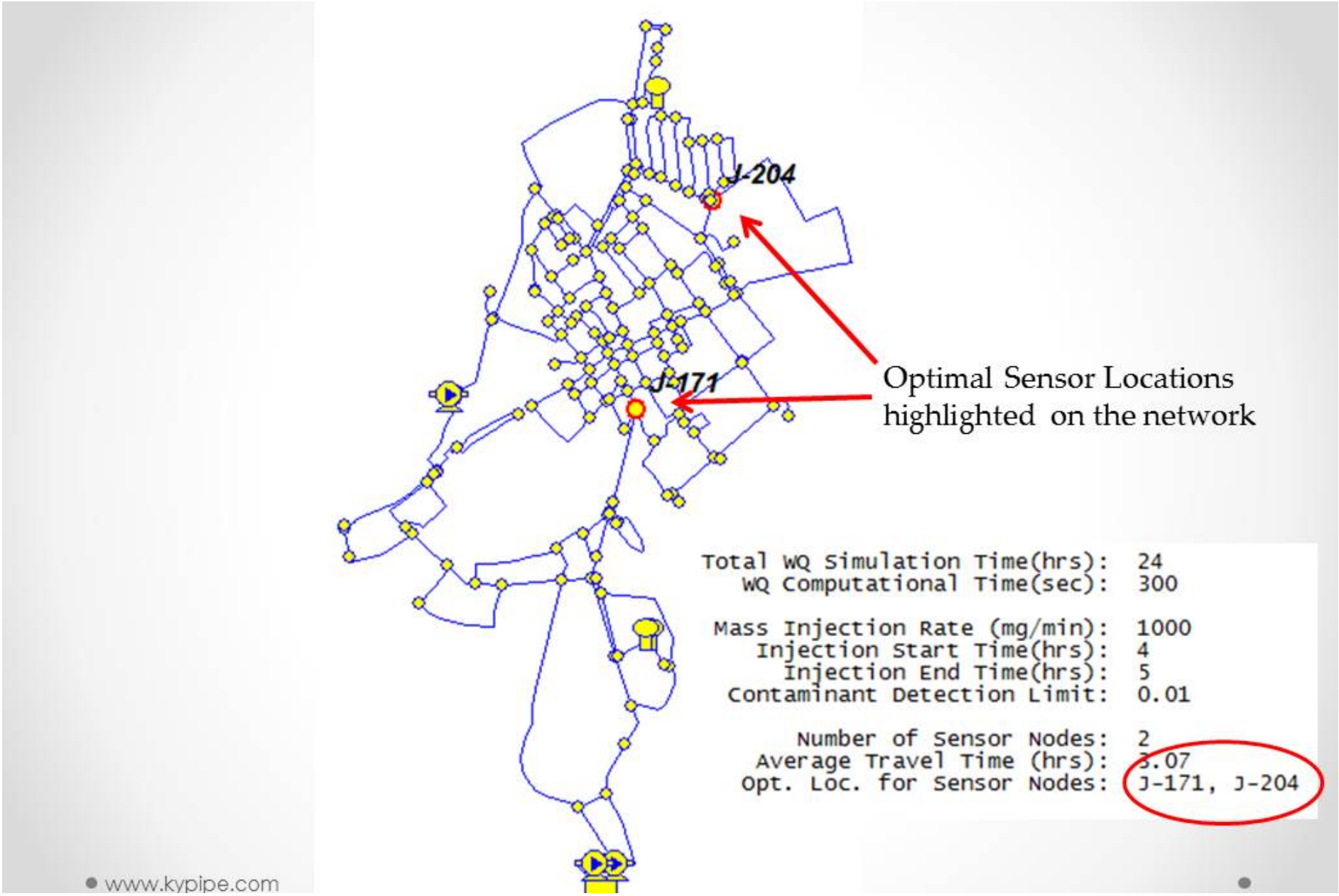




www.kypipe.com

.

_____

.


J-204
J-171
Optimal Sensor Locations
highlighted on the network
Total WQ Simulation Time(hrs): 24
WQ Computational Time(sec): 300
Mass Injection Rate (mg/min): 1000
Injection Start Time(hrs): 4
Injection End Time(hrs): 5
Contaminant Detection Limit: 0.01
Number of Sensor Nodes: 2
Average Travel Time (hrs): 3.07
Opt. Loc. for Sensor Nodes: J-171, J-204
www.kypipe.com

.

## 1.4.2 Extended Period Simulation

## 1.4.2.1 Extended Period Simulations (EPS)

Extended Period Simulations (EPS)

See also EPS and the EPS videos on the KYPipe web site.

See Demand Patterns for an example 24-hr demand pattern.

---

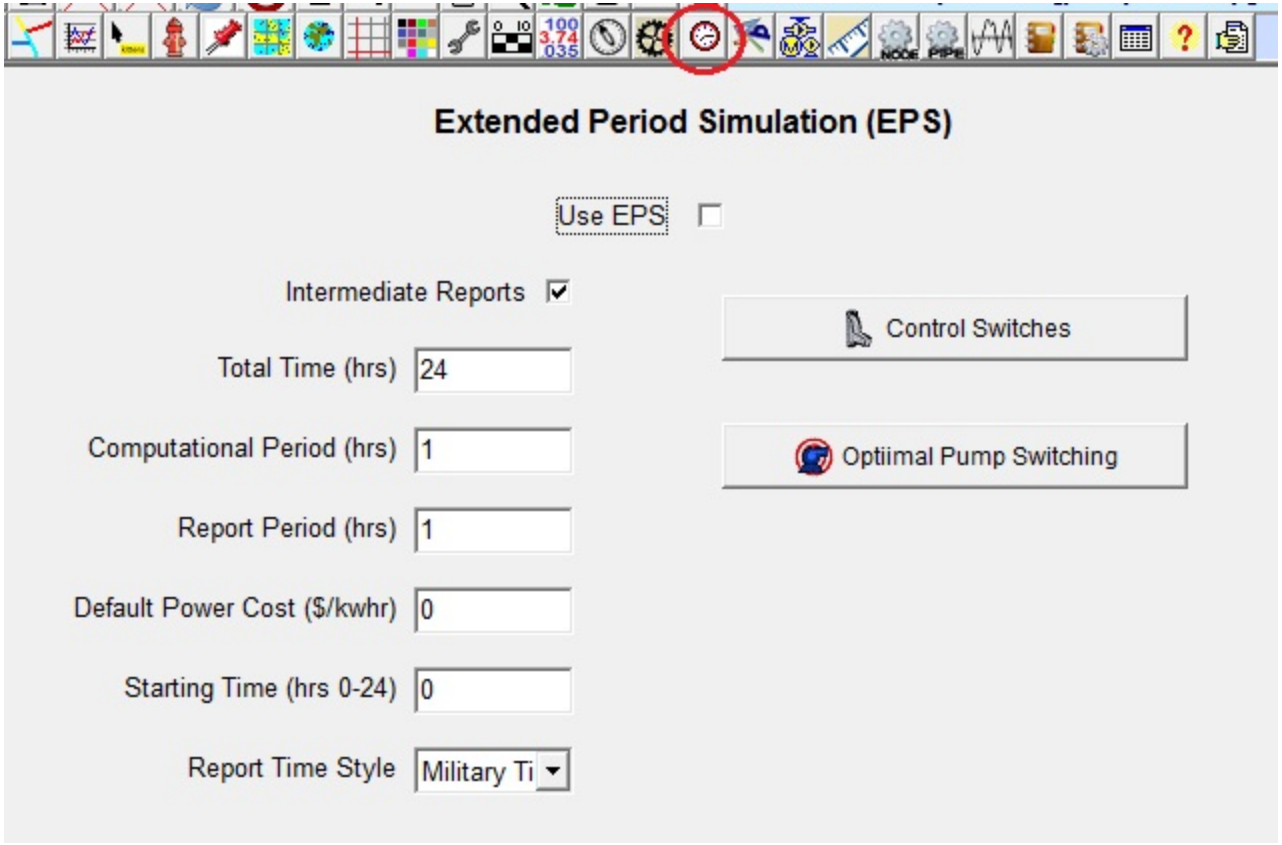
Once you have developed your model and can run a regular simulation, it is relatively easy to set up an EPS. There are 4 types of additional data which may be required. These include:

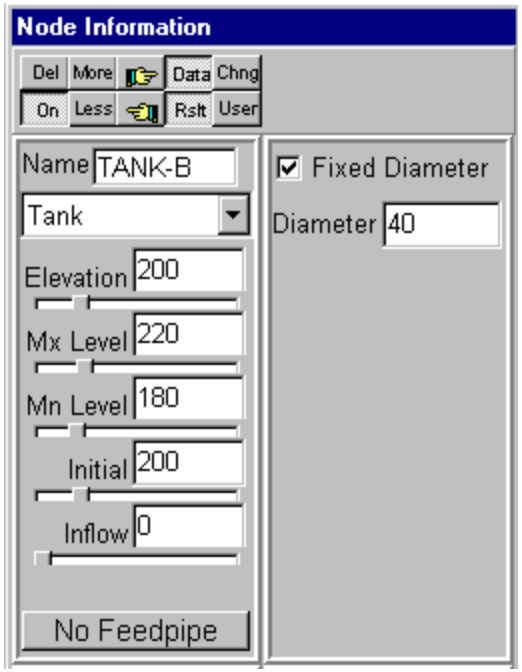
1. EPS Data (under EPS icon (KYnetic) or System Data tab (Classic) - always required)
2. Tank Data (in the Node Information Window)
3. Demand Pattern Data (under Demand Pattern icon or Setup/Defaults tab (Classic)
4. Control Switch Data (under EPS icon (KYnetic) or System Data tab (Classic)).

We will use Example 1 from the Reference Manual and Mex_EPS in your DataFiles subdirectory to illustrate several EPS set ups. The user is intended to follow along with the below examples and make the noted changes to the system in order to create the EPS simulations. These are also illustrated by the Audio/Video clips (Extended Period Simulations (EPS, EPS - Tanks, and EPS - Control Switches).

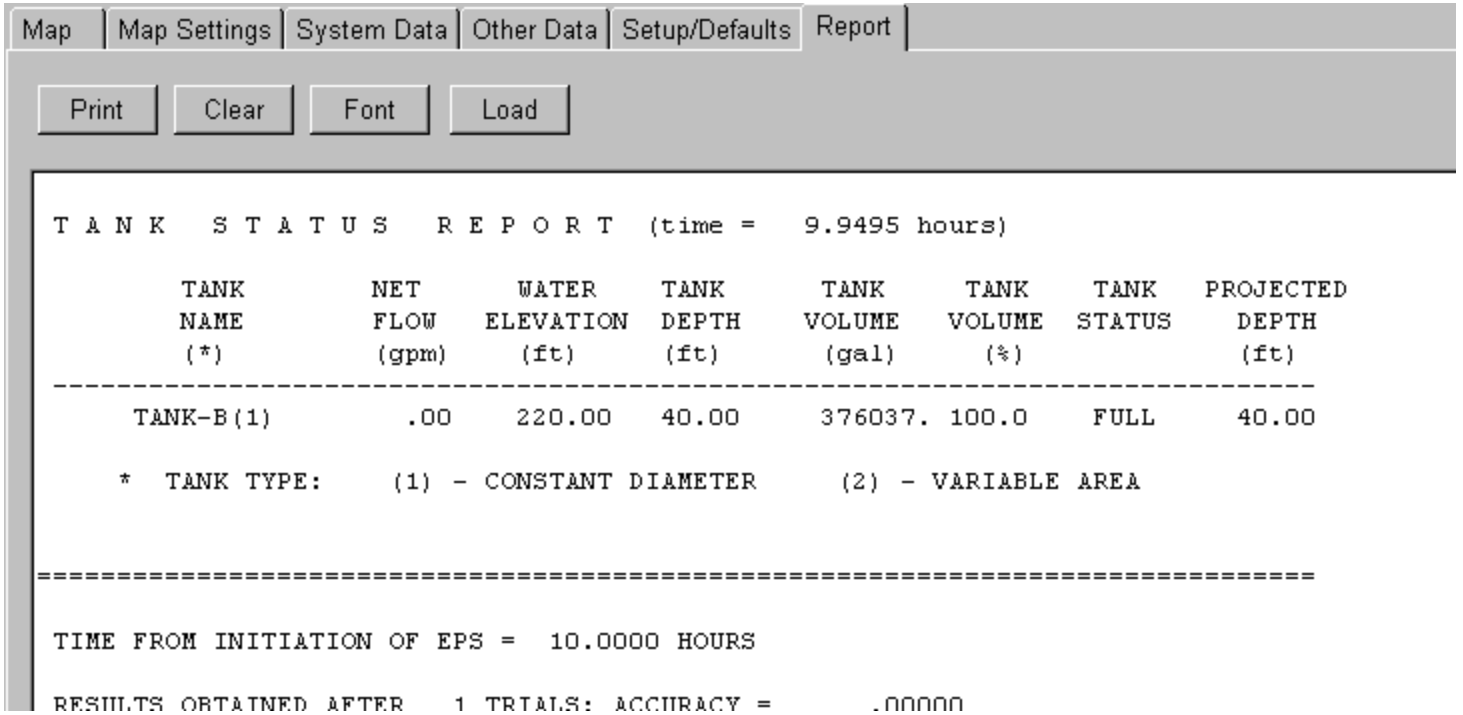
### Case1:

We will set up a 24-hour simulation where the demand remains constant and the tank levels change. We need EPS Data and Tank Data to accomplish this. The tank is a 40' diameter cylinder with the maximum, minimum, and initial levels as shown.  In Classic this is under System Data | EPS.


Extended Period Simulation (EPS)
Use EPS
Intermediate Reports
Total Time (hrs)
24
Computational Period (hrs)
1
Report Period (hrs)
1
Default Power Cost ($/kwhr)
0
Starting Time (hrs 0-24)
0
Report Time Style
Military Ti
Control Switches
Optiimal Pump Switching

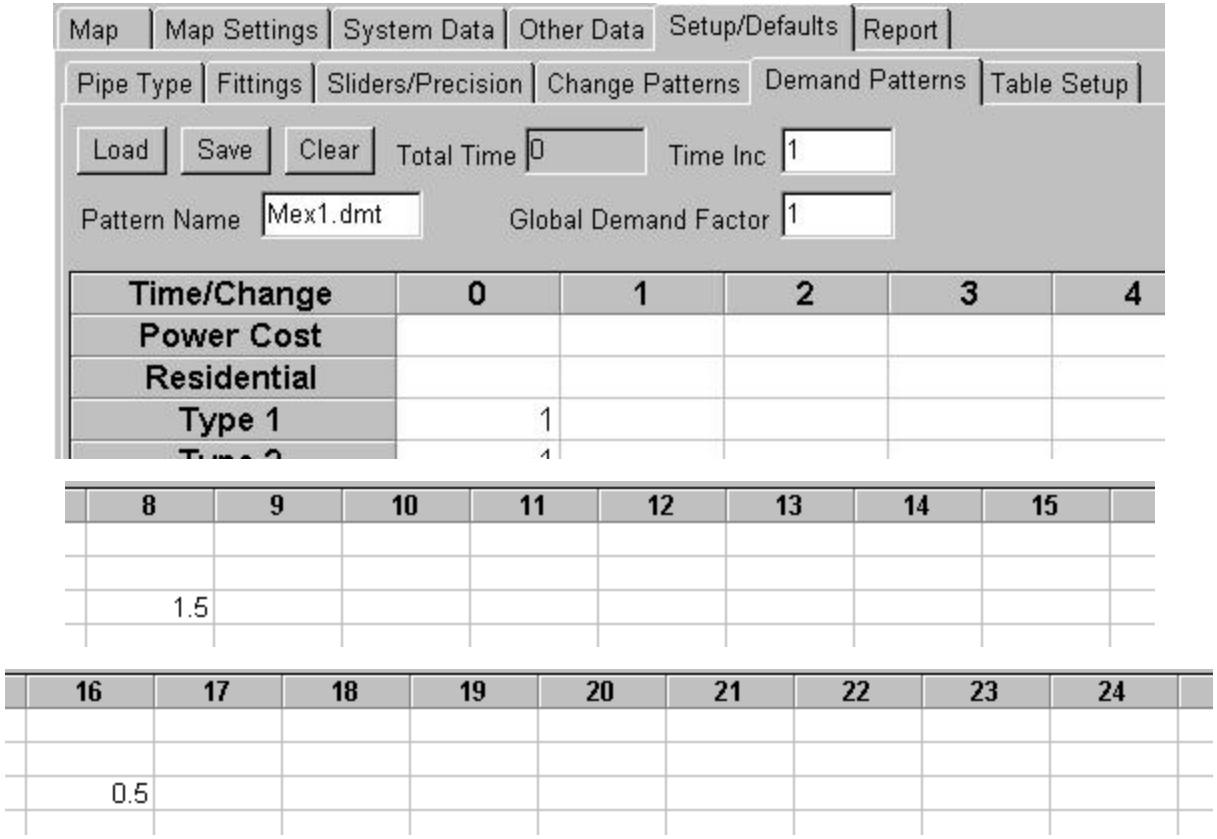



The analysis can now be carried out and the results studied (Report tab). This shows that the tank fills up in about 10 hours and remains full for the rest of the simulation.

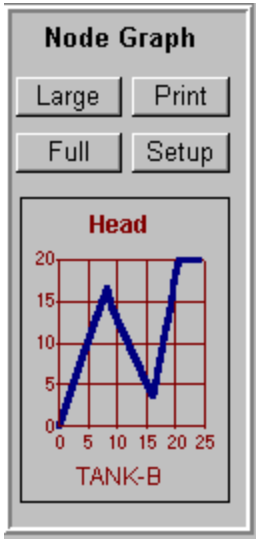



## Case 2:

We set up a Demand Pattern where the GDF (Global Demand Factor) = 1.0 for 8 hours, 1.5 for the next 8 hours, and 0.5 for the last 8 hours. This Demand Pattern set up is shown below:




The EPS data will remain set up as in Case 1. The analysis calculates the tank action in response to this Demand Pattern. This is shown in the tank head plot below:




## Case 3:

The pump is controlled by the tank level. It will come on when the level drops to 185' and go off when it exceeds 215'. A control switch is set as shown below to accomplish this.

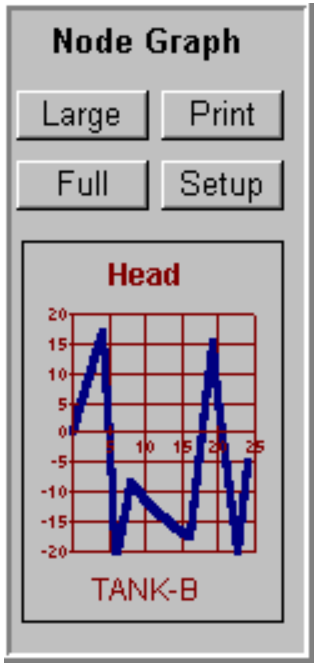
Map | Map Settings | System Data | Other Data | Setup/Defaults | Report

Control Switches | Constraints | Calibration | Quality | Meters

Switch Units: HGL (ft or m)

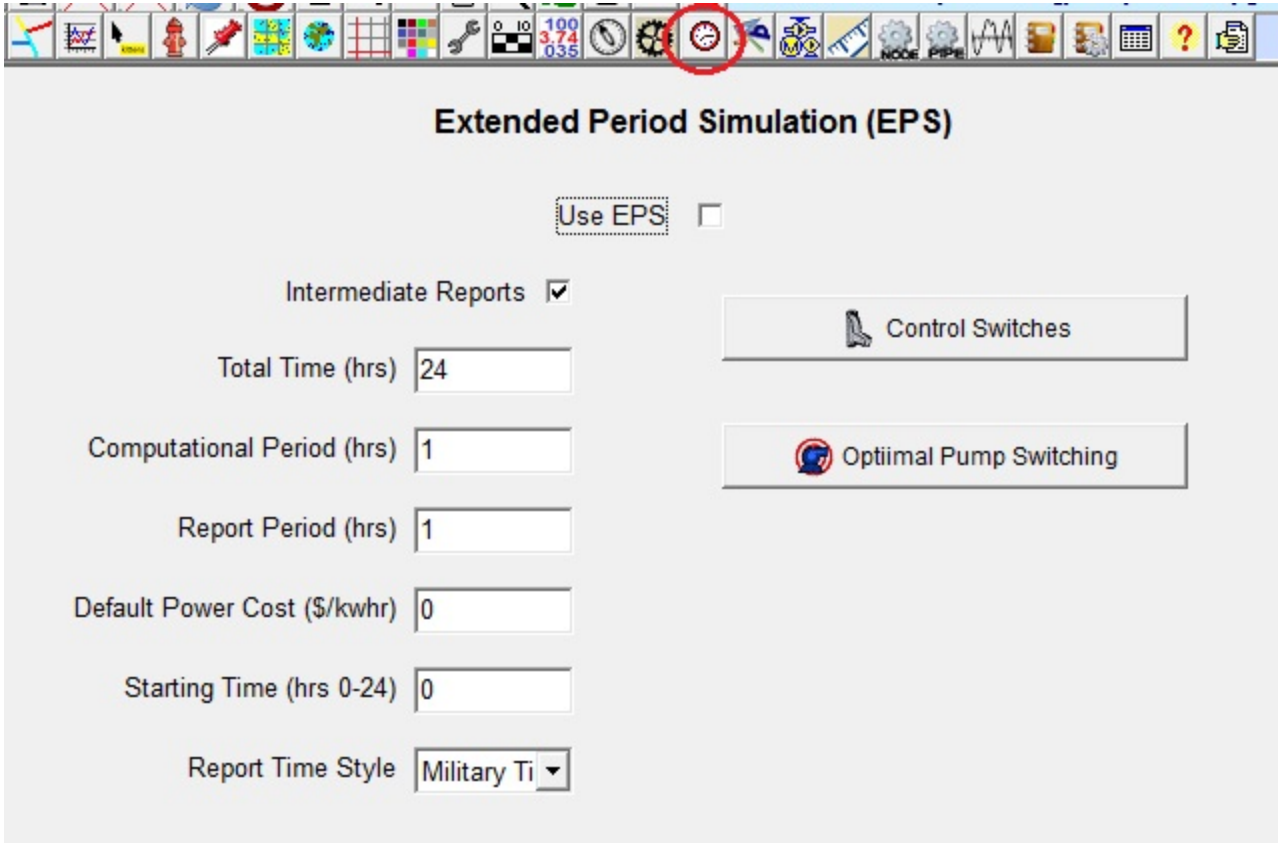
| controlled element | is | on/ off | when | sensing node | is | below/ between | low level | and | on/ off | when | above | high level |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| PUMP-1 | is | on | when | TANK-B | is | below | 185 | and | off | when | above | 215 |

Also, a diameter of the line to the tank has been increased to 12" since the 4" line is too small to supply the system when the pump is off. The tank head plot is shown below and depicts the effect of the pump action as a result of the control switch which produces several on/off cycles for the pump.




### 1.4.2.2 EPS (Extended Period Simulation)

EPS (Extended Period Simulation)




See also Extended Period Simulations (EPS) for examples and the EPS videos on the KYPipe web site.

See Demand Patterns for an example 24-hr demand pattern.

---

**This window sets the data that controls an extended period simulation. For more information on EPS please see the Reference Manual**

What is EPS?

Extended Period Simulation (EPS) refers to a hydraulic or water quality analysis carried out over a specified time period. Tank level variations will be calculated and control switches activated appropriately.

Use EPS

This check box determines if an extended period simulation will be performed.

Total Time

This is the total time (in hours) that the extended period simulation will cover (usually 24 hours).

Computational Period

This is the time period (in hours) between simulations (usually one hour).

Report Period

This is how often (in hours) reports should be generated during the EPS.

Default Power Cost

Sets the default power cost (in dollars per kilowatt hour) to perform cost analysis of pump operation.

Intermediate Reports

This check box determines if reports should be generated at intermediate events during the EPS simulation. For example if your Report Period is set to 1 hour and a tank were to empty at 1.5 hours this box being checked would result in a report being generated at 1.5 hours.

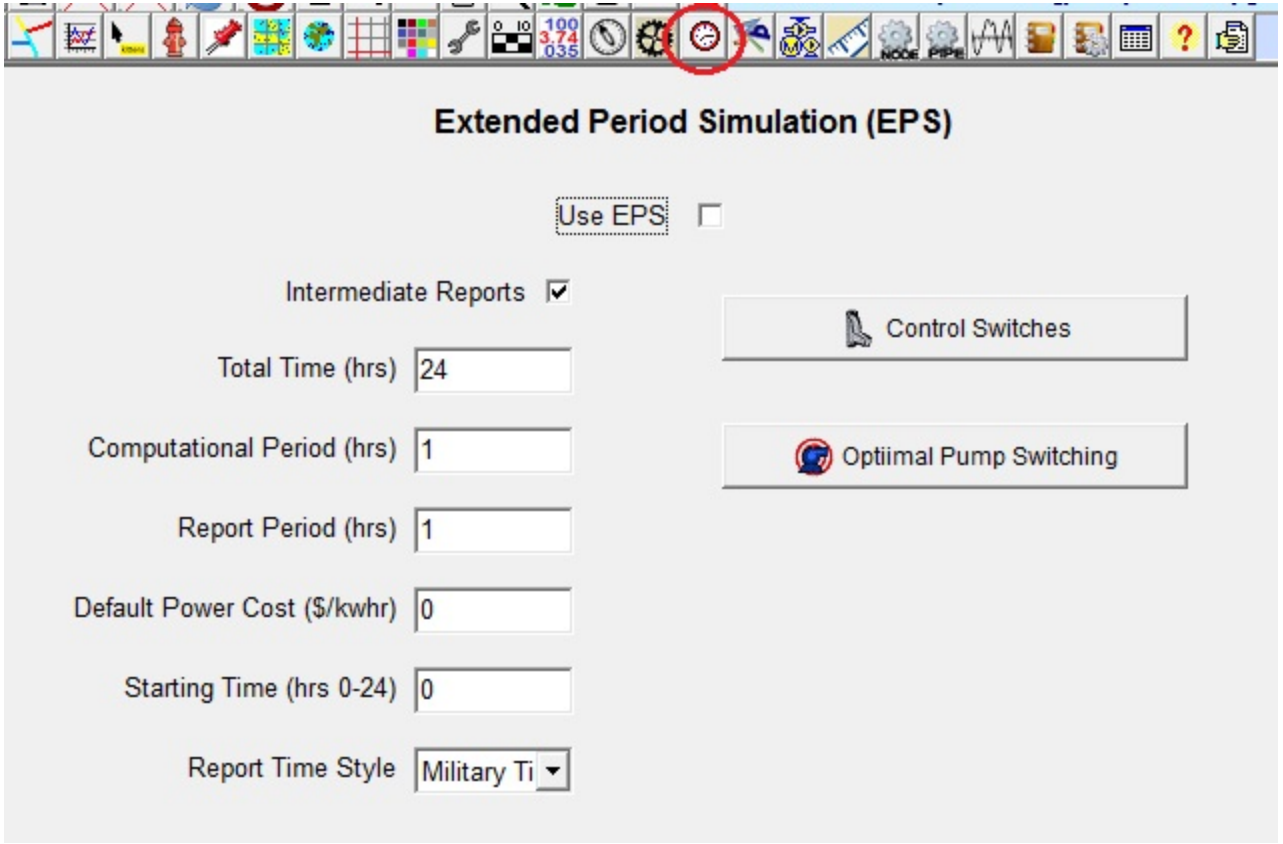
**Starting Time (hrs 0-24)**
Will note the specified start time in the Report results, next to the case number at the head of each results section.

**Report Time Style**
Will put the time, as noted in the Starting Time, in the selected style

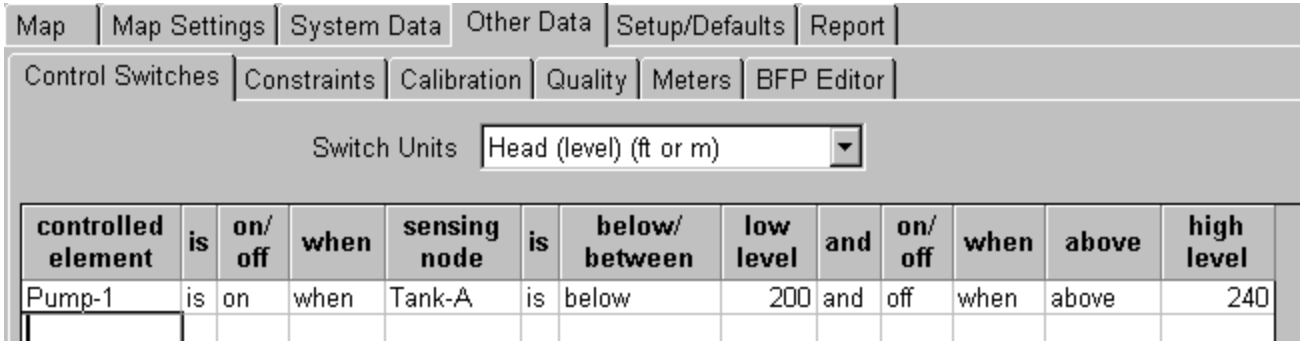
## 1.4.2.3 Control Switches Data

Control Switches Data

KYnetic:




Classic:




About Control/Pressure Switches

---

What is a Control Switch?

This modeling feature allows you to designate a switch to turn on / off a device (control element) or open (on) / close (off) a pipe based on the pressure or water level at some locations (sensing node) in the distribution system. This feature is usually applied for Extended Period Simulations but can be utilized for regular simulations.

A total range of capability is provided by accommodating four possible situations as illustrated by a pump controlled by a water level in a tank

1. The pump turns off when the level exceeds the high level and on when the level falls below the low level.
2. The pump turns on when the level exceeds the high level and off when the level falls below the low level.
3. The pump is on when the level is between the low level and the high level
4. The pump is off when the level is between the low level and the high level

For this example the pump is the controlled element and the tank is the sensing node. An interactive setup screen for control switches is provided in the Control Switches Tab.

Control Switch Data is accessed from the Other Data / Control Switches Tab. **Important:** You should first select the switch units from the choices of pressure head or HGL. For example, if you are using water level in a tank, you would normally select Head. Remember that Head is the water level referenced from the elevation of the sensing node (tank node). To set up a control switch provide the following input as shown in the above picture.

Switch Units

It is important for the user to choose the correct switch units used to define the switching value. The three choices are the pressure, head, and HGL units specified for the system

controlled element

This can be any pipe or one of the following end node types; pump, loss element, reservoir, sprinkler, or pressure supply

on/off

Select the appropriate choice

sensing node

This may be any node in the distribution system. Note however that the directional nodes have two sides and the name displayed refers to the upstream side (based on the directional indicator). You can choose the downstream side by inserting a tilde ~ before the name (~ Pump-1, for example is the opposite side of pump 1).

below / between

Select the appropriate choice for switching levels

1. low level - Enter low switch level in units selected above
2. high level - Enter high switch level in units selected above.

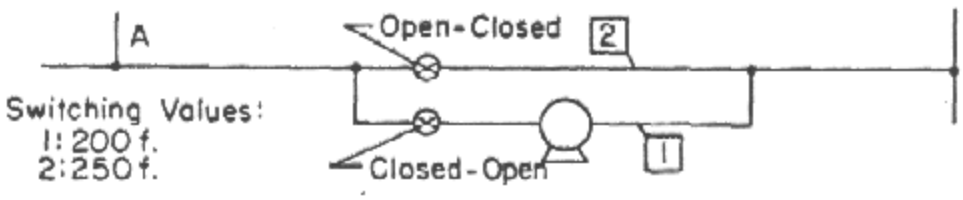
## 1.4.2.4 Pressure Switch

Pressure Switch

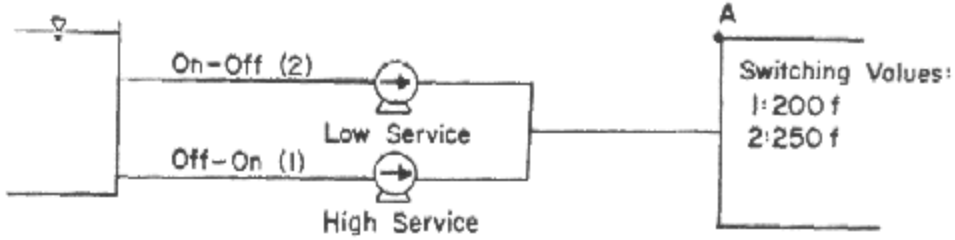
See Control Switches

---

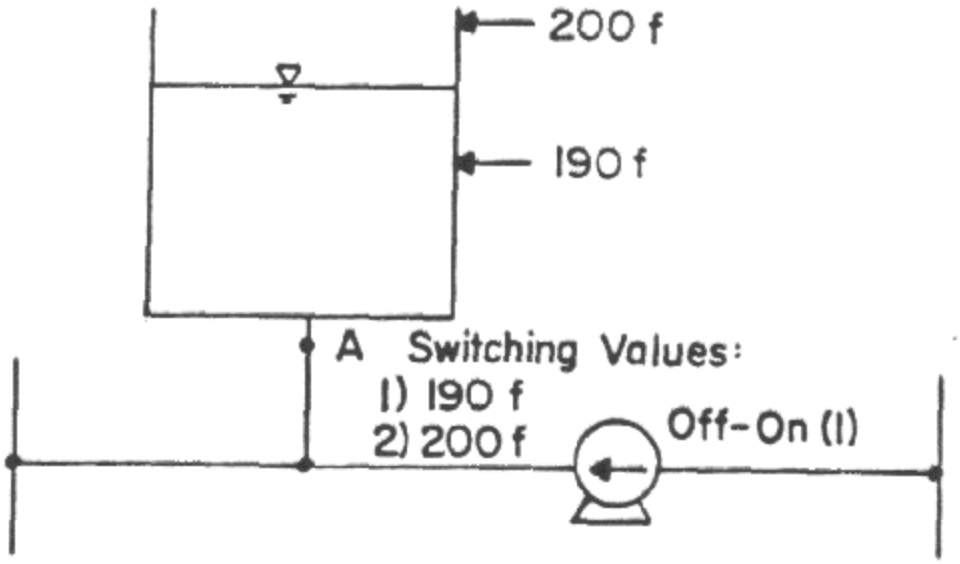
A feature provided for EPS applications is a pressure switch which allows the open-closed status of lines to be controlled by the HGL (elevation + pressure head) at a specified junction node. If the HGL at the specified junction node (reference node) goes through the defined switching value during an EPS, the open-closed status of the designated pipe (reference pipe) will change. When switching occurs a new value of HGL for the next switch can be designated or the same value can be employed. This feature can be used, for example, to turn on a booster pump if the pressure (or HGL) at some location falls below a specified value. This is depicted in Fig. below.




Line 2 with no pump is originally open and will stay open if the HGL at A is above the switching value (200 ft.). If the HGL at A falls below 200 ft., line 2 will close and line l will open bringing the booster pump on line. When this occurs the switching value is changed to 250 ft. so the booster pump will continue to run until the HGL at A reaches 250 ft. At that time, line 2 will open and line l will close and the switching value will change back to 200 ft. and the procedure continues. This feature can also be used to cycle pump operation by having pumps with different characteristics in parallel lines. This application is depicted below where the low service pump is switched off and the high service pump switched on if the HGL at A drops below 200 ft.




This status continues until the HGL exceeds 250 ft. where the high service pump is switched off and the low service on. A third application is depicted below where a pressure switch is used to control a pump to a storage tank based on the water level in the tank. A junction node located a short distance from the tank (A) will have a HGL nearly identical to the tank level elevation. In the case shown the booster pump which is originally off will come on if the water level drops to 190 ft. and will stay on until the water level reaches 200 ft.




## 1.4.3 Rural Analysis

### 1.4.3.1 Rural Water Systems

Rural Water Systems

See Rural Water Systems (Peak Demand Requirements)

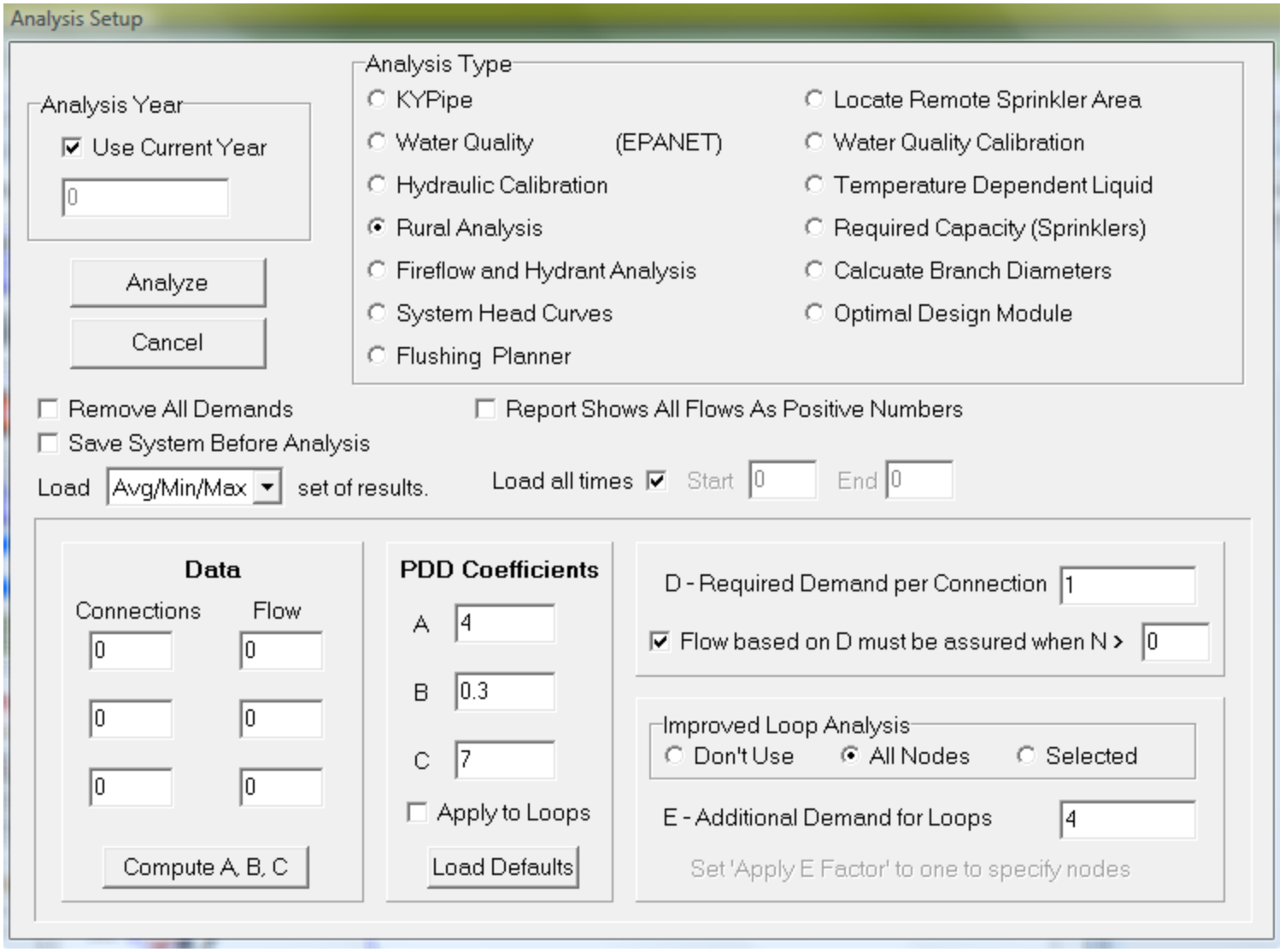
---

Pipe2018 Improved Loop Analysis options.

The rural analysis will be calculated as described in this topic. For loops, one pipe in each loop is specified to be excluded from the calculation to "break" the loop. All pipes in the model are subject to the Peak Demand requirements.

All Nodes - all the loops in the model will be analyzed using an unmodified hydraulic analysis and all branchlines will be subject to the Peak Demand Requirements. The "E" value will be applied to each node in the loop in turn to determine the most sensitive configuration.

Selected - The "E" value will be applied to only nodes selected using Group Mode or given a User attribute named "Apply E Factor" a value of "1".




**Using Pipe2018 for Branched Rural Water Systems**

## Overview

Pipe2018, with the KYPIPE4 module, includes the capability to carry out a special hydraulic analysis (**RURAL** Analysis) which addresses the special requirements of branched rural water systems. Two special requirements are incorporated into the hydraulic calculations:

1) A flow requirement for all branch lines is satisfied. The flow requirement is based on two factors.

   a) The total number of residential connections served by that line. A peak demand diversity curve is used to calculate the flow required to satisfy the peak domestic demand based on this number.

   b) The flow required to meet the total fixed demand requirements (livestock, etc.) served by that line.

2) The distribution system supplies must supply an amount equal to the sum of all the fixed demands plus a domestic flow based on the specified average 12-hour domestic demand (per connection) and the total number of domestic connections serviced by the distribution system. In addition, the sum of the fixed demands plus an average 12-hour domestic demand requirement must be delivered to all primary nodes supplying branched sections of the network. This requirement is also based on the total number of domestic connections served by all branch sections connecting that node.

The demand data is entered in two ways.

1) **domestic demands:** The number of domestic connections attached to each pipe is entered with the pipe data. Enter the number of residential meters (i.e., 15 residential meters are interpreted as 15 domestic connections attached to that pipe section).

2) **fixed demands:** These demands include livestock demands and are entered as normal demands at designated nodes and are handled in the normal manner.

## Domestic Flow Requirements

Based on the number of connections served by each pipe (Nc) the required domestic flow, Qd1 is calculated using a peak demand diversity relation of the form:

$$Q_D = A\sqrt{N_C} + B\,N_C + C$$

where the coefficients A, B and C have been determined using field data or other information.

The peak demand diversity (PDD) curve is based on probability considerations and recognizes that the domestic demand requirements can not be adequately described using a fixed average requirement for each connection. Design should be based on the expected peak domestic demand which depends on the number of domestic connections serviced by each pipe section. As the number of connections increase, the requirement per connection decreases because the probability of all users requiring maximum capacity simultaneously decreases. This is illustrated below in a table showing the calculations based on typical peak demand diversity curve coefficients.

Coefficients utilized for peak demand diversity curve:

A = 4 B = .3 C = 7

**Table 1 - Example Domestic Flow Requirements**

| No. Connections | Flow Requirements | RequirementsPer Connections |
|---|---|---|
| 1 | 11.3 | 11.3 |
| 2 | 13.3 | 6.7 |
| 3 | 14.8 | 4.9 |
| 4 | 16.2 | 4.1 |
| 5 | 17.4 | 3.5 |
| 10 | 22.6 | 2.3 |
| 20 | 30.9 | 1.5 |
| 30 | 37.9 | 1.3 |
| 40 | 44.3 | 1.1 |

| | | |
|---|---|---|
| 50 | 50.3 | 1.0 |
| 200 | 123.6 | .6 |
| 500 | 246.4 | .5 |
| 1000 | 433.5 | .4 |

In addition, an average domestic demand requirement of D gallons/minute per domestic connection must be specified for the distribution system. D is based on peak flow requirements and is applied to assure that adequate domestic flow is supplied to the overall system and to primary node serving each branch section of the distribution system.

## Pre-processing Module

If the user indicates that they wish to run a **RURAL** hydraulic analysis, then a preprocessing module does the following:

1) Locate all branch lines and calculate the total number of domestic connections serviced by each branch line and the associated peak domestic demand requirement (using the peak demand diversity curve) for each branch line.

2) Locate the primary node for a branch section and calculate the total number of domestic connections serviced through that node . Based on this number, calculate a domestic demand requirement for that node using the average domestic demand requirement (D).

3) Distribute domestic demand requirements for all non-branched pipes equally at the end nodes using the average domestic demand requirement (D).

The preprocessing module creates a domestic demand vector for junction nodes which assures that these requirements are met for each branch line. This vector is passed to and utilized by the KYPIPE4 program to allow the hydraulic analysis to made while meeting these requirements.

## KYPIPE4 - RURAL Analysis

The KYPIPE4 uses the domestic demand vector generated by the preprocessor to meet the following requirements:

a) Within non-branched regions all domestic connections are distributed equally to the end nodes and a domestic demand is imposed based on the average domestic flow requirement D.

b) At all primary nodes in a distribution system where a branched section starts, it is required that a domestic flow equal to $N_C * D$ is delivered to that connection. Nc represents the total number of domestic connections in that branch section of the distribution system. The domestic flow is in addition to the flow required to satisfy all fixed demands in the branched section.

c) Each branch line will be analyzed using a flowrate which includes a domestic flow requirement given by the peak demand diversity curve and based on the number of domestic connections serviced by that pipe. An additional flow based on all fixed demands serviced by that pipe will be included.

d) The sum of the inflow from all supplies will equal the sum of all the fixed demands plus a domestic flow requirement equal to Nc * D where Nc is the total number of domestic connections in the distribution system.

## Running the RURAL Analysis

The steps for running the modified programs are as follows:

1) Prepare the KYPIPE data file normally with the following guidelines:

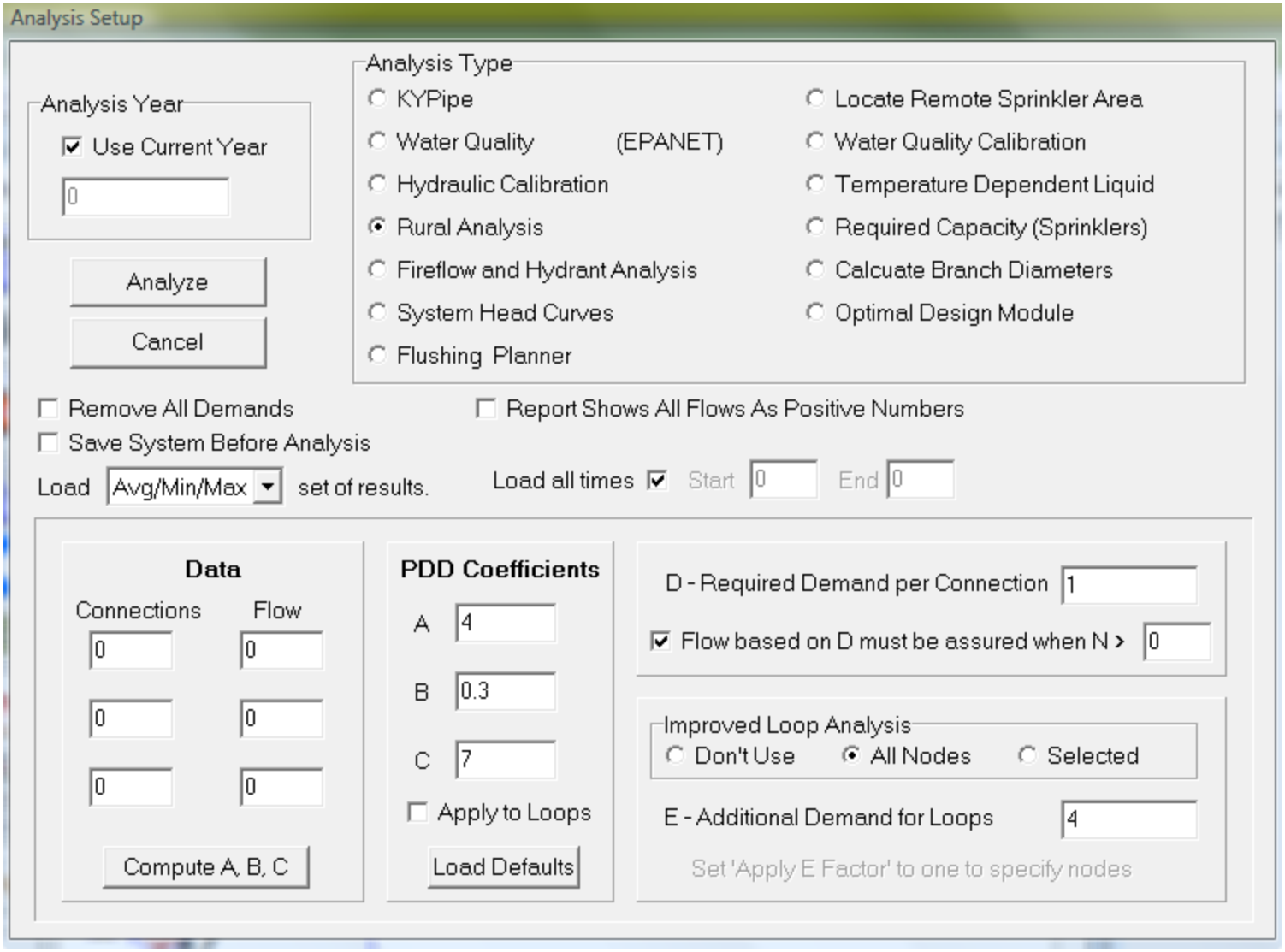
a) All fixed demands are entered at nodes.

b) Domestic connections in branched lines are entered by inputting the appropriate number of residential connections for each pipe section.

2) Click on **Analyze** and **Analyze** and select **Rural Analysis**. The screen below appears. You may use the coefficients displayed, or if you wish to override the default coefficients, provide the following data.

A, B, C - coefficients for the instantaneous peak demand flow curve

D - average domestic demand requirement (average peak flow per connection)




A calculator is provided to determine A, B, and C coefficients based on number of connections, N, and total flow, Q. Q is the flow requirement for N connections. A set of data recommended by the Health Department of Mississippi is as follows:

| N | Q (gpm) |
|---|---|
| 1 | 12.3 |
| 50 | 67.4 |
| 100 | 101 |

**“Flow based on D must be assured when N >“** – when this box is checked and a value for N is entered, the Q (flow) is equal to D whenever N (number of connections) exceeds that value.

**Apply to Loops** applies the allocation process to loops as well as branched lines.

3) Run the special hydraulic analysis (click on **Analyze**).

4) After the analysis is completed, you are returned to the Pipe2018 screen and can view and print tabular and graphical results normally.

## Special Considerations

If the distribution system contains parallel pipes or other loops in rural areas, it may be necessary to utilize a special feature to obtain the desired result. Branched sections are determined by starting at all dead ends and working back through the connections until either a loop or otherwise non-branching pipe sections are encountered. In this manner, all the branched sections are determined and the remaining portion of the distribution system is treated as non-branched. The presence of loops in the distribution system may result in a section which you wish to be handled as branched being identified as non-branched. Figure 1 illustrates several configurations which may cause this situation.

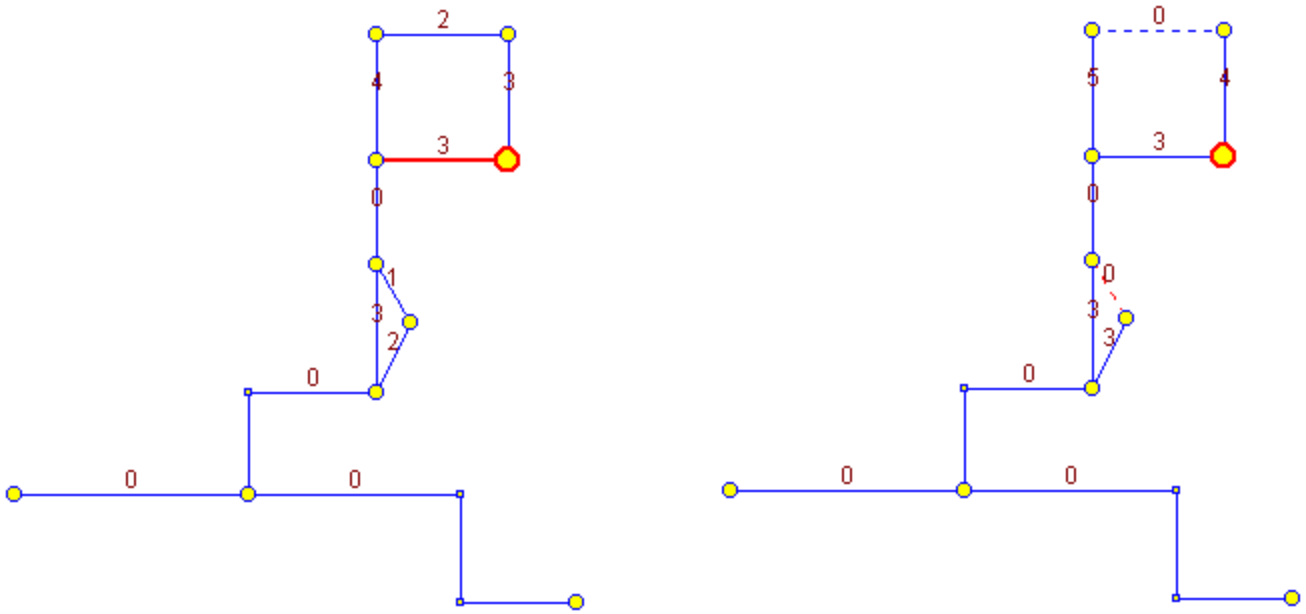



**Figure 1 Configuration for Excluded Pipes**

In order to handle loops in branched sections, selected pipes can be designated as using Calibration Group = 9 (User Data). These pipes will then be ignored (excluded pipes) when locating branch sections. If this is done, it is recommended the domestic connections be allocated to the other lines so that these requirements will be included in the overall branch line calculations. When the hydraulic analysis is then carried out, the flow is distributed hydraulically in the parallel pipes or loops while maintaining the total flow required by the peak demand diversity curve.

## Example Applications

Two example applications are presented. The first represents a branched rural system, while the second represents a combined municipal and rural system. In order to illustrate the effect of applying the special capabilities for analyzing branch lines, the examples are solved using the **RURAL Analysis** feature and using

conventional analysis where domestic requirements are based only on the average domestic demand per connection.

**Example 1 - Branched Rural Water System**

A simple example network is shown in Figure 2. This is meant to represent a typical Midwest rural water system. The system is supplied from a reservoir and pumping station (lower left) and has an elevated tank (center) and a booster pump (right). The labeling scheme is shown in Figure 3. Figure 4 is a schematic which shows the number of domestic connections for each pipe section and the fixed (livestock) demands considered. A small community with 40 domestic connections is served by one of the pipe links. The pipe lengths, diameters, roughness, elevations and other significant data is tabulated in the data summary which is provided (Table 2).

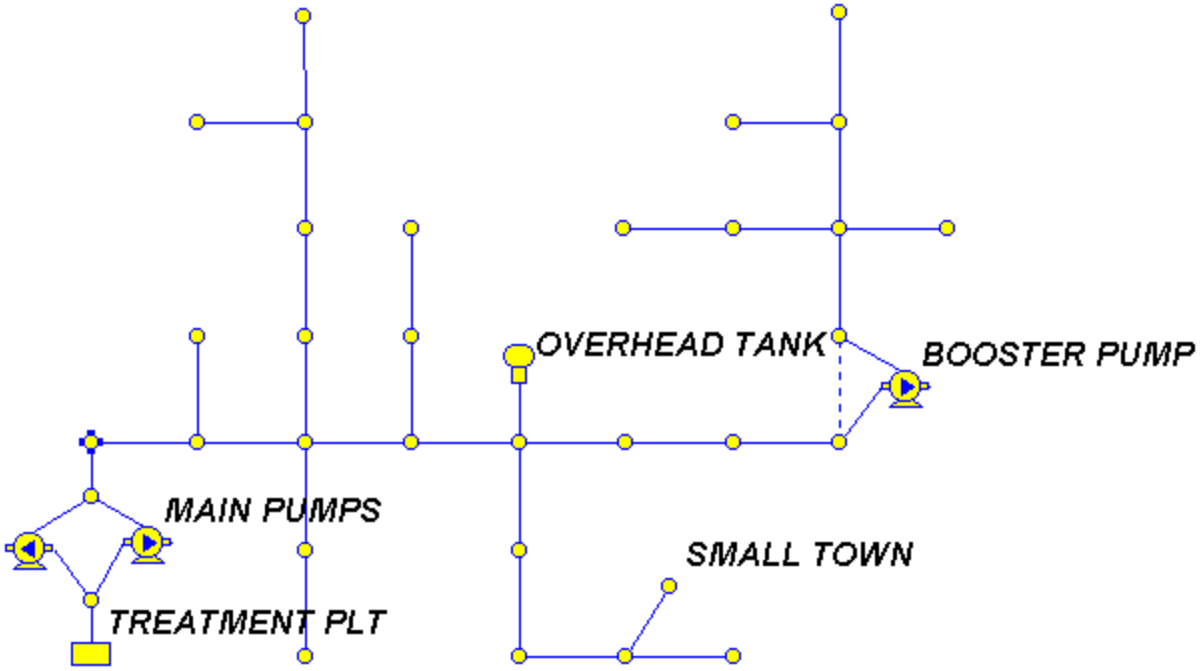



**Figure 2 Schematic for Example1**

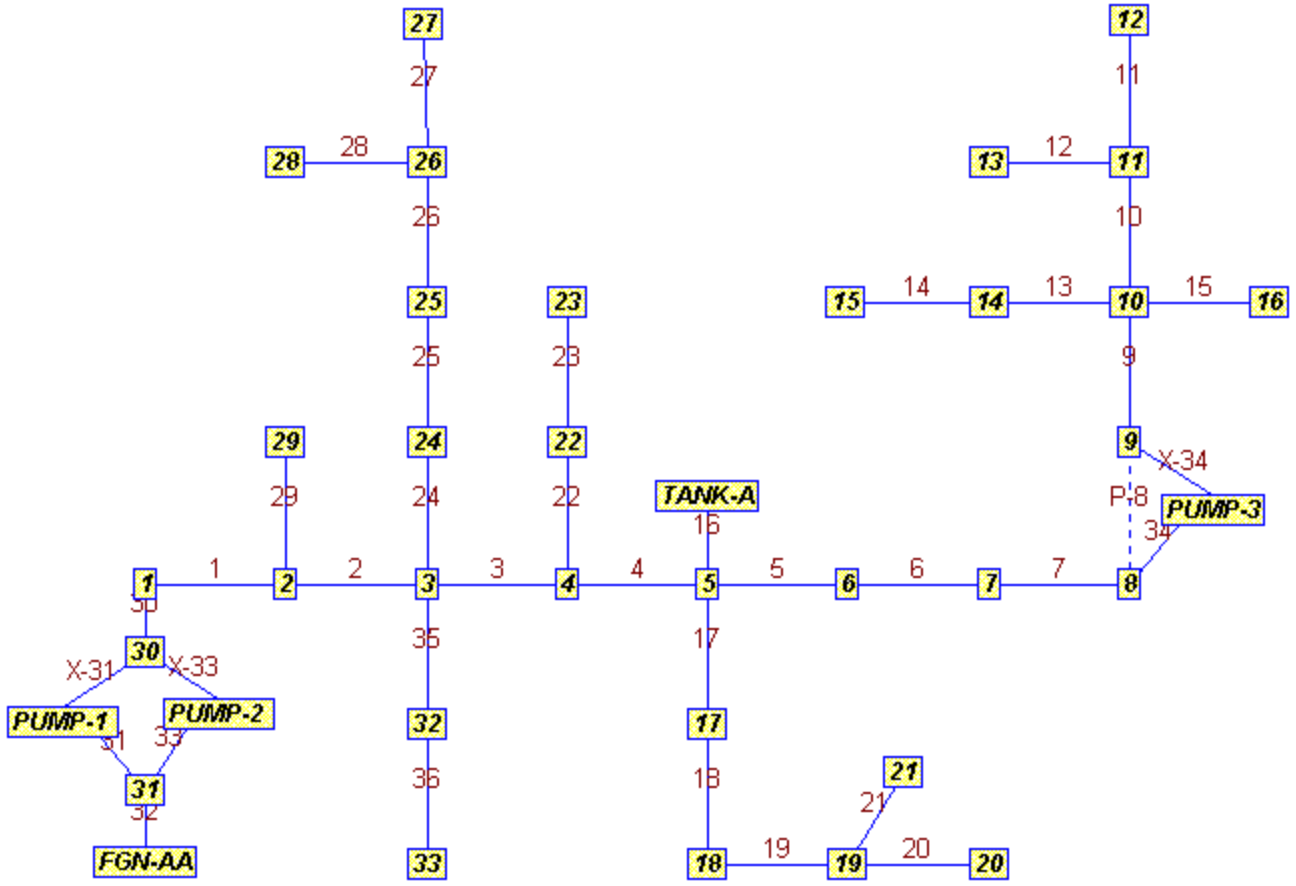



**Figure 3 Node and Pipe Labeling**

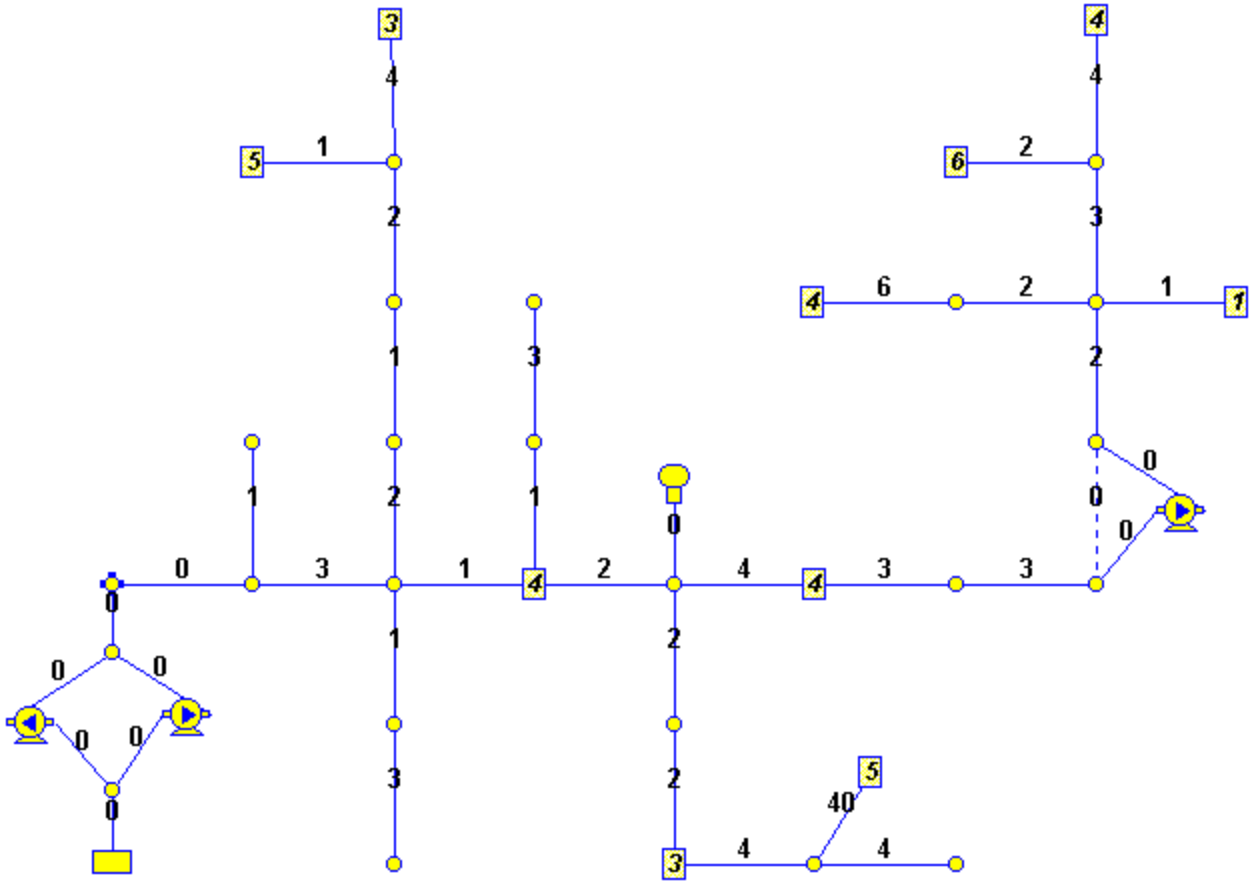



**Figure 4 Domestic Connections and Fixed Demands - Example 1**

**Table 2 Data Summary for Example 1 (KYRURAL)**

```
***********************************************
 S U M M A R Y   O F   O R I G I N A L   D A T A
***********************************************


U N I T S   S P E C I F I E D

   FLOWRATE ............ = gallons/minute
   HEAD (HGL) .......... = feet
   PRESSURE ............ = psig


P I P E L I N E   D A T A

  STATUS CODE:   XX -CLOSED PIPE    CV -CHECK VALVE
```

| PIPE NAME | NODE NAMES #1 | NODE NAMES #2 | LENGTH (ft) | DIAMETER (in) | ROUGHNESS COEFF. | MINOR LOSS COEFF. |
|---|---|---|---|---|---|---|
| 1 | | | 5300.00 | 4.00 | 150.0000 | .00 |
| 2 | | | 5300.00 | 4.00 | 150.0000 | .00 |
| 3 | | | 5300.00 | 4.00 | 150.0000 | .00 |
| 4 | | | 5300.00 | 4.00 | 150.0000 | .00 |
| 5 | | | 5300.00 | 4.00 | 150.0000 | .00 |
| 6 | | | 5300.00 | 4.00 | 150.0000 | .00 |
| 7 | | | 5300.00 | 4.00 | 150.0000 | .00 |
| 8-XX | | | 10.00 | 2.00 | 150.0000 | .00 |
| 9 | | | 5300.00 | 3.00 | 150.0000 | .00 |
| 10 | | | 5300.00 | 3.00 | 150.0000 | .00 |
| 11 | | | 5300.00 | 2.00 | 150.0000 | .00 |
| 12 | | | 5300.00 | 2.00 | 150.0000 | .00 |

| | | | | | | |
|---|---|---|---|---|---|---|
| 13 | | | 5300.00 | 2.00 | 150.0000 | .00 |
| 14 | | | 5300.00 | 2.00 | 150.0000 | .00 |
| 15 | | | 5300.00 | 2.00 | 150.0000 | .00 |
| 16 | | NK-A | 100.00 | 6.00 | 150.0000 | .00 |
| 17 | | | 5300.00 | 4.00 | 150.0000 | .00 |
| 18 | | | 5300.00 | 4.00 | 150.0000 | .00 |
| 19 | | | 5300.00 | 4.00 | 150.0000 | .00 |
| 20 | | | 5300.00 | 2.00 | 150.0000 | .00 |
| 21 | | | 2000.00 | 4.00 | 150.0000 | .00 |
| 22 | | | 5300.00 | 2.00 | 150.0000 | .00 |
| 23 | | | 5300.00 | 2.00 | 150.0000 | .00 |
| 24 | | | 5300.00 | 2.50 | 150.0000 | .00 |
| 25 | | | 5300.00 | 2.50 | 150.0000 | .00 |
| 26 | | | 5300.00 | 2.50 | 150.0000 | .00 |
| 27 | | | 5300.00 | 2.00 | 150.0000 | .00 |
| 28 | | | 5300.00 | 2.00 | 150.0000 | .00 |
| 29 | | | 5300.00 | 2.00 | 150.0000 | .00 |
| 30 | | | 10.00 | 2.00 | 150.0000 | 15.00 |
| 31 | | MP-1 | 5.00 | 2.00 | 150.0000 | .00 |
| 32 | N-AA | | 10.00 | 6.00 | 150.0000 | .00 |
| 33 | | MP-2 | 5.00 | 2.00 | 150.0000 | .00 |
| 34 | | MP-3 | 5.00 | 2.00 | 150.0000 | 10.00 |
| 35 | | | 5300.00 | 2.00 | 150.0000 | .00 |
| 36 | | | 5300.00 | 2.00 | 150.0000 | .00 |
| X-31 | MP-1 | | 5.00 | 2.00 | 150.0000 | .00 |
| X-33 | MP-2 | | 5.00 | 2.00 | 150.0000 | .00 |
| X-34 | MP-3 | | 5.00 | 2.00 | 150.0000 | 10.00 |

P U M P/L O S S  E L E M E N T  D A T A

THERE IS A DEVICE AT NODE PUMP-1 DESCRIBED BY THE FOLLOWING DATA: (ID= 1)

| HEAD (ft) | FLOWRATE (gpm) | EFFICIENCY (%) |
|---|---|---|
| 400.00 | .00 | 75.00 (Default) |
| 350.00 | 50.00 | 75.00 (Default) |
| 200.00 | 100.00 | 75.00 (Default) |

THERE IS A DEVICE AT NODE PUMP-2 DESCRIBED BY THE FOLLOWING DATA: (ID= 1)

| HEAD (ft) | FLOWRATE (gpm) | EFFICIENCY (%) |
|---|---|---|
| 400.00 | .00 | 75.00 (Default) |
| 350.00 | 50.00 | 75.00 (Default) |
| 200.00 | 100.00 | 75.00 (Default) |

THERE IS A DEVICE AT NODE PUMP-3 DESCRIBED BY THE FOLLOWING DATA: (ID= 2)

| HEAD (ft) | FLOWRATE (gpm) | EFFICIENCY (%) |
|---|---|---|
| 200.00 | .00 | 75.00 (Default) |
| 180.00 | 40.00 | 75.00 (Default) |
| 100.00 | 75.00 | 75.00 (Default) |

E N D  N O D E  D A T A

| NODE NAME | NODE TITLE | EXTERNAL DEMAND (gpm) | JUNCTION ELEVATION (ft) | EXTERNAL GRADE (ft) |
|---|---|---|---|---|
| 1 | | .00 | 400.00 | |
| 2 | | .00 | 430.00 | |
| 3 | | .00 | 460.00 | |
| 4 | | 4.00 | 480.00 | |
| 5 | | .00 | 500.00 | |
| 6 | | 4.00 | 490.00 | |
| 7 | | .00 | 470.00 | |
| 8 | | .00 | 470.00 | |
| 9 | | .00 | 470.00 | |
| 10 | | .00 | 520.00 | |
| 11 | | .00 | 550.00 | |
| 12 | | 4.00 | 560.00 | |
| 13 | | 6.00 | 560.00 | |
| 14 | | .00 | 540.00 | |
| 15 | | 4.00 | 550.00 | |
| 16 | | 1.00 | 490.00 | |
| 17 | | .00 | 460.00 | |
| 18 | | 3.00 | 430.00 | |
| 19 | | .00 | 420.00 | |
| 20 | | .00 | 420.00 | |
| 21 | | 5.00 | 430.00 | |
| 22 | | .00 | 480.00 | |
| 23 | | .00 | 470.00 | |
| 24 | | .00 | 460.00 | |
| 25 | | .00 | 450.00 | |
| 26 | | .00 | 440.00 | |

| | | | |
|---|---|---|---|
| 27 | 3.00 | 440.00 | |
| 28 | 5.00 | 420.00 | |
| 29 | .00 | 430.00 | |
| 30 | .00 | 400.00 | |
| 31 | .00 | 400.00 | |
| 32 | .00 | 450.00 | |
| 33 | .00 | 420.00 | |
| FGN-AA | ---- | 400.00 | 420.00 |
| PUMP-1 | .00 | 400.00 | |
| PUMP-2 | .00 | 400.00 | |
| PUMP-3 | .00 | 470.00 | |
| TANK-A | ---- | 500.00 | 620.00 |

a) Using **RURAL Analysis** to Calculate Domestic Flow Requirements

The data input is prepared normally with the exception that the connection data is entered as number of residential meters. The RURAL Analysis is selected and the preprocessor is first executed to analyze the number of connections served for each branch line and the resulting domestic flow requirement. The preprocessor output is shown in Table 3 along with the coefficients used for Peak Demand Diversity Curve and the average domestic demand (1 g.p.m.). A summary of the calculations for the branch lines is shown.

**Table 3 Connections and Domestic Flow Requirements - Example 1 (KYRURAL)**

```
          DOMESTIC DEMAND ALLOCATION FOR RURAL WATER SYSTEMS

        Preprocessor for Pipe2000 Rural Water Systems Analysis

                        copyright June 1999
                          by Don J. Wood


   The Current Data File Is C:\PROGRA~1\PIPE2000\HELPST~1\RWS1.DT2
 COEFFICIENTS OF DEMAND DIVERSITY CURVE:
   A =  4
   B =  0.3
   C =  7

AVERAGE DOMESTIC DEMAND REQUIREMENT (gpm) =  1


***** SUMMARY OF BRANCH LINE DOMESTIC FLOW REQUIREMENTS *****
```

| PIPE NO. | NODE #1 | NODE #2 | No. Connections Served | Required Domestic Flow |
|---|---|---|---|---|
| 5 | 5 | 6 | 30 | 37.9 |
| 6 | 6 | 7 | 26 | 35.1 |
| 7 | 7 | 8 | 23 | 33 |
| 9 | 9 | 10 | 20 | 30.8 |
| 10 | 10 | 11 | 9 | 21.7 |
| 11 | 11 | 12 | 4 | 16.2 |
| 12 | 11 | 13 | 2 | 13.2 |
| 13 | 10 | 14 | 8 | 20.7 |
| 14 | 14 | 15 | 6 | 18.5 |
| 15 | 10 | 16 | 1 | 11.3 |
| 17 | 5 | 17 | 52 | 51.4 |
| 18 | 17 | 18 | 50 | 50.2 |
| 19 | 18 | 19 | 48 | 49.1 |
| 20 | 19 | 20 | 4 | 16.2 |
| 21 | 19 | 21 | 40 | 44.2 |
| 22 | 4 | 22 | 4 | 16.2 |
| 23 | 22 | 23 | 3 | 14.8 |
| 24 | 3 | 24 | 10 | 22.6 |
| 25 | 24 | 25 | 8 | 20.7 |
| 26 | 25 | 26 | 7 | 19.6 |
| 27 | 26 | 27 | 4 | 16.2 |
| 28 | 26 | 28 | 1 | 11.3 |
| 29 | 2 | 29 | 1 | 11.3 |
| 34 | 8 | 42 | 20 | 30.8 |
| 35 | 3 | 32 | 4 | 16.2 |
| 36 | 32 | 33 | 3 | 14.8 |
| 39 | 37 | 9 | 20 | 30.8 |
| 43 | 42 | 37 | 20 | 30.8 |

The **Hydraulic Analysis** was then executed. The resulting pressures and flows are shown in Figure 5 along with the tabulated output (Table 4).

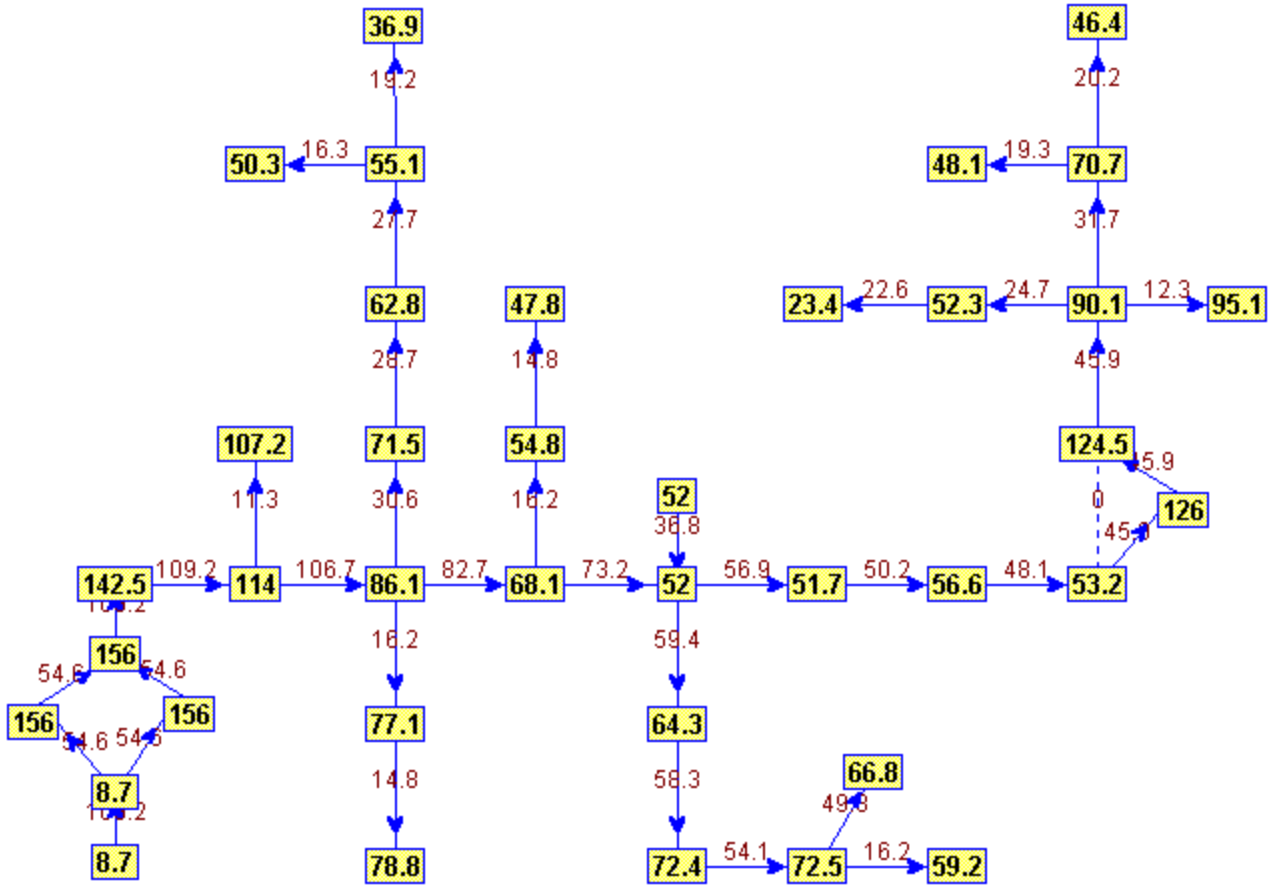



**Figure 5 Plot of Flowrates and Pressures - Example 1a**

**Table 4 Tabulated Results - Example 1a (KYRURAL)**

S I M U L A T I O N  D E S C R I P T I O N  (L A B E L)

Wood Rural Water System
This is a small sample system to demonstrate rural
water analysis.
A=4, B=0.3+0.5, C=7, D=0.8  "B includes domestic
and avg. livestock"

P I P E L I N E  R E S U L T S

STATUS CODE:  XX -CLOSED PIPE  CV -CHECK VALVE

| PIPE NAME | NODE NUMBERS #1 | #2 | FLOWRATE (gpm) | HEAD LOSS (ft) | MINOR LOSS (ft) | LINE VELO. (ft/s) | HL/ 1000 (ft/ft) |
|---|---|---|---|---|---|---|---|
| 1 | 1 | 2 | 109.17 | 35.93 | .00 | 2.79 | 6.78 |
| 2 | 2 | 3 | 106.67 | 34.42 | .00 | 2.72 | 6.49 |
| 3 | 3 | 4 | 82.67 | 21.47 | .00 | 2.11 | 4.05 |
| 4 | 4 | 5 | 73.17 | 17.13 | .00 | 1.87 | 3.23 |
| 5 | 5 | 6 | 56.90 | 10.75 | .00 | 1.45 | 2.03 |
| 6 | 6 | 7 | 50.19 | 8.52 | .00 | 1.28 | 1.61 |
| 7 | 7 | 8 | 48.08 | 7.87 | .00 | 1.23 | 1.48 |
| 8-XX | 8 | 9 | | | | | |
| 9 | 9 | 10 | 45.89 | 29.29 | .00 | 2.08 | 5.53 |
| 10 | 10 | 11 | 31.70 | 14.77 | .00 | 1.44 | 2.79 |
| 11 | 11 | 12 | 20.20 | 46.18 | .00 | 2.06 | 8.71 |
| 12 | 11 | 13 | 19.26 | 42.26 | .00 | 1.97 | 7.97 |
| 13 | 10 | 14 | 24.71 | 67.08 | .00 | 2.52 | 12.66 |
| 14 | 14 | 15 | 22.60 | 56.84 | .00 | 2.31 | 10.72 |
| 15 | 10 | 16 | 12.30 | 18.42 | .00 | 1.26 | 3.48 |

| | | | | | | | |
|---|---|---|---|---|---|---|---|
| 16 | 5 | TANK-A | -36.82 | .01 | .00 | .42 | .13 |
| 17 | 5 | 17 | 59.44 | 11.66 | .00 | 1.52 | 2.20 |
| 18 | 17 | 18 | 58.28 | 11.24 | .00 | 1.49 | 2.12 |
| 19 | 18 | 19 | 54.11 | 9.79 | .00 | 1.38 | 1.85 |
| 20 | 19 | 20 | 16.20 | 30.68 | .00 | 1.65 | 5.79 |
| 21 | 19 | 21 | 49.30 | 3.11 | .00 | 1.26 | 1.56 |
| 22 | 4 | 22 | 16.20 | 30.68 | .00 | 1.65 | 5.79 |
| 23 | 22 | 23 | 14.83 | 26.05 | .00 | 1.51 | 4.91 |
| 24 | 3 | 24 | 30.65 | 33.71 | .00 | 2.00 | 6.36 |
| 25 | 24 | 25 | 28.71 | 29.87 | .00 | 1.88 | 5.64 |
| 26 | 25 | 26 | 27.68 | 27.92 | .00 | 1.81 | 5.27 |
| 27 | 26 | 27 | 19.20 | 42.03 | .00 | 1.96 | 7.93 |
| 28 | 26 | 28 | 16.30 | 31.03 | .00 | 1.66 | 5.86 |
| 29 | 2 | 29 | 11.30 | 15.74 | .00 | 1.15 | 2.97 |
| 30 | 30 | 1 | 109.17 | 1.98 | 28.94 | 11.15 | 3092.48 |
| 31 | 31 | PUMP-1 | 54.58 | .27 | .00 | 5.57 | 54.91 |
| 32 | FGN-AA | 31 | 109.17 | .01 | .00 | 1.24 | .94 |
| 33 | 31 | PUMP-2 | 54.58 | .27 | .00 | 5.57 | 54.91 |
| 34 | 8 | PUMP-3 | 45.88 | .20 | 3.41 | 4.69 | 721.58 |
| 35 | 3 | 32 | 16.20 | 30.68 | .00 | 1.65 | 5.79 |
| 36 | 32 | 33 | 14.83 | 26.05 | .00 | 1.51 | 4.91 |
| X-31 | PUMP-1 | 30 | 54.58 | .27 | .00 | 5.57 | 54.91 |
| X-33 | PUMP-2 | 30 | 54.58 | .27 | .00 | 5.57 | 54.91 |
| X-34 | PUMP-3 | 9 | 45.88 | .20 | 3.41 | 4.69 | 721.58 |

P U M P/L O S S  E L E M E N T  R E S U L T S

| NAME | FLOWRATE (gpm) | INLET HEAD (ft) | OUTLET HEAD (ft) | HEAD CHANGE (ft) | EFFIC-ENCY (%) | USEFUL POWER (Hp) | INCREMTL COST ($) | TOTAL COST ($) |
|---|---|---|---|---|---|---|---|---|
| PUMP-1 | 54.58 | 19.72 | 360.13 | 340.4 | ---- | ----- | --- | ---- |
| PUMP-2 | 54.58 | 19.72 | 360.13 | 340.4 | ---- | ----- | --- | ---- |
| PUMP-3 | 45.88 | 119.24 | 290.82 | 171.6 | ---- | ----- | --- | ---- |

E N D  N O D E  R E S U L T S

| NODE NAME | NODE TITLE | EXTERNAL DEMAND (gpm) | HYDRAULIC GRADE (ft) | NODE ELEVATION (ft) | PRESSURE HEAD (ft) | NODE PRESSURE (psi) |
|---|---|---|---|---|---|---|
| 1 | | .00 | 728.93 | 400.00 | 328.93 | 142.54 |
| 2 | | .00 | 693.00 | 430.00 | 263.00 | 113.97 |
| 3 | | .00 | 658.58 | 460.00 | 198.58 | 86.05 |
| 4 | | 4.00 | 637.11 | 480.00 | 157.11 | 68.08 |
| 5 | | .00 | 619.99 | 500.00 | 119.99 | 51.99 |
| 6 | | 4.00 | 609.24 | 490.00 | 119.24 | 51.67 |
| 7 | | .00 | 600.72 | 470.00 | 130.72 | 56.64 |
| 8 | | .00 | 592.85 | 470.00 | 122.85 | 53.23 |
| 9 | | .00 | 757.21 | 470.00 | 287.21 | 124.46 |
| 10 | | .00 | 727.92 | 520.00 | 207.92 | 90.10 |
| 11 | | .00 | 713.15 | 550.00 | 163.15 | 70.70 |
| 12 | | 4.00 | 666.98 | 560.00 | 106.98 | 46.36 |
| 13 | | 6.00 | 670.89 | 560.00 | 110.89 | 48.05 |

| | | | | | |
|---|---|---|---|---|---|
| 14 | .00 | 660.84 | 540.00 | 120.84 | 52.36 |
| 15 | 4.00 | 604.00 | 550.00 | 54.00 | 23.40 |
| 16 | 1.00 | 709.49 | 490.00 | 219.49 | 95.11 |
| 17 | .00 | 608.33 | 460.00 | 148.33 | 64.28 |
| 18 | 3.00 | 597.09 | 430.00 | 167.09 | 72.41 |
| 19 | .00 | 587.30 | 420.00 | 167.30 | 72.50 |
| 20 | .00 | 556.62 | 420.00 | 136.62 | 59.20 |
| 21 | 5.00 | 584.19 | 430.00 | 154.19 | 66.82 |
| 22 | .00 | 606.43 | 480.00 | 126.43 | 54.79 |
| 23 | .00 | 580.39 | 470.00 | 110.39 | 47.83 |
| 24 | .00 | 624.87 | 460.00 | 164.87 | 71.45 |
| 25 | .00 | 595.00 | 450.00 | 145.00 | 62.83 |
| 26 | .00 | 567.08 | 440.00 | 127.08 | 55.07 |
| 27 | 3.00 | 525.05 | 440.00 | 85.05 | 36.86 |
| 28 | 5.00 | 536.05 | 420.00 | 116.05 | 50.29 |
| 29 | .00 | 677.26 | 430.00 | 247.26 | 107.15 |
| 30 | .00 | 759.85 | 400.00 | 359.85 | 155.94 |
| 31 | .00 | 419.99 | 400.00 | 19.99 | 8.66 |
| 32 | .00 | 627.90 | 450.00 | 177.90 | 77.09 |
| 33 | .00 | 601.86 | 420.00 | 181.86 | 78.80 |
| FGN-AA | ---- | 420.00 | 400.00 | 20.00 | 8.67 |
| PUMP-1 | .00 | 419.72 | 400.00 | 19.72 | 8.54 |
| PUMP-2 | .00 | 760.13 | 400.00 | 360.13 | 156.06 |
| PUMP-3 | .00 | 760.82 | 470.00 | 290.82 | 126.02 |
| TANK-A | ---- | 620.00 | 500.00 | 120.00 | 52.00 |
| PUMP-1 | .00 | 760.13 | 400.00 | 360.13 | 156.06 |
| PUMP-2 | .00 | 419.72 | 400.00 | 19.72 | 8.54 |
| PUMP-3 | .00 | 589.24 | 470.00 | 119.24 | 51.67 |

Some comments on the results are helpful for clarifying the special handling of the domestic flow requirements.

1) The flowrate of 14.8 g.p.m. in pipe 23 is based on three domestic connection served by that line and corresponds to the value given in Table 1.

2) The flowrate of 20.2 g.p.m. in pipe 11 represents 16.2 g.p.m. for the 4 domestic connections serviced by that line plus 4 g.p.m. for the fixed (livestock) demand assigned to the downstream node (12).

3) Nodes 2,3,4 and 5 all represent the primary connection to one or more branched sections and the average 12-hour domestic requirement of 1 g.p.m. per branch section connection served must be delivered to those nodes. This is in addition to any fixed demands and non-branch line domestic demands served by those nodes. At node 4, for example, a net demand of 9.5 g.p.m. (82.7 - 73.2) is provided by the non-branch connections (pipes 3 and 4). This requirement is based on the following data:

branch section serving 4 connections - 4.0 g.p.m.

fixed demand at node 4 - 4.0 g.p.m.

1/2 domestic flow requirements for pipes 3 and 4 - 1.5 g.p.m.

9.5 g.p.m.

4) The total flow supplied by the reservoir and tank is 146 g.p.m. This represents the sum of the fixed demands (37 g.p.m.) and the domestic demand of 107 g.p.m. based on 107 total connections requiring 1 g.p.m. each.

b) Using domestic demands based only on average demand requirements.

In order to evaluate the impact of this approach, the same example is analyzed using the conventional approach of employing only the average demand requirement for domestic demands applied at end nodes for the number of domestic connections serviced by that line. The demand pattern used is shown in Figure 6. Note that the total demand imposed is 146 g.p.m. which includes 107 g.p.m. based on the 107 domestic connections (using an average 12-hour demand of g.p.m./connection) plus 39 g.p.m. fixed (livestock) demand. The domestic demand is applied throughout as a fixed demand at the downstream node of the applicable pipe link. The demands shown in Figure 6 include the original fixed demands.

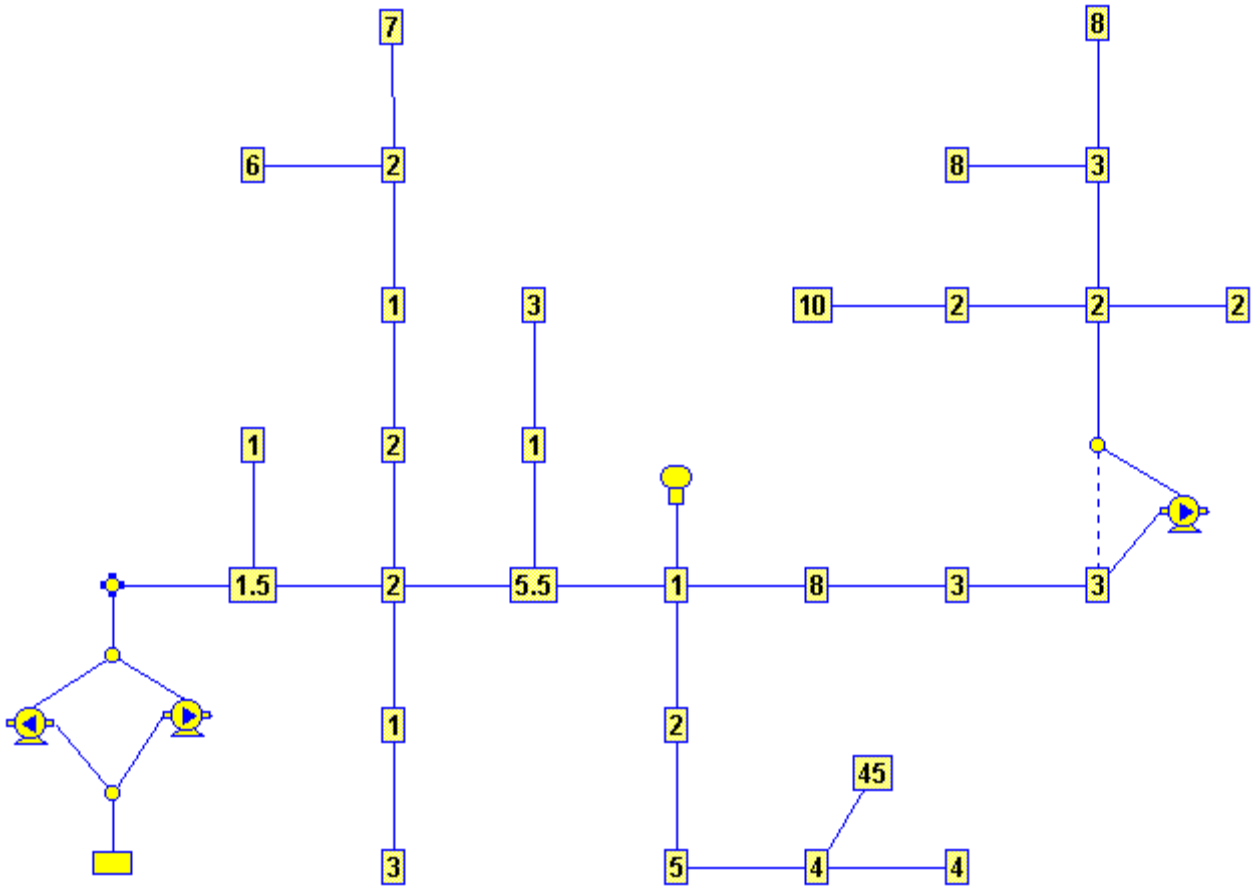



**Figure 6 Demands for Conventional Approach - Example 1b**

The regular KYPIPE4 analysis is then run (**Analyze**). A plot of the resulting pressures and flows for this hydraulic analysis are shown in Figure 7. It can be noted that even though the same total supply is provided, the differences in flowrates and pressures are very significant and this approach calculates much higher pressures in many areas. This is because, when using the conventional approach, the peak domestic flows are severely underestimated for the pipes serving just a few domestic connections when compared to the requirement calculated using the peak demand diversity curve. It is possible

that designs based on the conventional approach will be inadequate resulting in low pressures.

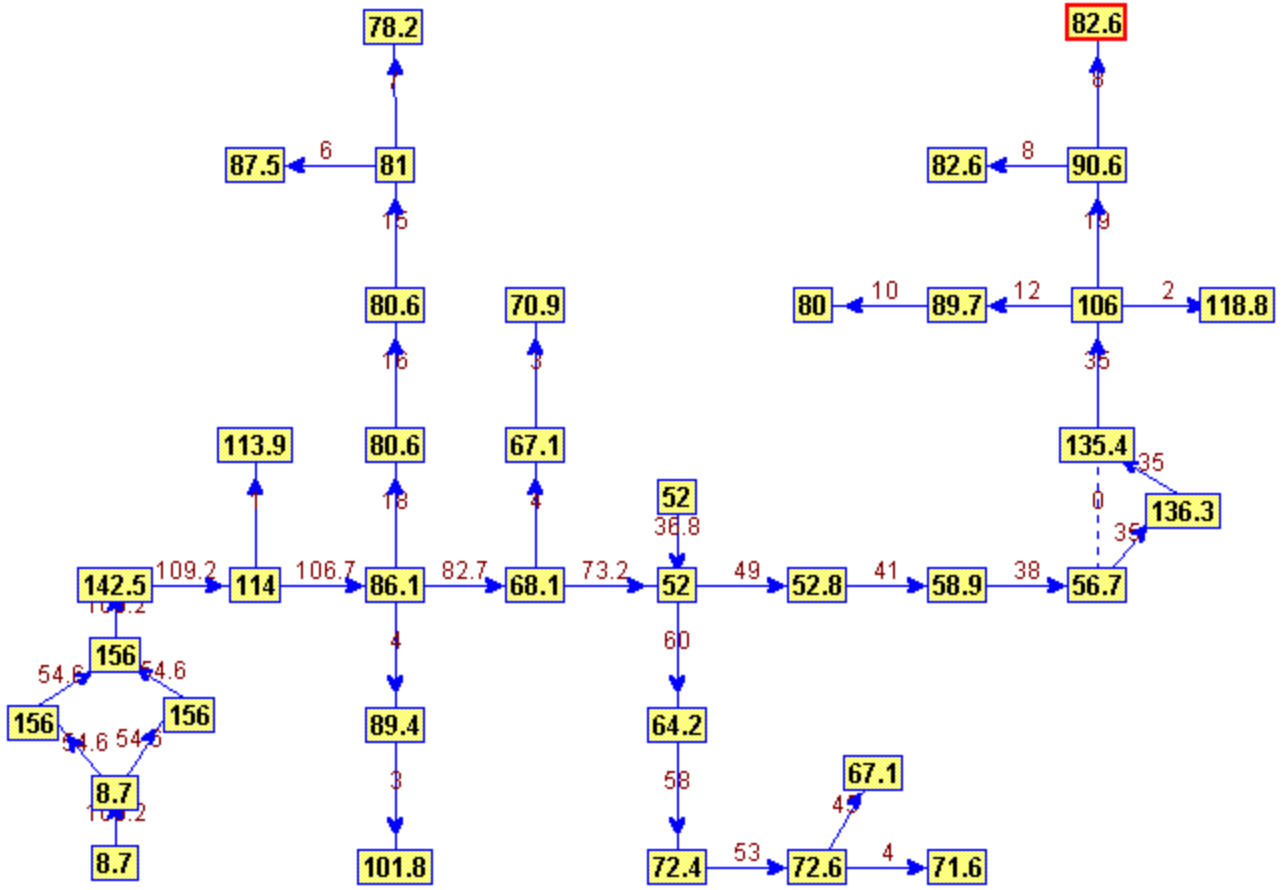



**Figure 7 Plot of Pressures and Flows - Example 1b**

### Example 2 - Combined Municipal and Branched Rural System

Figure 8 shows a schematic of this example with pipe and node labels. A small municipal system is represented on the left side of the schematic, which is not to scale. Figure 9 shows the domestic connection and fixed demands. There are a total of 196 domestic connections in the municipal region with an additional fixed

demand of 15 g.p.m. Pipe 19 leads to a rural and mostly branched region which services an additional 76 domestic connections and a fixed (livestock) demand of 50 g.p.m. The pipe lengths, diameters, roughness, node elevations and other significant data is tabulated in the data summary which is provided (Table 5).

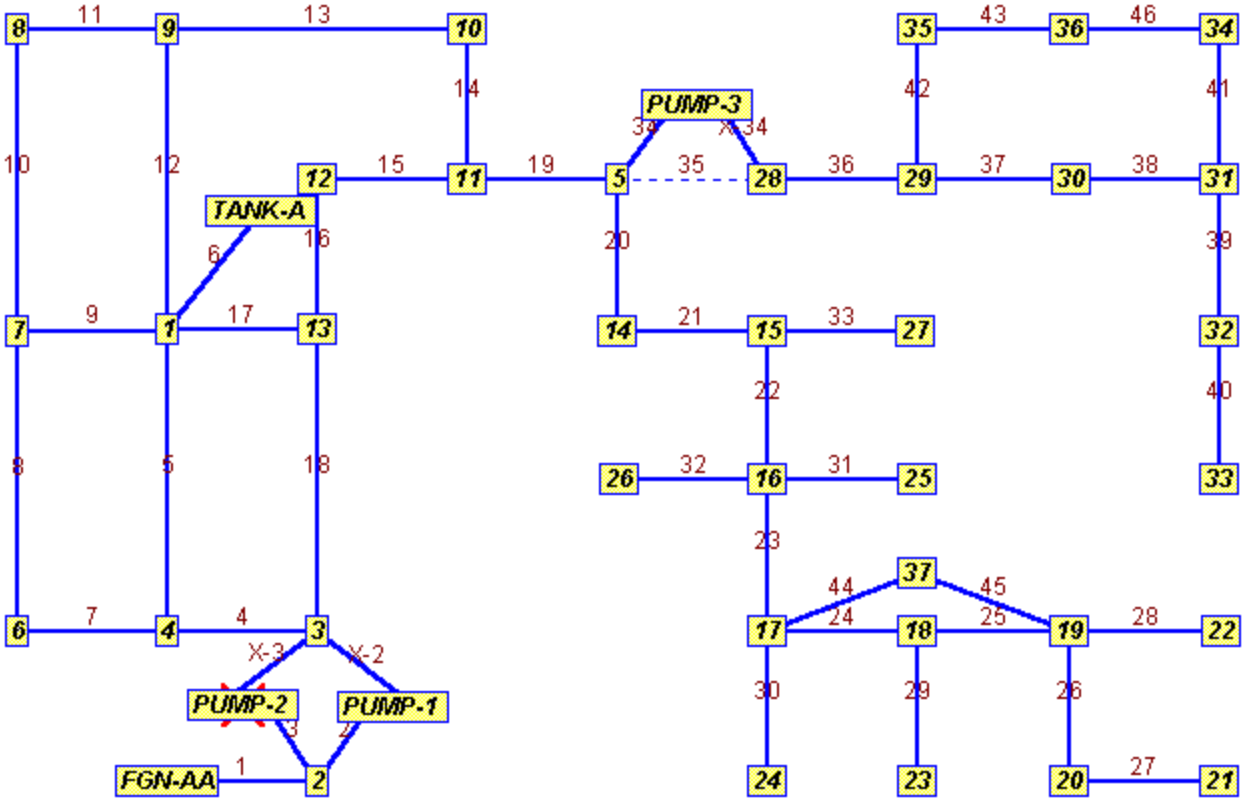



**Figure 8 Schematic for Example 2**

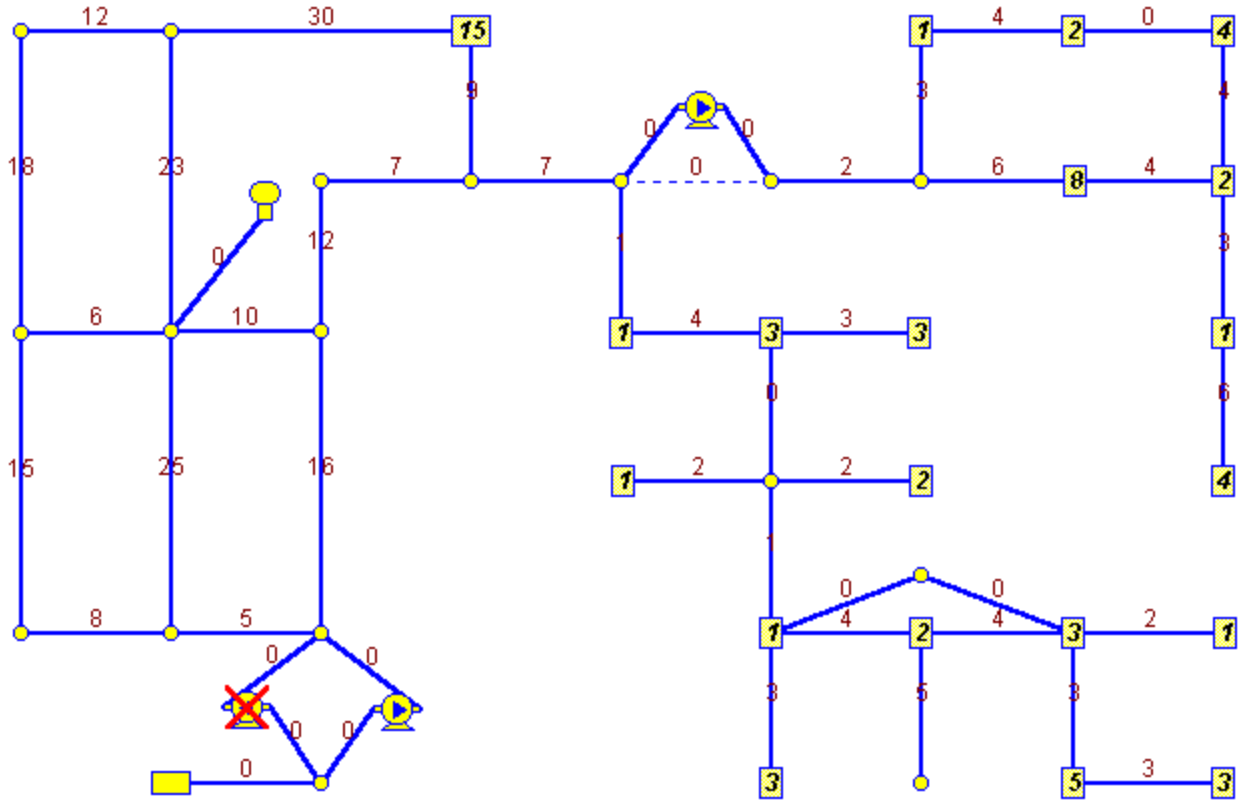



**Figure 9 Domestic Connections and Fixed Demands**

**Table 5 Data Summary for Example 2 (KYRURAL)**

**********
S U M M A R Y  O F  O R I G I N A L  D A T A
**********

U N I T S  S P E C I F I E D

FLOWRATE ..... = gallons/minute
HEAD (HGL) ..... = feet
PRESSURE ..... = psig

P I P E L I N E  D A T A

STATUS CODE: XX -CLOSED PIPE CV -CHECK VALVE

| PIPE NAME | NODE NAMES #1 | NODE NAMES #2 | LENGTH (ft) | DIAMETER (in) | ROUGHNESS COEFF. | MINOR LOSS COEFF. |
|---|---|---|---|---|---|---|
| 1 | N-AA | | 50.00 | 8.00 | 100.0000 | .00 |
| 2 | | MP-1 | 5.00 | 4.00 | 100.0000 | .00 |
| 3 | | MP-2 | 5.00 | 4.00 | 100.0000 | .00 |
| 4 | | | 800.00 | 6.00 | 100.0000 | .00 |
| 5 | | | 1600.00 | 6.00 | 100.0000 | .00 |
| 6 | | NK-A | 100.00 | 8.00 | 100.0000 | .00 |
| 7 | | | 800.00 | 4.00 | 80.0000 | .00 |
| 8 | | | 1600.00 | 4.00 | 80.0000 | .00 |
| 9 | | | 800.00 | 4.00 | 80.0000 | .00 |
| 10 | | | 1600.00 | 4.00 | 80.0000 | .00 |
| 11 | | | 800.00 | 4.00 | 80.0000 | .00 |

| | | | | | |
|---|---|---|---|---|---|
| 12 | | 1600.00 | 6.00 | 100.0000 | .00 |
| 13 | | 1600.00 | 6.00 | 100.0000 | .00 |
| 14 | | 800.00 | 4.00 | 80.0000 | .00 |
| 15 | | 800.00 | 4.00 | 80.0000 | .00 |
| 16 | | 800.00 | 4.00 | 80.0000 | .00 |
| 17 | | 800.00 | 6.00 | 100.0000 | .00 |
| 18 | | 1600.00 | 6.00 | 100.0000 | .00 |
| 19 | | 5300.00 | 6.00 | 150.0000 | .00 |
| 20 | | 5300.00 | 4.00 | 150.0000 | .00 |
| 21 | | 5300.00 | 4.00 | 150.0000 | .00 |
| 22 | | 5300.00 | 4.00 | 150.0000 | .00 |
| 23 | | 5300.00 | 4.00 | 150.0000 | .00 |
| 24 | | 5300.00 | 2.50 | 150.0000 | .00 |
| 25 | | 5300.00 | 2.50 | 150.0000 | .00 |
| 26 | | 5300.00 | 3.00 | 150.0000 | .00 |
| 27 | | 5300.00 | 2.50 | 150.0000 | .00 |
| 28 | | 5300.00 | 2.50 | 150.0000 | .00 |
| 29 | | 5300.00 | 2.00 | 150.0000 | .00 |
| 30 | | 5300.00 | 2.00 | 150.0000 | .00 |
| 31 | | 5300.00 | 2.00 | 150.0000 | .00 |
| 32 | | 5300.00 | 2.00 | 150.0000 | .00 |
| 33 | | 5300.00 | 2.00 | 150.0000 | .00 |
| 34 | MP-3 | 5.00 | 3.00 | 150.0000 | .00 |
| 35-XX | | 10.00 | 3.00 | 150.0000 | .00 |
| 36 | | 5300.00 | 4.00 | 150.0000 | .00 |
| 37 | | 5300.00 | 4.00 | 150.0000 | .00 |
| 38 | | 5300.00 | 3.00 | 150.0000 | .00 |
| 39 | | 5300.00 | 2.50 | 150.0000 | .00 |
| 40 | | 5300.00 | 2.50 | 150.0000 | .00 |
| 41 | | 5300.00 | 2.50 | 150.0000 | .00 |
| 42 | | 5300.00 | 3.00 | 150.0000 | .00 |
| 43 | | 5300.00 | 2.50 | 150.0000 | .00 |
| 44 | | 5300.00 | 3.00 | 150.0000 | .00 |

| | | | | | |
|---|---|---|---|---|---|
| 45 | | 5300.00 | 3.00 | 150.0000 | .00 |
| 46 | | 5300.00 | 2.50 | 150.0000 | .00 |
| X-2 | MP-1 | 5.00 | 4.00 | 100.0000 | .00 |
| X-3 | MP-2 | 5.00 | 4.00 | 100.0000 | .00 |
| X-34 | MP-3 | 5.00 | 3.00 | 150.0000 | .00 |

P U M P/L O S S  E L E M E N T  D A T A

THERE IS A DEVICE AT NODE PUMP-1 DESCRIBED BY THE FOLLOWING DATA: (ID= 1)

| HEAD (ft) | FLOWRATE (gpm) | EFFICIENCY (%) |
|---|---|---|
| 210.00 | .00 | 75.00 (Default) |
| 160.00 | 200.00 | 75.00 (Default) |
| 60.00 | 300.00 | 75.00 (Default) |

THERE IS A DEVICE AT NODE PUMP-2 DESCRIBED BY THE FOLLOWING DATA: (ID= 2)

| HEAD (ft) | FLOWRATE (gpm) | EFFICIENCY (%) |
|---|---|---|
| 260.00 | .00 | 75.00 (Default) |
| 170.00 | 300.00 | 75.00 (Default) |
| 60.00 | 400.00 | 75.00 (Default) |

THERE IS A DEVICE AT NODE PUMP-3 DESCRIBED BY THE FOLLOWING DATA: (ID= 3)

| HEAD (ft) | FLOWRATE (gpm) | EFFICIENCY (%) |
|---|---|---|

| | | |
|---|---|---|
| 90.00 | .00 | 75.00 (Default) |
| 80.00 | 60.00 | 75.00 (Default) |
| 60.00 | 100.00 | 75.00 (Default) |

E N D  N O D E  D A T A

| NODE NAME | NODE TITLE | EXTERNAL DEMAND (gpm) | JUNCTION ELEVATION (ft) | EXTERNAL GRADE (ft) |
|---|---|---|---|---|
| 1 | | .00 | 610.00 | |
| 2 | | .00 | 550.00 | |
| 3 | | .00 | 550.00 | |
| 4 | | .00 | 570.00 | |
| 5 | | .00 | 590.00 | |
| 6 | | .00 | 590.00 | |
| 7 | | .00 | 600.00 | |
| 8 | | .00 | 580.00 | |
| 9 | | .00 | 600.00 | |
| 10 | | 15.00 | 590.00 | |
| 11 | | .00 | 580.00 | |
| 12 | | .00 | 590.00 | |
| 13 | | .00 | 590.00 | |
| 14 | | 1.00 | 580.00 | |
| 15 | | 3.00 | 600.00 | |
| 16 | | .00 | 590.00 | |
| 17 | | 1.00 | 580.00 | |
| 18 | | 2.00 | 570.00 | |
| 19 | | 3.00 | 560.00 | |
| 20 | | 5.00 | 530.00 | |

| | | | |
|---|---|---|---|
| 21 | 3.00 | 530.00 | |
| 22 | 1.00 | 560.00 | |
| 23 | .00 | 550.00 | |
| 24 | 3.00 | 560.00 | |
| 25 | 2.00 | 590.00 | |
| 26 | 1.00 | 570.00 | |
| 27 | 3.00 | 590.00 | |
| 28 | .00 | 590.00 | |
| 29 | .00 | 630.00 | |
| 30 | 8.00 | 650.00 | |
| 31 | 2.00 | 670.00 | |
| 32 | 1.00 | 650.00 | |
| 33 | 4.00 | 630.00 | |
| 34 | 4.00 | 680.00 | |
| 35 | 1.00 | 650.00 | |
| 36 | 2.00 | 660.00 | |
| 37 | .00 | 570.00 | |
| FGN-AA | ---- | 550.00 | 570.00 |
| PUMP-1 | .00 | 550.00 | |
| PUMP-2 | .00 | 550.00 | |
| PUMP-3 | .00 | 590.00 | |
| TANK-A | ---- | 610.00 | 720.00 |

a) Using **RURAL Analysis**

The rural section, which we wish to analyze using the demand diversity curve requirements, does have several loops that require special handling. As previously described, selected pipes can be designated as type 9 and will be ignored when determining branch line configurations and domestic flow requirements. To do this the domestic connections should be attributed to other links. Pipes 35, 44, 45 and 46 are input with no domestic connections and designated as type 9. Table 6 shows the results for the branch lines obtained by using **RURAL Analysis** and verifies that the entire region has been analyzed as desired ignoring the designated pipe links.

**Table 6 Connections and Domestic Flow Requirements - Example 2**

```
          DOMESTIC DEMAND ALLOCATION FOR RURAL WATER SYSTEMS

        Preprocessor for Pipe2000 Rural Water Systems Analysis

                        copyright June 1999
                          by Don J. Wood


   The Current Data File Is C:\PROGRA~1\PIPE2000\HELPST~1\COMBINED.DT2
 COEFFICIENTS OF DEMAND DIVERSITY CURVE:
   A =  4
   B =  0.3
   C =  7

AVERAGE DOMESTIC DEMAND REQUIREMENT (gpm) =  1
```

***** SUMMARY OF BRANCH LINE DOMESTIC FLOW REQUIREMENTS *****

| PIPE NO. | NODE #1 | NODE #2 | No. Connections Served | Required Domestic Flow |
|---|---|---|---|---|
| 3 | 2 | 40 | 0 | 7 |
| 19 | 11 | 5 | 76 | 64.6 |
| 20 | 5 | 14 | 37 | 42.4 |
| 21 | 14 | 15 | 36 | 41.8 |
| 22 | 15 | 16 | 29 | 37.2 |
| 23 | 16 | 17 | 25 | 34.5 |
| 24 | 17 | 18 | 21 | 31.6 |
| 25 | 18 | 19 | 12 | 24.4 |
| 26 | 19 | 20 | 6 | 18.5 |
| 27 | 20 | 21 | 3 | 14.8 |
| 28 | 19 | 22 | 2 | 13.2 |
| 29 | 18 | 23 | 5 | 17.4 |
| 30 | 17 | 24 | 3 | 14.8 |
| 31 | 16 | 25 | 2 | 13.2 |
| 32 | 16 | 26 | 2 | 13.2 |
| 33 | 15 | 27 | 3 | 14.8 |
| 34 | 5 | 46 | 32 | 39.2 |
| 36 | 28 | 29 | 32 | 39.2 |
| 37 | 29 | 30 | 23 | 33 |
| 38 | 30 | 31 | 17 | 28.5 |
| 39 | 31 | 32 | 9 | 21.7 |
| 40 | 32 | 33 | 6 | 18.5 |
| 41 | 31 | 34 | 4 | 16.2 |
| 42 | 29 | 35 | 7 | 19.6 |
| 43 | 35 | 36 | 4 | 16.2 |
| 44 | 17 | 37 | 0 | 7 |
| 48 | 45 | 3 | 0 | -7 |
| 49 | 41 | 28 | 32 | 39.2 |
| 53 | 46 | 41 | 32 | 39.2 |

The **Hydraulic Analysis** program was next executed and the resulting pressures and flows are shown in Figure 10 along with the tabulated results (Table 7).

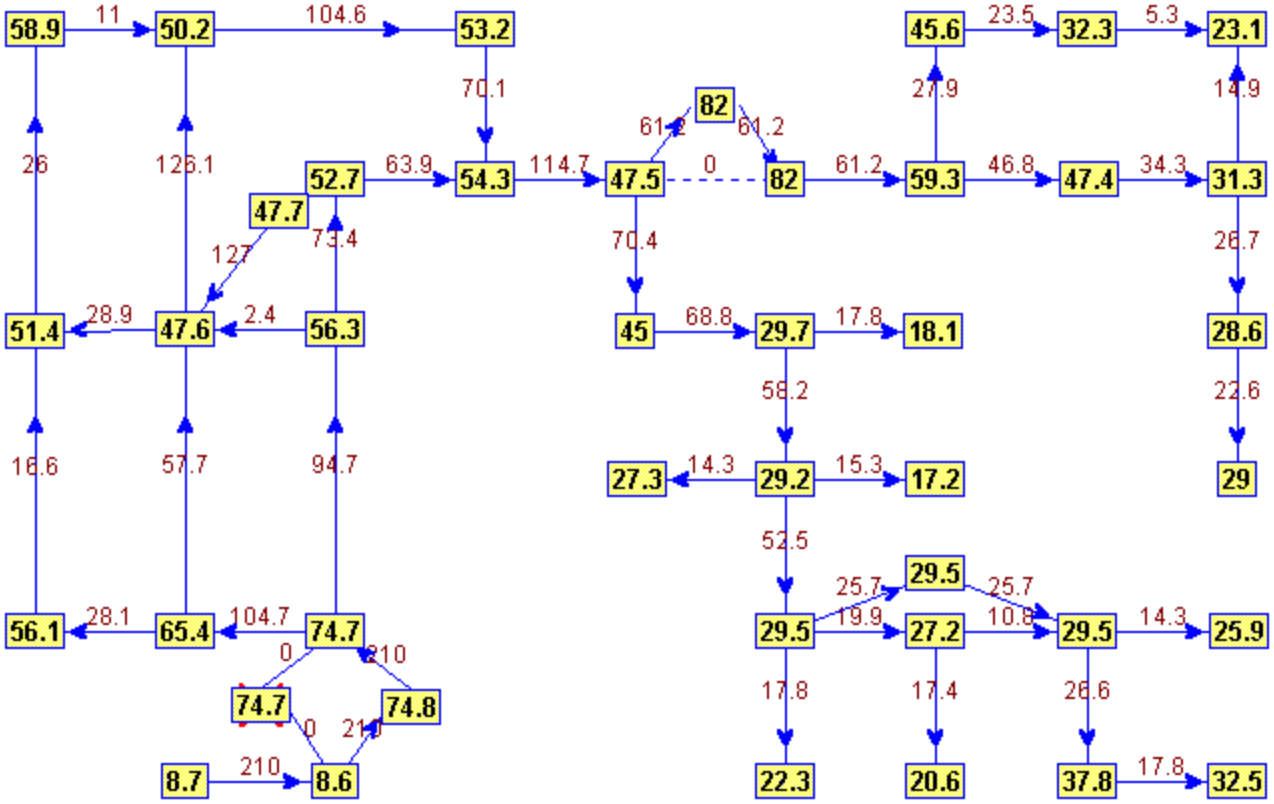



**Figure 10 Plot of Pressures and Flowrates - Example 2a**

**Table 7 Tabulated Results - Example 2a**

P I P E L I N E  R E S U L T S

STATUS CODE: XX -CLOSED PIPE CV -CHECK VALVE

| PIPE NAME | NODE NUMBERS #1 | #2 | FLOWRATE (gpm) | HEAD LOSS (ft) | MINOR LOSS (ft) | LINE VELO. (ft/s) | HL/ 1000 (ft/ft) |
|---|---|---|---|---|---|---|---|
| 1 | FGN-AA | 2 | 209.97 | .08 | .00 | 1.34 | 1.65 |
| 2 | 2 | PUMP-1 | 209.97 | .24 | .00 | 5.36 | 48.24 |
| 3 | 2 | PUMP-2 | 7.00 | .00 | .00 | .18 | .09 |
| 4 | 3 | 4 | 104.73 | 1.48 | .00 | 1.19 | 1.85 |
| 5 | 4 | 1 | 57.66 | .98 | .00 | .65 | .61 |
| 6 | 1 | TANK-A | -127.02 | .07 | .00 | .81 | .65 |
| 7 | 4 | 6 | 28.07 | 1.40 | .00 | .72 | 1.76 |
| 8 | 6 | 7 | 16.57 | 1.06 | .00 | .42 | .66 |
| 9 | 7 | 1 | -28.92 | 1.48 | .00 | .74 | 1.86 |
| 10 | 7 | 8 | 25.99 | 2.44 | .00 | .66 | 1.52 |
| 11 | 8 | 9 | 10.99 | .25 | .00 | .28 | .31 |
| 12 | 9 | 1 | -126.11 | 4.17 | .00 | 1.43 | 2.60 |
| 13 | 9 | 10 | 104.60 | 2.95 | .00 | 1.19 | 1.84 |
| 14 | 10 | 11 | 70.10 | 7.65 | .00 | 1.79 | 9.56 |
| 15 | 11 | 12 | -63.89 | 6.44 | .00 | 1.63 | 8.05 |
| 16 | 12 | 13 | -73.39 | 8.33 | .00 | 1.87 | 10.41 |
| 17 | 13 | 1 | 2.36 | .00 | .00 | .03 | .00 |
| 18 | 3 | 13 | 94.74 | 2.45 | .00 | 1.07 | 1.53 |
| 19 | 11 | 5 | 114.66 | 5.46 | .00 | 1.30 | 1.03 |
| 20 | 5 | 14 | 70.42 | 15.95 | .00 | 1.80 | 3.01 |
| 21 | 14 | 15 | 68.79 | 15.28 | .00 | 1.76 | 2.88 |
| 22 | 15 | 16 | 58.23 | 11.22 | .00 | 1.49 | 2.12 |
| 23 | 16 | 17 | 52.50 | 9.26 | .00 | 1.34 | 1.75 |
| 24 | 17 | 18 | 21.36 | 17.28 | .00 | 1.40 | 3.26 |
| 25 | 18 | 19 | 12.19 | 6.11 | .00 | .80 | 1.15 |

| | | | | | | | |
|---|---|---|---|---|---|---|---|
| 26 | 19 | 20 | 26.60 | 10.67 | .00 | 1.21 | 2.01 |
| 27 | 20 | 21 | 17.83 | 12.36 | .00 | 1.17 | 2.33 |
| 28 | 19 | 22 | 14.26 | 8.17 | .00 | .93 | 1.54 |
| 29 | 18 | 23 | 17.44 | 35.19 | .00 | 1.78 | 6.64 |
| 30 | 17 | 24 | 17.83 | 36.64 | .00 | 1.82 | 6.91 |
| 31 | 16 | 25 | 15.26 | 27.46 | .00 | 1.56 | 5.18 |
| 32 | 16 | 26 | 14.26 | 24.22 | .00 | 1.46 | 4.57 |
| 33 | 15 | 27 | 17.83 | 36.64 | .00 | 1.82 | 6.91 |
| 34 | 5 | PUMP-3 | 61.23 | .05 | .00 | 2.78 | 9.43 |
| 35-XX | 5 | 28 | | | | | |
| 36 | 28 | 29 | 61.23 | 12.31 | .00 | 1.56 | 2.32 |
| 37 | 29 | 30 | 46.82 | 7.49 | .00 | 1.20 | 1.41 |
| 38 | 30 | 31 | 34.33 | 17.12 | .00 | 1.56 | 3.23 |
| 39 | 31 | 32 | 26.70 | 26.11 | .00 | 1.74 | 4.93 |
| 40 | 32 | 33 | 22.60 | 19.17 | .00 | 1.48 | 3.62 |
| 41 | 31 | 34 | 14.94 | 8.91 | .00 | .98 | 1.68 |
| 42 | 29 | 35 | 27.94 | 11.69 | .00 | 1.27 | 2.21 |
| 43 | 35 | 36 | 23.46 | 20.55 | .00 | 1.53 | 3.88 |
| 44 | 17 | 37 | 31.26 | 14.39 | .00 | 1.42 | 2.72 |
| 45 | 37 | 19 | 24.26 | 9.00 | .00 | 1.10 | 1.70 |
| 46 | 36 | 34 | 5.26 | 1.29 | .00 | .34 | .24 |
| X-2 | PUMP-1 | 3 | 209.97 | .24 | .00 | 5.36 | 48.24 |
| X-3 | PUMP-2 | 3 | -7.00 | .00 | .00 | .18 | .09 |
| X-34 | PUMP-3 | 28 | 61.23 | .05 | .00 | 2.78 | 9.43 |

E N D  N O D E  R E S U L T S

| NODE NAME | NODE TITLE | EXTERNAL DEMAND (gpm) | HYDRAULIC GRADE (ft) | NODE ELEVATION (ft) | PRESSURE HEAD (ft) | NODE PRESSURE (psi) |
|---|---|---|---|---|---|---|
| 1 | | .00 | 719.93 | 610.00 | 109.93 | 47.64 |
| 2 | | .00 | 569.92 | 550.00 | 19.92 | 8.63 |
| 3 | | .00 | 722.39 | 550.00 | 172.39 | 74.70 |
| 4 | | .00 | 720.91 | 570.00 | 150.91 | 65.40 |
| 5 | | .00 | 699.71 | 590.00 | 109.71 | 47.54 |
| 6 | | .00 | 719.51 | 590.00 | 129.51 | 56.12 |
| 7 | | .00 | 718.45 | 600.00 | 118.45 | 51.33 |
| 8 | | .00 | 716.01 | 580.00 | 136.01 | 58.94 |
| 9 | | .00 | 715.77 | 600.00 | 115.77 | 50.17 |
| 10 | | 15.00 | 712.82 | 590.00 | 122.82 | 53.22 |
| 11 | | .00 | 705.17 | 580.00 | 125.17 | 54.24 |
| 12 | | .00 | 711.61 | 590.00 | 121.61 | 52.70 |
| 13 | | .00 | 719.94 | 590.00 | 129.94 | 56.31 |
| 14 | | 1.00 | 683.75 | 580.00 | 103.75 | 44.96 |
| 15 | | 3.00 | 668.48 | 600.00 | 68.48 | 29.67 |
| 16 | | .00 | 657.26 | 590.00 | 67.26 | 29.15 |
| 17 | | 1.00 | 648.00 | 580.00 | 68.00 | 29.47 |
| 18 | | 2.00 | 630.72 | 570.00 | 60.72 | 26.31 |
| 19 | | 3.00 | 624.61 | 560.00 | 64.61 | 28.00 |
| 20 | | 5.00 | 613.94 | 530.00 | 83.94 | 36.37 |
| 21 | | 3.00 | 601.58 | 530.00 | 71.58 | 31.02 |
| 22 | | 1.00 | 616.44 | 560.00 | 56.44 | 24.46 |
| 23 | | .00 | 595.53 | 550.00 | 45.53 | 19.73 |
| 24 | | 3.00 | 611.36 | 560.00 | 51.36 | 22.26 |
| 25 | | 2.00 | 629.80 | 590.00 | 39.80 | 17.25 |
| 26 | | 1.00 | 633.04 | 570.00 | 63.04 | 27.32 |
| 27 | | 3.00 | 631.84 | 590.00 | 41.84 | 18.13 |

```
          28                        .00       779.17    590.00    189.17    81.97
          29                        .00       766.86    630.00    136.86    59.31
          30                       8.00       759.37    650.00    109.37    47.39
          31                       2.00       742.25    670.00     72.25    31.31
          32                       1.00       716.13    650.00     66.13    28.66
          33                       4.00       696.96    630.00     66.96    29.02
          34                       4.00       733.34    680.00     53.34    23.11
          35                       1.00       755.17    650.00    105.17    45.57
          36                       2.00       734.62    660.00     74.62    32.34
          37                        .00       633.61    570.00     63.61    27.56
      FGN-AA                       ----       570.00    550.00     20.00     8.67
      PUMP-1                        .00       722.63    550.00    172.63    74.81
      PUMP-2                        .00       569.92    550.00     19.92     8.63
      PUMP-3                        .00       779.22    590.00    189.22    81.99
      TANK-A                       ----       720.00    610.00    110.00    47.67
      PUMP-1                        .00       569.68    550.00     19.68     8.53
      PUMP-2                        .00       722.39    550.00    172.39    74.70
      PUMP-3                        .00       699.66    590.00    109.66    47.52

S U M M A R Y   O F   I N F L O W S   A N D   O U T F L O W S

(+) INFLOWS INTO THE SYSTEM FROM SUPPLY NODES
(-) OUTFLOWS FROM THE SYSTEM INTO SUPPLY NODES

        NODE        FLOWRATE        NODE
        NAME         (gpm)          TITLE
     -----------------------------------------
       FGN-AA         209.97
       TANK-A         127.02

NET SYSTEM INFLOW  =    336.99
NET SYSTEM OUTFLOW =       .00
NET SYSTEM DEMAND  =     65.00
```

The following clarifying comments are provided.

1) A total inflow of 134 g.p.m. is provided at node 11 where the branch line connects into the looped network. This requirement is based on the following considerations:

branch section serving 76 connections - 76 g.p.m.
fixed demand in branch section - 50 g.p.m.
1/2 the domestic requirements for pipes 14 and 15 - 8 g.p.m.
134 g.p.m.

2) The flowrate in pipe 19 of 114.7 g.p.m. is based on the following:

76 connections requirement (demand diversity curve) - 64.7 g.p.m.
fixed demand requirement serviced by this pipe - 50.0 g.p.m.
114.7 g.p.m.

3) The combined flow in parallel pipes 25 and 45 is 36.5 g.p.m. This represents the sum of the domestic requirement of 24.5 g.p.m. attributed to pipe 25 (based on serving 12 connections) and a fixed demand requirement of 12 g.p.m. which also must be serviced through these pipes. The total requirement of 36.5 g.p.m. is distributed between the two parallel pipes based on hydraulic considerations. In this manner the domestic flow requirements specified using the demand diversity curve are maintained through parallel paths.

b) Using Conventional Approach

The demand pattern determined by placing domestic demands based on the conventional approach is shown if Figure 11. Figure 12 shows the pressures and flowrates obtained using this approach. Again, some of the pressures obtained are much higher than those obtained using **RURAL Analysis** to produce the domestic flows given by the peak demand diversity curve.

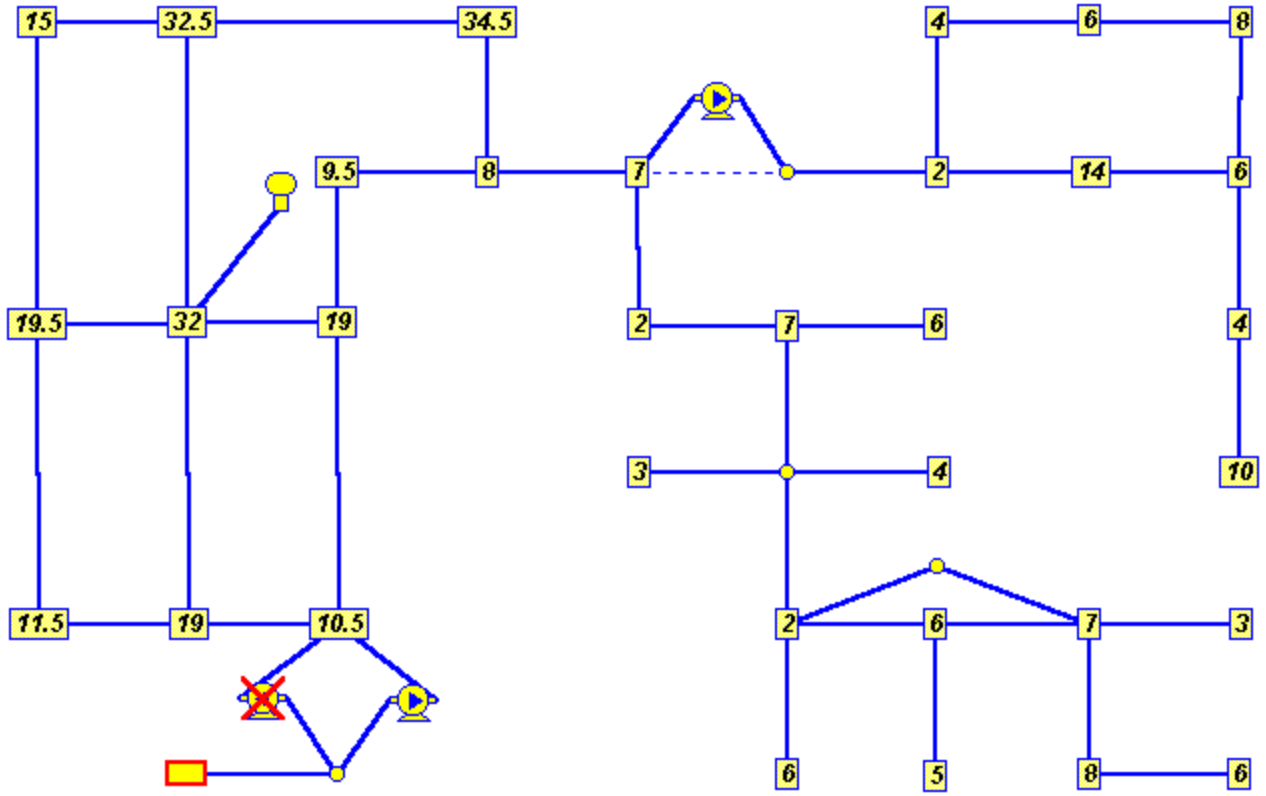



**Figure 11 Demands for Conventional Approach - Example 2b**

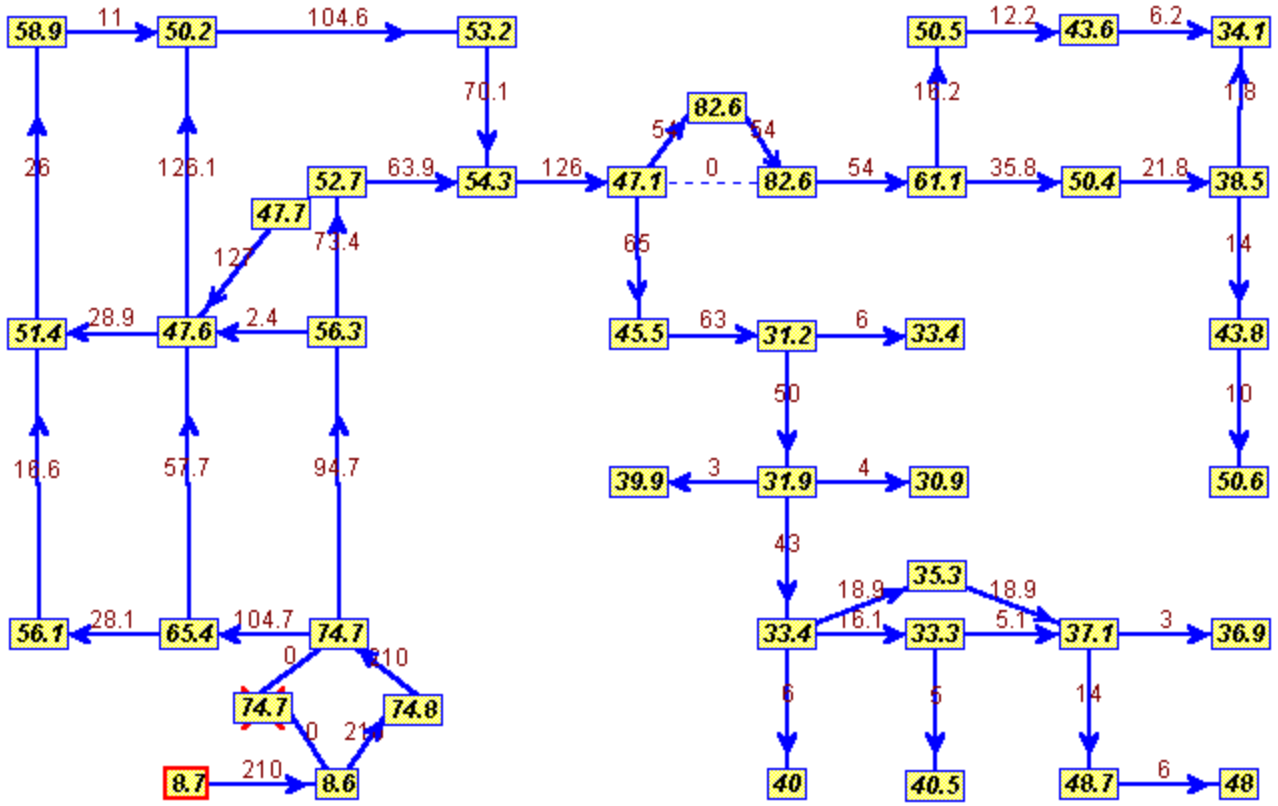



**Figure 12 Plot of Pressure and Flows - Example 2b**

## 1.4.3.2 Quick Guide to Running Rural Analysis

Quick Guide to Running Rural Analysis

---

1. Develop your Pipe2018 Model
2. Input number of meters to describe residential connections to each pipe.
3. Input fixed demands at Junctions
4. Set up the Rural Analysis
   a) Select Analyze | Analysis
   b) Check Rural Analysis
   c) Either enter desired coefficients or 3 data points (let software calculate coefficients).
   d) If you want to apply PDD allocations to looped areas also check Apply to Loops.
5. Click Analyze
6. You can now review the results on your system map or using the Report

Note: If you do not select the option to apply loops, the Rural program applies the PDD Curve Allocations only to branch lines. If you want to apply this approach in selected looped areas you must insert an active valve with the wide open resistance (R 100%) set to 0 at a location in the loop where you wish to break it. It will break the loop to perform the allocations and rejoin it to do the hydraulics which is the correct way to perform the analysis.

## 1.4.3.3 Rural Water Systems (Peak Demand Requirements)

Rural Water Systems (Peak Demand Requirements)

See Rural Water Systems
Quick Guide to Running Rural Analysis

---

### Rural Analysis (Peak Demand Requirements)

If rural water systems are not designed to provide fire flows, then special handling for long branch line serving few users may be necessary. Requirements for residential water delivered through branched lines depend on the number of residential (domestic) connections served by each branch. These requirements should be based on probability considerations and the requirement per connection served decreases as the number of connections increase. Relationships called Peak Demand Diversity Curves (PDD) are available to calculate these requirements as a function of the number of connections served by each branch line. Additional conventional demands can be added at junctions and are accommodated in the analysis. Branch line pipes should be sized to accommodate residential requirements calculated in this manner.

The conventional approach of a distributing residential demands throughout the system based on a fixed demand per connection and the number of connections served tends to greatly overstate overall requirements while severely understating requirements in branch lines serving few **Rural Analysis connections**. Designs based on this approach are flawed. Pipe2018 includes a special capability which identifies all branch lines, allows the number of connections per line to be entered as data and calculates the residential flow requirements for all branch lines. The flow distribution and pressure calculations are then made satisfying these requirements. This capability is tightly integrated into the Pipe2018 environment and uses the KYPIPE data file with the exception of coefficients for the PDD curve to be used, no additional data is required (see **Running the Rural Analysis**).

It should be noted that the purpose of the **RURAL Analysis** is to assure that the branch pipes are adequately sized for a peak demand situation. It is not appropriate to run the **RURAL Analysis** for EPS or multiple scenarios.

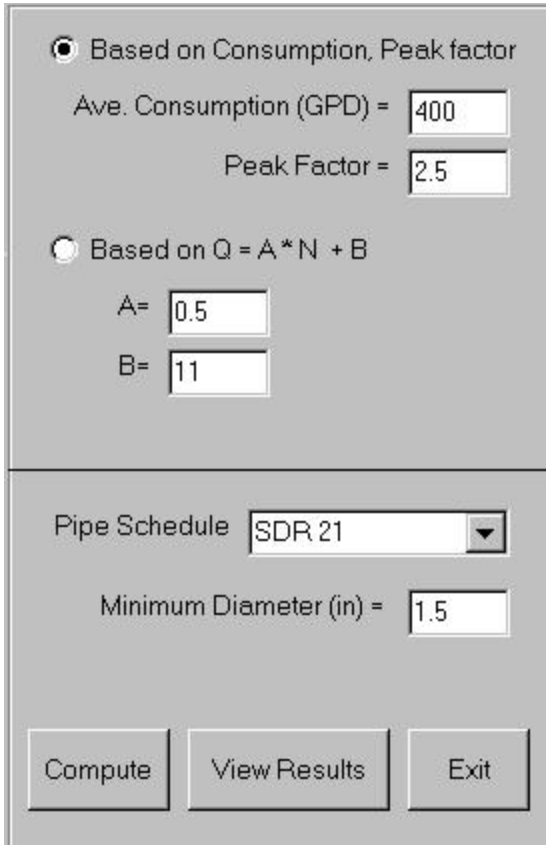
A detailed description of the use of the **RURAL Analysis** feature is presented in the topic, **Rural Water Systems**. This includes two examples (**Example 1** and **Example 2**). Details on the coefficients for the PDD curve are presented in the section entitled, "**Domestic Flow Requirements**". These should be customized to the users requirements.

## 1.4.3.4 Calculate Branch Diameters

Calculate Branch Diameters (Premium version feature)

---

This features gives the user the ability to modify the diameters of branch lines based on either average consumption and a peak factor or by providing an A and B factor for calculating equivalent flow. Note, the demand must be expressed in number of connection (residential meters) for use with this feature.




After results have been calculated, the new diameters may be viewed in a report as shown below. Upon exiting the user will be prompted whether the new calculated diameters should be applied to the pipes in the system. Answering 'Yes' will alter the baseline pipe data.

Branch Lines Flow and Pipe Sizing Calculations

Copyright: April 2005
by
Don J. Wood
Srini Lingireddy

Date: 05-23-2005
Time: 10:25:46

***** SUMMARY OF PEAK FLOW REQUIREMENTS *****

| PIPE NO. | NODE #1 | NODE #2 | No. Connections Served | Design Flow | Required Diameter | Design Velocity |
|---|---|---|---|---|---|---|
| 9 | 9 | 10 | 20 | 13 | 1.5 | 2.52 |
| 10 | 10 | 11 | 9 | 6 | 1.5 | 1.13 |
| 11 | 11 | 12 | 4 | 2 | 1.5 | 0.5 |
| 12 | 11 | 13 | 2 | 1 | 1.5 | 0.25 |
| 13 | 10 | 14 | 8 | 5 | 1.5 | 1 |
| 14 | 14 | 15 | 6 | 4 | 1.5 | 0.75 |
| 15 | 10 | 16 | 1 | 0 | 1.5 | 0.12 |
| 17 | 5 | 17 | 52 | 36 | 2 | 3.68 |
| 18 | 17 | 18 | 50 | 34 | 2 | 3.54 |

## 1.4.4 Hydrants, Fire Flows, Flushing

### 1.4.4.1 Fire Flows (Calculated)

Fire Flows (Calculated)

## Fire Flow Calculations

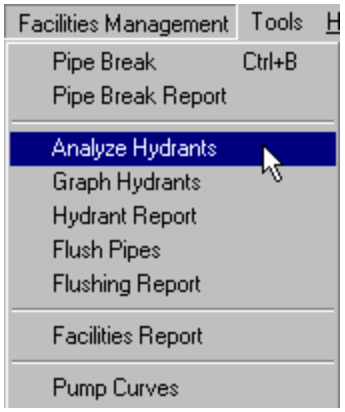
Fire flows may be calculated at hydrants, junctions or both. Therefore it is not necessary to include hydrants in your model to calculate fire flows. However, additional capablities to plot hydrant test data, maintain hydrant records, etc. are available if hydrants are incorporated into the model. Two pressures must be specified when performing fire flow calculations.

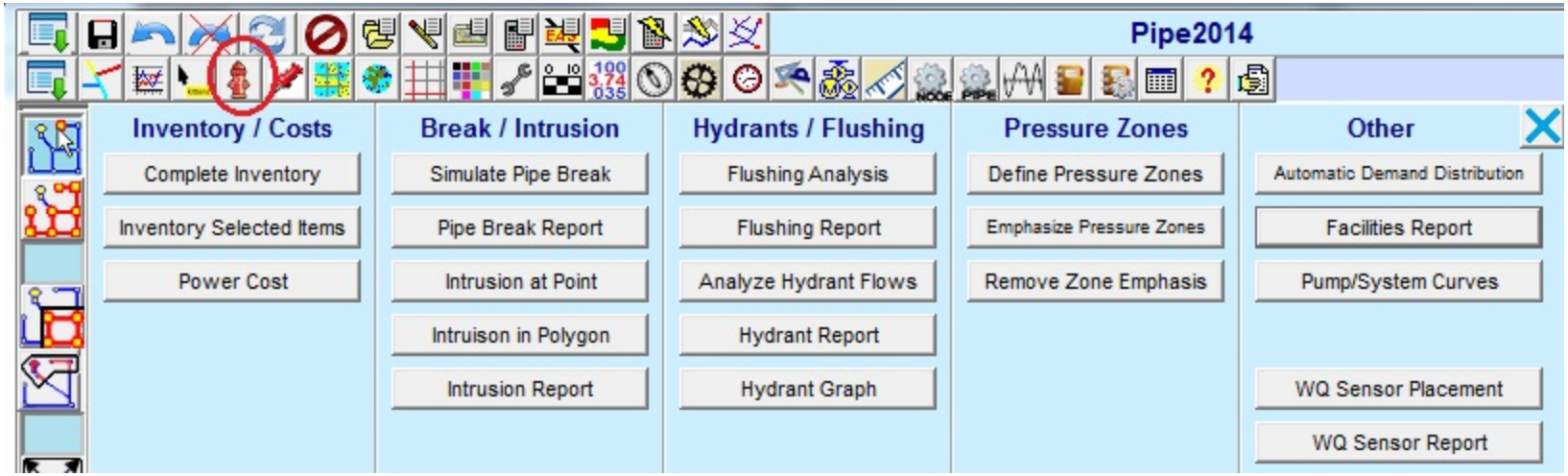
**Minimum pressure for fire flows:** This input is the lowest acceptable pressure at all applicable hydrants and nodes. This limit (usually 20 psi) will be reached at one node (usually the location of the hydrant or junction being analyzed) and will determine the maximum fire flow. All nodes are considered and the calculated fire flow will be adjusted accordingly.

**Static pressure limit:** This input defines a value of static pressure such that any nodes with a lower static pressure will not be used in the minimum pressure check. Thus a pump suction node or clearwell connection with a low static pressure will be excluded when checking the minimum pressure requirement.

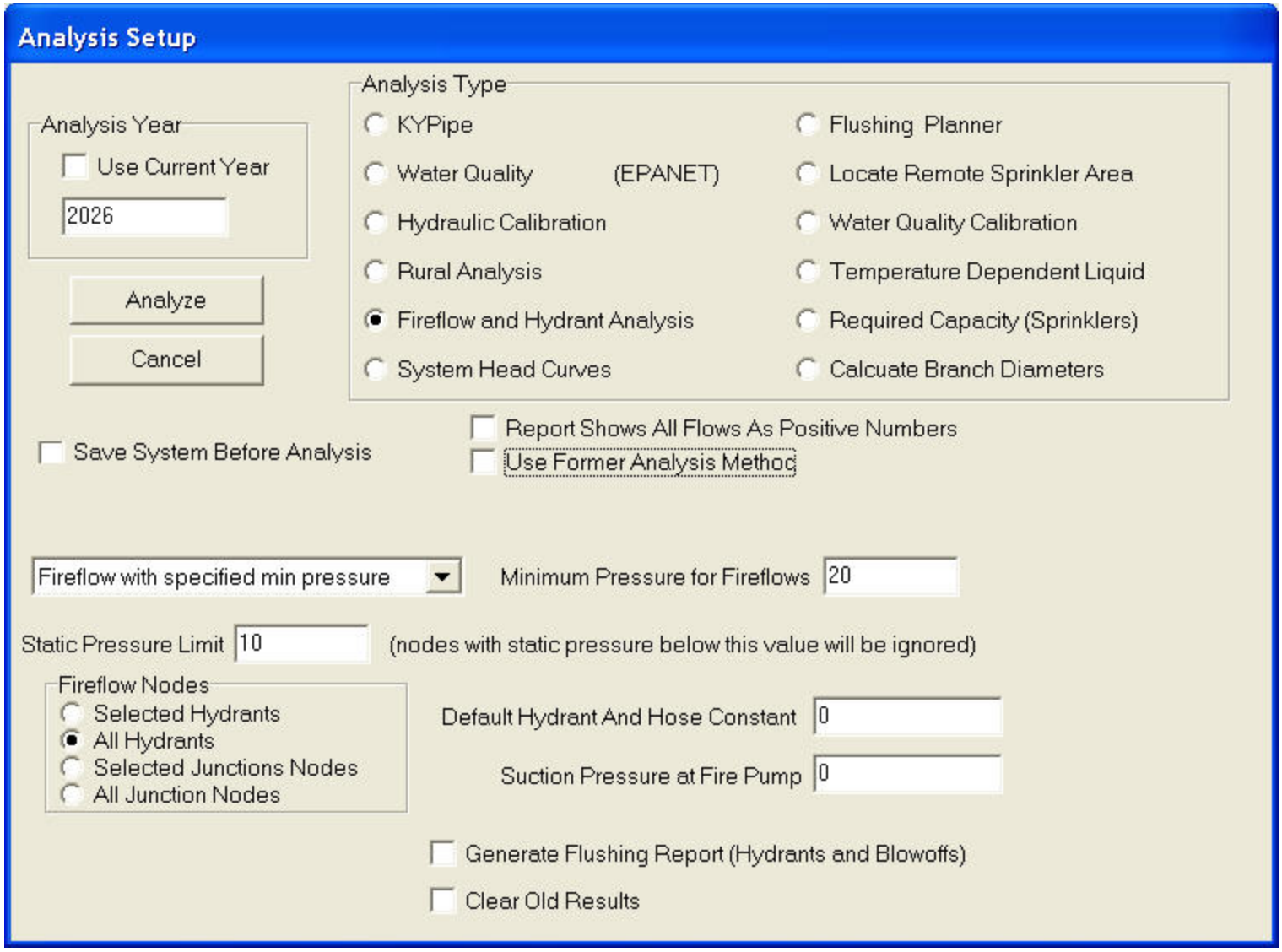
A Fire Flow Analysis may be conducted on a single hydrant or junction, on a group of hydrants or junctions selected using Group Mode, or on all the hydrants or junctions in a system.

To run a fire flow select (highlight) the hydrant(s) or junction(s) in question. If it is desired to run an analysis of all hydrants or junctions, there is no need to select any hydrants or junctions. The option to analyze all hydrants or junctions will be given in the Analysis Setup window as you proceed. Click on Facilities Management in the Main Menu. Choose Analyze Hydrants from the drop-down box.






The Analysis Setup box appears (you may also click on Analysis in the main menu directly and select Fireflow Analysis). Fireflow Analysis will be selected by default. Specify the minimum pressure to be maintained for the analysis in the data field at the bottom of the box (20 is the default). Then choose one of the four the options for Fireflow Nodes at the bottom of the window.




Click Analyze. Once the analysis is complete, there are several ways to view the results. They are slightly different for an analysis on hydrants than for an analysis on junction nodes.

### Optional Data

**Default Hydrant and Hose Constant / Suction Pressure at Fire Pump**- These two fields are used in conjuction. The conventional fire flow analysis determines how much flow can be pulled out of the system and not lower the pressure in the distribution system to < 20 psi (or other specified value). This approach may over-estimate the possible fire flow at locations where there is a large capacity. This is because the conventional analysis predicts fire flows so large they can't be delivered to the pumper at an acceptable pressure (e.g. > 0 psi). The losses through the hydrant and hose to the pumper are far too large. When a value is entered in this field, a second calculation is done which calculates the maximum flow which will maintain an acceptable suction at the fire pump. When this is lower than the first calculation then the first approach has over estimated the potential fire flow and the second flow is used.

Hydrant and Hose Constant is defined by the equation: $Qr = Kh (Pr)^{0.5}$ where Qr is the residual flow in gpm (rps) and Pr is the corresponding pressure from the fire flow test. The hydrant constant uses field data to characterize the properties of the hydrant and the connecting pipe. Use the Hose and Nozzle Constants tool to calculate.

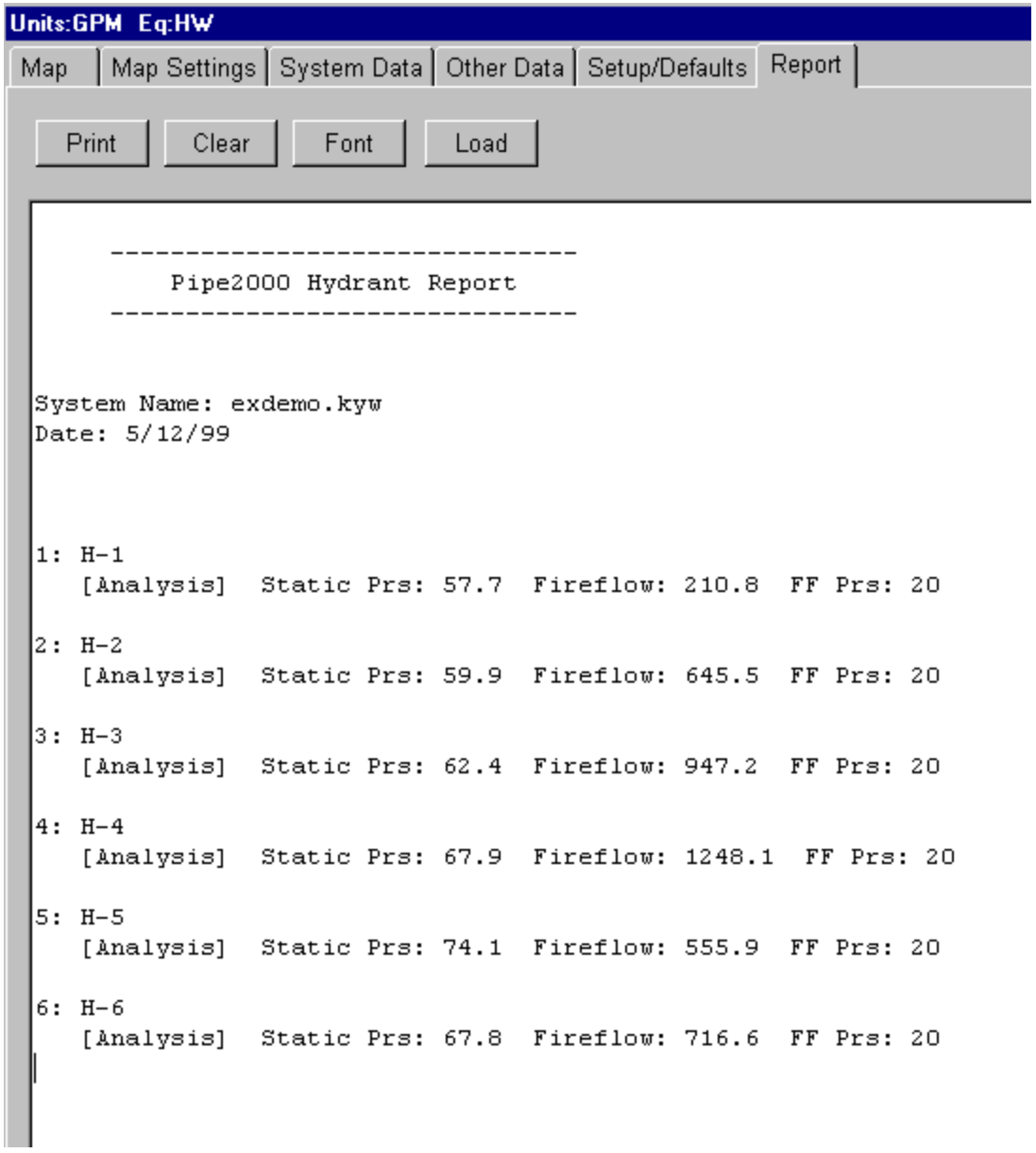
An appropriate example Suction Pressure is 5 psi.

If 0 is input, then the data is ignored (no fire pump suction pressure requirement). The calculation looks only at the 20 psi requirement in the distribution system. It is important to remember that some predicted fire flows may be too large and impossible to attain.

## Showing Fire Flow Results

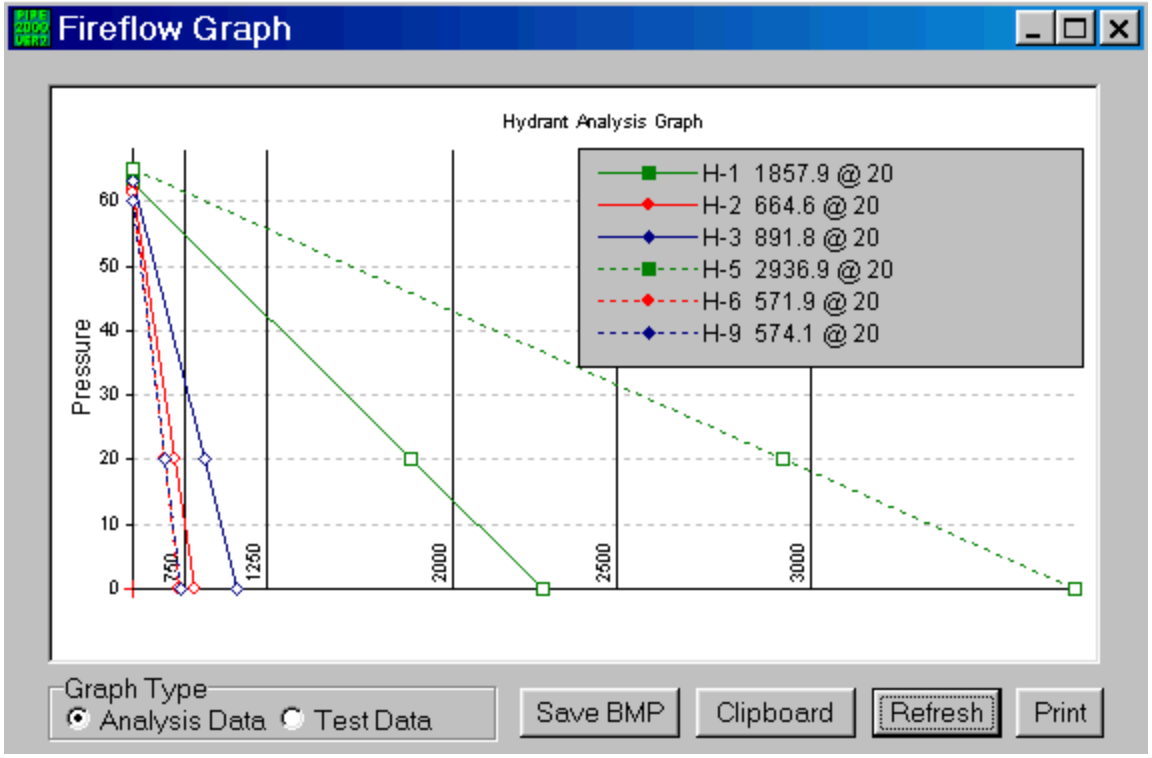
### Hydrant Report

There are a number of ways the fire flow results can be presented. Some of these will only apply to calculations for hydrant nodes while others are available for both hydrant and junction calculations. When an analysis has been done for hydrants (not junctions), Pipe2018 generates a Hydrant Report. To access this click Facilities Management in the Main Menu and select Hydrant Report. The report shown below appears. This report will contain test data and additional user data which is provided - address, manufacturer, etc.


Units:GPM Eq:HW
Map
Map Settings
System Data
Other Data
Setup/Defaults
Report
Print
Clear
Font
Load
Pipe2000 Hydrant Report
System Name: exdemo.kyw
Date: 5/12/99
1: H-1
[Analysis] Static Prs: 57.7 Fireflow: 210.8 FF Prs: 20
2: H-2
[Analysis] Static Prs: 59.9 Fireflow: 645.5 FF Prs: 20
3: H-3
[Analysis] Static Prs: 62.4 Fireflow: 947.2 FF Prs: 20
4: H-4
[Analysis] Static Prs: 67.9 Fireflow: 1248.1 FF Prs: 20
5: H-5
[Analysis] Static Prs: 74.1 Fireflow: 555.9 FF Prs: 20
6: H-6
[Analysis] Static Prs: 67.8 Fireflow: 716.6 FF Prs: 20

### Fireflow Graphs

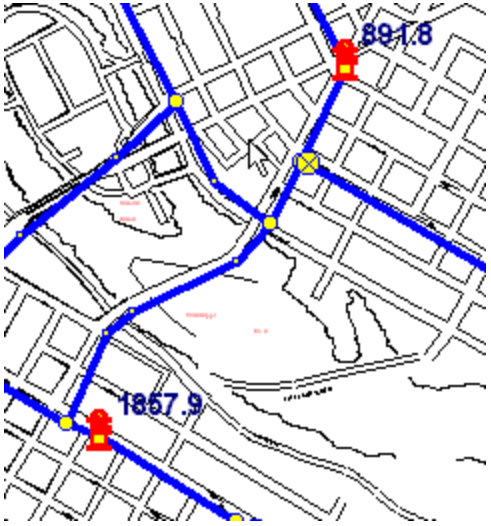
For a hydrant analysis, Pipe2018 generates a Fireflow Graph. Click on Facilities Management and select Graph Hydrants. The graph shown below appears. You can plot either calculated (analysis) or test data.




### Fireflow Labels

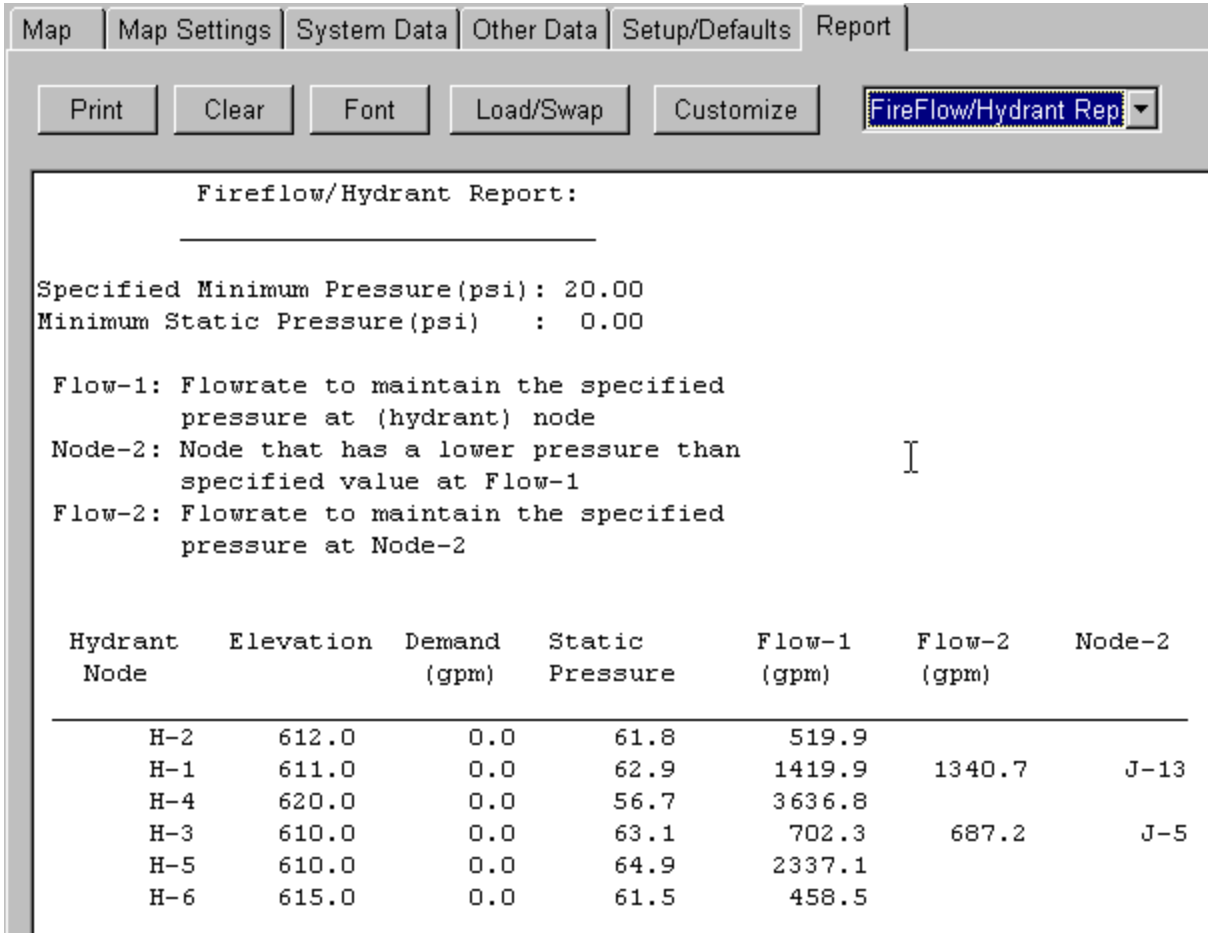
Another useful way to display Fire Flow Analysis results is as follows. In the Map screen, click Labels in the Main Menu and display Node Results A. Using the Results Selector bar for Nodes, display the Flow results.

The hydrant flow results will appear next to each hydrant or junction for which an analysis was conducted.



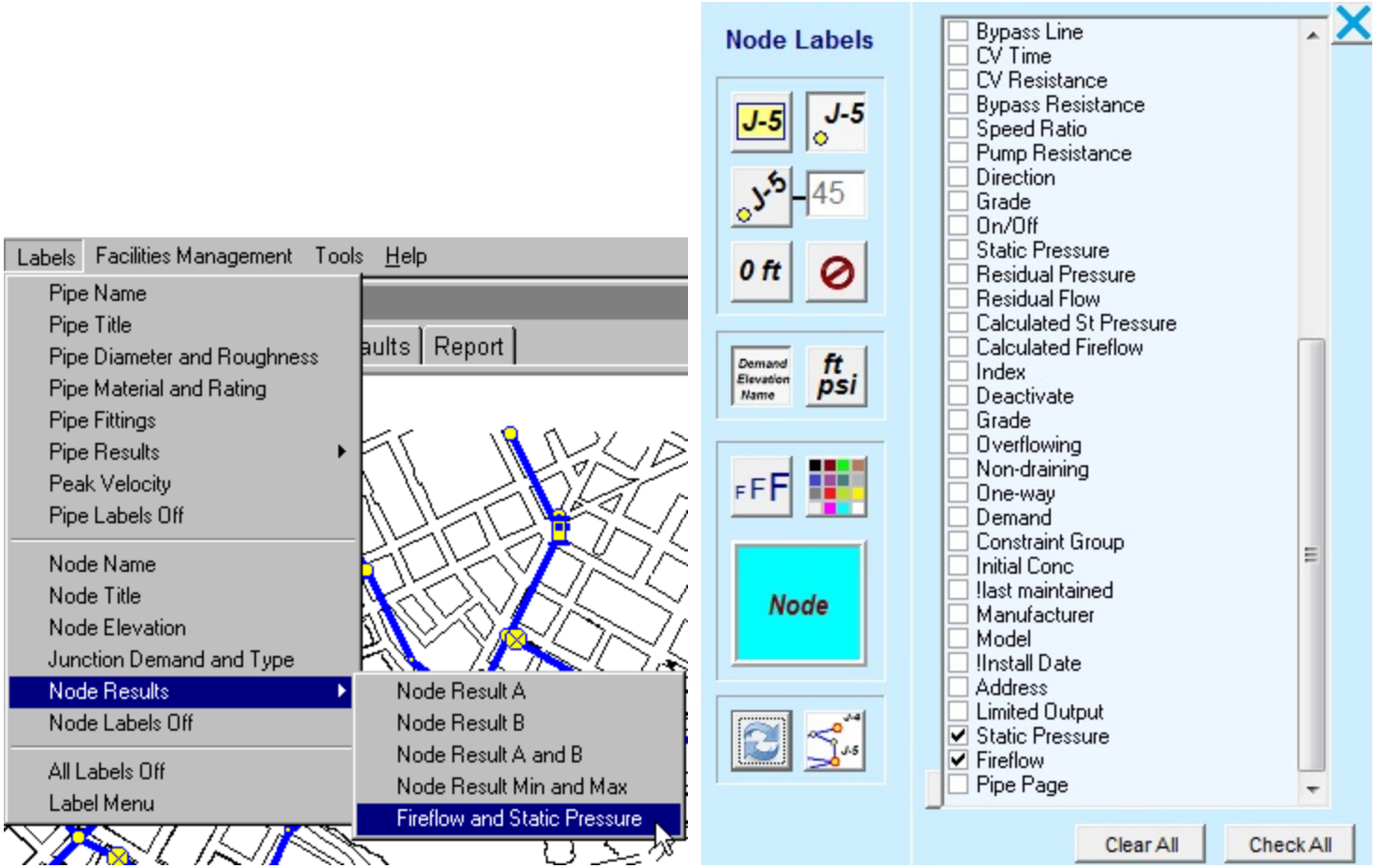

### Fireflow/Hydrant Report

There is a Fireflow/Hydrant Report that is included in the Report as shown below. This same report is generated for fireflow calculations at junction nodes and hydrants. When a junction in the system other than the specified hydrant has a lower pressure than is specified as the "Minimum Pressure for Fire Flows" (e.g. usually 20 psi) then that node and the flow for that node are noted in the last two columns.



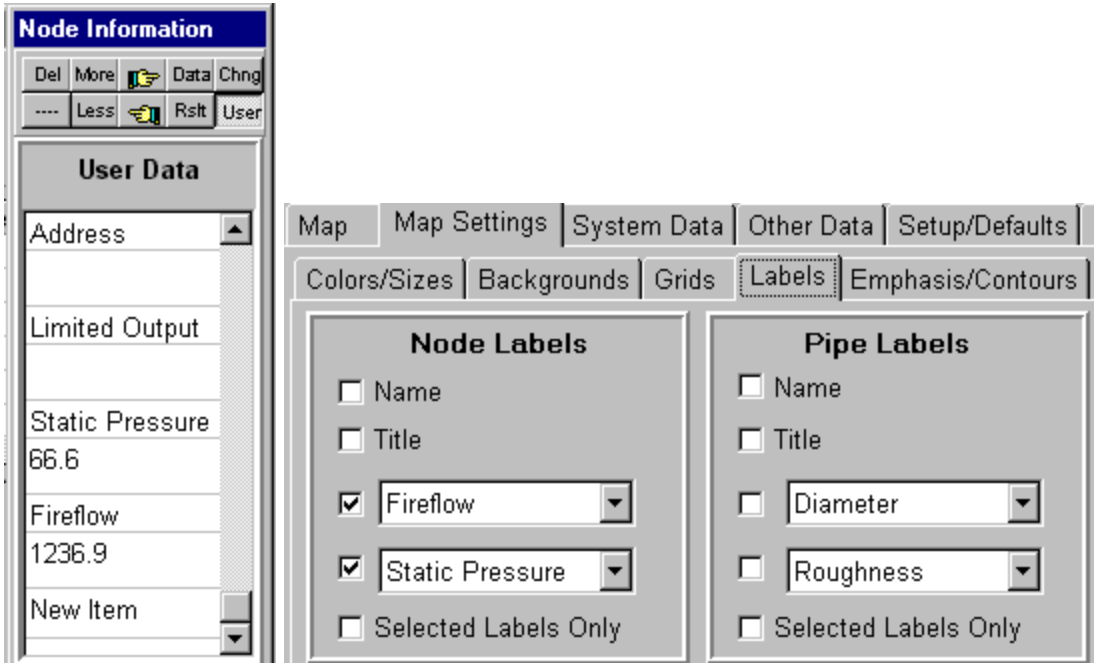

### Additional Considerations for Fireflows at Junction Nodes

For an analysis conducted on Junction Nodes, there are several ways to view these results. One of the easiest is to view these as map labels. Click on Labels (in the Main Menu) | Node Results | Fireflow and Static Pressure as shown below:

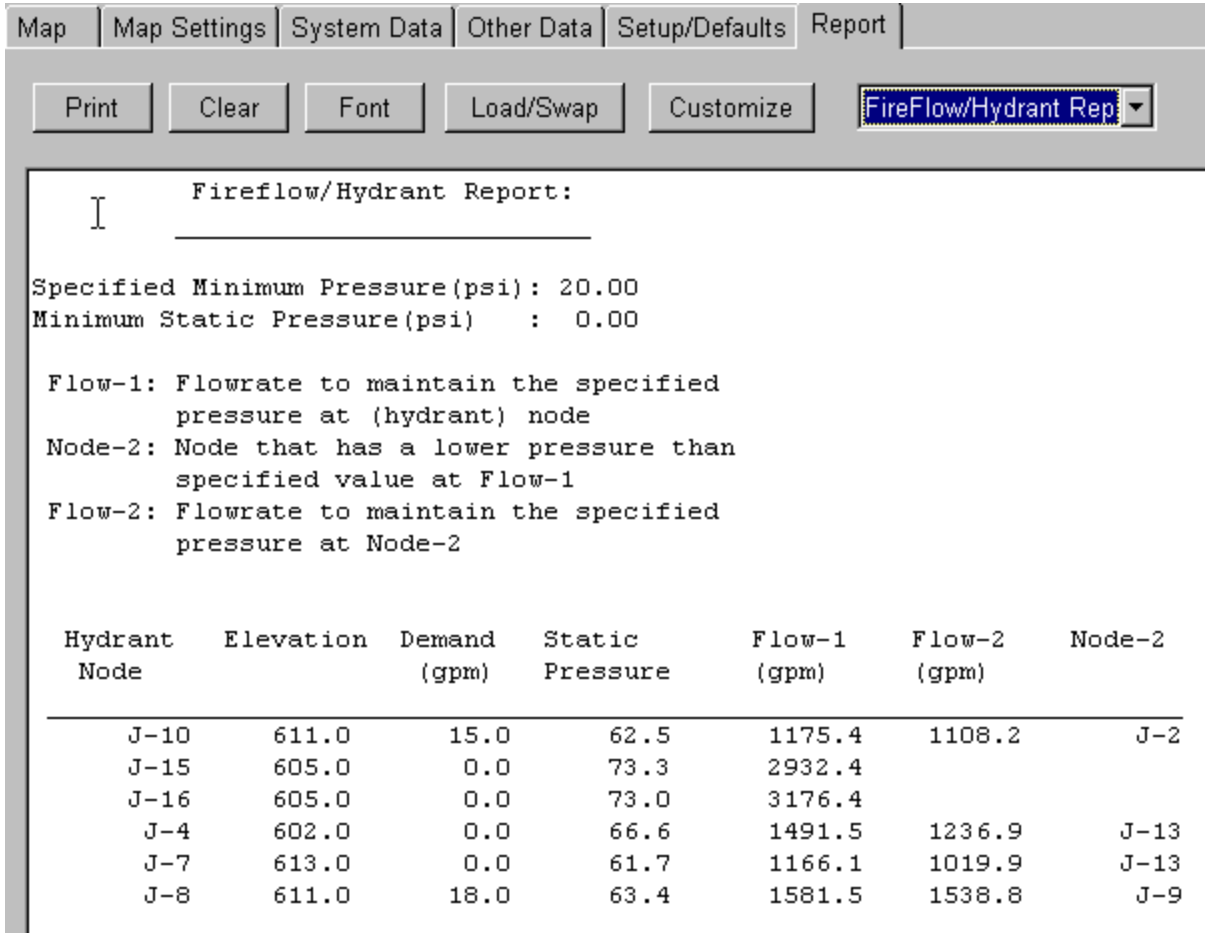



The Fireflow and Static Pressure results labels will automatically appear.

When a junction fireflow analysis is conducted, two User Data items are generated, Static Pressure and Fireflow. The results are stored in these User Data items. Displaying the map labels in the manner described above is a shortcut method of displaying the User Data items, Fireflow and Static Pressure, on the map. One could also go to Map Settings | Labels and select Fireflow and/or Static Pressure as the Node Labels to be displayed, as shown below. This options allows more versatility, such as the ability to combine other labels.




Lastly, there is a Fireflow/Hydrant Report for junction nodes that is included in the Report as shown below.




| Hydrant Node | Elevation | Demand (gpm) | Static Pressure | Flow-1 (gpm) | Flow-2 (gpm) | Node-2 |
|---|---|---|---|---|---|---|
| J-10 | 611.0 | 15.0 | 62.5 | 1175.4 | 1108.2 | J-2 |
| J-15 | 605.0 | 0.0 | 73.3 | 2932.4 | | |
| J-16 | 605.0 | 0.0 | 73.0 | 3176.4 | | |
| J-4 | 602.0 | 0.0 | 66.6 | 1491.5 | 1236.9 | J-13 |
| J-7 | 613.0 | 0.0 | 61.7 | 1166.1 | 1019.9 | J-13 |
| J-8 | 611.0 | 18.0 | 63.4 | 1581.5 | 1538.8 | J-9 |

## NFPA Hydrant Color Coding

The last column in the Fireflow/Hydrant Report shows the NFPA Hydrant Color Coding. If you are using flow units other than gpm, the value is converted to gpm for the purpose of assigning the color coding. The color ranges are as follows:

< 500 gpm = Red

<= 1000 gpm = Orange

< 1500 gpm = Green

>= 1500 gpm = Blue

To display the NFPA Hydrant Color Coding as a node emphasis on the map screen, go to Map Settings | Emphasis/Contours and press the "NFPA Hydrant Color Coding" button. You may also click Ctrl Alt H.

## 1.4.4.2 Flushing Pipes

Flushing Pipes

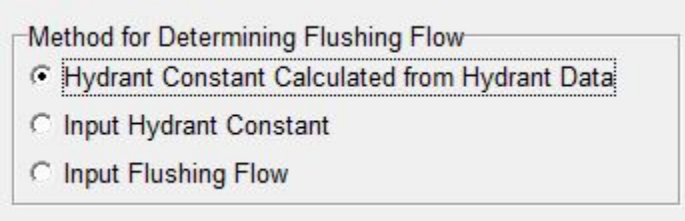
---

### Flushing Module

Pipe2018 provides a flushing analysis which identifies pipes which attain a specified velocity when a flushing scenario is analyzed. To utilize this capability the user should do the following:

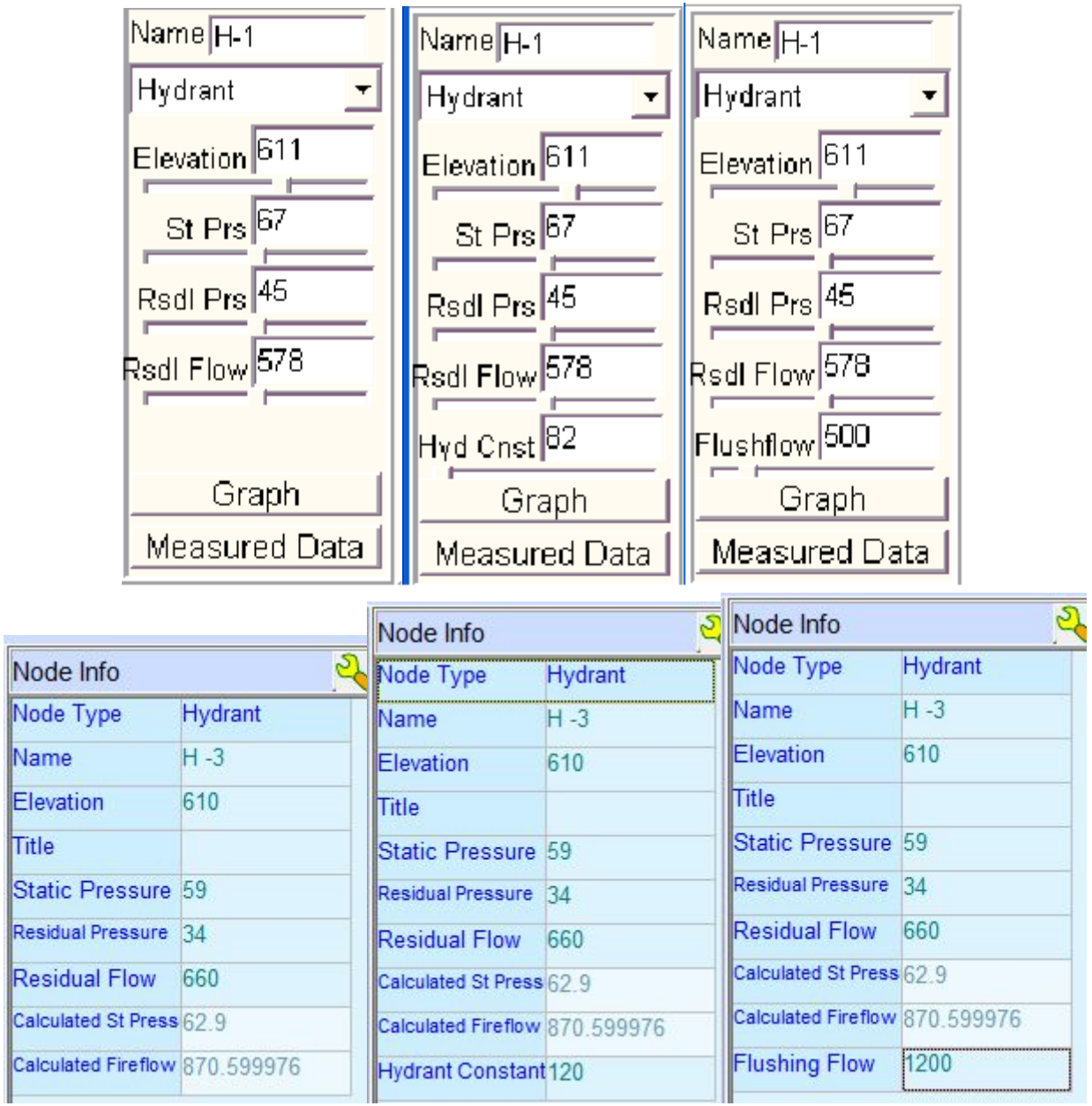
1) Select the method for calculating the flushing flow. This selection is made on the System/Other data screen as shown below.
2) Close valves to isolate desired areas
3) In Group Mode choose hydrants (one or more) to be flowed
4) Run the Flush Pipe Analysis (Analysis/Other screen)
5) Review results. If necessary make adjustments and rerun analysis.

The methods available for calculating the flushing flow are shown below.




**Data Requirements**

Each of these methods have unique input data requirements which are illustrated on the Hydrant Data screens shown below respectlively.


Name H-1
Hydrant
Elevation 611
St Prs 67
Rsdl Prs 45
Rsdl Flow 578
Graph
Measured Data
Name H-1
Hydrant
Elevation 611
St Prs 67
Rsdl Prs 45
Rsdl Flow 578
Hyd Cnst 82
Graph
Measured Data
Name H-1
Hydrant
Elevation 611
St Prs 67
Rsdl Prs 45
Rsdl Flow 578
Flushflow 500
Graph
Measured Data
Node Info
Node Type Hydrant
Name H -3
Elevation 610
Title
Static Pressure 59
Residual Pressure 34
Residual Flow 660
Calculated St Press 62.9
Calculated Fireflow 870.599976
Node Info
Node Type Hydrant
Name H -3
Elevation 610
Title
Static Pressure 59
Residual Pressure 34
Residual Flow 660
Calculated St Press 62.9
Calculated Fireflow 870.599976
Hydrant Constant 120
Node Info
Node Type Hydrant
Name H -3
Elevation 610
Title
Static Pressure 59
Residual Pressure 34
Residual Flow 660
Calculated St Press 62.9
Calculated Fireflow 870.599976
Flushing Flow 1200

### 1. Hydrant Constant Calculated from Hydrant Data

For this approach the hydrant test data is used to calculate a hydrant constant. The hydrant constant is defined by the equation

$Qr = K_{h\,(}P_{r)\wedge 0.5}$

where $Q_r$ is the residual flow in gpm (rps) and Pr is the corresponding pressure from the fire flow test. The hydrant constant uses field data to characterize the properties of the hydrant and the connecting pipe. For the example shown the hydrant constant is

$K_h = 578/((45)\wedge 0.5) = 86$

### 2. Input Hydrant Constant

If test data is not available, a hydrant constant can be calculated using the properties of the hydrant and connecting pipe. This is a function of the diameter, length, and number of elbows in the connection to the hydrant, the size of the hydrant orifice and the elevation difference between the connection to the distribution system and the hydrant orifice. A tool is available to calculate the hydrant constant based on this data.

### 3. Input Flushing Flow

This option requires the user to input a value for flushing flow (gpm) for each hydrant that is used in the flushing analysis. Note that flushing flow should not exceed the amount which can be obtained by opening the hydrant.

### Running a Flushing Scenario

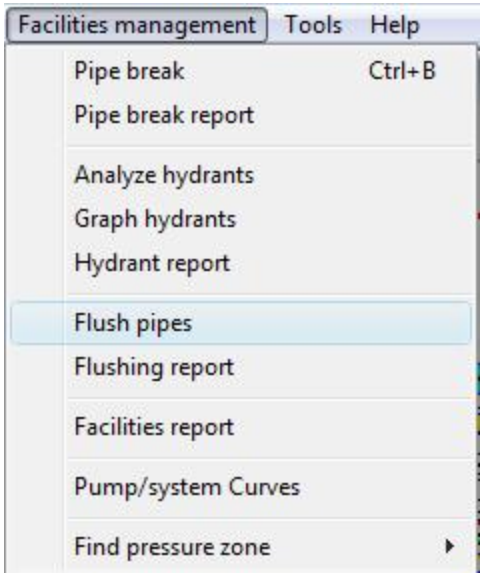
For all three methods the procedure is the same and the set up is illustrated below.

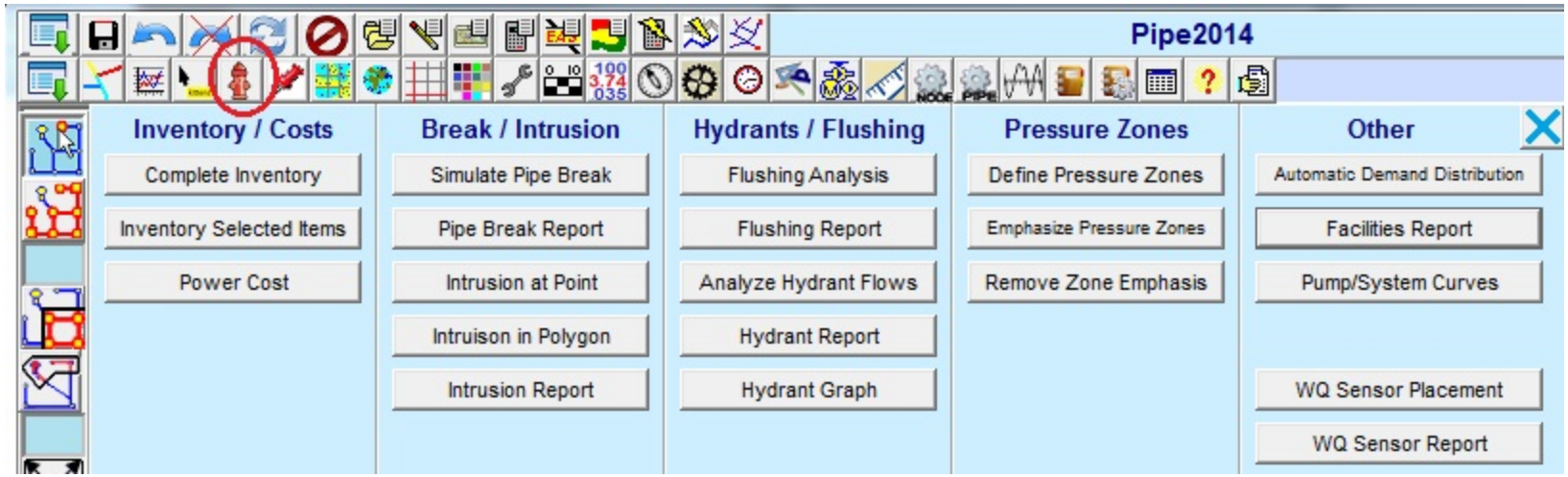
**1) Close pipes to isolate desired region**. The closed valves are noted with the red x and dashed pipe. To close a valve select the valve and click the ON/OFF switch (Node Information). Note this can be done effectively in Group Mode. In they mode select all valves to be closed and then select the off switch. Note that a pipe can also be closed by selecting the pipe and clicking on the Closed button. This closed pipe will

appear as dashed.

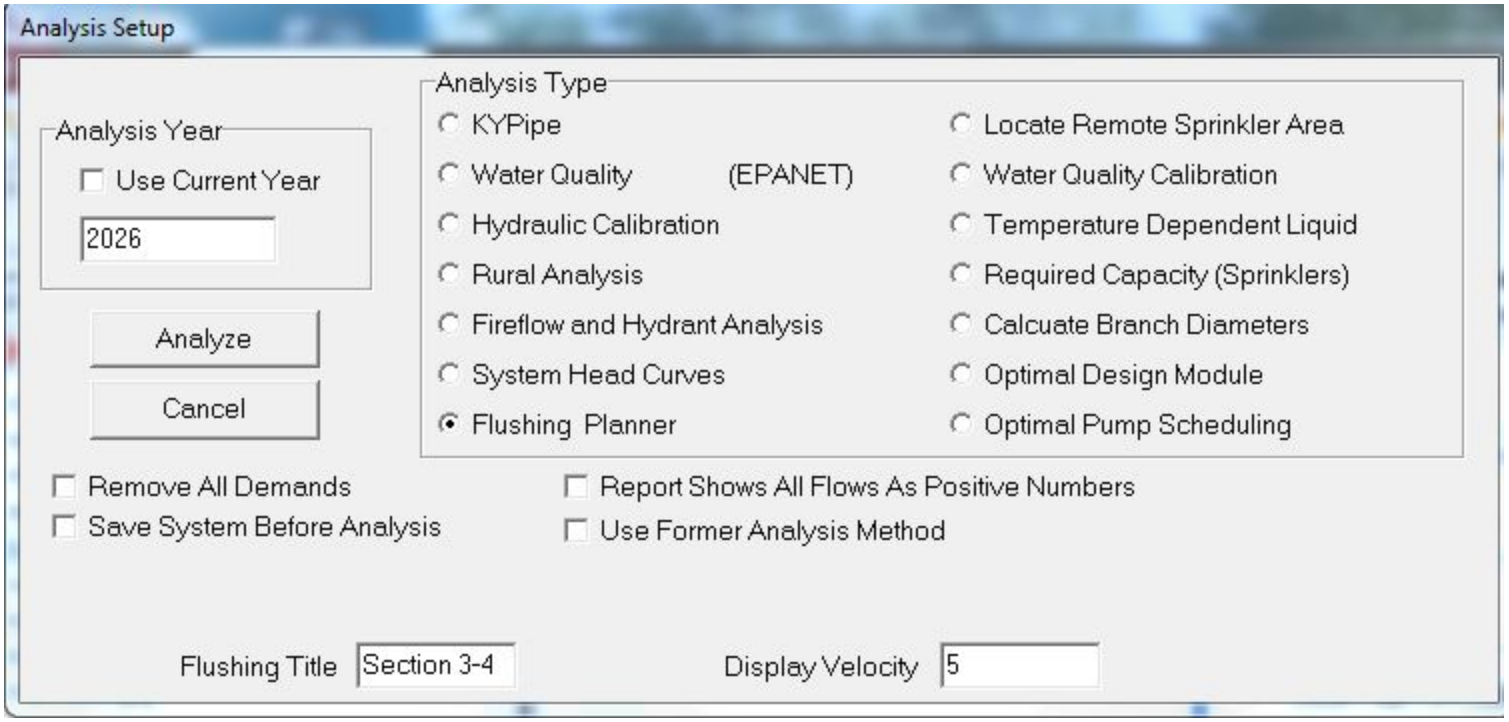
**2) Go to Group Mode and select one or more hydrants to be flowed**. You are now ready to run the analysis.

**3) Run the Flush Pipe Analysis (Analyze | Analyze).** You will get the screen shown below. At this time you need to select the desired flushing velocity (Display Velocity) and provide a title for this scenario. Once you run the analysis you will note a red **Clear Flush** displayed in the upper left corner. While in this mode you can adjust conditions which include:



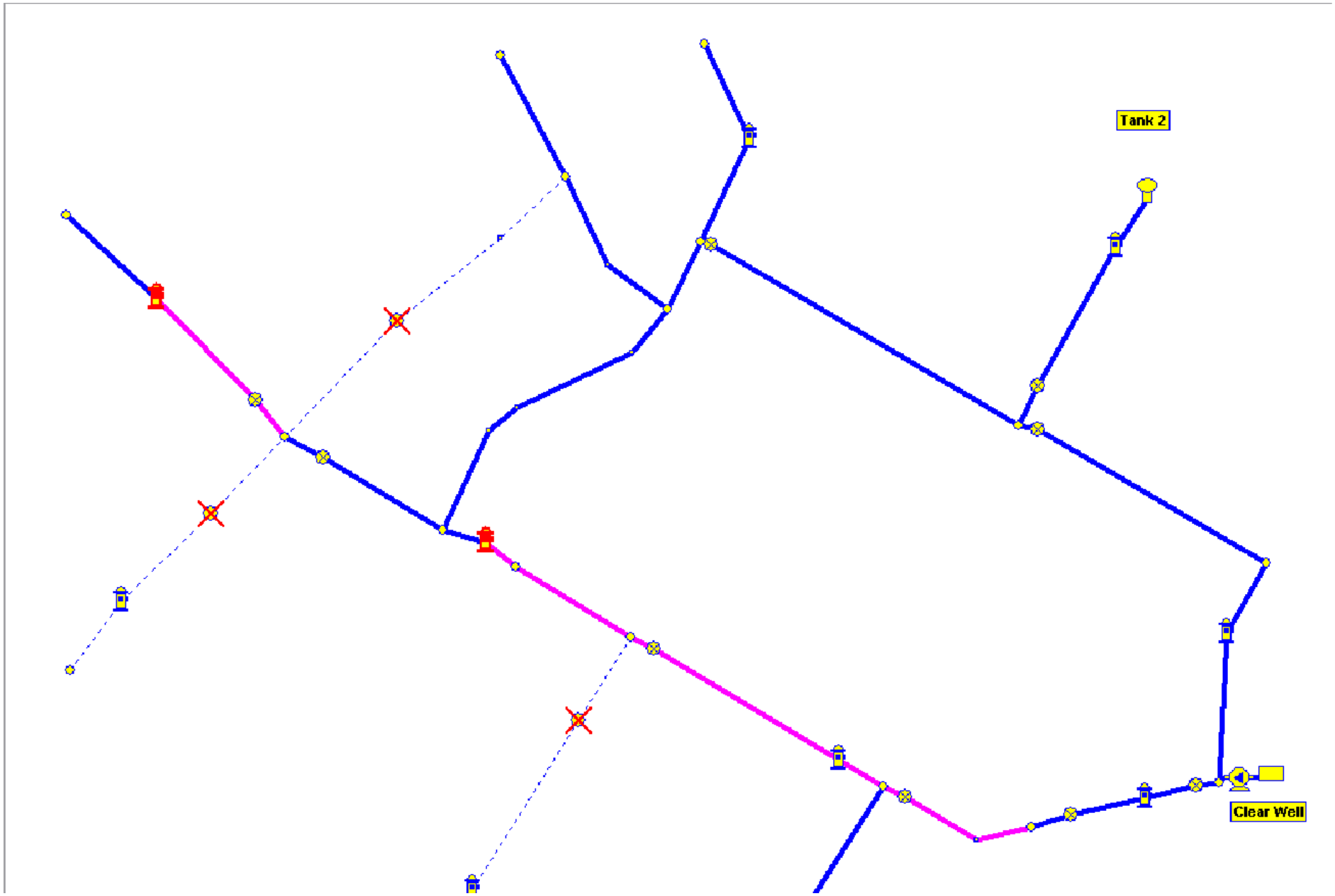


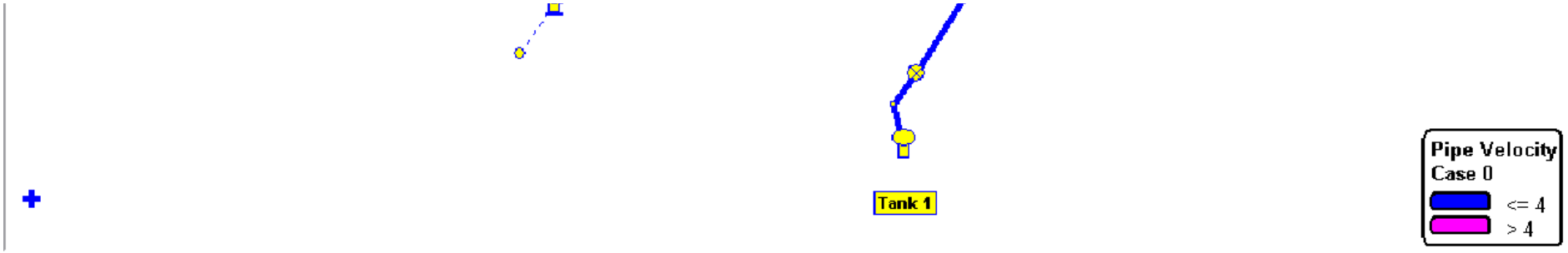

**Flushing Pipes Analysis Set Up**




1) which valves are closed
2) which hydrants are selected
3) the hydrant data (hydrant constant a flush flow)

The analysis can be rerun and adjustments made until the desired results are attained. At this time you can print a display of the flushed pipes shown below.

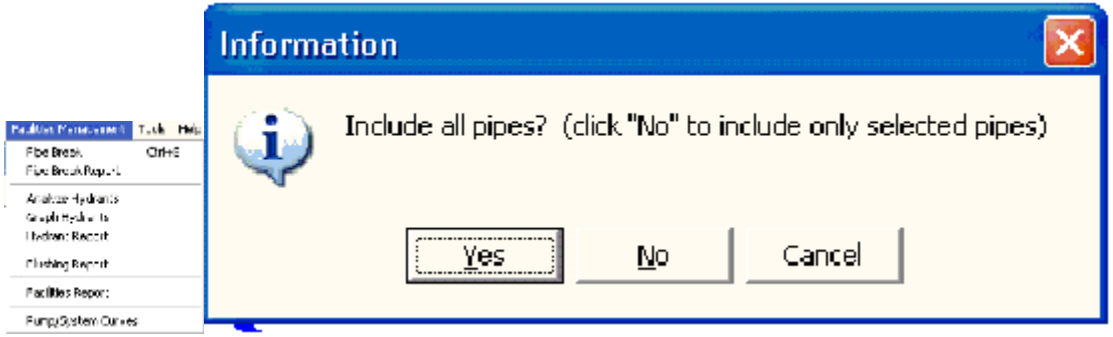

Tank 2
Clear Well




**Pipe2018 Display Showing Flushed Pipes**

You can print the Pipe2018 flushing report (Facilties Management/Flushing Report) which lists the flushed pipes. Because other pipes in the distribution system may attain the flushing velocity you will want to limit the report to pipes in the region of interest. To do this you should select the region of interest before accessing the Flushing Report and choose the option to include only "Selected Pipes" as shown on the next page.

When you have finished a scenario you can then click on **Clear Flush** and initiate a new scenario.



```
     ---------------------------------
       Pipe2000 Pipe Flushing Report
     ---------------------------------

section 3-4

System Name: Flush.p2k

Date: 11/10/2002

Selected Hydrants:

    Name                 Flow
    ----                 ----
    H-1                  582
    H-2                  434


Desired Flushing Velocity: 4


The following pipes were flushed:

    Pipe                 Travel Time(sec)        Velocity
    ----                 ----------------        --------
    P-11                 313                     5.4
    P-12                 349                     4.2
    P-22                 296                     4.1
    P-23                 81                      4.1
    P-8                  612                     4.3
```

```
The following valves were turned off:

1: V-11
   1165 Main

2: V-12
   2134 Main

3: V-13
   4321 Third
```

**Pipe2018 Flushing Report**

## 1.4.4.3 Hydrants, Fire Flows, and Flushing Pipes

Hydrants, Fire Flows, and Flushing

---

**Hydrants, Fire Flows, and Flushing**

Pipe2018 models fire hydrants and provides the capability of plotting field hydrant flow data, calculating fire flows and maintaining records using the model.

- Hydrant Test Data and Fire Flow Plots
- Fire Flows (calculated)
- Flushing Pipes Module

## 1.4.5 Low Pressure Sewer Models



# Setting up a LPS Pipe System

A number of features are included in Pipe2018 for the purpose of speeding up and simplifying the development of a LPS pipe system model. These features are illustrated in this document. The following steps are recommended for setting up your LPS model.

**1)** Select and Open New Pipe2018 file with desired Pipe Type Table and pump ID's.
2) Add background map showing lots and elevation contours
3) Layout LPS pipe collection system with following:
   4) all main collection pipes
   5) dead end lines with junction node for selected pumps
   6) Collection reservoir
7) Input elevations for all nodes
8) Input number of Residential Connections (meters) for each pipe (these inputs should add up to the total number of lots)
9) (Optional) Use the **Calculate Branch Diameters Module (Premium version feature)** to calculate design flows, size pipes and create an updated data file with these diameters assigned
10) Use **Select End Node Junctions (Edit)** feature to select all pump locations
11) Use Group Edit to change selected nodes **to pumps**
12) Select a subset of pumps (**Group Select – Select from Group**) which will all be assigned the same ID
13) Use Group Edit to assign pump ID from ones available with this template. The elevation and grade of the pumps will be assigned the node elevation
14) Repeat until all pumps are assigned an ID.
15) Evaluate pump performances
   16) Turn off selected pumps and run a KYPIPE (hydraulic analysis)
   17) Show Pump Status (Ctrl-Alt-P) This shows ON (green), OFF (red) and ON with no flow (yellow). Details appear in the Pump Report

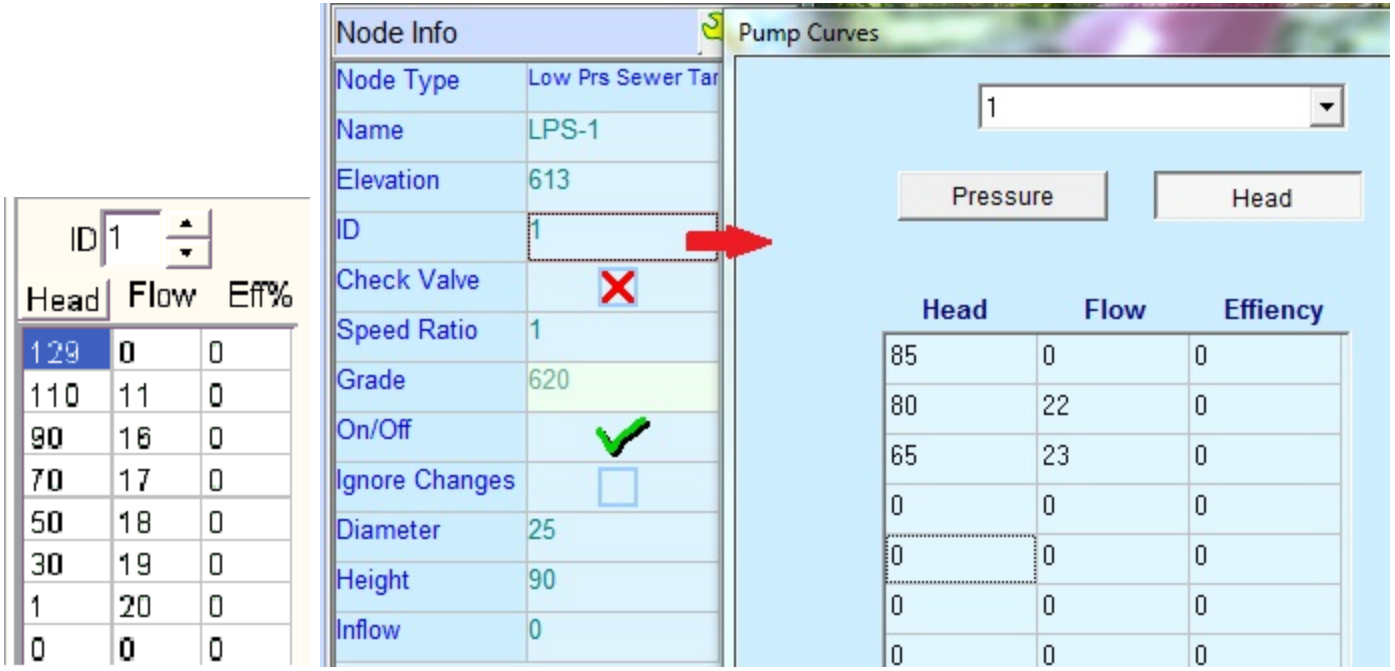
# Modeling LPS Systems Using Pipe2018

The following example and detailed descriptions provide a Users Guide for LPS model development using Pipe2018:

**1) Select and Open New Pipe2018 file with desired Pipe Type Table and pump ID's.** Go to Setups and Defaults/Pipe Type to load in desired table.

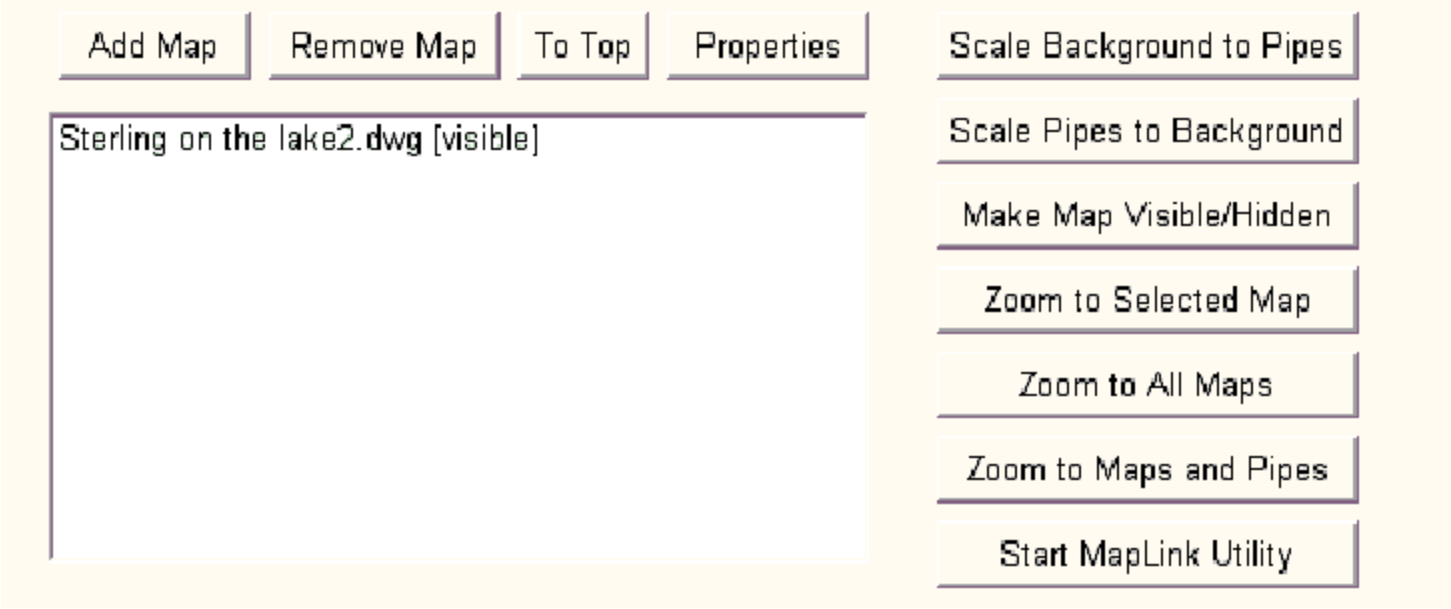
| | Material | Rating | Nominal Diameter | Actual Diameter | Unit Cost | Reference Roughness |
|---|---|---|---|---|---|---|
| 1 | | | | | | |
| 2 | | | | | | |
| 3 | DR-11 | 160 | 1.25 | 1.34 | | 120 |
| 4 | DR-11 | 160 | 1.5 | 1.533 | | 120 |
| 5 | DR-11 | 160 | 2 | 1.917 | | 120 |
| 6 | DR-11 | 160 | 2.5 | 2.411 | | 120 |
| 7 | DR-11 | 160 | 3 | 2.826 | | 120 |
| 8 | DR-11 | 160 | 4 | 3.633 | | 120 |
| 9 | DR-11 | 160 | 5 | 4.49 | | 120 |
| 10 | DR-11 | 160 | 6 | 5.349 | | 120 |
| 11 | DR-11 | 160 | 8 | 6.963 | | 120 |
| 12 | DR-11 | 160 | 10 | 8.679 | | 120 |
| 13 | DR-11 | 160 | 12 | 10.293 | | 120 |

**Pipe Type Table for LPS**

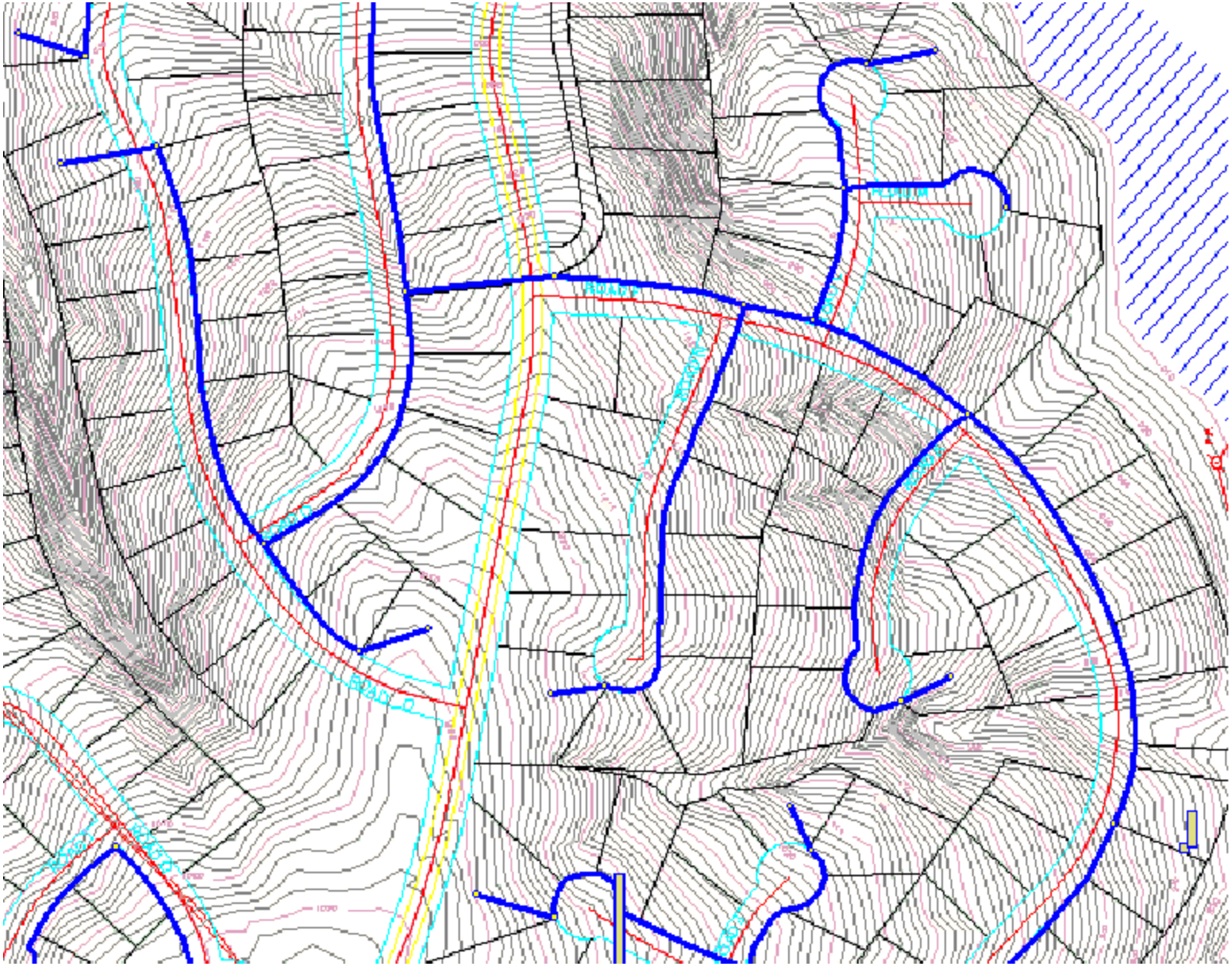


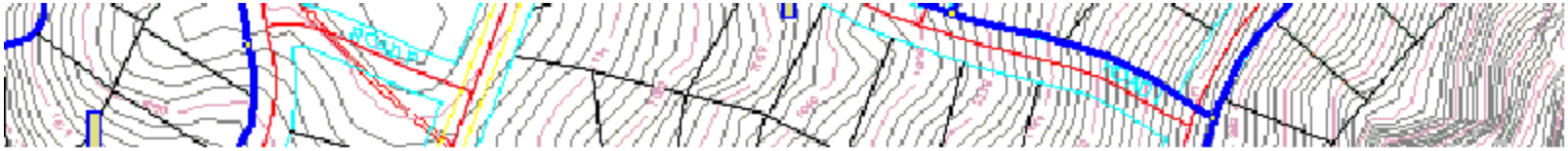

**Pump ID Classic and KYnetic**

2) **Add background map(Map Settings/Backgrounds) showing lots and elevation contours.** Click Add Map and locate with Browser



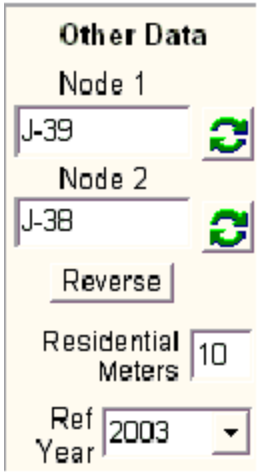

3) **Layout LPS pipe collection system with following:**
    2) **all main collection pipes**
    3) **dead end lines with junction node for selected pumps**
    4) **Collection reservoir**



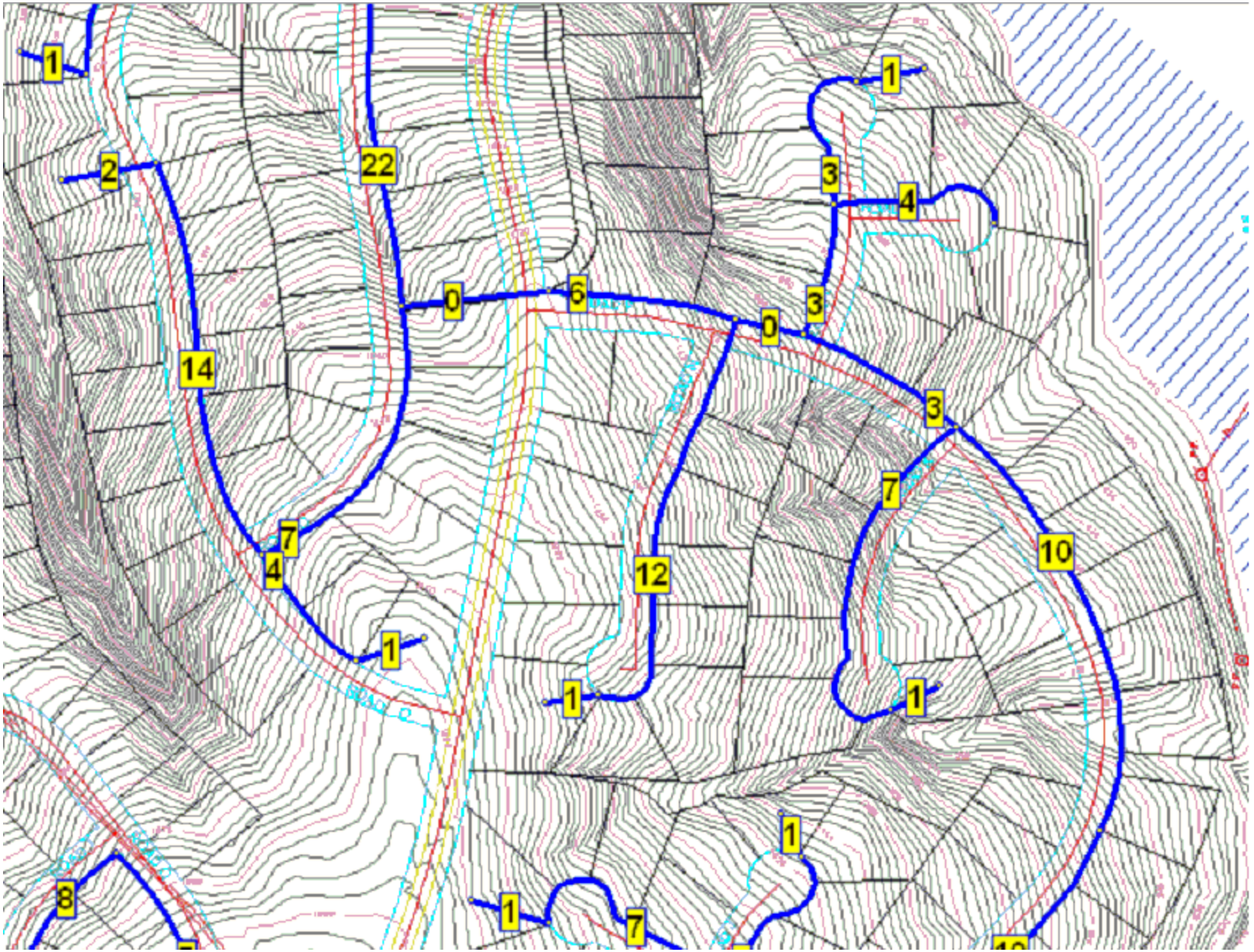


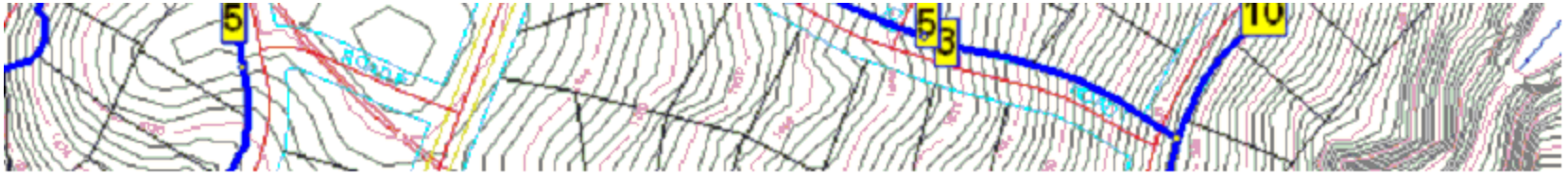
**LPS Collection System (1)**

3) **Input elevations for all nodes**
4) **Input number of Residential Connections (meters) for each pipe (these inputs should add up to the total number of lots)**



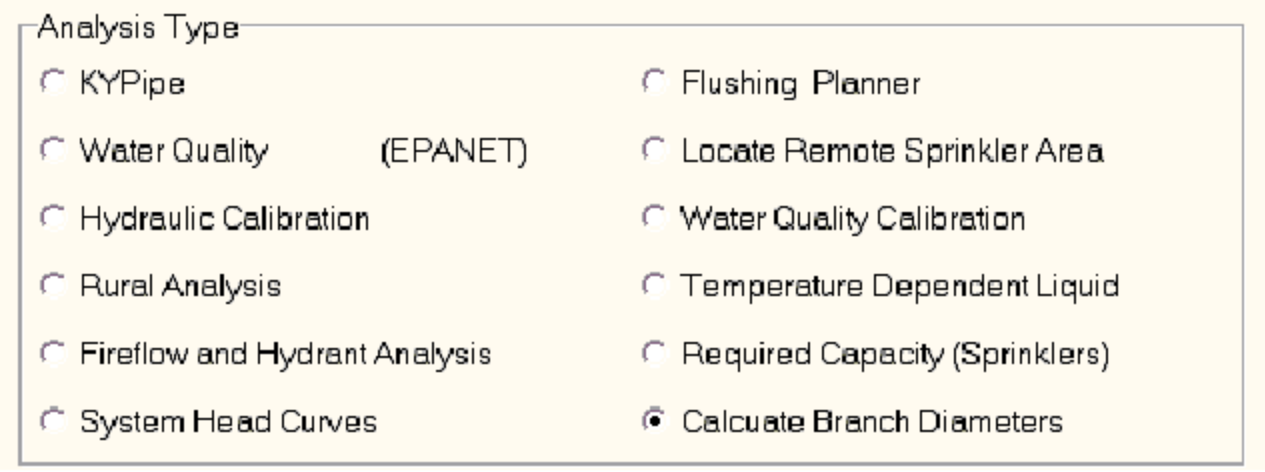

**Pipe Data - Input number of Residential connections (meters)**


1
2
22
14
0
6
0
3
4
1
3
3
7
10
7
4
12
1
1
1
1
8
1
7




**Pipe2018 plot showing Number of connections to each pipe in LPS**

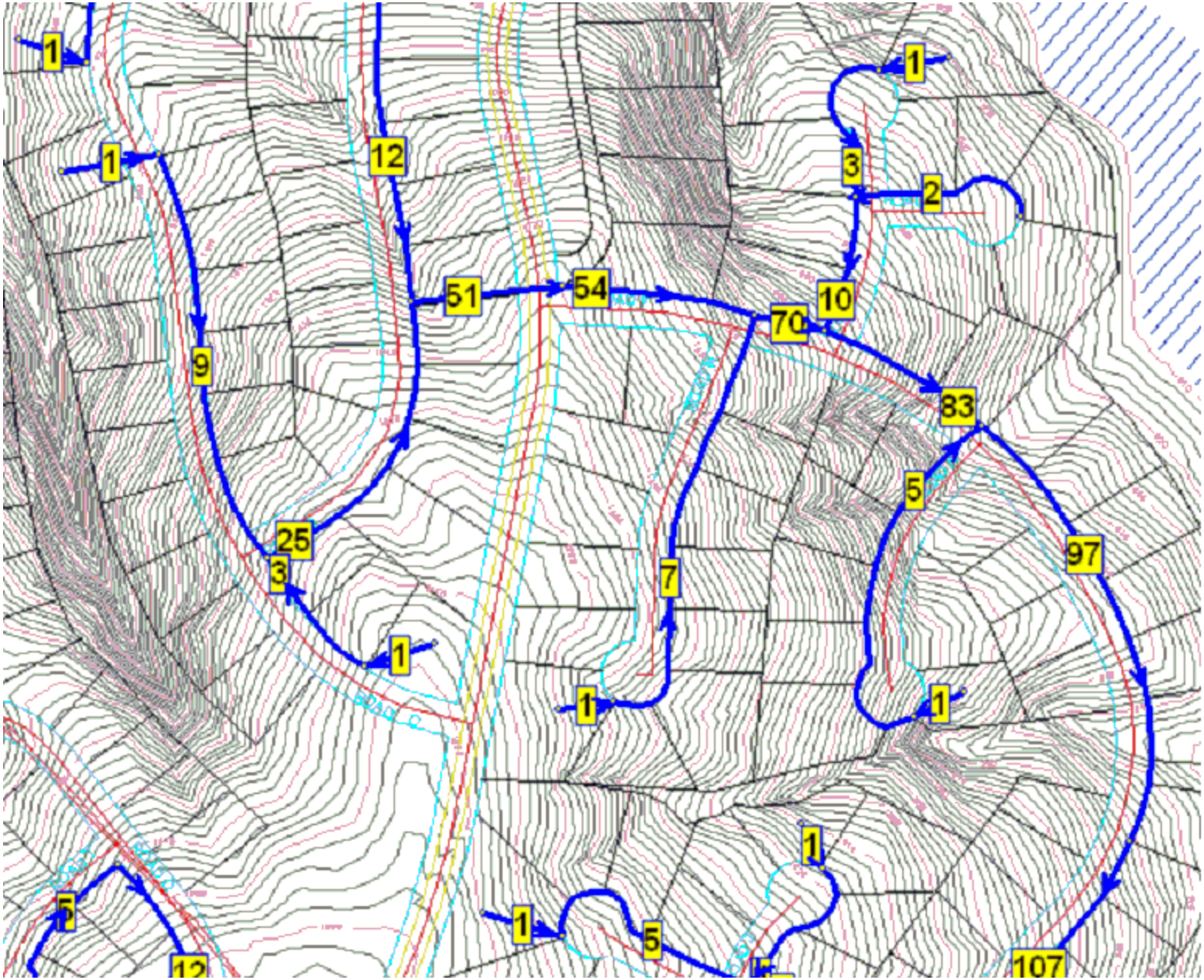
5) **Use the Calculate Branch Diameters Module to calculate design flows, size pipes and create an updated data file with these diameters assigned.**

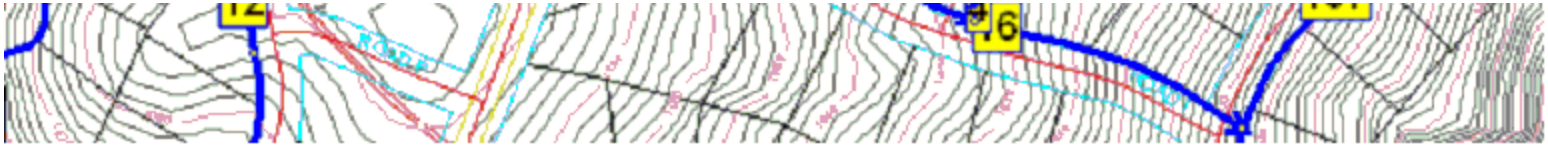



**Calculate Branch Diameters Module (6)**

**In the Calculate Branch Diameters Module Setup Screen, choose the Method to be used to calculate peak flow:**

**1) Flow = (Ave. Consumption*Peak Factor/1440)* N**
**2) Flow = A*N + B**
**N = number connections**


1
1
12
51
54
9
25
3
1
7
1
70
10
3
2
1
83
5
97
1
1
5
1
5
107




**Number of connections served by each pipe (calculated by the Calculate Branch Diameters Module**

**Branch Lines Flow and Pipe Sizing Calculations**

Copyright: April 2005
by
Don J. Wood
Srini Lingireddy

Date: 04-22-2005
Time: 10:09:52

***** SUMMARY OF PEAK FLOW REQUIREMENTS *****

| PIPE NO. | NODE #1 | NODE #2 | No. Connections Served | Design Flow | Required Diameter | Design Velocity |
|---|---|---|---|---|---|---|
| P-1 | J-2 | J-4 | 10 | 6 | 1.5 | 1.26 |
| P-10 | J-13 | J-7 | 502 | 348 | 6 | 3.95 |
| P-11 | J-7 | J-51 | 516 | 358 | 6 | 4.06 |
| P-12 | J-50 | J-49 | 28 | 19 | 1.5 | 3.53 |
| P-13 | J-11 | J-5 | 15 | 10 | 1.5 | 1.89 |
| P-14 | J-9 | J-11 | 5 | 3 | 1.5 | 0.63 |
| P-15 | J-92 | J-11 | 5 | 3 | 1.5 | 0.63 |
| P-16 | J-84 | J-92 | 1 | 0 | 1.5 | 0.12 |
| P-17 | J-6 | J-13 | 484 | 336 | 6 | 3.81 |
| P-18 | J-15 | J-13 | 12 | 8 | 1.5 | 1.51 |
| P-19 | J-85 | J-15 | 1 | 0 | 1.5 | 0.12 |
| P-2 | J-66 | J-2 | 1 | 0 | 1.5 | 0.12 |
| P-20 | J-17 | J-6 | 479 | 332 | 6 | 3.77 |
| P-21 | J-10 | J-14 | 52 | 36 | 2 | 3.68 |
| P-22 | J-18 | J-14 | 2 | 1 | 1.5 | 0.25 |
| P-23 | J-14 | J-34 | 133 | 92 | 3 | 4.19 |

| | | | | | | |
|---|---|---|---|---|---|---|
| P-24 | J-1 | J-14 | 75 | 52 | 2.5 | 3.4 |
| P-25 | J-21 | J-22 | 88 | 61 | 2.5 | 3.99 |
| P-26 | J-24 | J-19 | 14 | 9 | 1.5 | 1.76 |
| P-27 | J-23 | J-21 | 36 | 25 | 1.5 | 4.53 |
| P-28 | J-20 | J-27 | 25 | 17 | 1.5 | 3.15 |
| P-29 | J-19 | J-21 | 52 | 36 | 2 | 3.68 |
| P-3 | J-8 | DUPLEX | 597 | 414 | 6 | 4.7 |
| P-30 | J-87 | J-23 | 2 | 1 | 1.5 | 0.25 |
| P-31 | J-88 | J-26 | 2 | 1 | 1.5 | 0.25 |
| P-32 | J-22 | J-17 | 181 | 125 | 4 | 3.2 |
| P-33 | J-25 | J-20 | 6 | 4 | 1.5 | 0.75 |
| P-34 | J-89 | J-24 | 1 | 0 | 1.5 | 0.12 |

**Results for Pipe Sizing Module (Method 1)**

Branch Lines Flow and Pipe Sizing Calculations

Copyright: April 2005
by
Don J. Wood
Srini Lingireddy

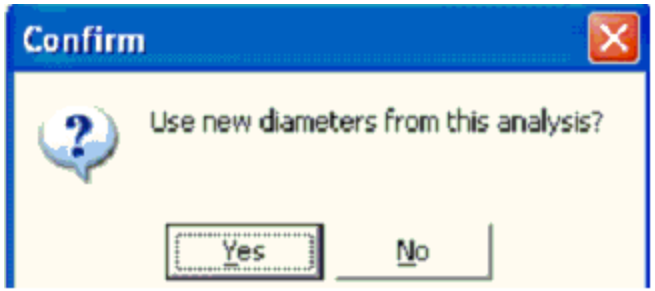
Date: 05-28-2005
Time: 09:47:38

***** SUMMARY OF PEAK FLOW REQUIREMENTS *****

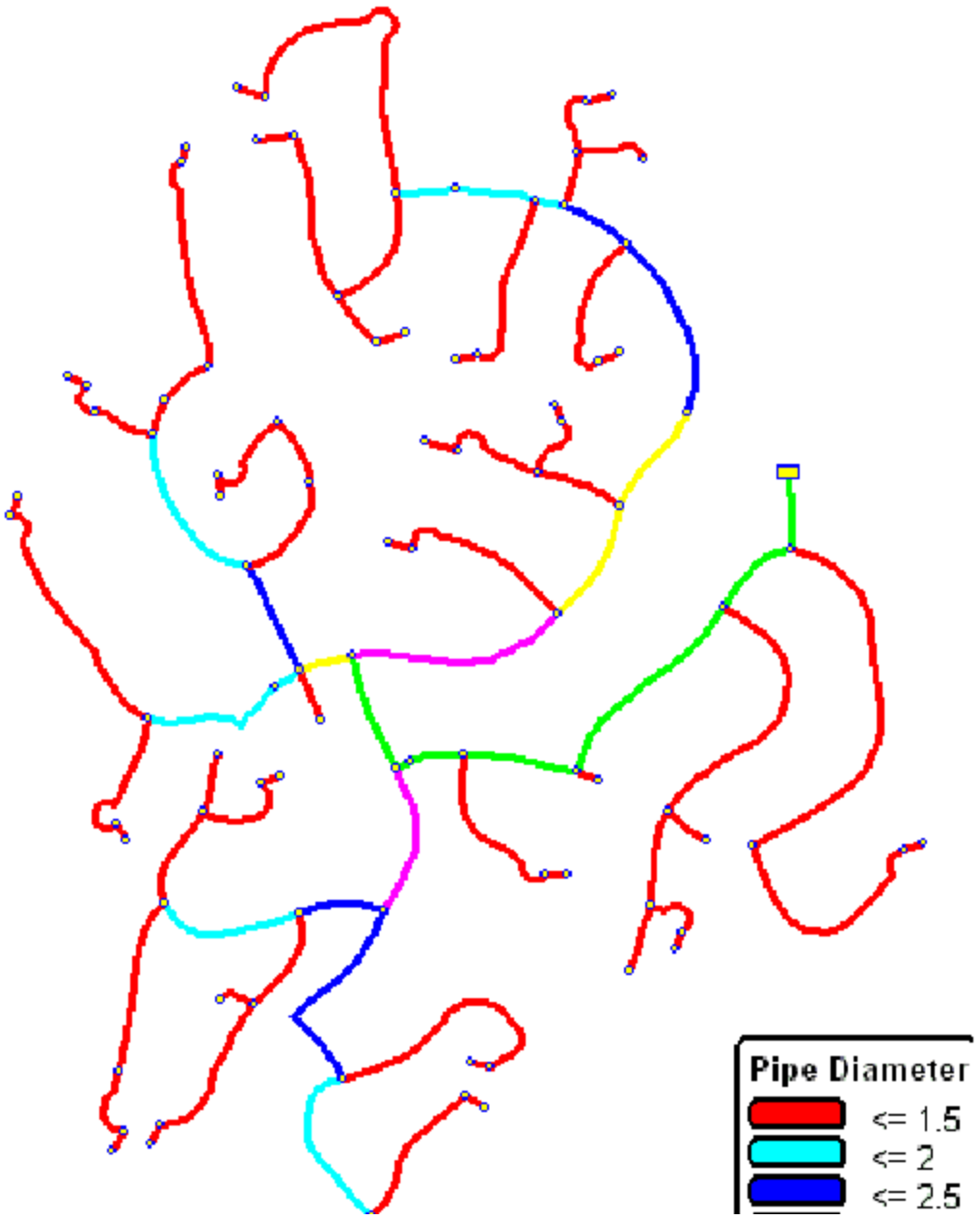
| PIPE NO. | NODE #1 | NODE #2 | No. Connections Served | Design Flow | Required Diameter | Design Velocity |
|---|---|---|---|---|---|---|
| P-1 | J-2 | J-4 | 10 | 16 | 1.5 | 2.9 |
| P-10 | J-13 | J-7 | 502 | 262 | 6 | 2.97 |
| P-11 | J-7 | J-51 | 516 | 269 | 6 | 3.05 |
| P-12 | J-50 | J-49 | 28 | 25 | 1.5 | 4.53 |
| P-13 | J-11 | J-5 | 15 | 18 | 1.5 | 3.35 |
| P-14 | J-9 | J-11 | 5 | 13 | 1.5 | 2.45 |
| P-15 | J-92 | J-11 | 5 | 13 | 1.5 | 2.45 |
| P-16 | J-84 | J-92 | 1 | 11 | 1.5 | 2.08 |
| P-17 | J-6 | J-13 | 484 | 253 | 6 | 2.87 |
| P-18 | J-15 | J-13 | 12 | 17 | 1.5 | 3.08 |
| P-19 | J-85 | J-15 | 1 | 11 | 1.5 | 2.08 |
| P-2 | J-66 | J-2 | 1 | 11 | 1.5 | 2.08 |
| P-20 | J-17 | J-6 | 479 | 250 | 6 | 2.84 |
| P-21 | J-10 | J-14 | 52 | 37 | 2 | 3.77 |
| P-22 | J-18 | J-14 | 2 | 12 | 1.5 | 2.17 |
| P-23 | J-14 | J-34 | 133 | 77 | 3 | 3.51 |

| | | | | | | |
|---|---|---|---|---|---|---|
| P-24 | J-1 | J-14 | 75 | 48 | 2 | 4.95 |
| P-25 | J-21 | J-22 | 88 | 55 | 2.5 | 3.59 |
| P-26 | J-24 | J-19 | 14 | 18 | 1.5 | 3.26 |
| P-27 | J-23 | J-21 | 36 | 29 | 2 | 2.96 |
| P-28 | J-20 | J-27 | 25 | 23 | 1.5 | 4.26 |
| P-29 | J-19 | J-21 | 52 | 37 | 2 | 3.77 |
| P-3 | J-8 | DUPLEX | 597 | 309 | 6 | 3.51 |
| P-30 | J-87 | J-23 | 2 | 12 | 1.5 | 2.17 |
| P-31 | J-88 | J-26 | 2 | 12 | 1.5 | 2.17 |
| P-32 | J-22 | J-17 | 181 | 101 | 3 | 4.6 |
| P-33 | J-25 | J-20 | 6 | 14 | 1.5 | 2.54 |
| P-34 | J-89 | J-24 | 1 | 11 | 1.5 | 2.08 |

**Results for Pipe Sizing Module (Method 2)**




**Assign diameters based on Design Flows**

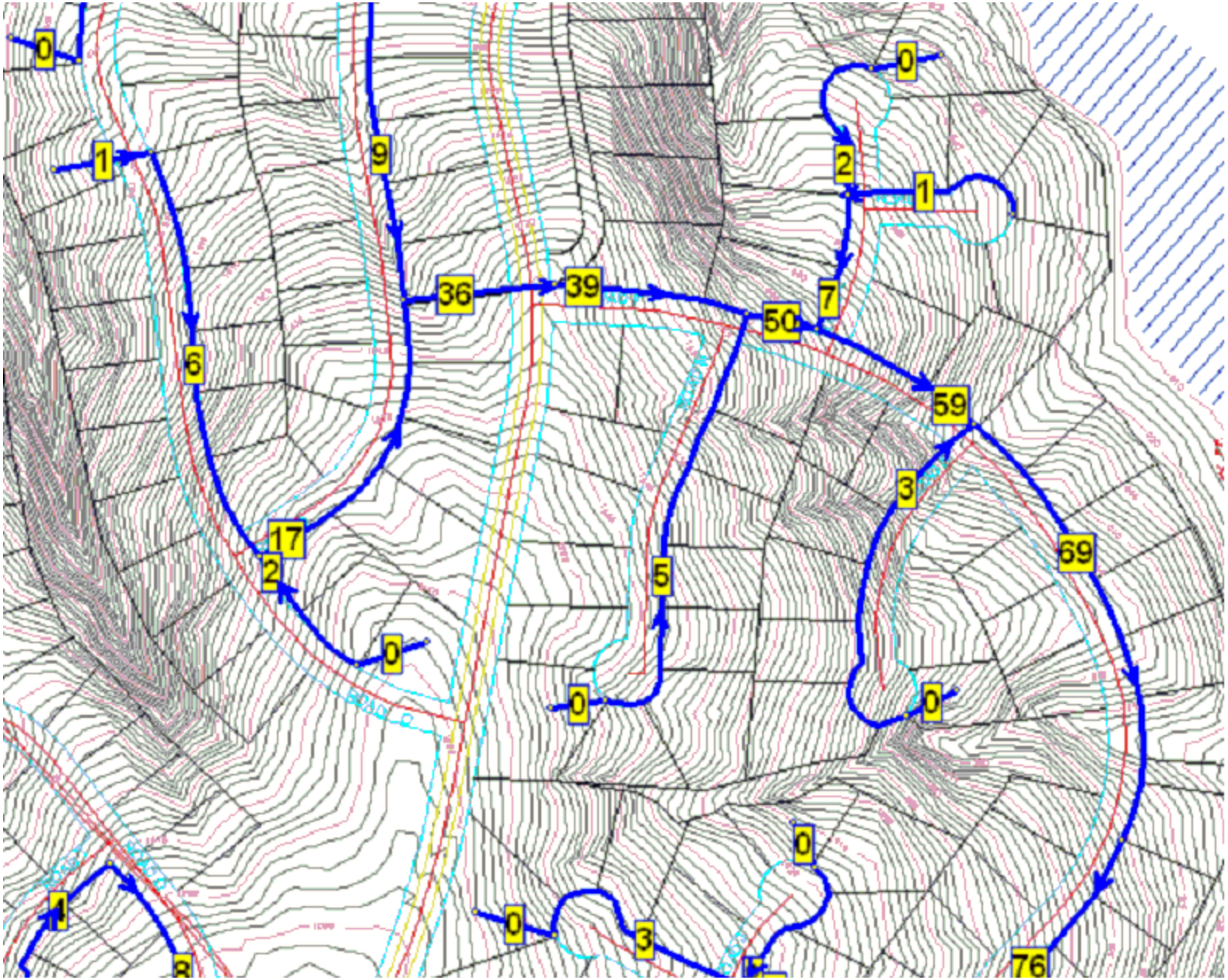

Pipe Diameter
<= 1.5
<= 2
<= 2.5

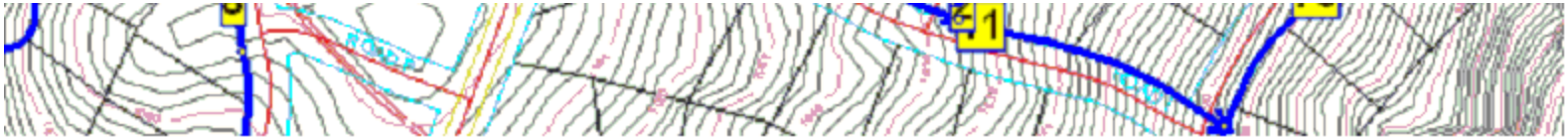
**Diameters assigned by Calculate Branch Diameters Module (Method 1)**

**Diameters assigned by Calculate Branch Diameters Module (Method 2)**

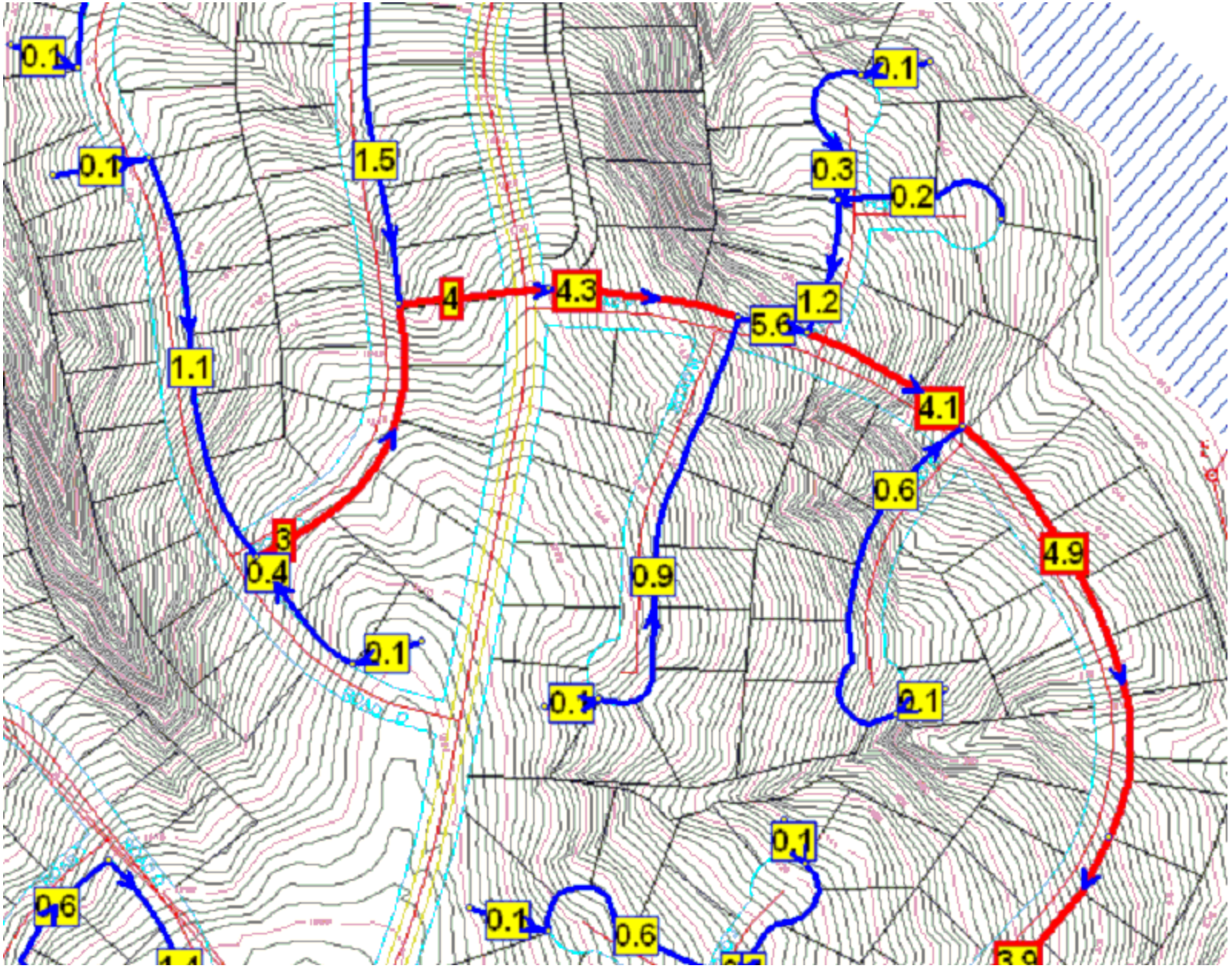
## Analyzing LPS systems Using Peak Flow Conditions

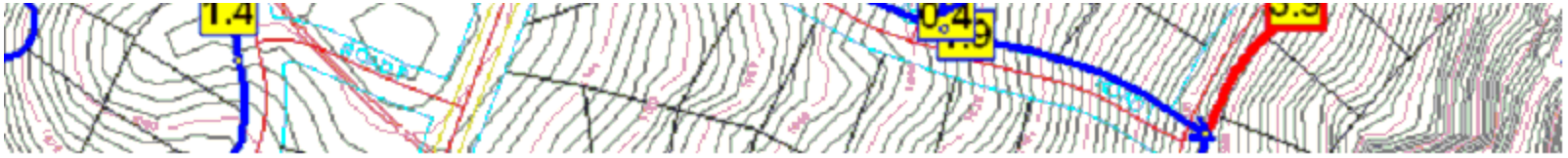
Average Residential Meter Demand -0.714


0
1
9
36
39
50
7
2
0
1
6
59
3
69
17
2
0
5
0
0
0
4
0
3
76



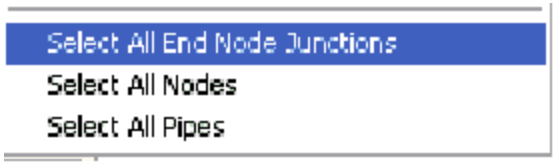

**Peak Flow in LPS Based on 0.714 gpm per connection**


0.1
0.1
1.5
0.1
0.3
0.2
4
4.3
1.2
5.6
1.1
4.1
0.6
3
0.4
0.9
4.9
2.1
0.1
2.1
0.1
0.6
0.6
0.1

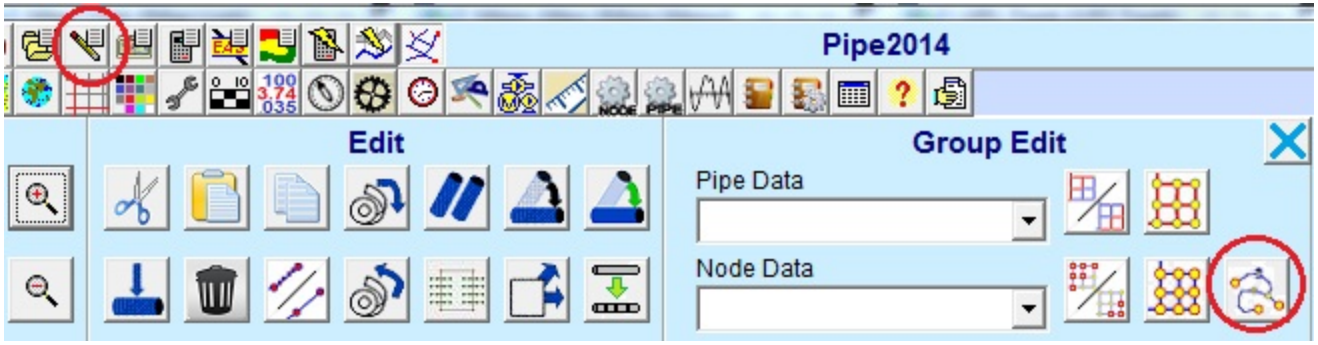


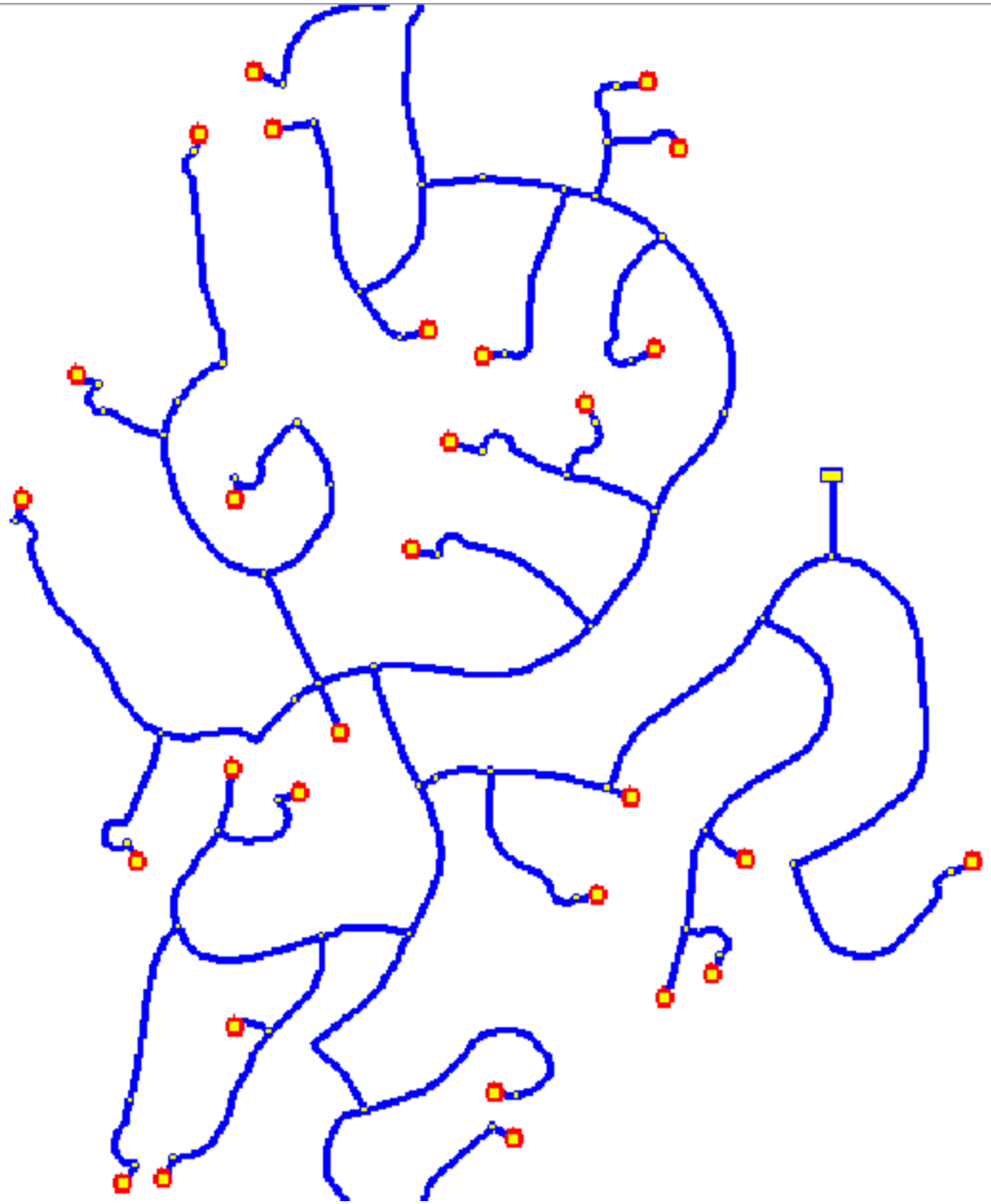

**Velocities in Main LPS Collection Pipes Due to Peak Flow**

3) **Use Select All End Node Junctions (Edit) feature to select all pump locations**

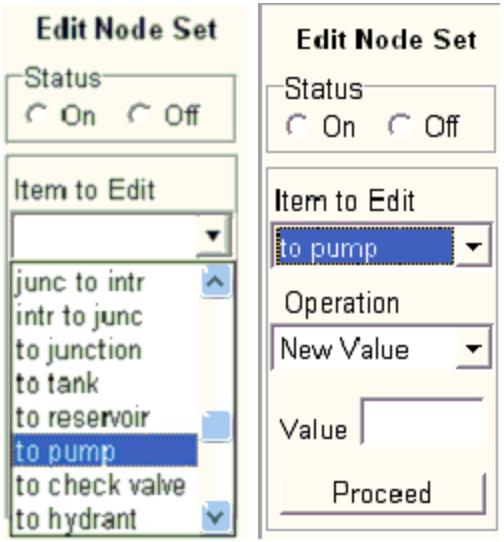

Select All End Node Junctions
Select All Nodes
Select All Pipes

In KYnetic:

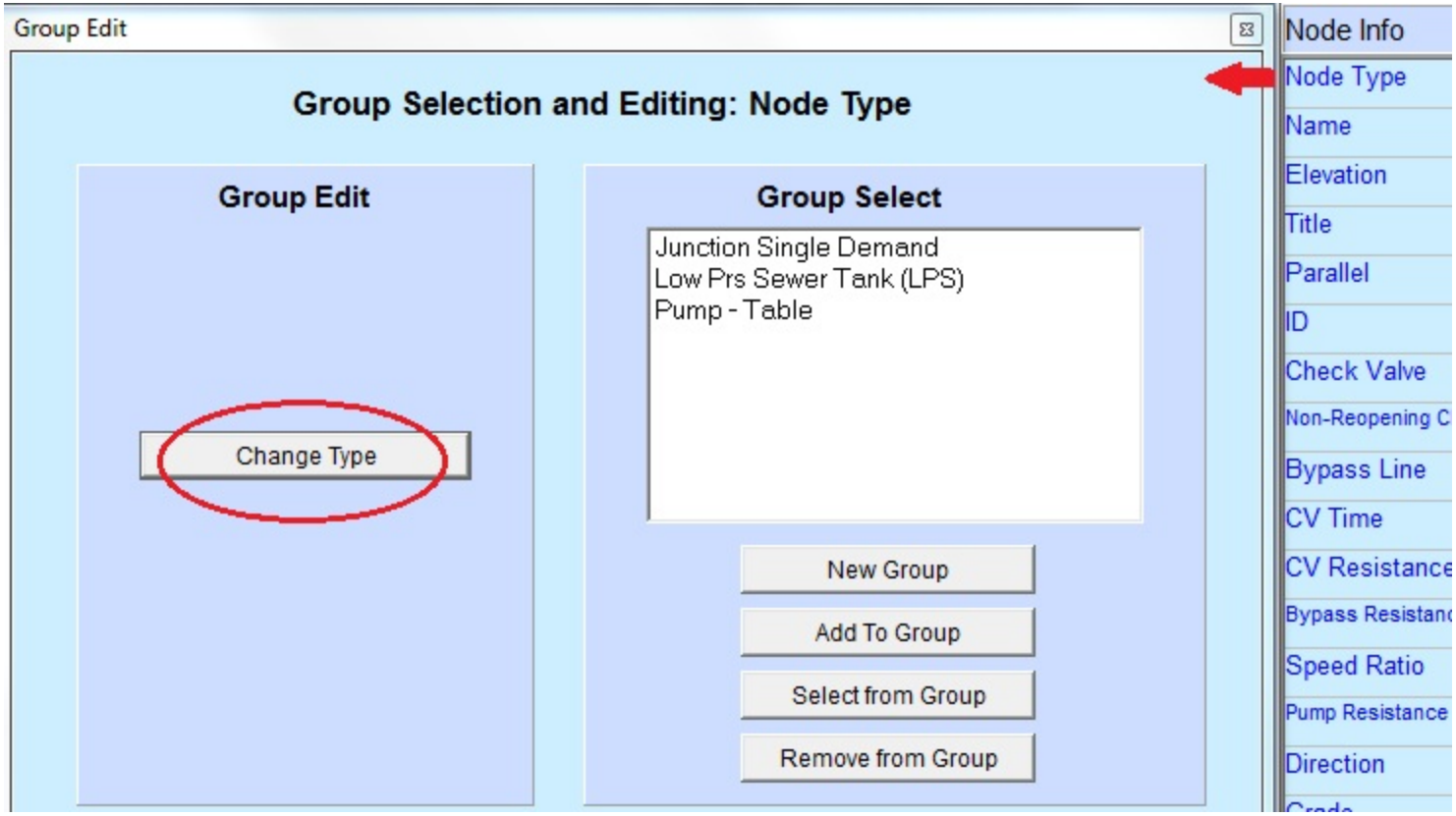

Pipe2014
Edit
Group Edit
Pipe Data
Node Data

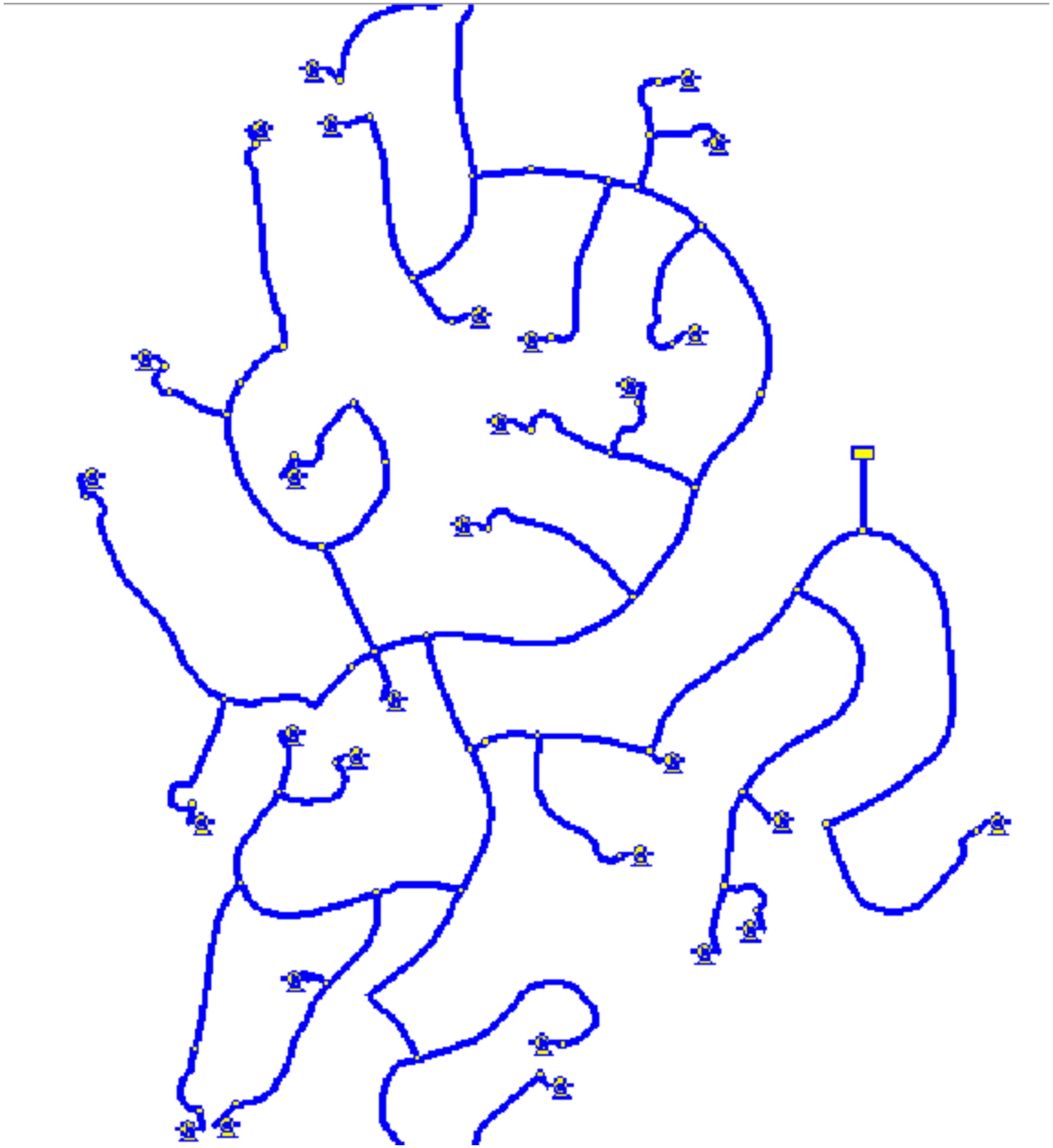


Under Edit select **[Select all End Node Junctions]**. The end nodes will be selected as shown above

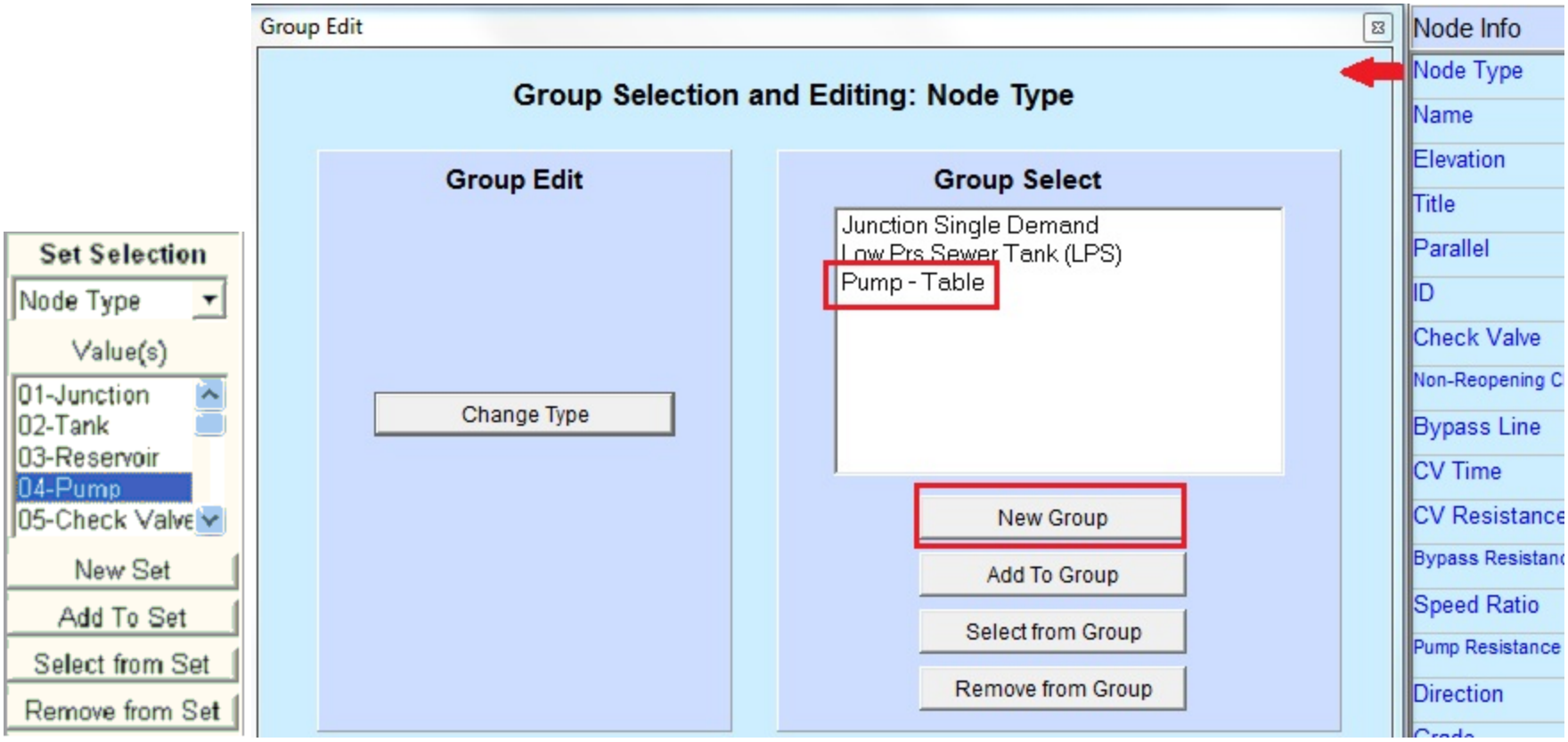



In KYnetic, click "Node Type" row heading in Node Info window

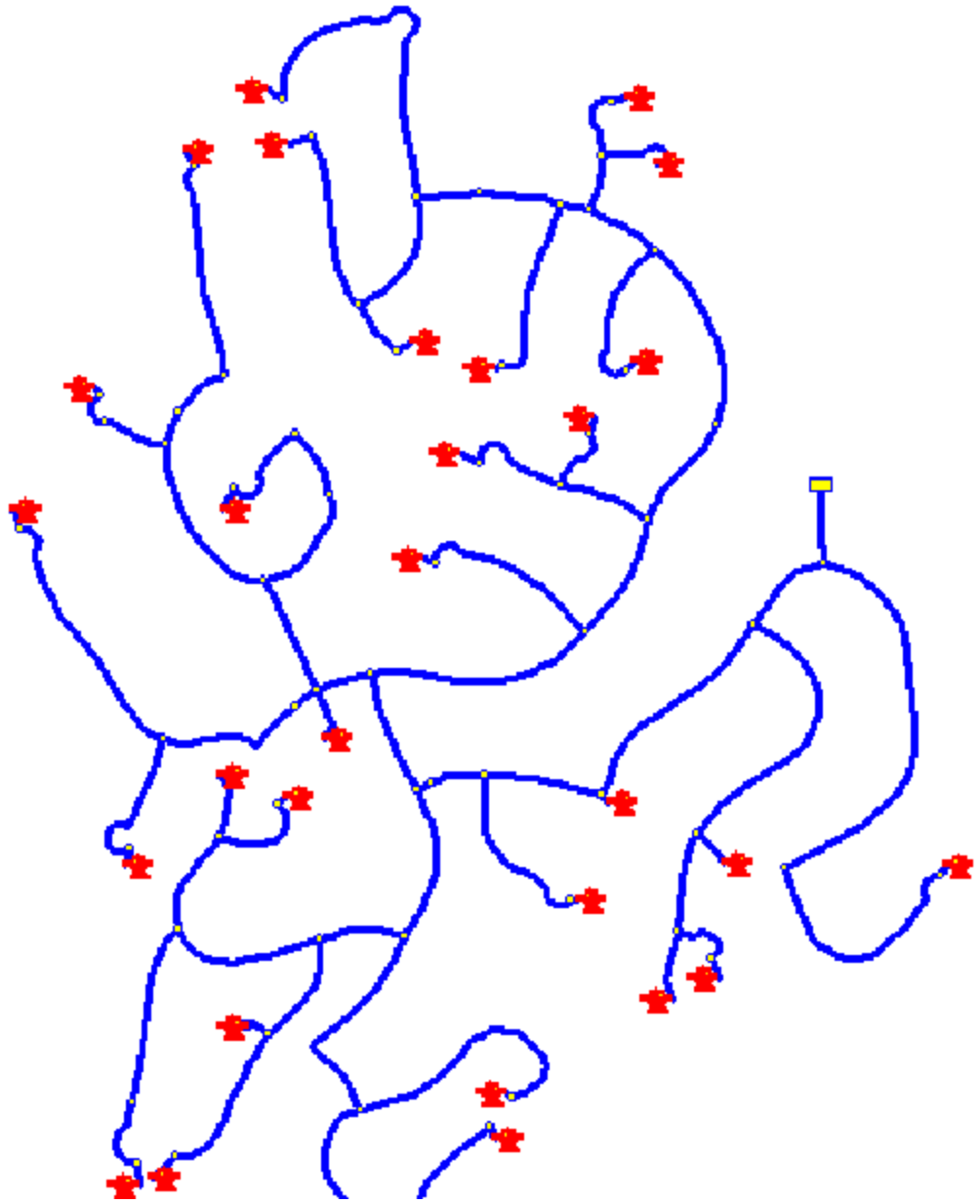

Group Edit
Group Selection and Editing: Node Type
Group Edit
Change Type
Group Select
Junction Single Demand
Low Prs Sewer Tank (LPS)
Pump - Table
New Group
Add To Group
Select from Group
Remove from Group
Node Info
Node Type
Name
Elevation
Title
Parallel
ID
Check Valve
Non-Reopening C
Bypass Line
CV Time
CV Resistance
Bypass Resistanc
Speed Ratio
Pump Resistance
Direction



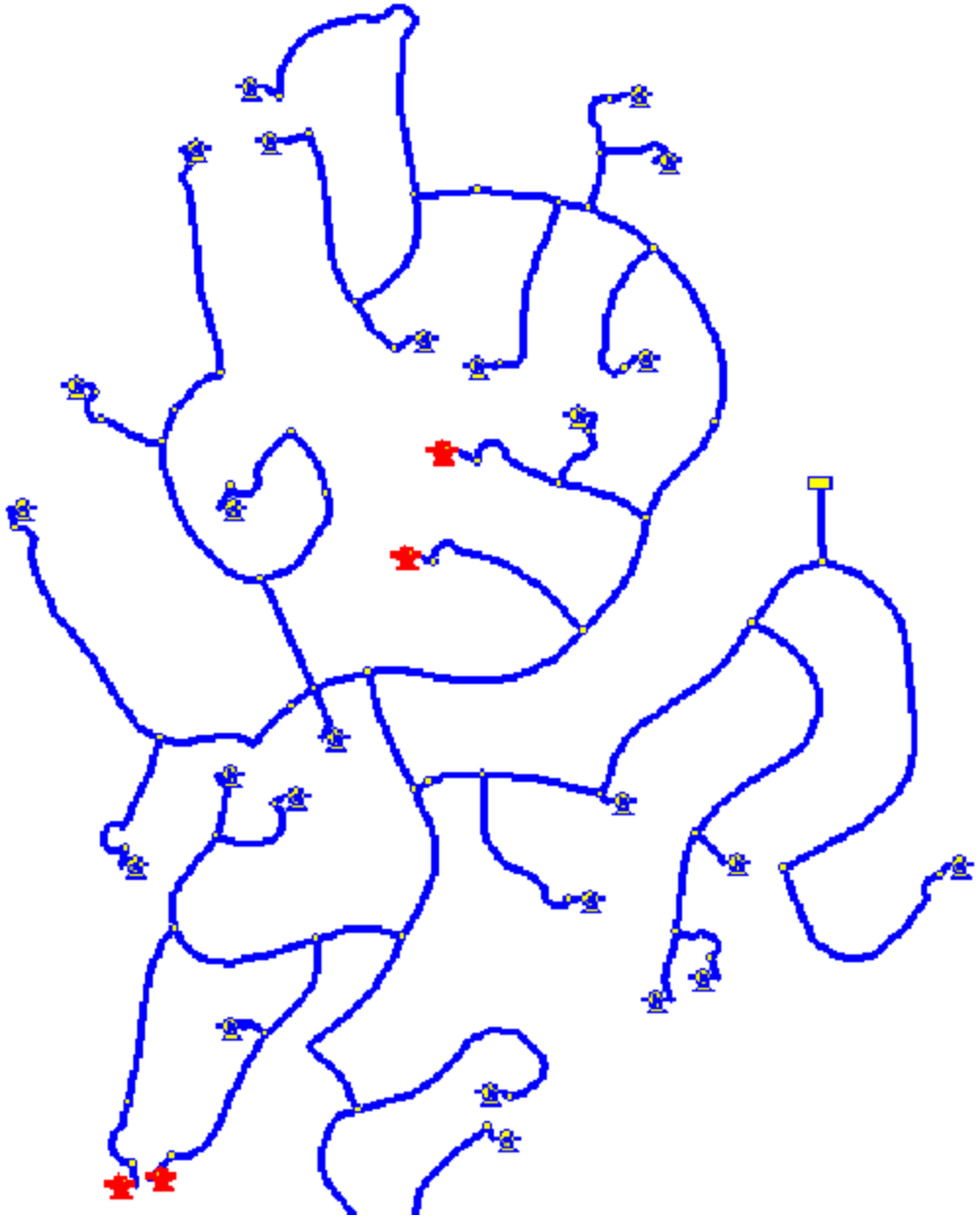
3) **Use Group Edit to change selected nodes to pumps. Change the selected end nodes to pumps. In Group Mode ender Edit Node Set select to pump and then click Proceed as shown above**




In KYnetic, same as above except select

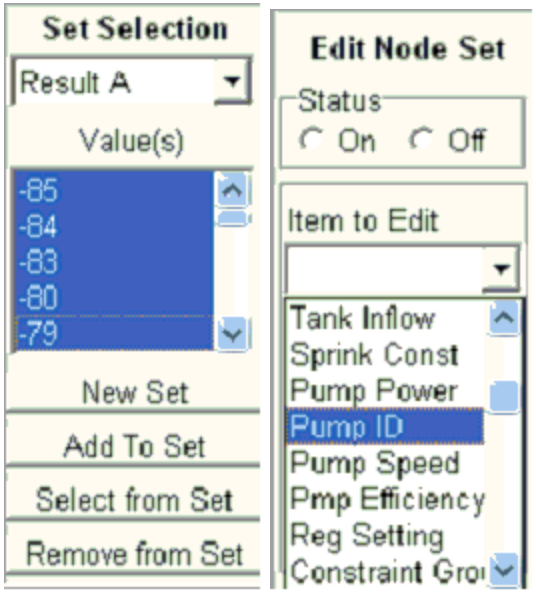


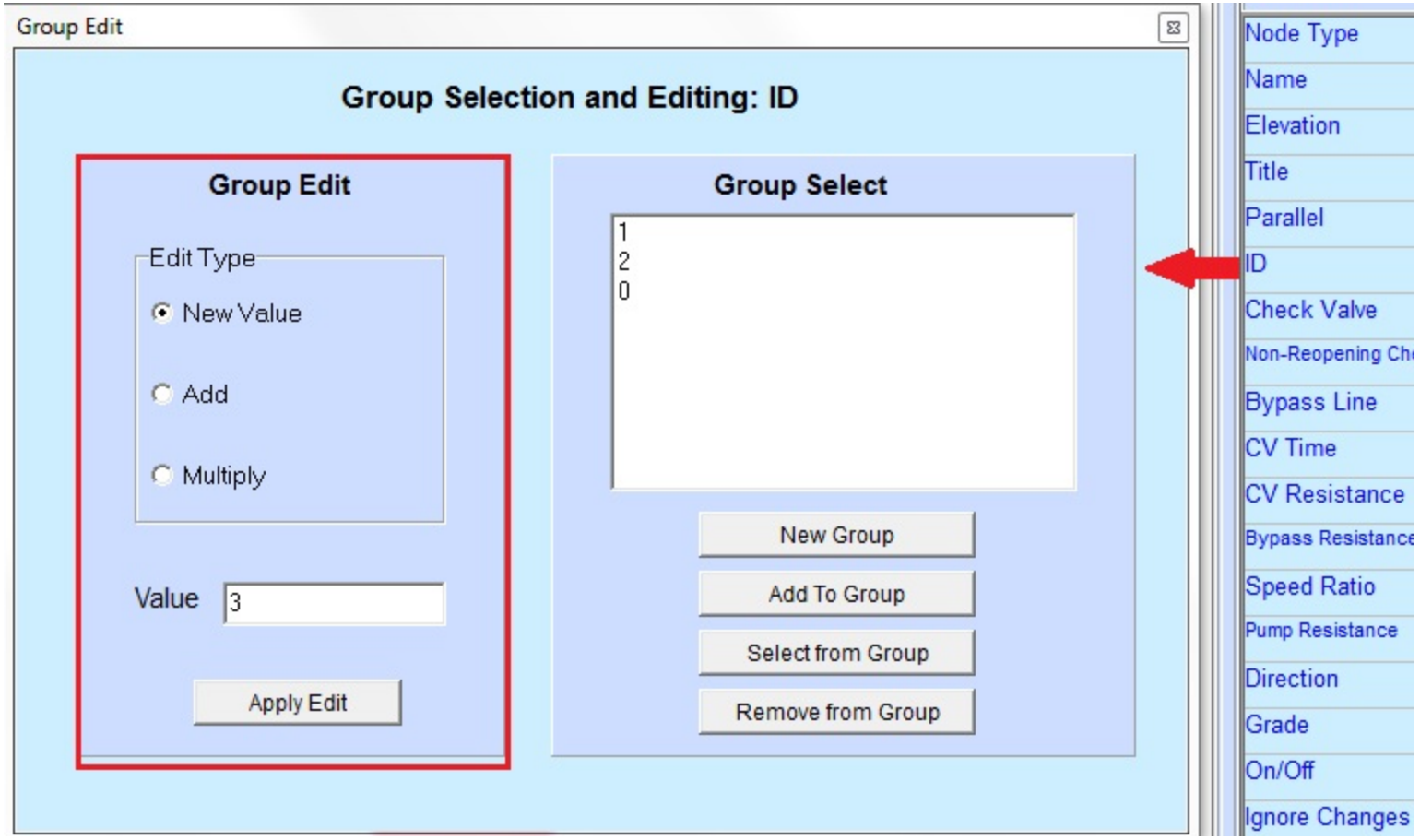
9) **Select a subset of pumps (Group Select – Select from Group) which will all be assigned the same ID**



**10) Use Group Edit to assign pump ID from ones available with this template. The elevation and grade of the pumps will be assigned the node elevation**

To edit the pump ID's select all the pumps using the Group Set Selection as shown above. If you want to select a sub group of pumps based on some parameter such as elevation, Static Head or Required Head for Peak Flow you can select a sub group of pumps as shown below. You can then assign the pump ID for this group of pumps


Set Selection
Result A
Value(s)
-85
-84
-83
-80
-79
New Set
Add To Set
Select from Set
Remove from Set
Edit Node Set
Status
On
Off
Item to Edit
Tank Inflow
Sprink Const
Pump Power
Pump ID
Pump Speed
Pmp Efficiency
Reg Setting
Constraint Gro



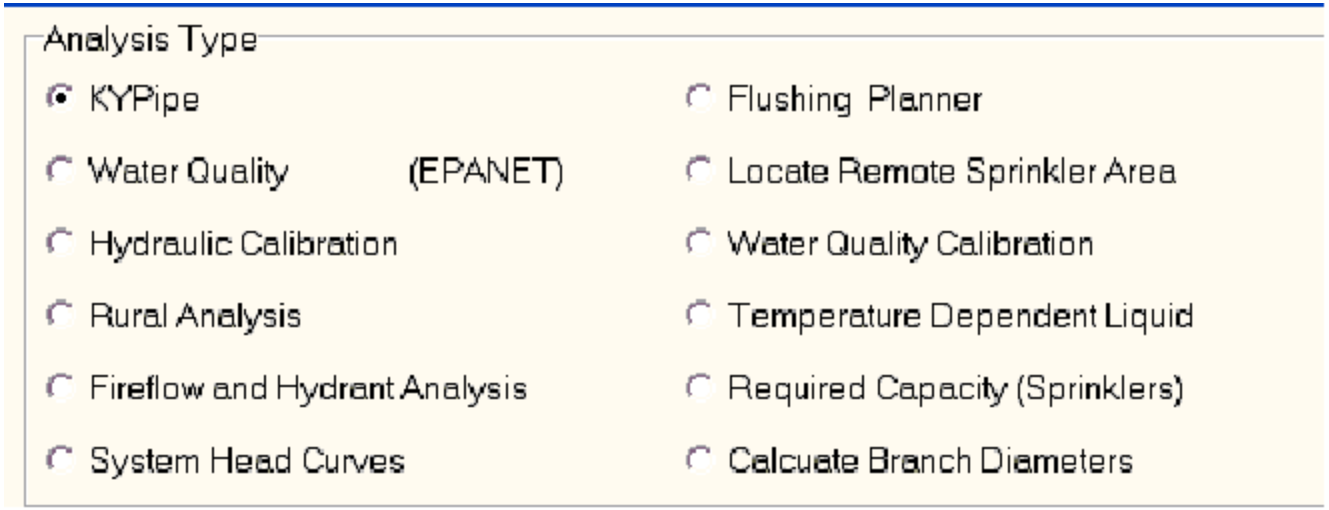

**Set of pumps selected by Peak Demand Head Requirement**

**11) Repeat until all pumps are assigned an ID**

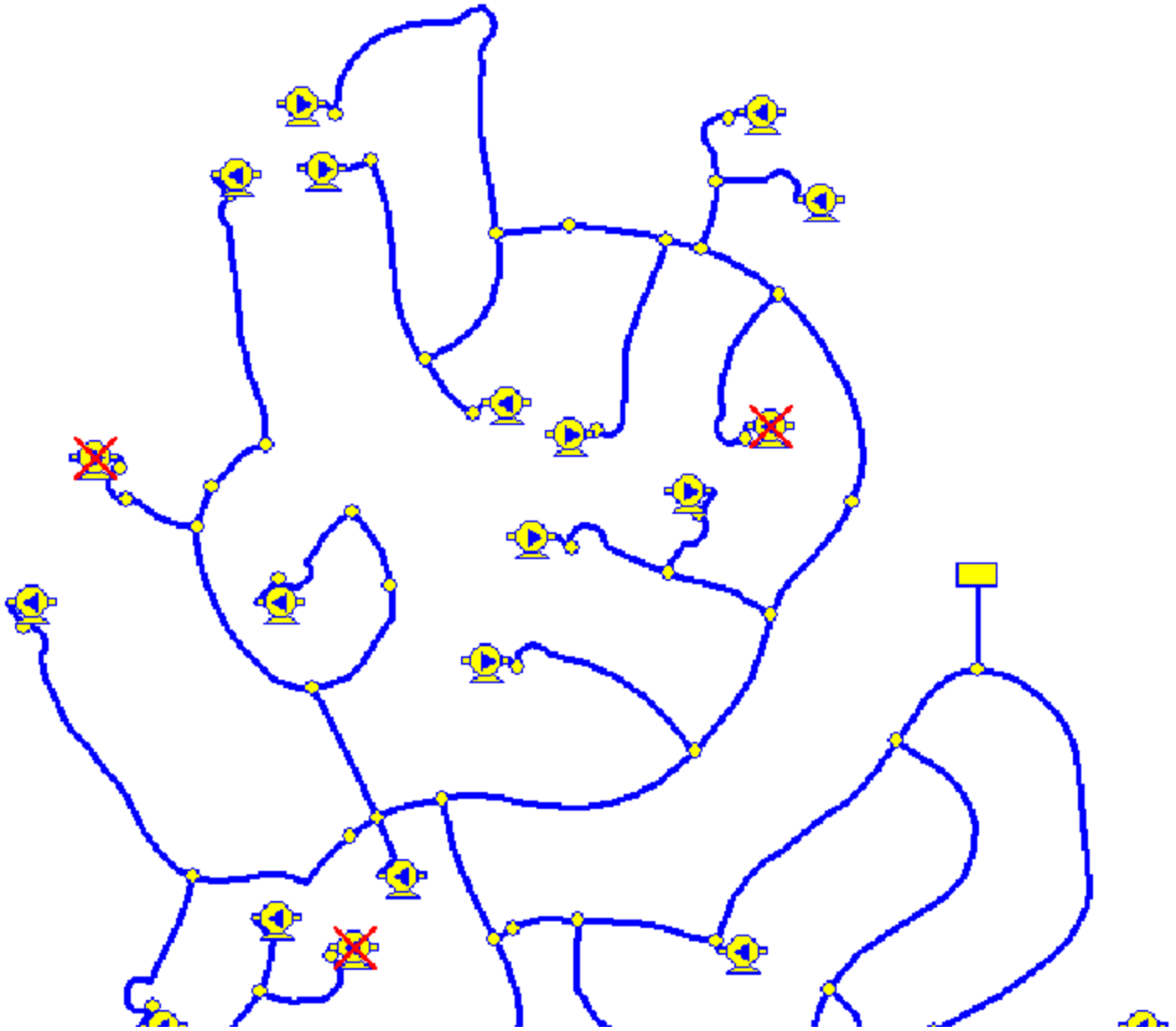
**12) Evaluate pump performances**

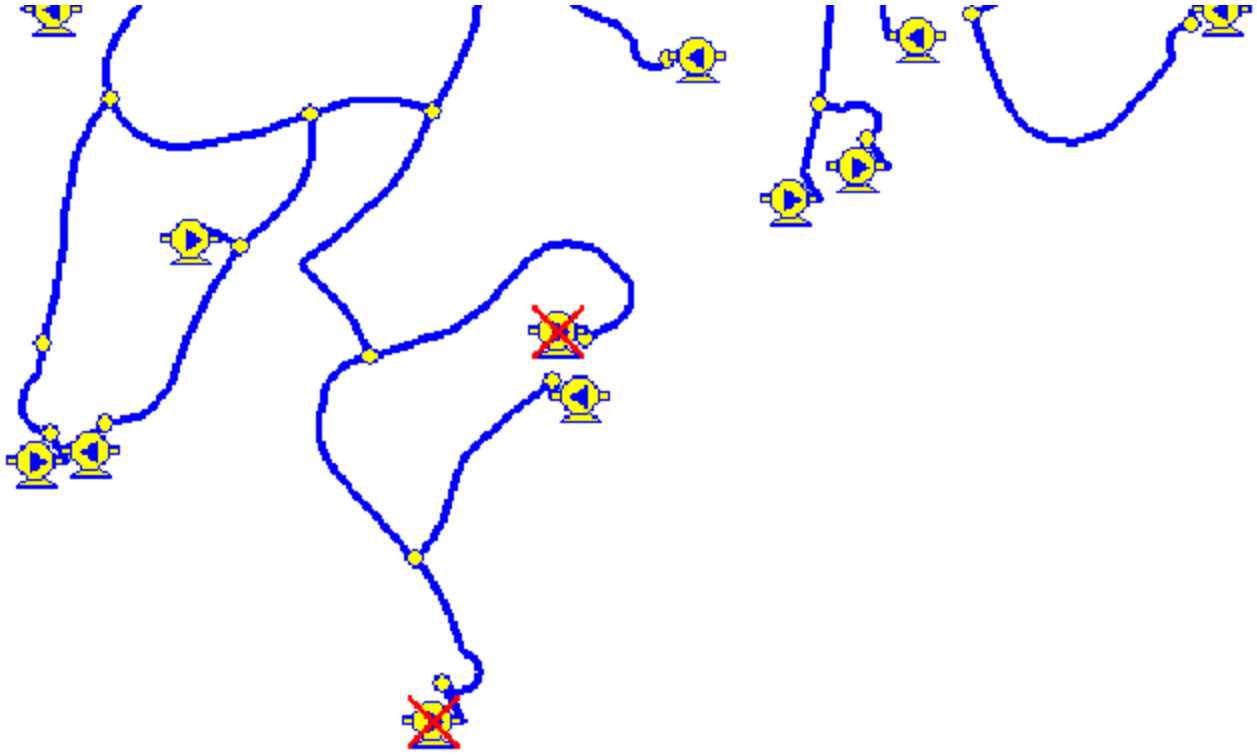
2) **Turn off selected pumps and run a KYPIPE (hydraulic analysis)**
3) **Show Pump Status (Ctrl-Alt-P) This shows ON (green), OFF (red), and ON with no flow (yellow). Details**

**appear in the Pump Report**

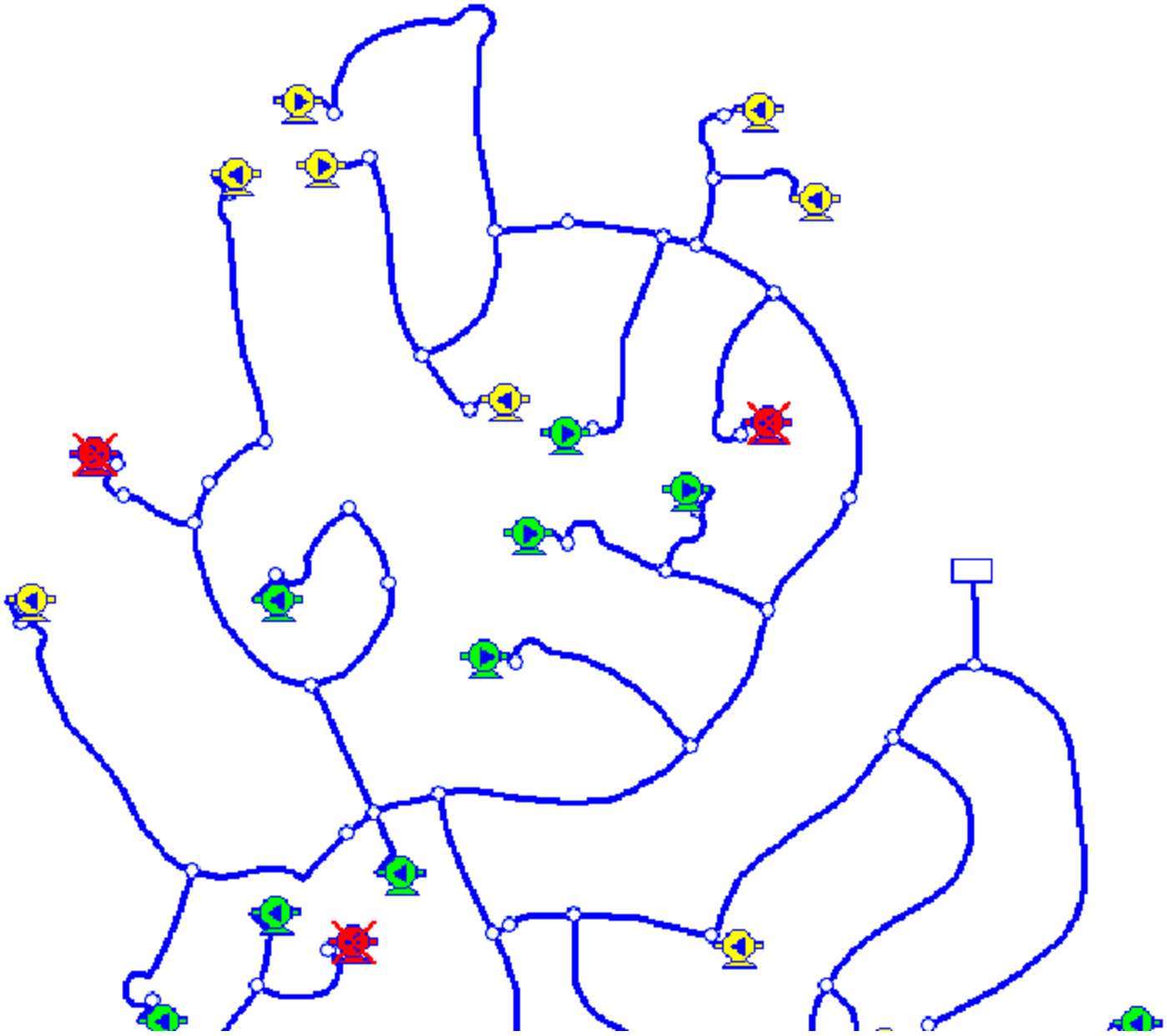


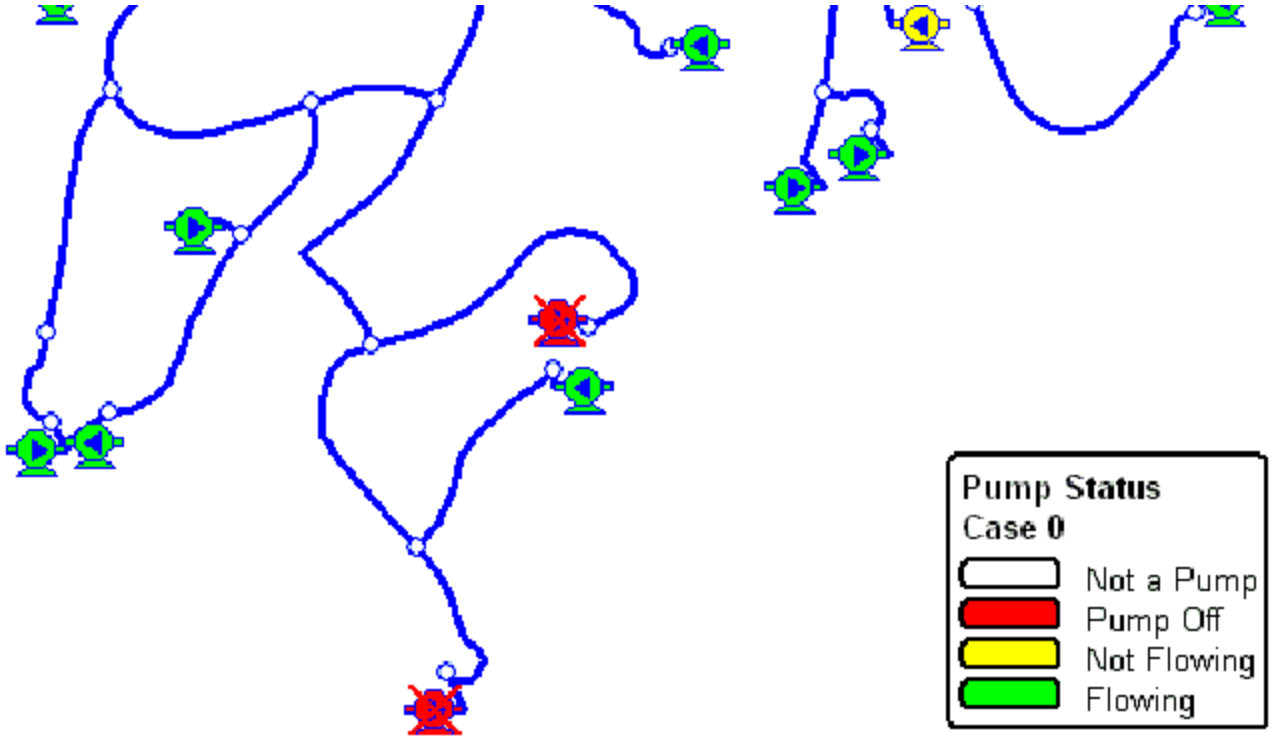

**Run KYPIPE Analysis**





**Turn off selected pumps and run a KYPIPE (hydraulic analysis)**






**Show Pump Status (Ctrl-Alt-P) This shows ON (green), OFF (red), and ON with no flow (yellow).**

P U M P/L O S S  E L E M E N T  R E S U L T S

| NAME | FLOWRATE (gpm) | INLET HEAD (ft) | OUTLET HEAD (ft) | PUMP HEAD (ft) |
|---|---|---|---|---|
| Pump-1 | 17.99 | 0.00 | 50.21 | 50.2 |
| Device "Pump-10" is closed | | | | |
| Device "Pump-11" is closed | | | | |
| Pump-12 | 16.45 | 0.00 | 82.18 | 82.2 |
| Pump-13 | 5.95 | 0.00 | 123.16 | 123.2 |
| Pump-14 | 16.75 | 0.00 | 75.04 | 75.0 |
| Pump-15 | 18.90 | 0.00 | 32.63 | 32.6 |
| Pump-16 | 5.26 | 0.00 | 124.39 | 124.4 |
| Device "Pump-17" is closed | | | | |
| Pump-18 | 15.58 | 0.00 | 95.63 | 95.6 |
| Device "Pump-19" is closed | | | | |
| Pump-2 | 2.27 | 0.00 | 128.08 | 128.1 |
| Device "Pump-20" is closed | | | | |
| Device "Pump-21" is closed | | | | |
| Device "Pump-22" is closed | | | | |
| Device "Pump-23" is closed | | | | |
| Pump-24 | 5.02 | 0.00 | 124.78 | 124.8 |
| Pump-25 | 17.25 | 0.00 | 65.09 | 65.1 |
| Device "Pump-26" is closed | | | | |
| Device "Pump-27" is closed | | | | |
| Pump-28 | 7.29 | 0.00 | 120.37 | 120.4 |
| Device "Pump-29" is closed | | | | |

```
Device "Pump-3" is closed
 Pump-30        6.78       0.00     121.50    121.5
  Pump-4       11.58       0.00     108.04    108.0
Device "Pump-5" is closed
Device "Pump-6" is closed
  Pump-7        8.16       0.00     118.30    118.3
  Pump-8       14.15       0.00     106.53    106.5
  Pump-9       15.17       0.00     100.03    100.0
```

**Details appear in the Pump Report**

## 1.4.6 Temperature Dependent Liquid

Temperature Dependent Liquid Analysis (KYPipe)

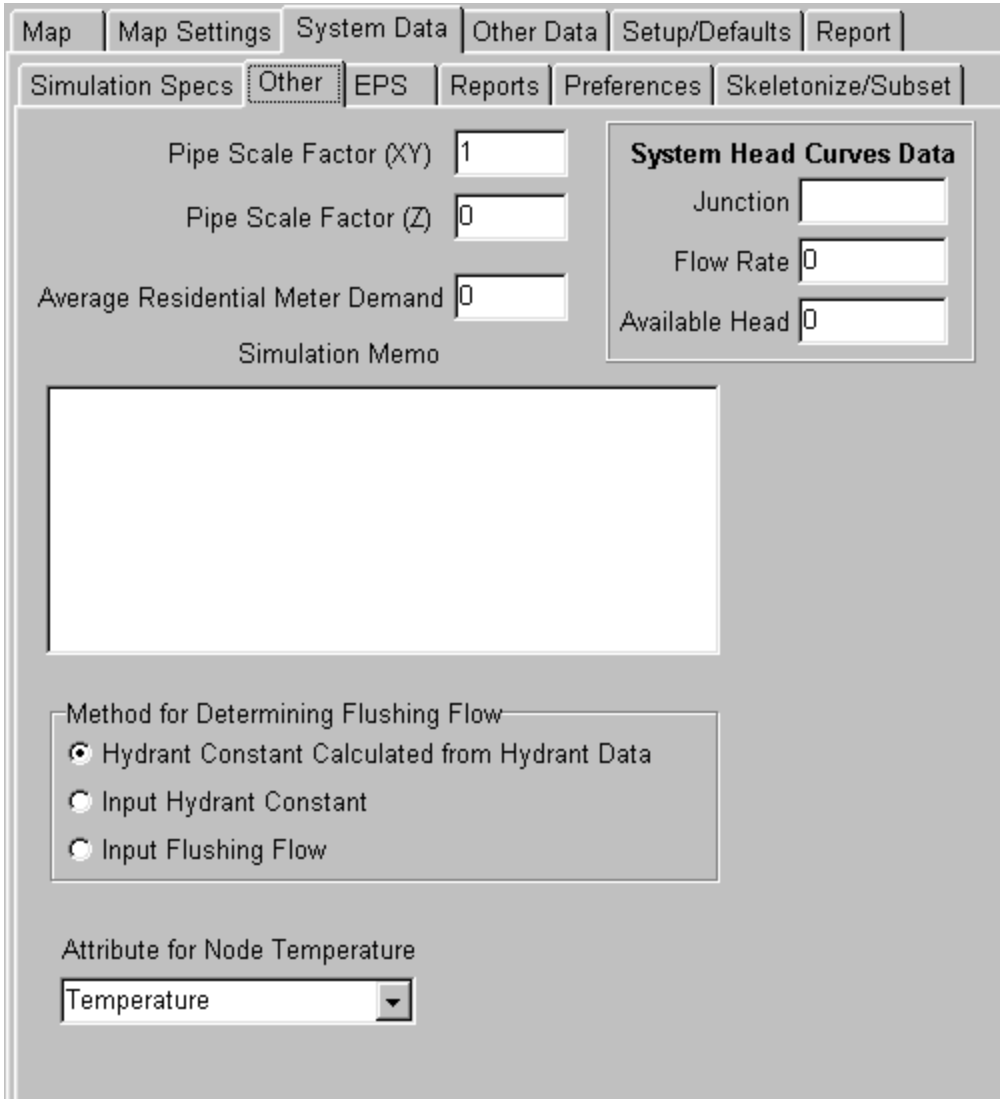
---

Temperature Dependant Liquid analysis is applicable to KYPipe (liquid) systems only.

This feature allows for modeling liquids with significant changes in density and viscosity at different parts of the distribution system. Based on the user specified node temperature, the program first calculates the right viscosity and density for each pipeline. Then the program calculates the appropriate Darcy-Wiesbach friction factors taking into account the correct viscosity and density for each pipe. Conventional KYPIPE analysis assumes constant density and viscosity throughout the distribution system. This may NOT be acceptable if the fluid temperature varies drastically from one part of the network to other thereby necessitating the use of this feature of the Pipe2018 program. It should be realized that the program adjusts the density and viscosity based on the user specified nodal temperatures ONLY and the program can NOT calculate heat losses along the pipelines.

Note: This analysis does not calculate heat loss. For heat loss calculation, see Superheat for Pipe2018: Steam.

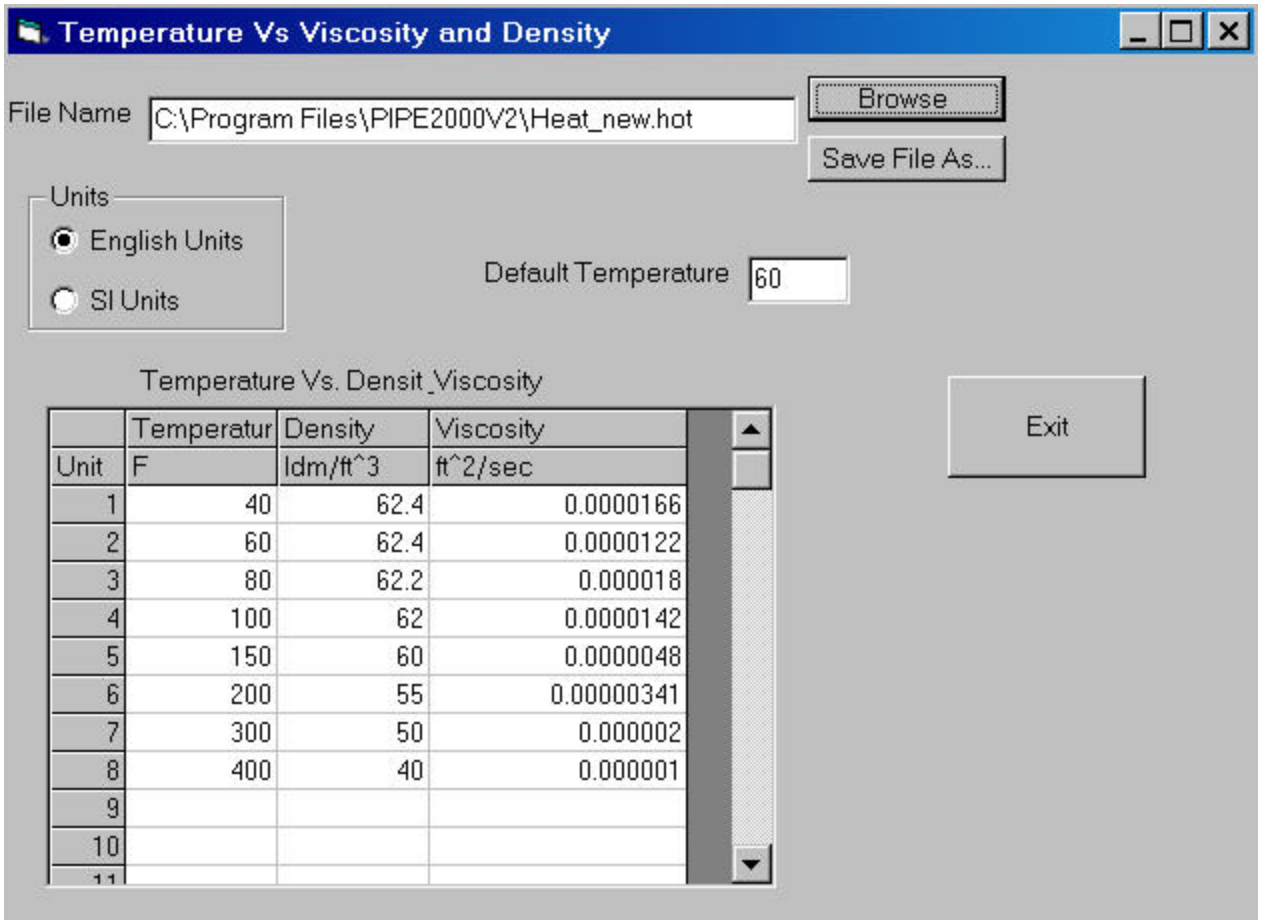
For an existing system, create a User Data item called Temperature. Next go to the Other Data icon (KYnetic) or Sysetm Data | Other menu and specify the Attribute for Node Temperature box as shown below. See User Data.




Now temperature data may be assigned for each node.

Next, go to Tools | Density and Viscosity Table. Enter Temperature and Viscosity values of the range of temperatures expected in your analysis and save this table using the same filename as the model. You may also use enclosed hotwater.exe file to create/edit the viscosity/density vs temp file. Run this program outside Pipe2018 menu (from explorer). This program creates a file with extension HOT. Note: This file MUST be in the same folder where the P2K file is and should have

the same name (please limit the name to 8 chars)




| | Temperatur | Density | Viscosity |
|---|---|---|---|
| Unit | F | ldm/ft^3 | ft^2/sec |
| 1 | 40 | 62.4 | 0.0000166 |
| 2 | 60 | 62.4 | 0.0000122 |
| 3 | 80 | 62.2 | 0.000018 |
| 4 | 100 | 62 | 0.0000142 |
| 5 | 150 | 60 | 0.0000048 |
| 6 | 200 | 55 | 0.00000341 |
| 7 | 300 | 50 | 0.000002 |
| 8 | 400 | 40 | 0.000001 |
| 9 | | | |
| 10 | | | |

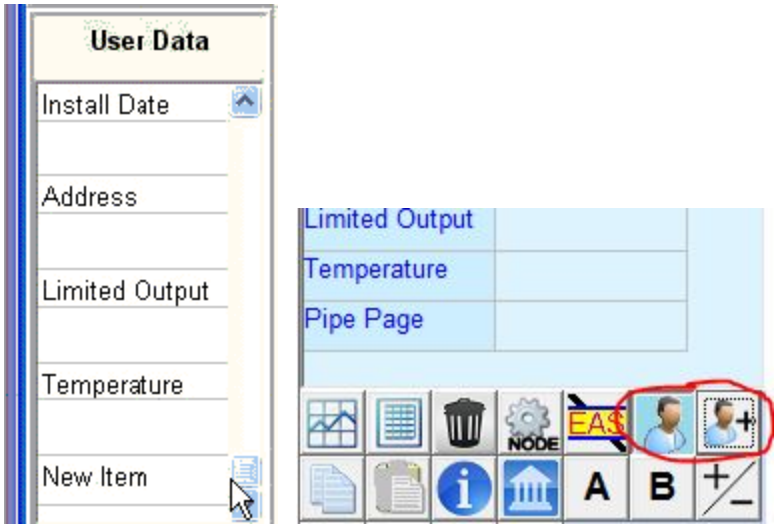
To run the analysis, click Analyze | Analyze and select Temperature Dependant Analysis. To view the results, use Map Labels and/or create a temperature contour. The results are also included in the Report as added columns of Node and Pipeline Results Data.

Note: By default, flow rates displayed are in Volumetric flowrate units (irrespective of whether the demand are specified in volumetric units or mass units). To display the flowrates in Mass units(on map), enter maximum number of trials (in system data) with a negative sign.

_____

To specify temperatures at different nodes:

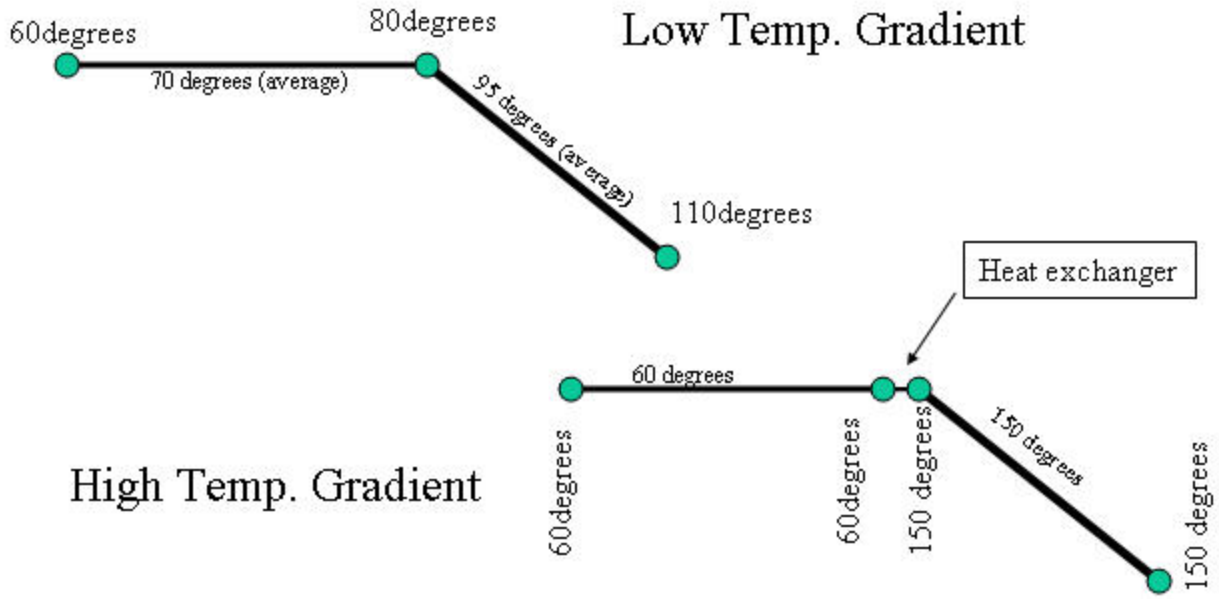
Click the User button at the top of the Node Information window. Enter a value under Temperature or create a New Item called Temperature




Group operations may be used to specify a particular temperature for a group of nodes

- Click “Group” button
- Select all the nodes in the group
- Click on the “item to edit” pull down menu in “Edit Node Set” window and select “Temperature”
- Assign a value and click "OK".

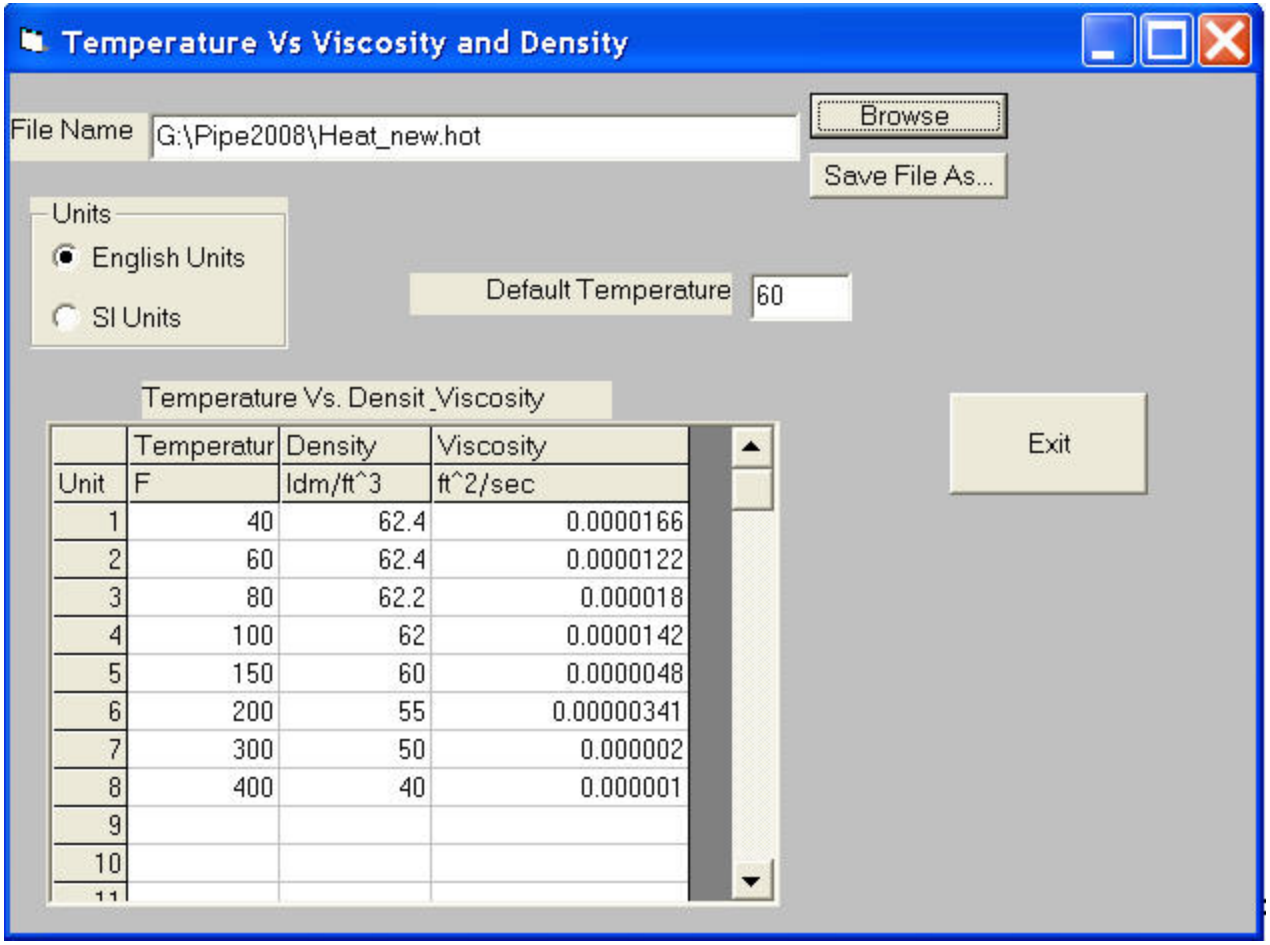
Adjacent Pipes with High Temperature Gradient
This illustrates modeling pipes adjacent to heat exchangers.




## 1.4.6.1 Density and Viscosity Table Tool

**Density and Viscosity Table**

Enter Temperature and Viscosity values of the range of temperatures expected in your Temperature Dependant Analysis and save this table using the same filename as the model. You may also use the enclosed hotwater.exe file to create/edit the viscosity/density vs temp file. Run hotwater.exe outside Pipe2018 menu (from Windows Explorer). This program creates a file with extension HOT. Note: This file MUST be in the same folder where the P2K file is and should have the same name (please limit the name to 8 chars)




## 1.4.7 Age Based Roughness

Age Based Roughness

See the Pipe Type video at www.kypipe.com.

---

See Pipe Type Data for more information on using this feature.

This feature ties in the roughness of the pipe to the age of the pipe for whatever head loss expression you choose (Hazen-Williams, Darcy Weisbach, or Manning).

This not only allows the model roughness to be automatically updated each year, but also allows future simulations to be run which automatically use roughness values appropriate for the year designated. This feature improves calibration and the subsequent adjustment of model data because roughness adjustments are directly related to pipe age. To utilize this feature you must provide the reference roughness (new pipe) and an estimate of the roughness value after 10 years. There is also a Tool available with Pipe2018 to aid in this calculation.

## Age-Based Roughness Calculations

The principal data available for aging has been obtained by Pitometer Associates and has been presented as plots of the Hazen Williams C values over time with pipe type. The aging rate depends on the type of pipe and characteristics of the water and will vary greatly for different situations. Some typical data is shown in Figure 1. A relationship to describe the non linear variation of C with pipe age was developed for use in Pipe2018. It is calculated by using a roughness value for new pipe and a value for 10-year-old pipe. Plots based on this technique are shown in Figure 2 using 140 for the new pipe roughness and 130, 120 , and 110 for the 10 year values. Figure 2 also shows the data for the three cities and verifies that the technique we have incorporated into Pipe2018 closely represents the field data trends. This is further illustrated in Figure 3 which compares the field data curve and the one generated by Pipe2018 for one of the cases.

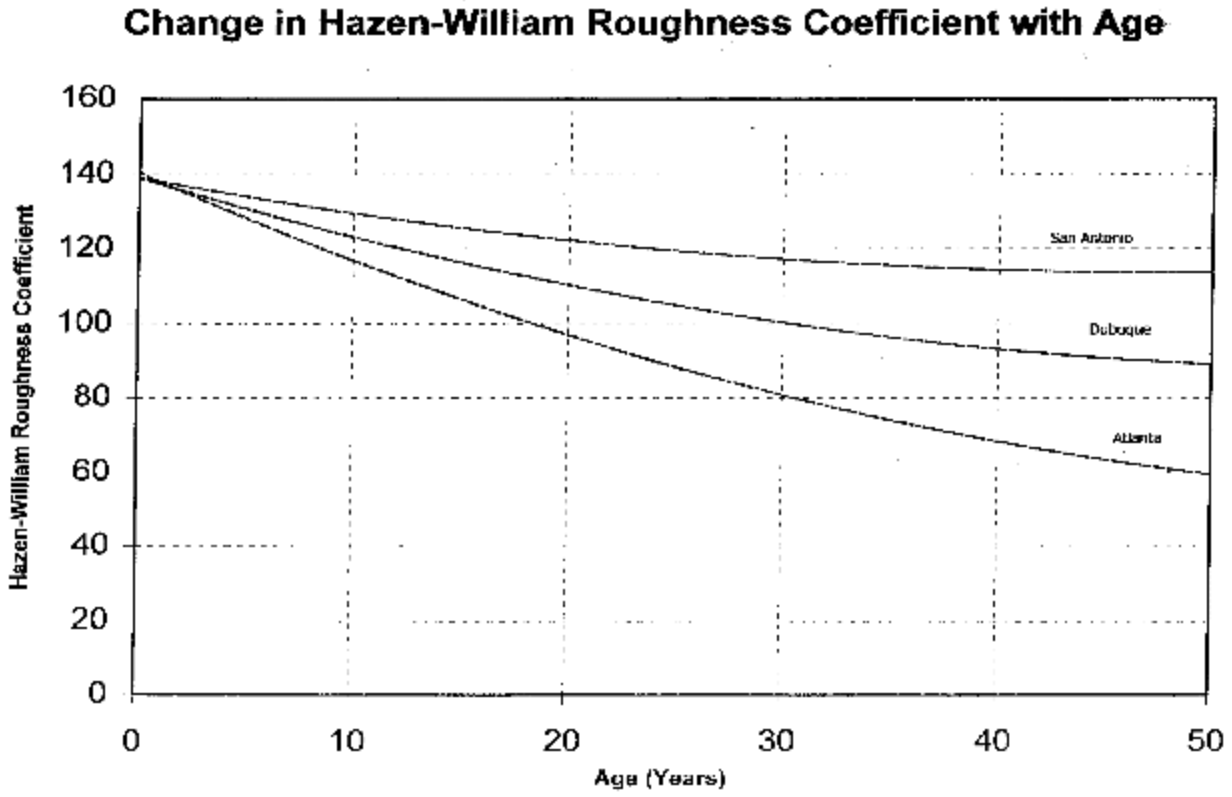



Figure 1. Historical Data (Pitometer Associates)

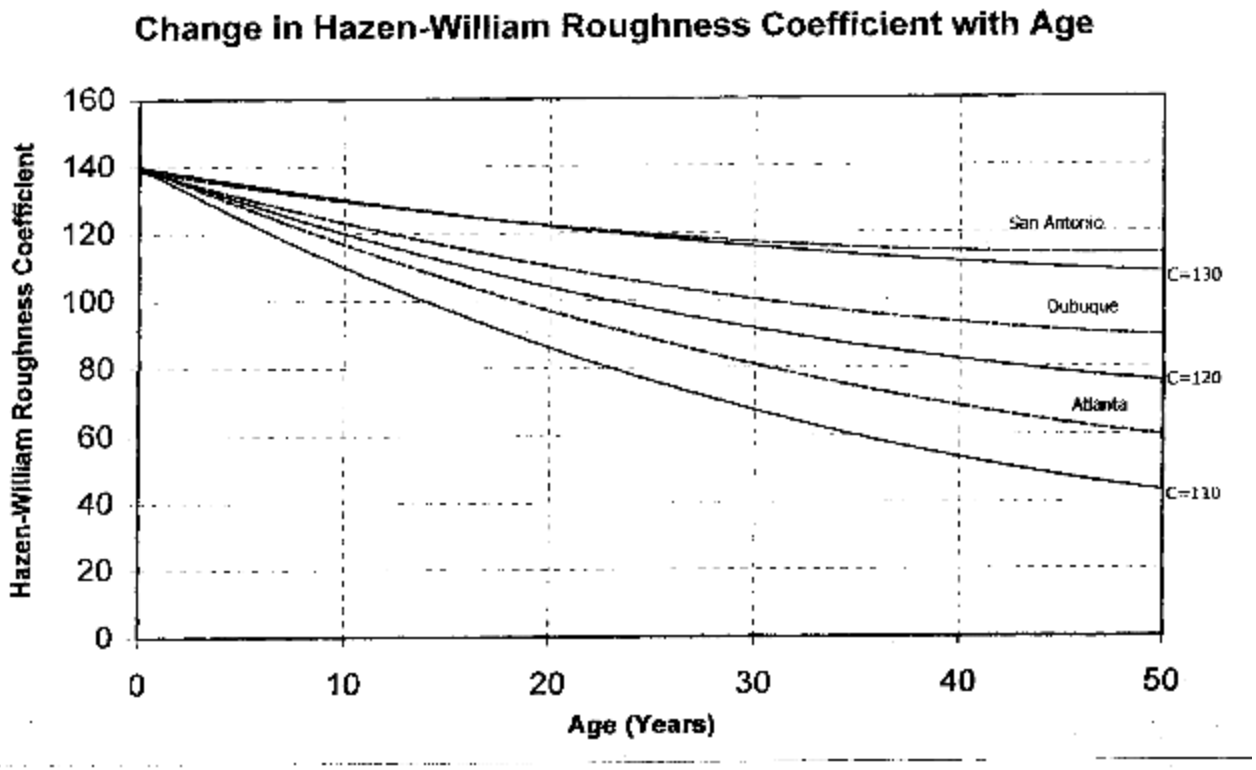



Figure 2. PIPE2000 Roughness Aging Model (based on 10 year estimate)

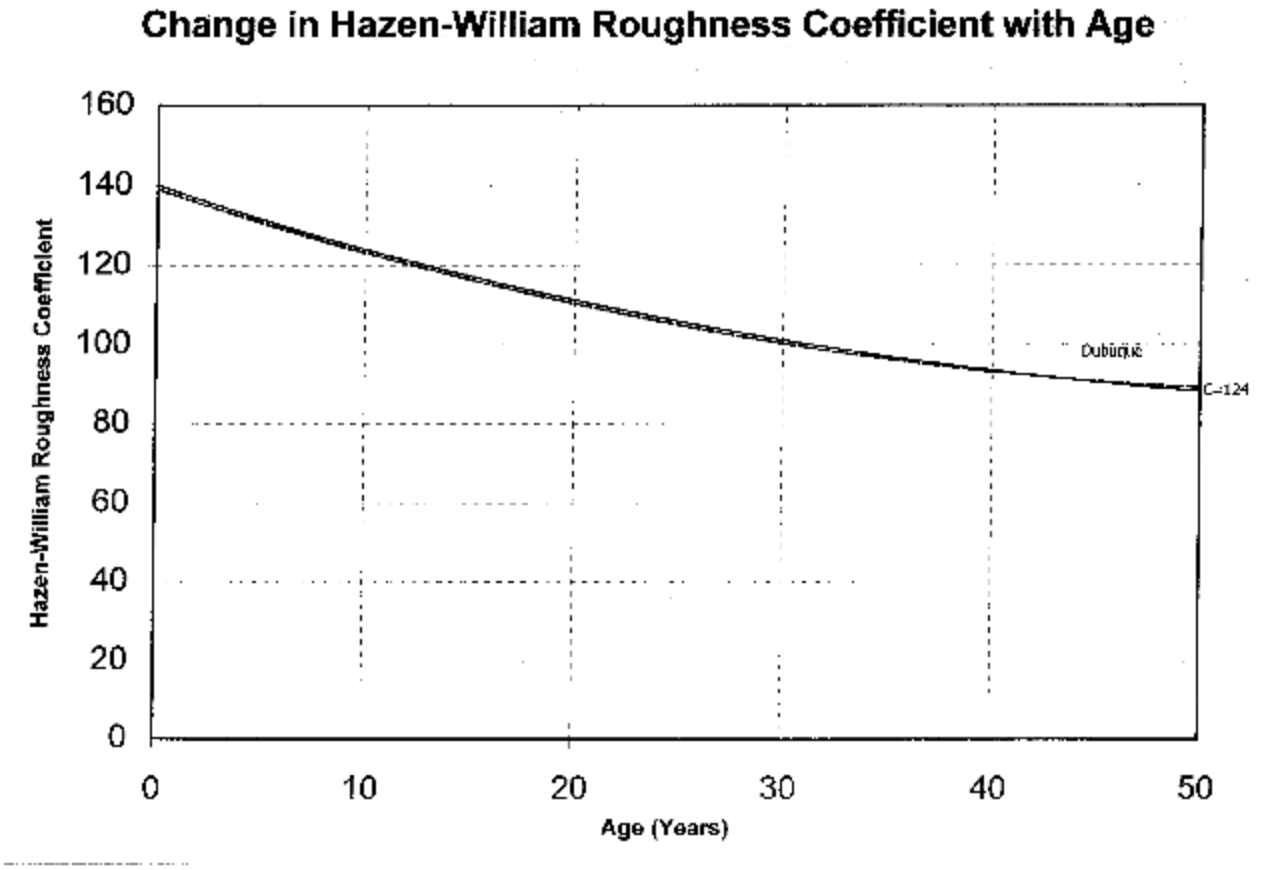



Figure 3. Comparison of PIPE2000 Model and Historical Data

Age-based roughness models were developed for use in Pipe2018 for the Darcy Weisbach and Manning equations which give results for head loss calculations similar to ones based on the Hazen Williams equation and field data. All calculations use the value for the roughness for new pipe and an estimated value for 10-year-old pipe. The following comparison shows that the age-based roughness models developed for Pipe2018 give similar results for all these head loss equations.

### Comparison of Pipe2018 Aging Calculations

Example data

D = 12 in.

Q = 4 cfs

L = 1000 ft.

Based on an initial Hazen Williams roughness of 140, the following corresponding initial roughness values were chosen for the Darcy Weisbach and Manning equations so that the initial head loss calculations would be in agreement.

$C_0$ = 140 (Hazen Williams)

$e_0$ = 0.3 millifeet (Darcy Weisbach)

$n_0$ = 0.0094 (Manning)

Ten-year roughness values were chosen to provide an aging rate so that the three head loss relations would give comparable results in 50 years. These values are:

$C_{10}$ = 122

$e_{10}$ = 0.9

$n_{10}$ = 0.0106

The following table compares the head loss calculations for the three different relations. The roughness values shown were determined using the automatic aging calculations incorporated into Pipe2018.

| | Hazen Williams | | Darcy Weisbach | | Man ning | |
|---|---|---|---|---|---|---|
| Time | C | $h_l$ | e | $h_l$ | n | $h_l$ |
| 0 | 140 | 6.5 | 0.3 | 6.6 | 0.0094 | 6.6 |
| 10 | 122 | 8.4 | 0.9 | 8.0 | 0.0106 | 8.4 |
| 20 | 108 | 10.6 | 2.7 | 10.4 | 0.0118 | 10.4 |
| 30 | 97 | 13.0 | 5.7 | 12.9 | 0.0130 | 12.7 |
| 40 | 88 | 15.4 | 9.9 | 15.3 | 0.0143 | 15.1 |
| 50 | 82 | 17.6 | 15.3 | 17.8 | 0.0155 | 17.8 |

## Estimating the 10 Year Roughness (C Value)

Pipe2018 uses a 10-year roughness value to calculate a roughness for age-based roughness simulations. You can use the table below to quickly calculate the 10-year roughness using the C value for new pipe and a second C value based on any age.

| Age | Factor | Age | Factor | Age | Factor | Age | Factor | Age | Factor |
|---|---|---|---|---|---|---|---|---|---|
| 1 | 9.1200 | 21 | 0.5349 | 41 | 0.3417 | 61 | 0.2890 | 81 | 0.2740 |
| 2 | 4.6063 | 22 | 0.5161 | 42 | 0.3373 | 62 | 0.2876 | 82 | 0.2737 |
| 3 | 3.1021 | 23 | 0.4990 | 43 | 0.3332 | 63 | 0.2864 | 83 | 0.2735 |
| 4 | 2.3504 | 24 | 0.4834 | 44 | 0.3294 | 64 | 0.2852 | 84 | 0.2733 |
| 5 | 1.8996 | 25 | 0.4692 | 45 | 0.3258 | 65 | 0.2841 | 85 | 0.2731 |
| 6 | 1.5993 | 26 | 0.4560 | 46 | 0.3223 | 66 | 0.2830 | 86 | 0.2729 |
| 7 | 1.3850 | 27 | 0.4440 | 47 | 0.3191 | 67 | 0.2821 | 87 | 0.2728 |
| 8 | 1.2244 | 28 | 0.4328 | 48 | 0.3160 | 68 | 0.2811 | 88 | 0.2726 |
| 9 | 1.0996 | 29 | 0.4225 | 49 | 0.3132 | 69 | 0.2803 | 89 | 0.2725 |
| 10 | 1.0000 | 30 | 0.4129 | 50 | 0.3104 | 70 | 0.2795 | 90 | 0.2725 |
| 11 | 0.9186 | 31 | 0.4041 | 51 | 0.3079 | 71 | 0.2788 | 91 | 0.2724 |
| 12 | 0.8509 | 32 | 0.3958 | 52 | 0.3055 | 72 | 0.2781 | 92 | 0.2723 |

| | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|
| 13 | 0.7937 | 33 | 0.3881 | 53 | 0.3032 | 73 | 0.2775 | 93 | 0.2723 |
| 14 | 0.7448 | 34 | 0.3809 | 54 | 0.3010 | 74 | 0.2769 | 94 | 0.2723 |
| 15 | 0.7025 | 35 | 0.3741 | 55 | 0.2990 | 75 | 0.2763 | 95 | 0.2722 |
| 16 | 0.6656 | 36 | 0.3678 | 56 | 0.2970 | 76 | 0.2759 | 96 | 0.2722 |
| 17 | 0.6331 | 37 | 0.3619 | 57 | 0.2952 | 77 | 0.2754 | 97 | 0.2722 |
| 18 | 0.6043 | 38 | 0.3564 | 58 | 0.2935 | 78 | 0.2750 | 98 | 0.2722 |
| 19 | 0.5787 | 39 | 0.3512 | 59 | 0.2919 | 79 | 0.2746 | 99 | 0.2722 |
| 20 | 0.5556 | 40 | 0.3463 | 60 | 0.2904 | 80 | 0.2743 | 100 | 0.2722 |

## Factor for 10-Year Roughness Calculations

The 10-year roughness is given by:

$$C_{10} = C_0 - (C_0 - C_x) \times Factor_x$$

Where $C_0$ is the new pipe roughness and $C_x$ is the roughness after x years. $Factor_x$ is the factor from the above table based on the age of x years.

Example:

$$C_0 = 140$$

$$C_x = 120$$

$$x \text{ age} = 25 \text{ years}$$

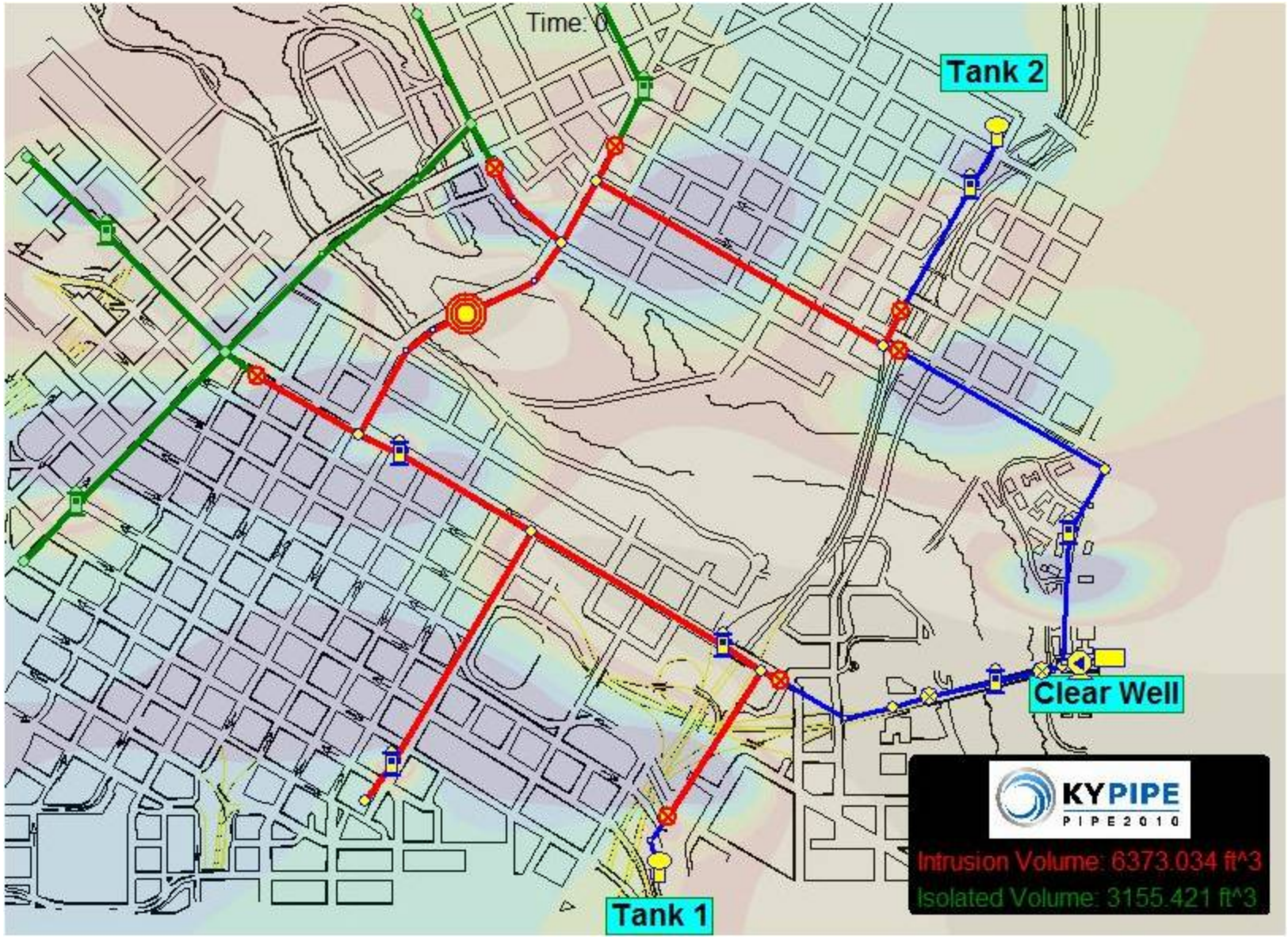
From the table, the factor at 25 years = 0.4692

$$C_{10} = 140 - (140 - 120) \times 0.4692$$

$$C_{10} = 130.6$$

### 1.4.8 Intrusion Calculator

A location or area where intrusion takes place is defined by the user.




Under Facilities Management, choose one of the following options:

Intrusion at a point - click on pipe at the location where the break occurs

Intrusion in a polygon - draw a polygon around the intruded area.

The Intrusion Calculator indicates valves used to isolate an intruded pipe. These are highlighted on the system Map. Volume, number of customers and total demand are reported for both the area that experiences intrusion and the areas that are isolated from a source by the closed valves.

Intrusion Volume and color-coded area is the affected volume/area which is contained by the closed valves. Water in the Intrusion area could potentially drain out of the compromised pipe, and may not be under pressure due to the closed valves.

Isolated Volume and color-coded areas are volumes/areas not directly affected by intrusion but which are no longer connected to a source (tank, water plant) due to the closed valves

Under Facilities Management, select Intrusion Report. The Contamination Report includes the following:

Intrusion Volume
Intrusion Customers (residential connections only, does not report junction demands)
Intrusion Demand (residential meters plus junction demands)
Isolated Volume
Isolated Customers
Isolated Demand
Valves to use for isolation
Hydrants in the contaminated area, indicates lowest elevation hydrant
Summary of impacted pipes.

Note: After simulation, valves may be clicked to open or close. Volumes will be recalculated.

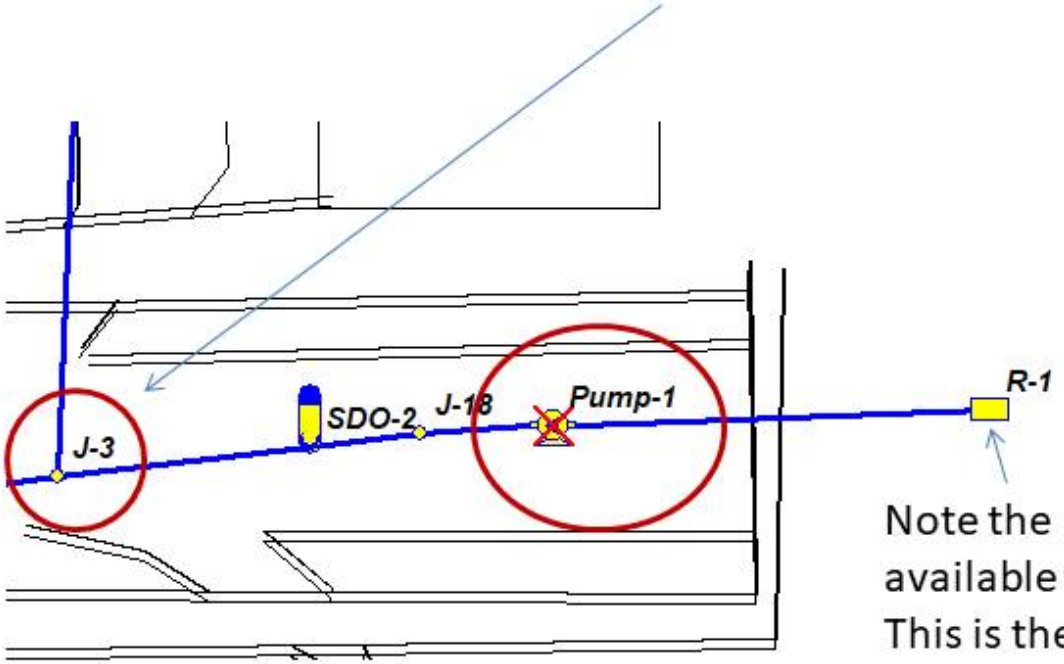
If an on/off valve is deactivated it is ignored in intrusion or pipe break simulation.

## 1.6 Surge

### 1.5.1 System Curve for Surge

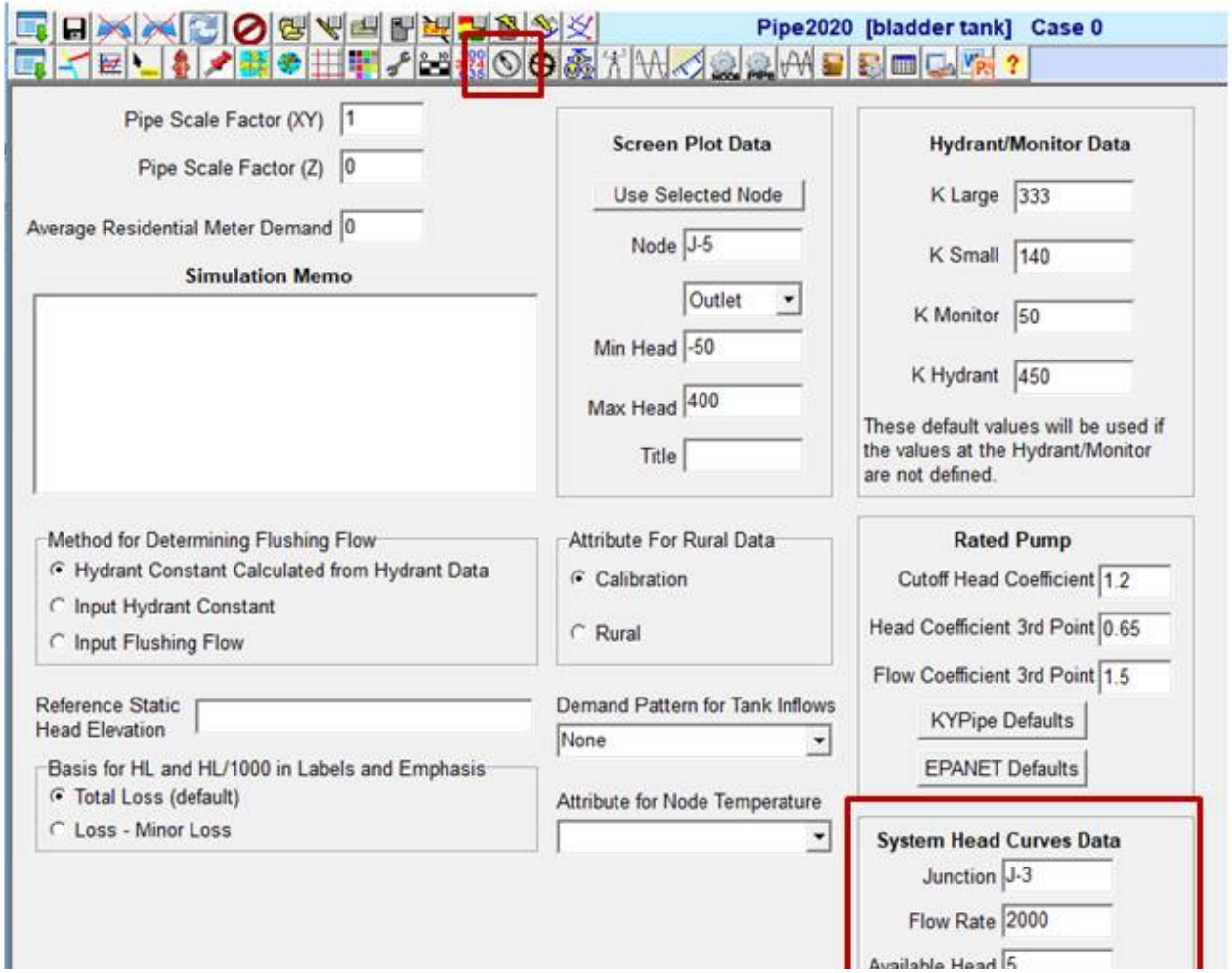
# System Curve Analysis in Surge

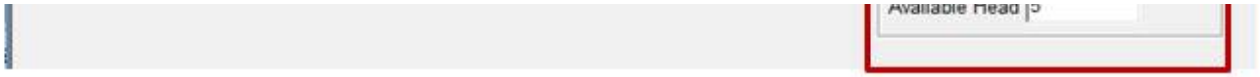
Make sure the pump is turned off and the downstream junction node is identified.



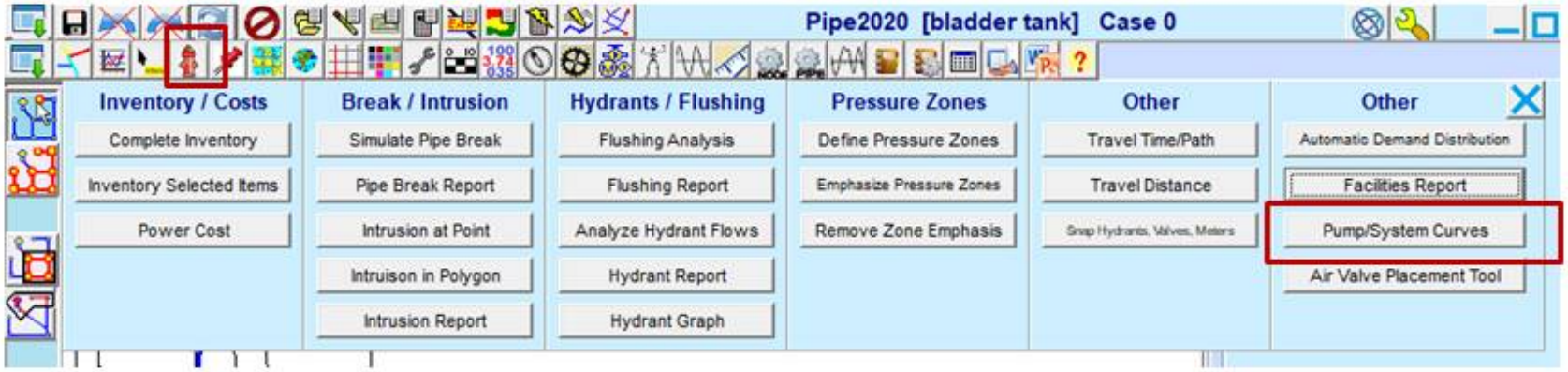

Note the amount of head available for the pump. This is the Available Head.

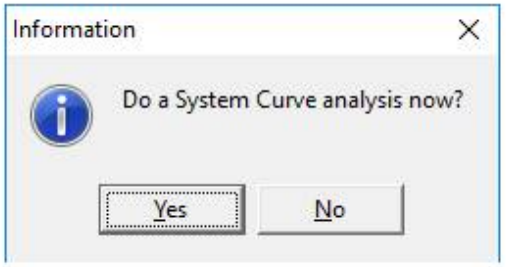
# Go to Other Data menu.
# Enter System Head Curves Data




Available Head

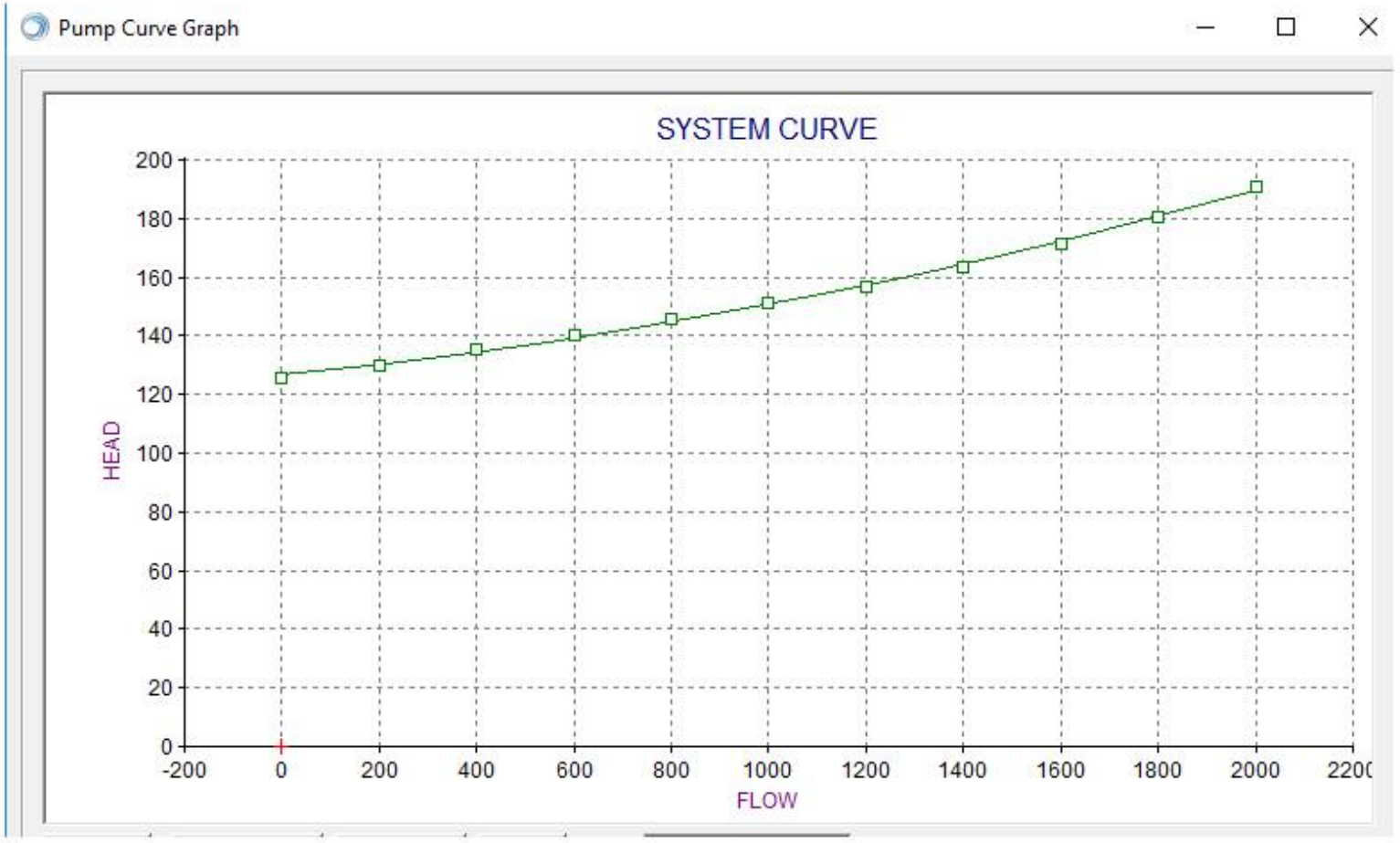
Go to Facilities Management.
Select Pump/System Curves



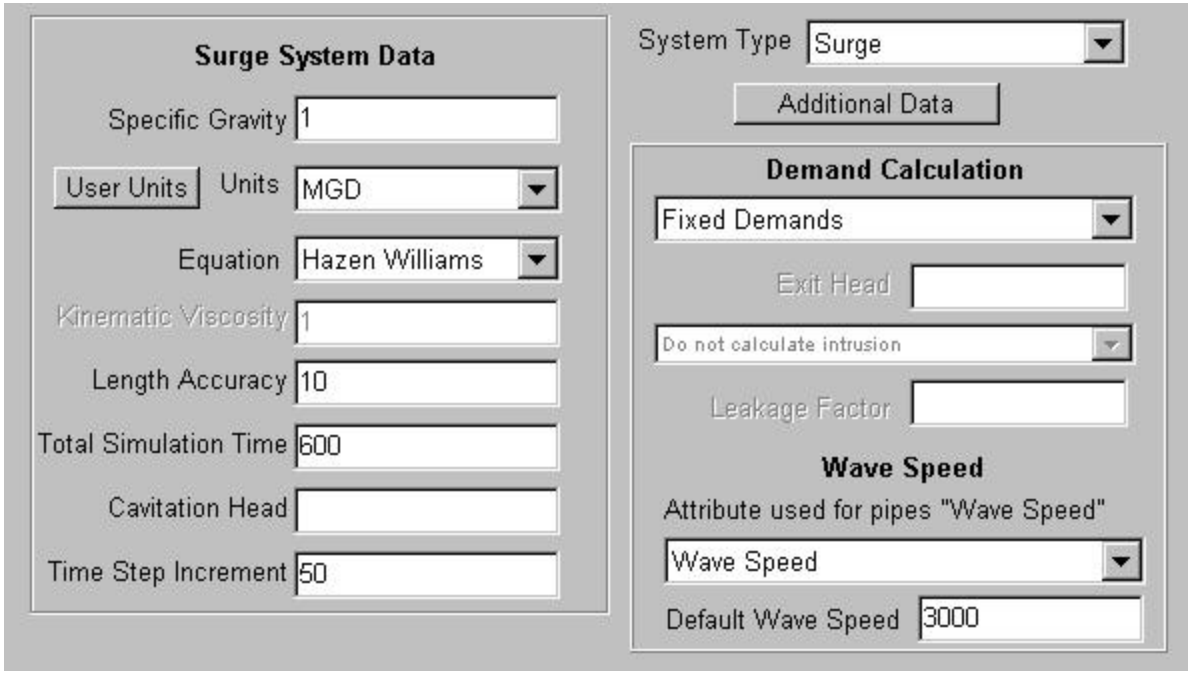




Answer Yes.

System Curve is calculated and plotted.



## 1.5.2 Surge - System Data / Simulation Specs




**Specific Gravity** - ratio of density of liquid to density of water. See also Specific Gravity.

**Units - Flow Units** CFS (cubic feet/second)

GPM (gallons/minute)

MGD (million gallons/day)

Liters/Sec (liters/second)

CMS (cubic meters/second)

Liters/Min (liters/minute)

Lb/s (pounds/second)

BPH (barrels/hour)

kg/s (kilograms/second)

USER (user defined units)

**Equation** - Head Loss Equation. Choose between Hazen Williams, Darcy Weisbach and Constant Resistance. The appropriate roughness must be entered (Pipe Information - Data) for the equation chosen. See Hazen Williams Table and Darcy Weisbach Table. For Constant Resistance the resistance is defined as the head drop (ft or m) divided by the flow squared (CFS or CMS)

**Kinematic Viscosity** - Required only for Darcy Weisbach equation (ft^2/s or m^2/s).

**Length Accuracy** - This input defaults to 10 ft (3 m) and represents the maximum difference between the actual pipe lengths and the ones chosen for the model. Note the calculation time increment and required computational time are affected by this selection and decreasing the length accuracy by a factor of two will double the required computational time.

Pipe lengths (or wave speeds) in the model must be adjusted so each pipe will be a length – wave speed combination such that the pressure wave will traverse the pipe in a time which is an exact integer multiple of the computational time increment. Lengths will be rounded to the nearest multiple of the Length Accuracy (not including 0), therefore the maximum difference between adjusted pipe lengths in the model and actual system is usually Length Accuracy/2. For example if we use Length Accuracy = 20 the lengths will be rounded to the nearest multiple of 20 and the largest difference between the model adjusted lengths and actual length is 10 feet (say 380 feet for a 389 foot pipe). This does not hold for pipes which are shorter than the Length Accuracy. The adjusted pipe length will be equal to the Length Accuracy so that the maximum difference is <= Length Accuracy. For example if a pipe is 0.5 feet long and the Length Accuracy is 10, then the adjusted pipe length will be 10 ft or 9.5 feet of difference.

It is important therefore for the Length Accuracy to be similar in value to the length of the smaller pipes in the model.

Note for very large systems, there is a limit to total number of time increments which is effected by Length Accuracy. Sometimes it is necessary to increase the Length Accuracy value to stay within this limit.

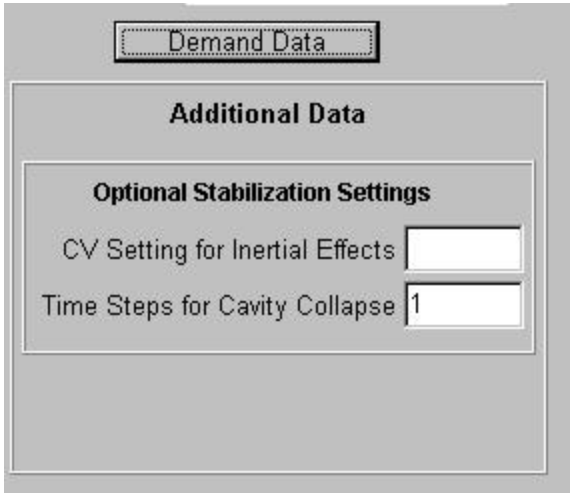
**Total Simulation Time** - The time for a wave to travel down a pipeline and back is 2L/c, where L= length (or longest path through a looped system) and c=wave speed. A good approximation for Total Simulation Time would be to allow this cycle to happen 5 times or 10L/c. Once the Surge analysis is run, if the pressure waves have not dampened to a new steady state conditon, increase the Total Simulation Time. If it dampens rapidly, the Total Simulation Time can be decreased. .

**Cavitation Head** - This defaults to values applicable at standard pressure and can be entered here to override the default. For water, if left blank the program assumes caviation head to be -14.4 psi (-99.3 kPa) or -33.2 feet (-10.1 m). If the fluid is other than water then this is divided by Specific Gravity, SG.

**Time Step Increment** - This must be an integer and will result in a smaller time increment being used. New time increment = calculated time increment / time step increment. Hence the input 5 will result in a new time increment on 1/5 of the original and will increase the required computational time by a factor of 5. This input may be required for high frequency periodic disturbances.

**System Type** - Should say 'Surge'

**Additional Data** - This will bring up the following box. See also Stabilization Settings.




**CV Setting for Inertial Effects** - With check valves that are opening and closing repeatedly, this allows the user to incorporate inertial effects into the modeling of the check valve. CV Setting for Inertial Effects is a number between 0 and 1. The default is 0 and will result in no inertial effects. If the CV is in the closing mode (reverse flow through the element) and the CV open ratio is greater than CV Setting, then Pipe2018 continues to close the CV until it is completely closed EVEN if the flow changes sign (flow in the forward direction). In other words, it is assumed that the CV continues to close once it reaches certain closing position due to the inertial effects of the disc or flap. The use of this setting will reduce pressure spiking due to check valve action. The default of zero should be applied for CVs that are initially closed.

In general no inertia effect (Zero) results in a "rattling" of the flap. The intended effect of the CV Setting for Inertial Effects is to slow down the rate (frequency) of the "rattling" flap by requiring the CV action (opening or closing) to be completed before swing the opposite direction. Due to inertia the swing will complete after reaching a certain % of the opening or closing before the flap can go in the opposite direction. This tends to reduce spiking somewhat because the fluctuation rate is slowed.

**Time Step for Cavity Collapse** - Time Step for Cavity Collapse is an integer representing the number of time increments for complete collapse of a vapor cavity. The default is 1, implying that the cavity collapses in one time increment, however small the time increment is. The purpose of this setting is to "soften" the effects of cavity collapse. Since air is normally released when cavitation occurs a softening effect is expected. Without allowing for this, excessive pressure spiking may occur. The factor is applied to the simulation time step. Therefore if your simulation time step is 1/100 second (this value is shown in the Report), then a value of 10 here would have the cavity collapse in 1/10 second.

**Demand Calculation** - Choose between Fixed Demands and Pressure Sensitive Demands. For Fixed Demands the demands remain constant throughout the

transient analysis and no additional input is required. For Pressure Sensitive Demands the demands vary with the pressure difference in the pipe and the exit region.

**Exit Head** - The assumed head in the exit region for pressure sensitive demands. This defaults to zero (atmospheric pressure).

**Calculate Intrusion** - If Pressure Sensitive Demands are selected, three options are available:

**Do not calculate intrusion**

**Calculate intrustion using a leakage factor**

**Calculate intrusion using a leakage contstant.**

If one of the last two are selected then intrusion of liquid back into the pipe system will be calculated when the exit head exceeds the head inside the pipe. The user will need to then enter the Leakage Constant or Leakage Factor for that calculation. See below.

**Leakage Coefficient** - If intrusion is calculated, this input determines the ratio of flow which can intrude for the same differential head. For example, a Leakage Coefficient of 0.1 will calculate an intrusion rate 0.1 times as great as the corresponding outflow at the same head difference. The Leakage Coefficient may correlate to the lost water percent.

The Leakage Factor is the %/100 for leakage in the distribution system. For example, a Leakage Factor = 0.05 means that the leakage is 5% of the demand. From this factor a leakage coefficient is calculated for each demand junction which will discharge 5% of the flow at the initial pressure. Therefore, at a junction where the demand is 10 gpm at 64 psi (difference) we assume that 0.5 gpm is leaking through an orifice and the leakage constant is computed as Lc = Q(leak)/(pressure difference))^0.5 or Lc = 0.5/(64)^0.5 = 0.0625. This constant is used with the relationship for leak flow (Q(gpm) = Lc*(pressure)^0.5 when the pressure gradient is negative (Pexit > Pline). For example if the line pressure is 1psi less than the exit pressure the intrusion = 0.0625 gpm.

The Leakage Constant is defined above using the relation Q(leak) = Lc*(pressure difference). If we use Lc = 0.1 we will get a leak of 0.1 gpm with a pressure differential of 1 psi. The leakage constant is applied to every node (not just demand nodes) and only is used for intrusion calculations.

**Wave Speed Attribute** - This should normally display the User Data "wave speed" although another User Data attribute can be used (but this is not recommended). For new files this should automatically appear. For imported files the "wave speed" attribute may not be defined and will not appear on the drop down list of User Data attributes. In this case the user should access the User Data and make a "wave speed" attribute and then select this in the drop down list.

**Default Wave Speed** - If a value is entered here, this will be used for all pipes for which a wave speed is not specified in the Pipe Information window data box (Map Screen) or in the Wave Speed User Data item defined in the Wave Speed Attribute field above.

## 1.5.3 Surge Modeling Requirements

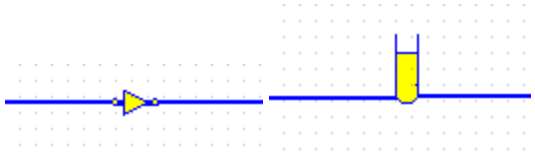
Surge Modeling Requirements

## Using Surge

### Modeling Requirements

A Surge model is comprised of pipe sections connecting junctions, components, and surge control nodes. The following restriction applies:

Only two pipe connections are allowed for regulators and surge control devices. These include side discharge orifice's (SDO's), surge tanks, relief valves, sprinklers, surge anticipation valves and air/vacuum valves.



The Surge error check will check this requirement and produce an error message if the requirement is not met.

Number of changes - Up to 99 elements may have changes. For example, in a single model, up to 99 different pumps may be set up to trip.

# 1.5.4 Surge Calculations and Sign Conventions

Surge Calculations and Sign Conventions

See also Surge Modeling Requirements

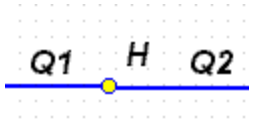
Surge Node Results

### Calculated Parameters

For surge analysis, calculations (heads, flows, volumes) are made at each element for each time step, and flows for pipes at each time step. This produces very large files of results. Elements consist of junctions (including reservoirs and dead ends), components (usually pumps and valves) and surge control devices. For each time step the following values are calculated for each element. Note that there are no results presented for pipes since both head/pressure and flow vary along the pipe.

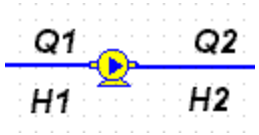
| | |
|---|---|
| junction: | head/pressure (one value) |

flow (each connection)




component head/pressure (each side)

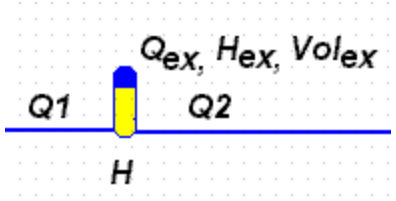
flow (each side)




surge control (SDO) head/pressure (inside and external)

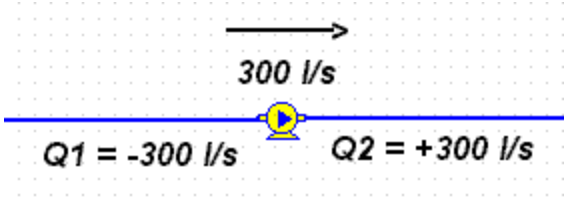
flow (side 1, side2, external)

air or cavity volume (external)




### Sign Convention for Flow

Flow away from a node is positive and flow toward the node is negative. For example, if 300 l/s is flowing in the normal direction through a pump the outlet flow is +300 l/s while the inlet flow is -300 l/s. Therefore, when plotting and tabulating results you can choose to utilize the outlet value to get positive flow. See Surge Node Results




### Notes on algorithm

Air cavities - Surge, as with all surge modeling programs, assumes that cavities and air pockets are located at a point and the pipe segments remain full of water and transmit pressure waves. In other words the lengths of adjacent water columns will not change. There is no attempt to allow vapor cavities and air pockets to change lengths of adjacent water columns and does not account for the volume occupied by air pockets or consider the changes in wave transmission times which would result.

Intermediate nodes - Because of the effect of friction on pressure waves a calculation is made for each wave in each pipe section. When a wave goes through a pipe with intermediate nodes the effect of friction is made with one calculation. If there are junctions instead and two or more pipe sections the effect of friction is calculated for each pipe section. This can cause differences - especially if there are very long pipe sections. We do not recommend that a model have very long pipe sections for surge because of the handling of the friction.

## 1.5.5 Surge Elements and Data Requirements

Surge Elements and Data Requirements

---

**See also** **Surge Components**
**Surge Control and Control Devices**
**Surge Variable Input Data**

# Surge Elements and Data Requirements

**Pipes**

**Wave speed**

**Nodes**

**Junctions**
**Demands**
**Reservoirs and Tanks**
**Pump Table**
**Pump File**
**Pump Constant Power**
**Pump Rated**
**Sprinkler/Leak**

Regulators
Loss Element
Pressure Supply
Active Valve
Side Orifice
Open Surge Tank
Spilling Tank
One Way (Feed) Tank
Closed Surge Tank
Hybrid Tank
Bladder Tank
Pressure Relief Valve
Rupture Disk
Surge Anticipation Valve
Air Vacuum Valves
1/2/3 Stage Valves

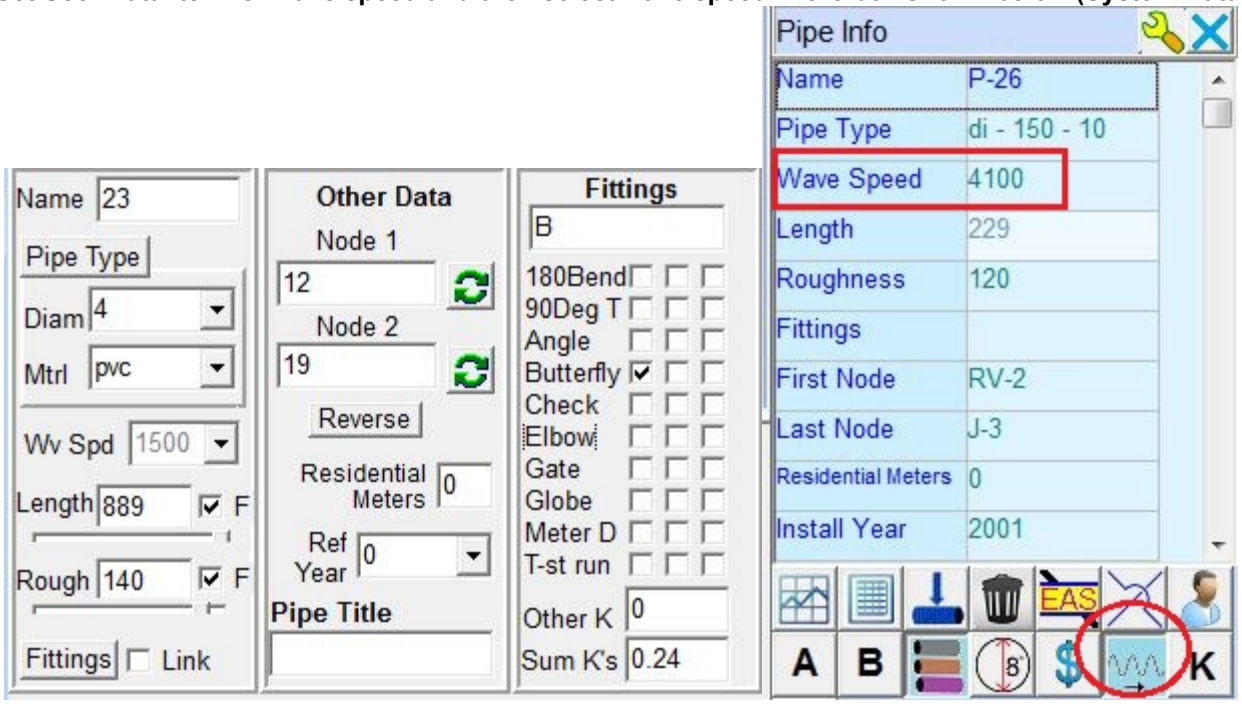
**Creating a Transient**

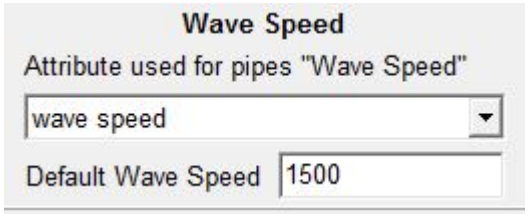
Pumps
Valves
Demand Changes
Reservoir Head Changes
Periodic Variations

# Surge – Pipes

See also Surge PipeData/Wave Speed

**Set User Data item for wave speed and then select wave speed in the box shown below (System Data/Simulation Specifications)**




Pipe Data

Diam(eter) – in. or mm

Wave Speed – ft./s or m/s.

Length – ft. or m.

Roughness – depends on HL equ.

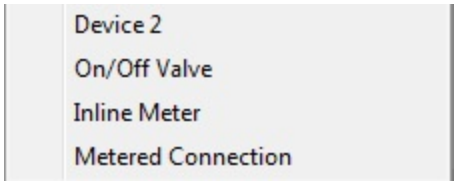
Number of Residential Meters –

Fittings data – sum of K's

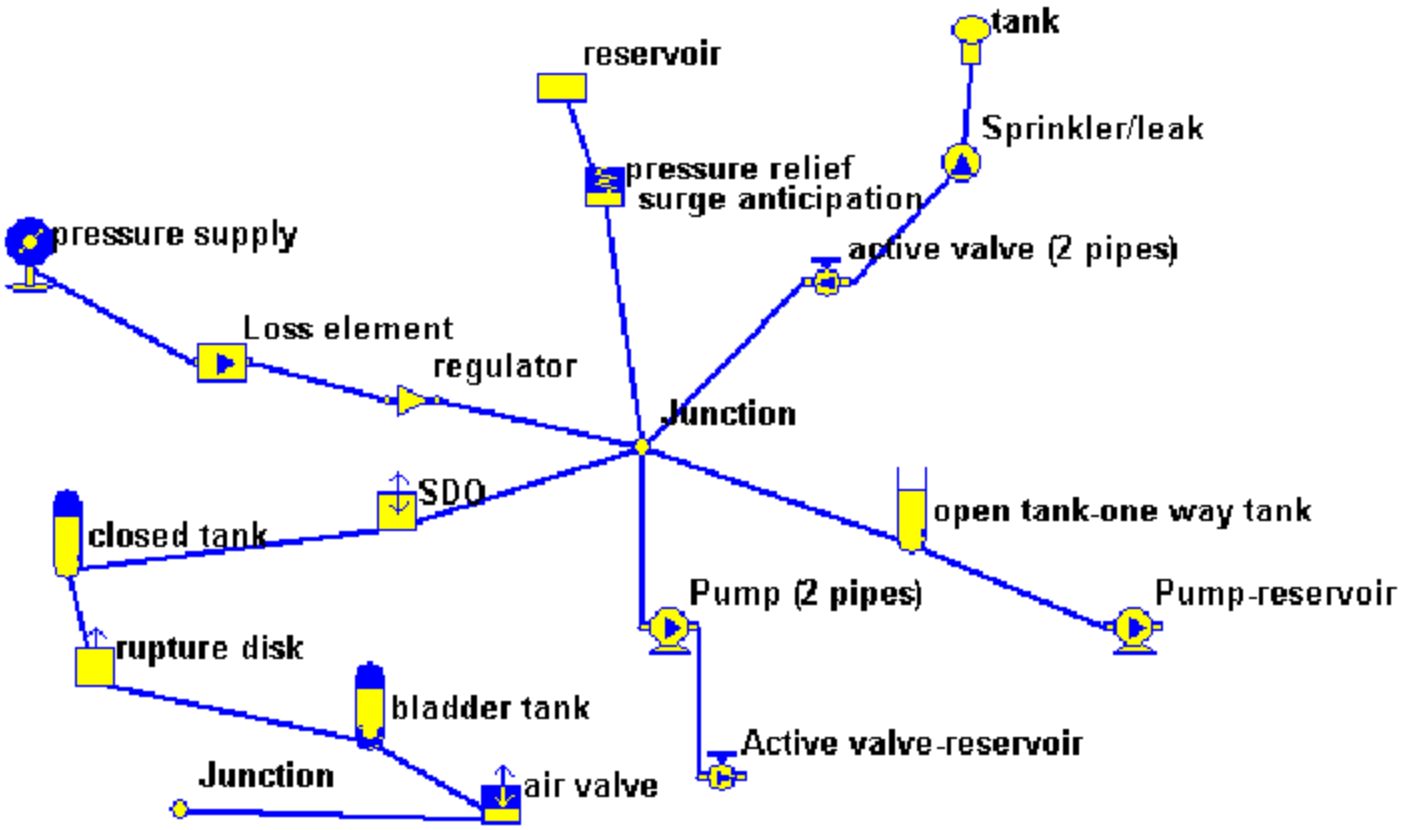
**Pipes connect two nodes and can have intermediate nodes for alignment changes. Because of the importance of the location of closed valves, for Surge models pipes cannot be closed. Closing pipes must be done using a closed Active Valve. Also for Surge models pipes do not incorporate internal node type check valves as can be done in steady state models. For Surge models Check Valves are designated at nodes (such as Pumps and Active Valves) or using the Check Valve Element. If a Check Valve is closed for the steady state then the element which incorporates the Check Valve will be initially closed. If conditions are appropriate the element Check Valve will open during the transient analysis**

## Steady State (SS) and Surge Nodes

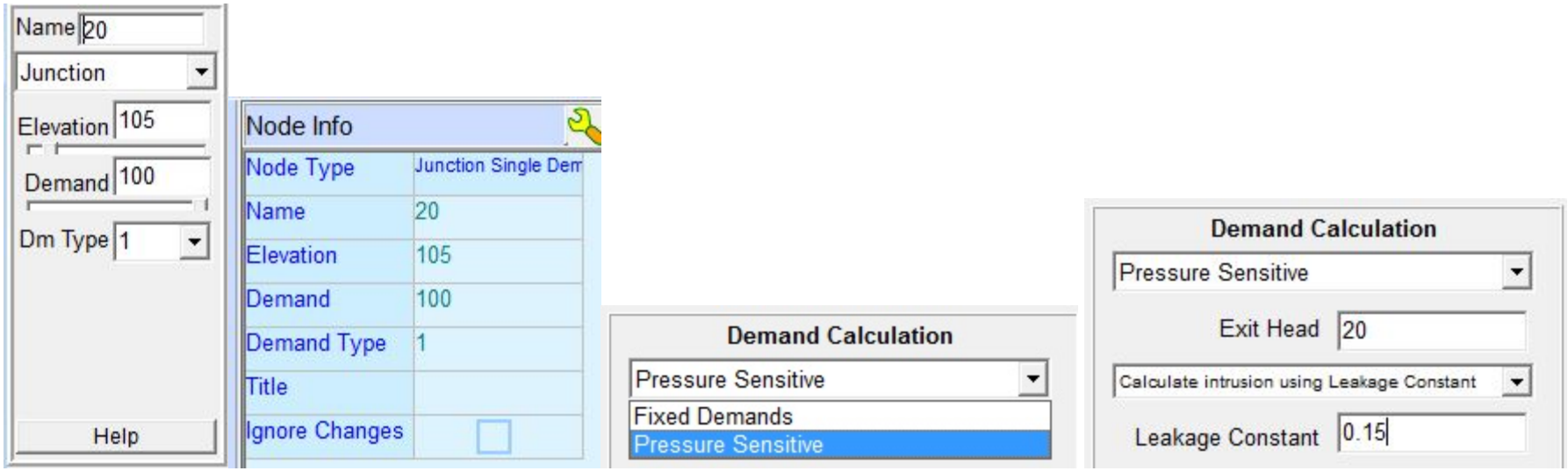
Junction Single Demand
Junction Multi Demand
Tank (Surge)
Reservoir (Grade)
Reservoir (Pressure)
Reservoir (Depth)
Pumps
Library BFP
Library Pump
Library Air Valve
Sprinkler/Leak
Regulator
Loss Element
Rated Pressure Supply
Variable Pressure Supply
Active Valve
Turbine
Wicket Gate
Low Prs Sewer Tank (LPS)
Hydrotank
Hydant Monitor
Zero Velocity Valve
Check Valve Element
Holding Tank
SDOs
Modules
Intermediate Node
Device 1




The list above and the network below show the devices that can be included in a Surge model. Some of these devices are utilized in both the initial steady state and the surge analysis. These devices have specific modeling requirements (number of pipe connections) for Surge which are illustrated in the network shown below and discussed on the following pages. Each of these devices requires unique input data, which includes the elevation (in feet or meters). The additional data requirements for each device are presented on separate pages along with notes regarding their use.




## SS & Surge - Junctions



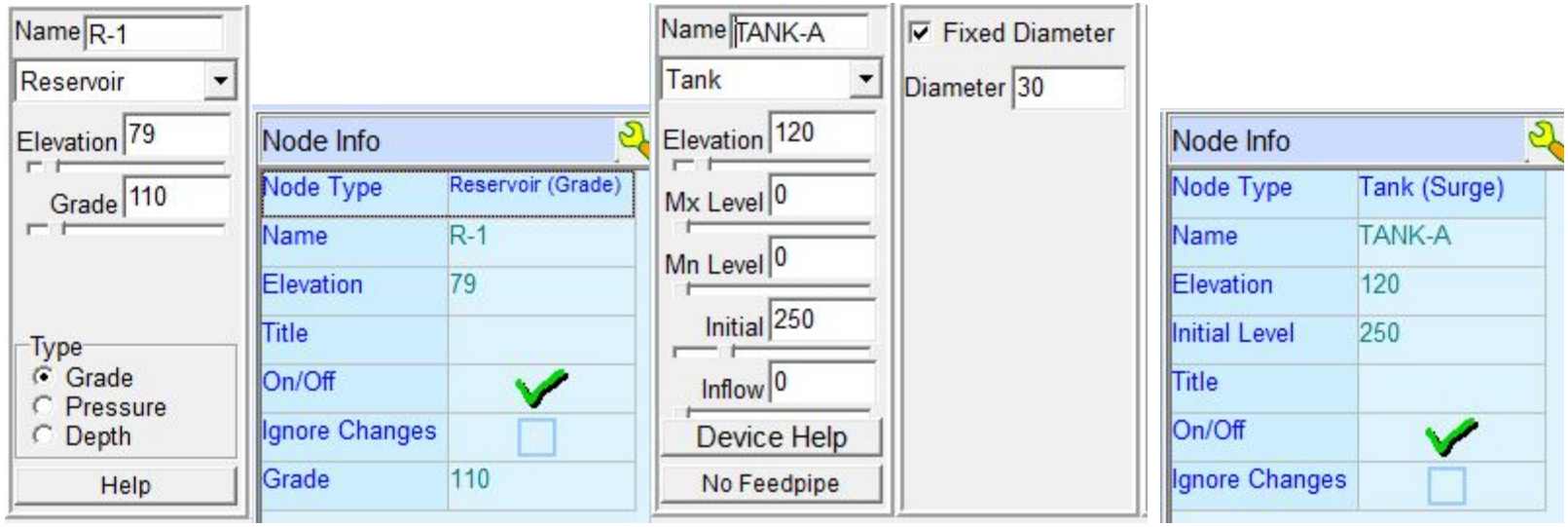

Connection Restrictions: No restrictions for junctions

**Demands** – each junction may be assigned a Demand by the user or through meters connected to pipes connected to the junction. The **Demand Type** is not
utilized in the surge analysis.

Under System Data menu icon or System Data | Simulation Specs (Classic):
**Demand Calculation** – select calculation method:
- **Fixed Demands** – demands remain constant throughout transient analysis
- **Pressure Sensitive Demands** – demands vary with pressure according to the orifice relation

**Intrusion Calculation** – select calculation method
- Do not calculate intrusion
- Calculate using Leakage Factor or Leakage Constant

# SS & Surge – Reservoirs and Tanks




Reservoir and Tank Data

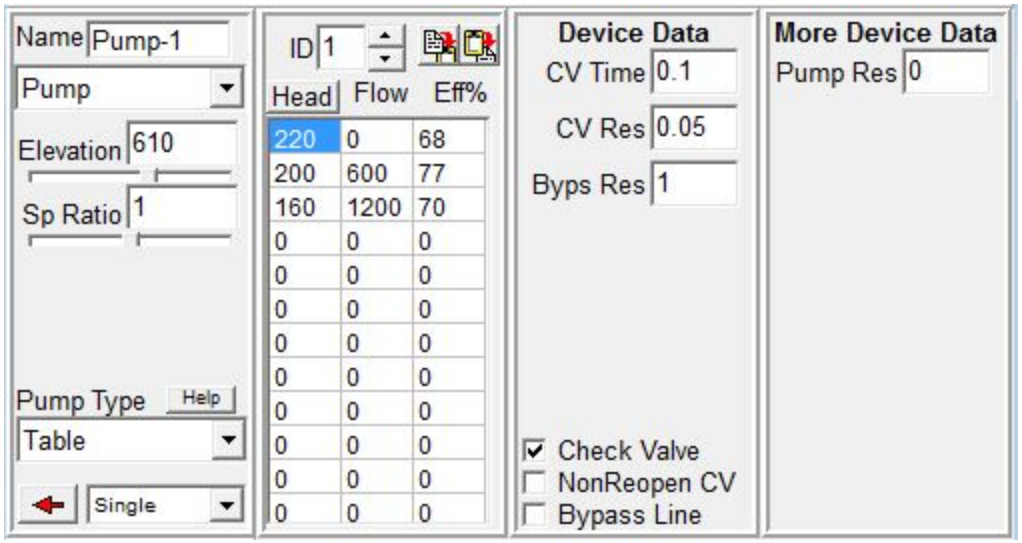
On/off switch
Grade – ft. or m.
Initial (level) – ft. or m.

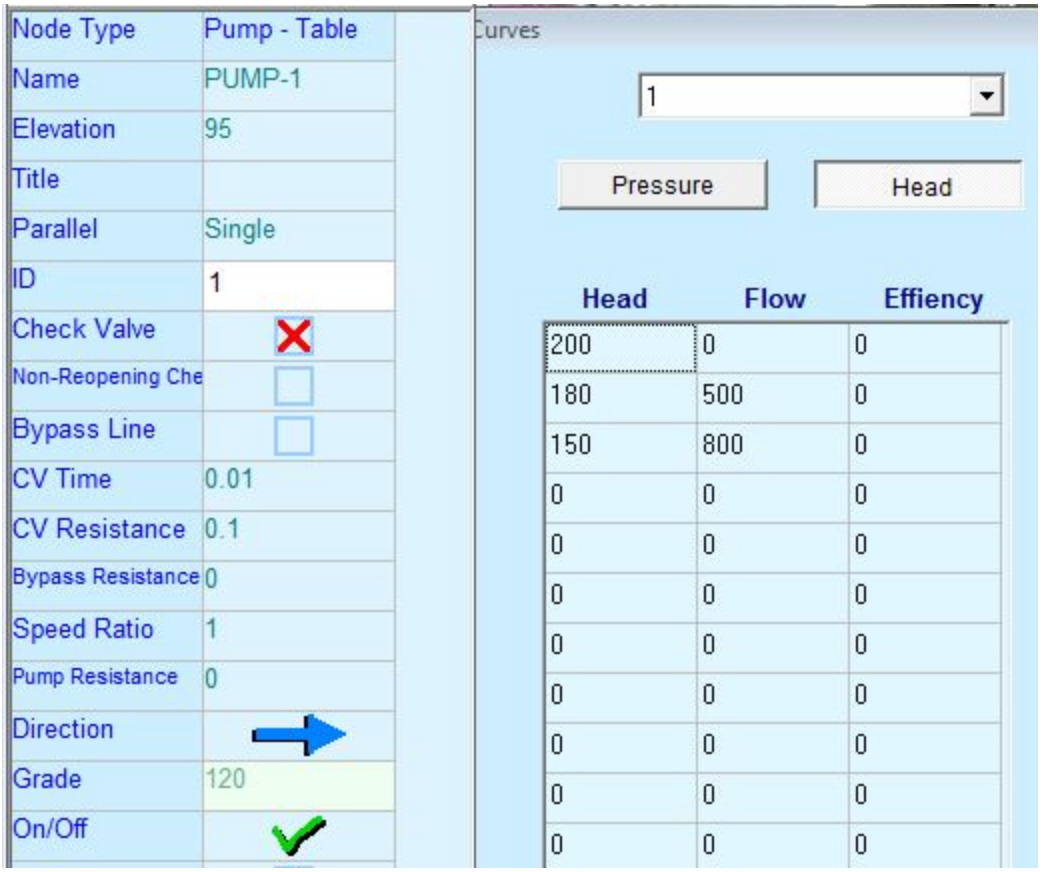
Connection Restrictions: Only one pipe can connect these devices

**Reservoir** – A constant grade is maintained at a reservoir. The grade maintained is input. **Grade is defined as elevation and pressure head (in feet or meters).**

**Tank** – A supply tank is treated as a constant head reservoir for surge analysis. Besides the Elevation the (Initial) grade (elevation of level) is the only input value required for a tank. A Feedpipe is a steady-state feature, which does not apply to Surge Analysis.

## SS & Surge – Pump Table (Head/Flow Data)






Pump Table Data

On/off switch

Speed (ratio) – rpm/rated speed

Head – ft. or mt.

Or

Pressure – psi or kPa

Flow – specified flow units

Eff(iciency) - %/100 (not used in Surge)

Connection Restrictions: one pipe connected to each side or one pipe connected to one side and reservoir on other side

This Pump Type is selected as Table. It is ideal to input into the head (pressure)/flow table exactly 3 data points including cutoff head (at zero flow), although more data points are allowed.

Pumps may be specified as multiple pumps (e.g. 2 parallel pumps). Note that results are for combined pump configuration, not for individual pumps.

All pumps can include a Check Valve, Non Reopening Check Valve and/or Bypass line. These options perform the following functions.

**Check Valve** – prevents flow reversal through the pump

CV Time = time it takes check valve to fully open (or close) after flow reversal initiates

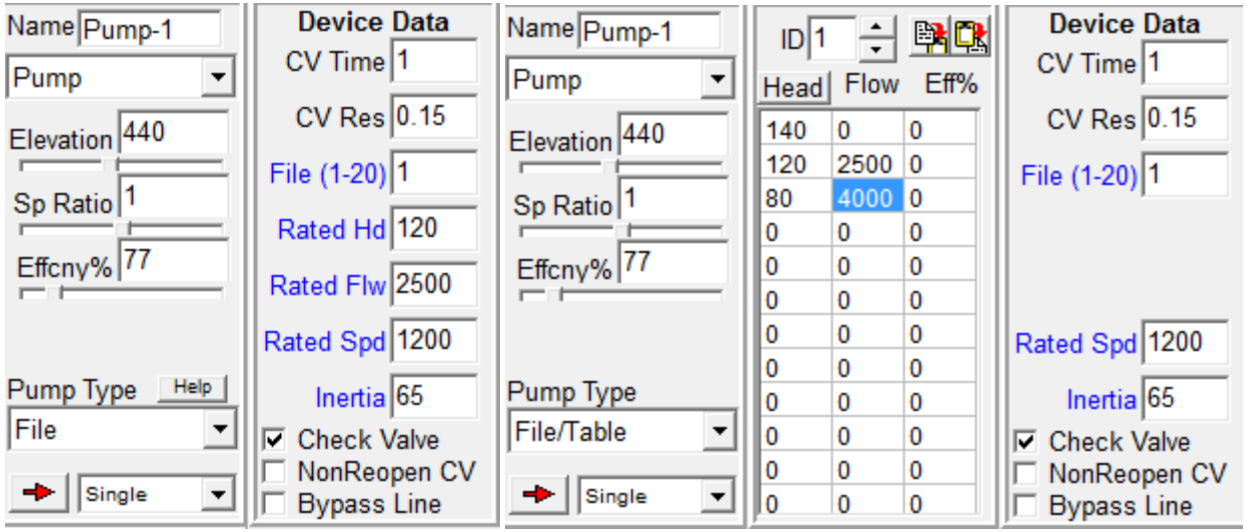
CV Res(istance) = wide open resistance for the check valve

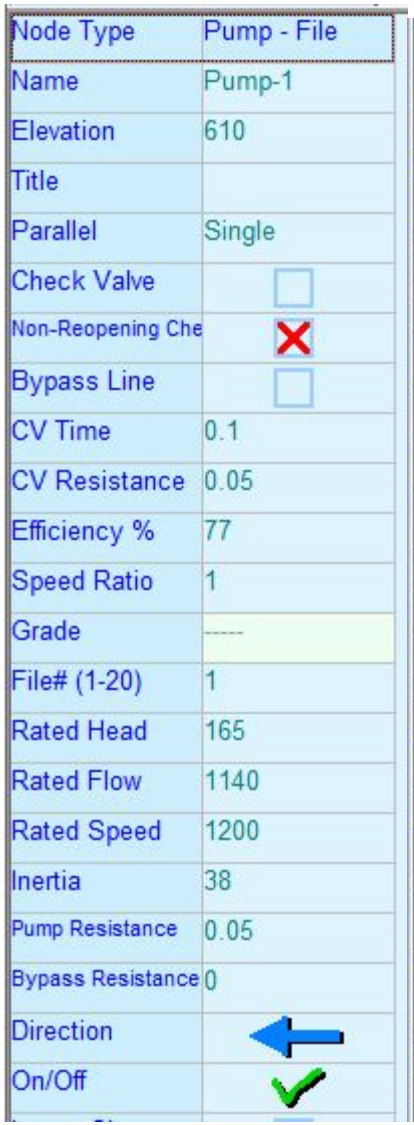
**Non-Reopen(ing) CV** – This check valve will not open once it has closed.

**Bypass Line** – This allows flow to bypass the pump when the suction head exceeds the discharge head.
The bypass line has an internal check, independant of whether a check valve is specified for the pump. Byps Res(istance) = resistance of the Bypass line

**Pump Resistance** - With parallel or series pumps, it may be desired to define a resistance to account for pump piping. For this purpose, Pump Res in the More Device Data box may be used (click the 'More' button or pointing hands to view, if necessary). A pump resistance for one individual pump is eneter here and is defined in units of headloss/(flow)^2. A special tool is available to calculate the resistance, Resistance Calculations. In the Resistance Calculation Tool, specify 'Piping for Parallel and Series Pumps.'

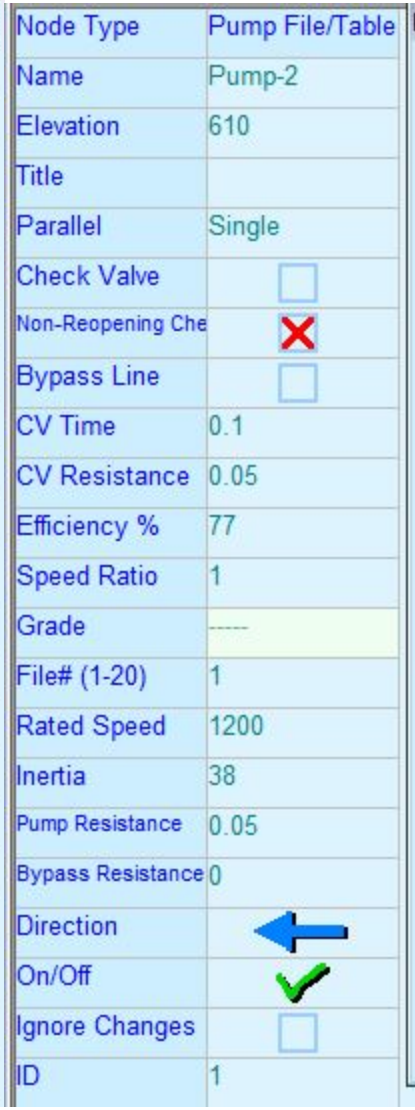
# SS & Surge – Pump File





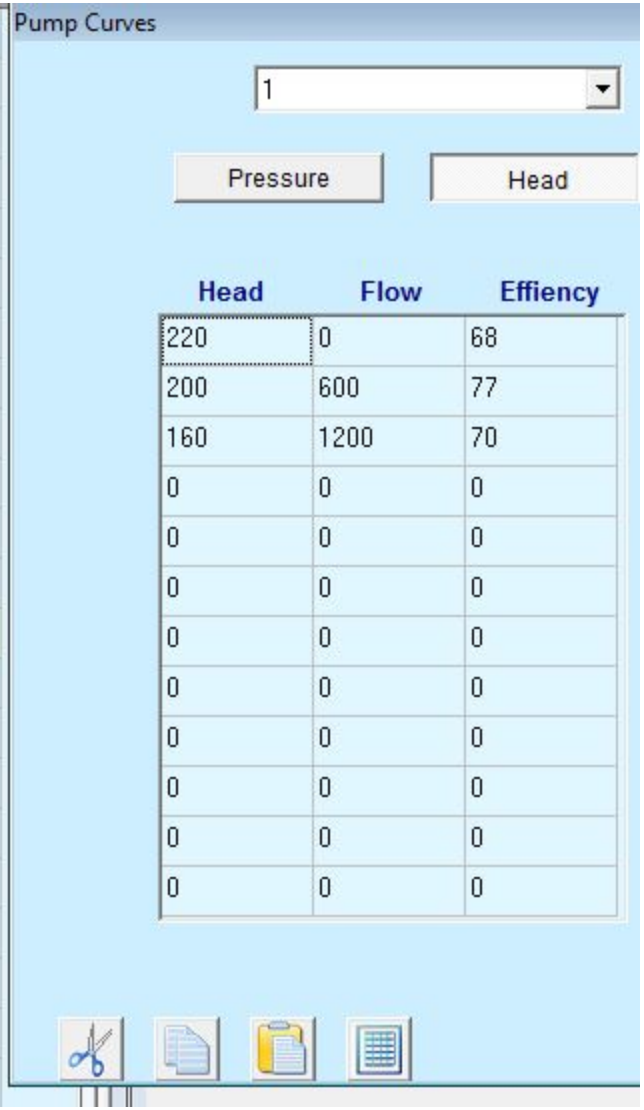
| Node Type | Pump - File |
|---|---|
| Name | Pump-1 |
| Elevation | 610 |
| Title | |
| Parallel | Single |
| Check Valve | |
| Non-Reopening Che | |
| Bypass Line | |
| CV Time | 0.1 |
| CV Resistance | 0.05 |
| Efficiency % | 77 |
| Speed Ratio | 1 |
| Grade | ----- |
| File# (1-20) | 1 |
| Rated Head | 165 |
| Rated Flow | 1140 |
| Rated Speed | 1200 |
| Inertia | 38 |
| Pump Resistance | 0.05 |
| Bypass Resistance | 0 |
| Direction | |
| On/Off | |

File Pump



| Node Type | Pump File/Table |
|---|---|
| Name | Pump-2 |
| Elevation | 610 |
| Title | |
| Parallel | Single |
| Check Valve | |
| Non-Reopening Che | |
| Bypass Line | |
| CV Time | 0.1 |
| CV Resistance | 0.05 |
| Efficiency % | 77 |
| Speed Ratio | 1 |
| Grade | ----- |
| File# (1-20) | 1 |
| Rated Speed | 1200 |
| Inertia | 38 |
| Pump Resistance | 0.05 |
| Bypass Resistance | 0 |
| Direction | |
| On/Off | |
| Ignore Changes | |
| ID | 1 |

File/Table Pump




| Head | Flow | Effiency |
|---|---|---|
| 220 | 0 | 68 |
| 200 | 600 | 77 |
| 160 | 1200 | 70 |
| 0 | 0 | 0 |
| 0 | 0 | 0 |
| 0 | 0 | 0 |
| 0 | 0 | 0 |
| 0 | 0 | 0 |
| 0 | 0 | 0 |
| 0 | 0 | 0 |
| 0 | 0 | 0 |
| 0 | 0 | 0 |

Pump File Data

On/off switch

Speed (ratio) – rpm/rated speed

Efficiency - %/100 (at rated flow)

File (number) – 1 to 8

Rated Head – ft. or mt.

Rated Flow – specified flow unit

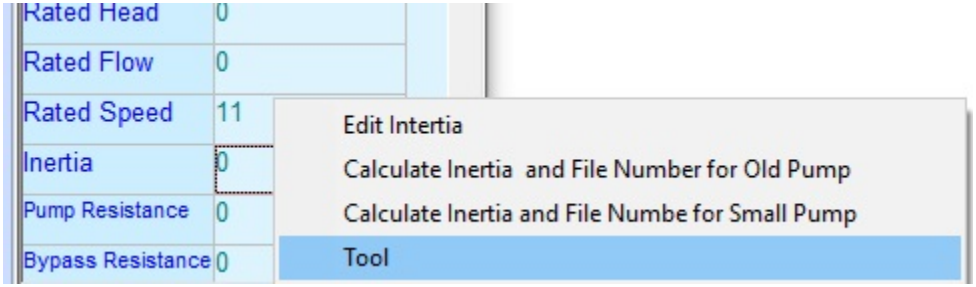
Rated Speed – rpm

Inertia (more info below) - # $ft^2$ or $nmt^2$

Connection Restrictions: one pipe connected to each side or one pipe connected to one side and reservoir on other side

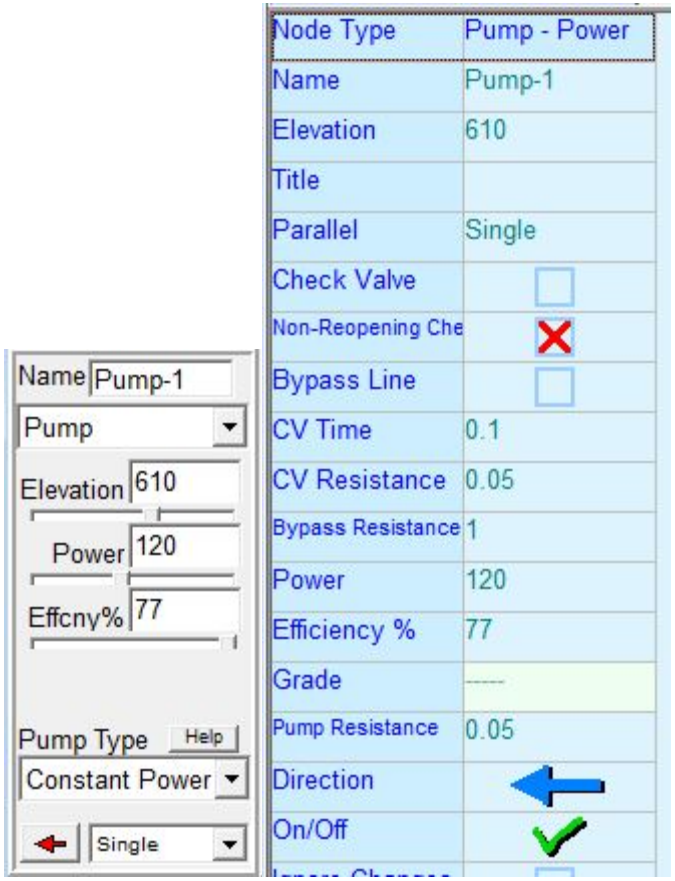
The **Pump Type** is selected as **File**. This description should be employed only if the pump will be tripped and therefore operated abnormally during the transient (flow reversal, turbining, etc). The other three descriptions should be used for transient analysis only if the pumps always operate in the normal zone of operation (positive head-positive flow). To use this description, you must select file (1-8) based on the specific speed (use the **Select Pump File/Calculate Inertia** tool) and provide the rated head (feet or meters), flow and speed. The motor and pump inertia is normally difficult to obtain from the manufacturer and a reasonable estimate can be calculated using **Inertia/Specific Speed** tool.

Inertia - (Pipe2018 and later) Inertia may be automatically calculated.



**Selecting an Inertia value using the Tool "Select Pump File/Calculate Inertia"**: Generally, the Combined (motor + pump) from the tool is used. Since pumps have gotten lighter, newer pumps have a lower inertia so the user should select older or newer based on the age of the pump. Inertia selection is approximate, just a best guess. Inertia data from the manufacturer is always best, but is often not available.

# SS & Surge – Pump (Constant Power)




Constant Power Pump Data

On/off switch

Power – horsepower or kilowatts

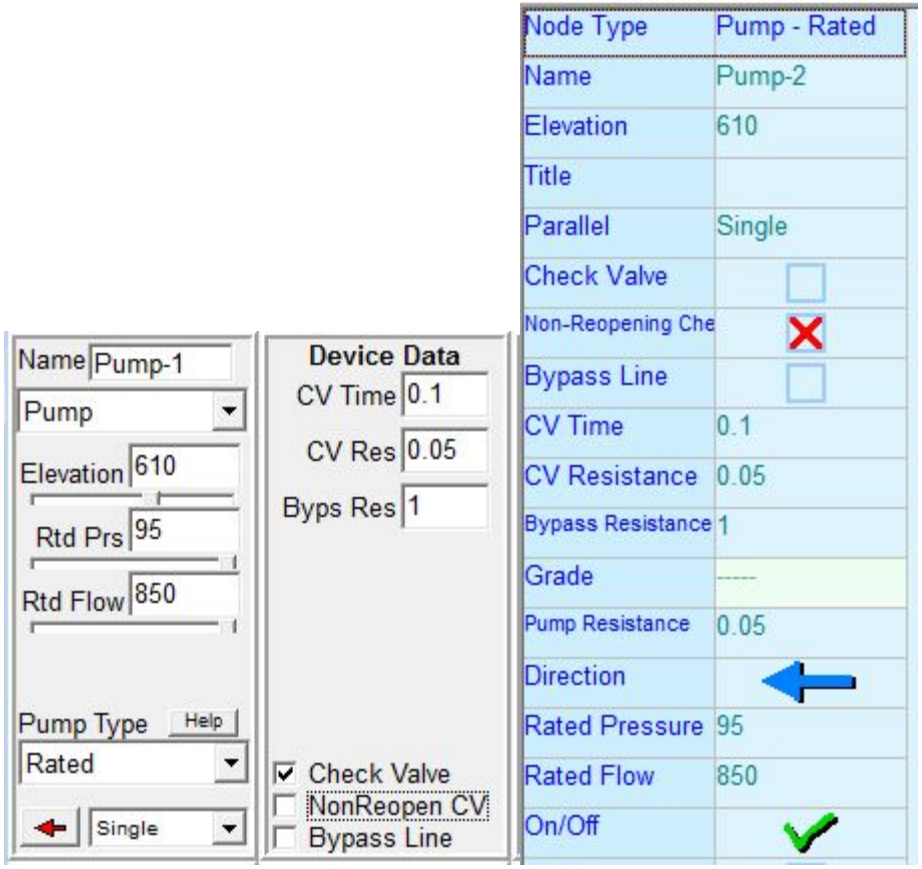
Efficiency - %/100 (at rated flow)

(not used in Surge)

Connection Restrictions: one pipe connected to each side or one pipe connected to one side and reservoir on other side

This **Pump Type** is selected as **Constant.** All the user is required to enter is the useful power (horsepower or kilowatts). For the surge analysis a head/flow curve is generated based on using the steady state operating point as rated conditions employing the same procedure as that described for the Rated Pump. The constant power pump description is not recommended for Surge Analysis because of the assumptions required to apply it. This description also cannot be used for pumps that are initially off since the appropriate rated conditions cannot be determined.

## SS & Surge – Pump (Rated)




Rated Pump Data

On/off switch

Rated pressure – psi or kPa

Rated flow – specified units

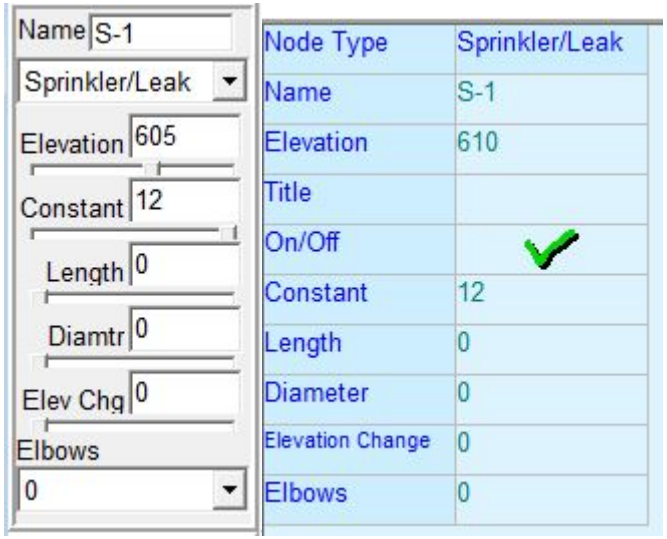
Connection Restrictions: one pipe connected to each side or one pipe connected to one side and reservoir on other side

This **Pump Type** is selected as **Rated**. The user is required to enter the Rated Pressure (in psi or kPa) and Rated Flow (selected flow units). A normal operating range pump curve is generated using the following pressure/flow data:

| Pressure | Flow |
|---|---|
| 1.4 $P_R$ | 0 |
| $P_R$ | $Q_R$ |
| .65 $P_R$ | 1.5$Q_R$ |

where $P_R$ is the rated pressure and $Q_R$ is the rated flow. This relationship may be customized in the Other Data menu or System Data | Other (Classic).

## SS & Surge – Sprinkler/Leak



Sprinkler Data

On/off switch

Constant – flow in gpm for 1 psi

or

flow in l/s for 1 kPa

Length – (of connector) ft. or m.

Diameter – (of connector) in. or mm

Elevation Change – ft. or mt.

Number Elbows in connector

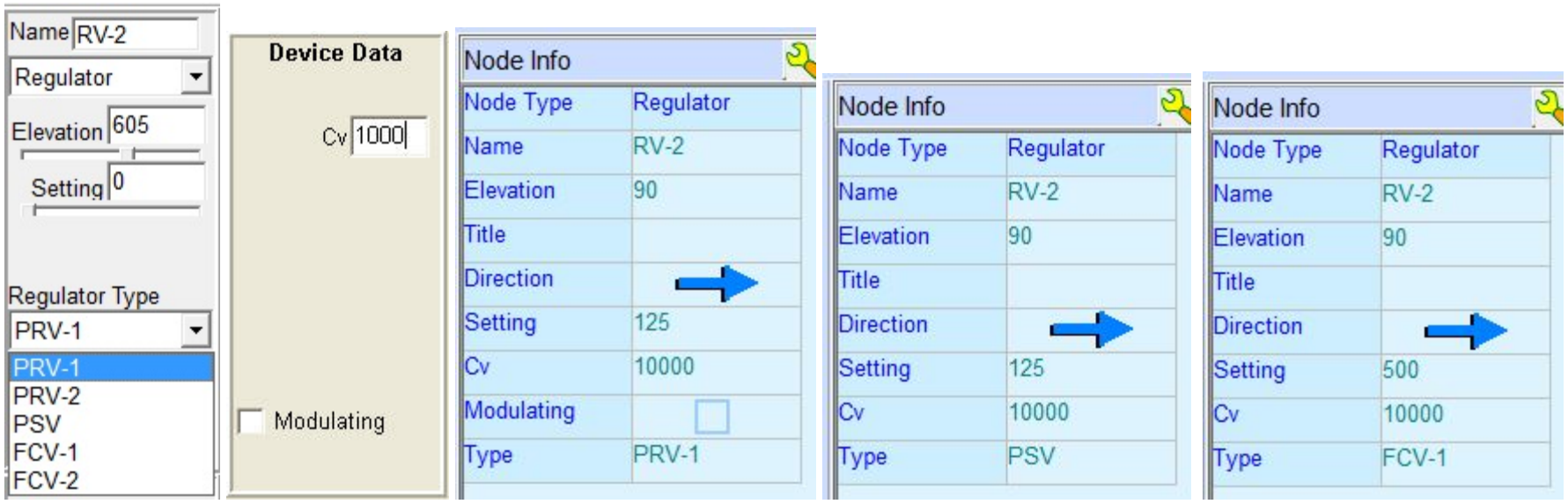
Connection Restrictions: one pipe connected to each side

This element models flow through a sprinkler orifice. The flow obeys the classic sprinkler relation where Q is the flowrate (in gpm or liters/s), $K_S$ is the sprinkler constant and P is the pressure difference (in psi or kPa) between the inside and exit of the sprinkler.

$$Q = K_S \sqrt{\Delta P}$$

A connecting pipe between the pipe in the model and the sprinkler orifice can be modeled by inputting data for the characteristics of the connections (length, diameter, elevation change from pipe centerline to orifice (negative if sprinkler is above pipe centerline) and number of elbows in the connection.

## SS & Surge – Regulators




Regulator Data

Setting

PRV psi or kPa

PSV psi or kPa

FCV specified flow

Resistance - R = head drop (in ft. or m) over the flow squared (in cfs or cms)

or Flow Coefficient - Cv = flowrate in gpm (m^3/hr) required to cause a 1 psi (1 bar) drop in pressure (units gpm/psi^0.5 or cmh/bar^0.5)

Connection Restrictions: one pipe connected to each side

This element models flows through regulating valves (pressure regulating, pressure sustaining and flow regulating). You can

select PRV-1 (pressure regulating), PSV (pressure sustaining) and FCV-1 (flow regulating) from the drop down list. The other two (PRV-2 and FCV-2) should not be selected since they model abnormal conditions.

For transient analysis, the regulating valves are modeled as a constant resistance based on the steady state conditions. If the regulator is wide open or closed for the steady state it will remain in the same condition for the transient analysis. If you wish to model a regulating valve, which changes its stem position during the transient analysis, you need to use an Active Valve or check the Modulating option (PRV-1 only).

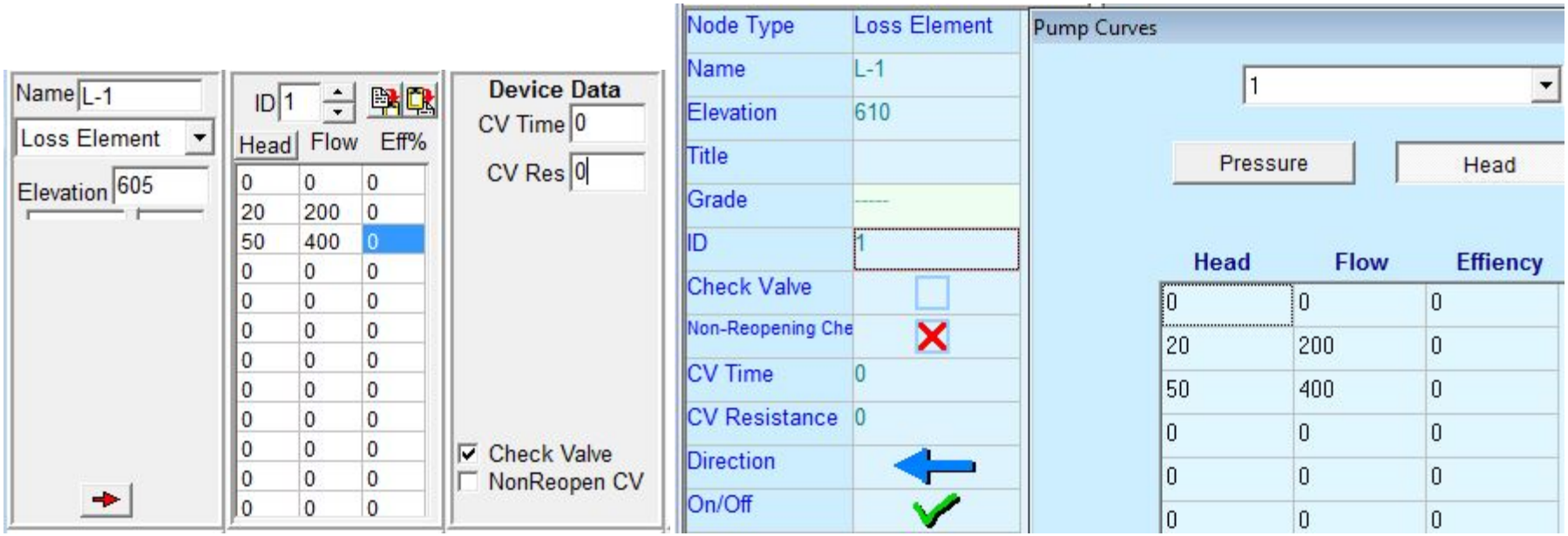
**Modulating Regulating Valve** - to make the regulating valve modulate, check the Modulating box under Device Data and enter the Response Time. 'Response time' is how long the regulator takes to respond to the transient. Specifically, it is the time it takes for the regulator to change the pressure by the amount specified in the regulator setting.

The modulating valve feature of the Pipe2018 : Surge program works as follows.

1. The steady state portion of the program computes head drop across the PRV.
2. Based on this head drop and the flowrate through the PRV, a resistance for this PRV is calculated
3. The PRV is then modeled as an active valve and a resistance calculated in step 2 is assigned to the active valve
4. If the "Modulating Valve" option is NOT checked, then this active valve resistance remains CONSTANT for the entire transient simulation.
5. If the "Modulating Valve" option is CHECKED, then the active valve resistance is VARIED during the transient run in the following manner:

If the pressure at downstream side of PRV continues to INCREASE then the valve resistance is increased otherwise valve resistance is decreased. The rate at which the valve resistance changes is equal to the initial valve resistance (step 3) divided by the response time. In other words, the program assumes that the PRV cannot quickly respond to pressure variations during a transient run but at the same time it will NOT stay at the same initial throttled condition and responds in an user specified time.

# SS & Surge – Loss Element




Loss Element Data

On/off switch

Head loss – ft. or mt.

or

Pressure loss – psi or kPa

Flow – specified units

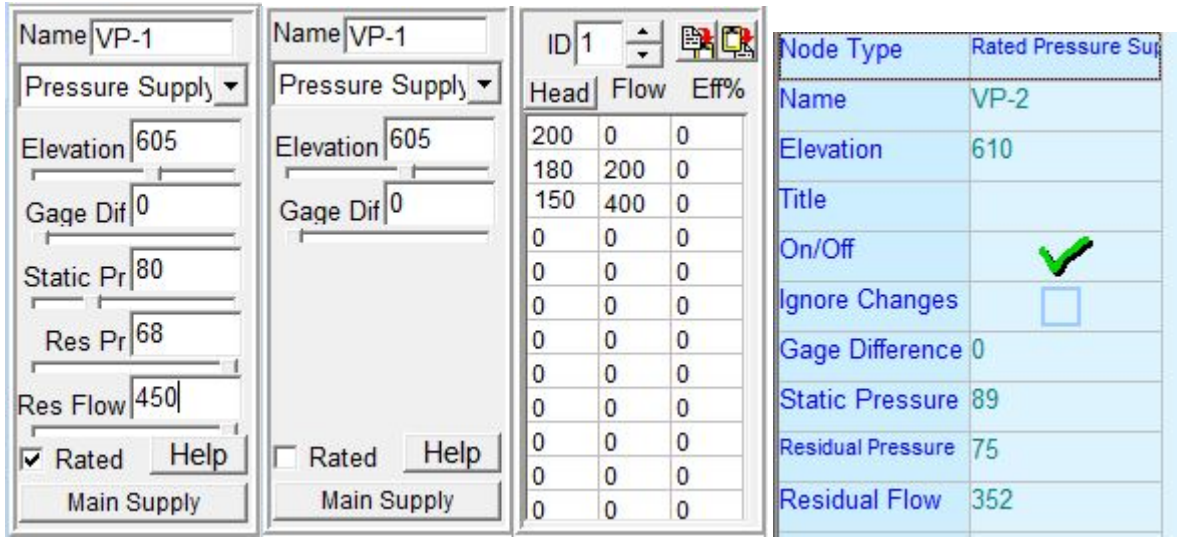
Eff(iciency) – not applicable

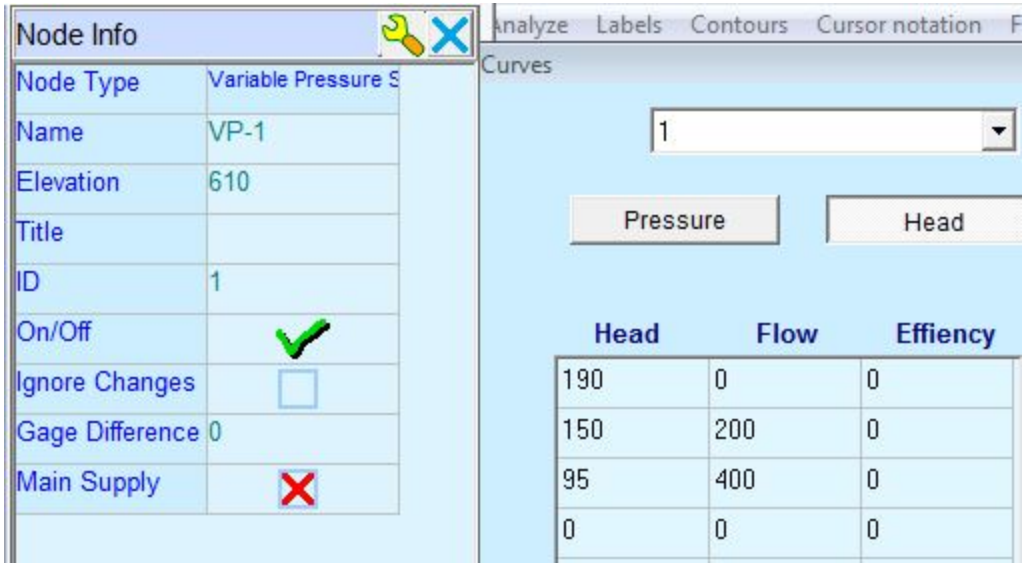
Two sets of head/flow data not allowed, must use 1 or 3+

Connection Restrictions: one pipe connected to each side

This element models a device for which head loss/flow data is available. The modeling is identical to a pump described by head/flow data except that a head loss (instead of a head gain) occurs across the element. The Loss Element can have a Check Valve to prevent flow reversal.

## SS & Surge – Pressure Supply






Pressure Supply Data

On/off switch

Gauge Dif(ference) – distance gauge is

above CL pipe (ft. or m)

Static Pressure – psi or kPa

Residual Pressure – psi or kPa

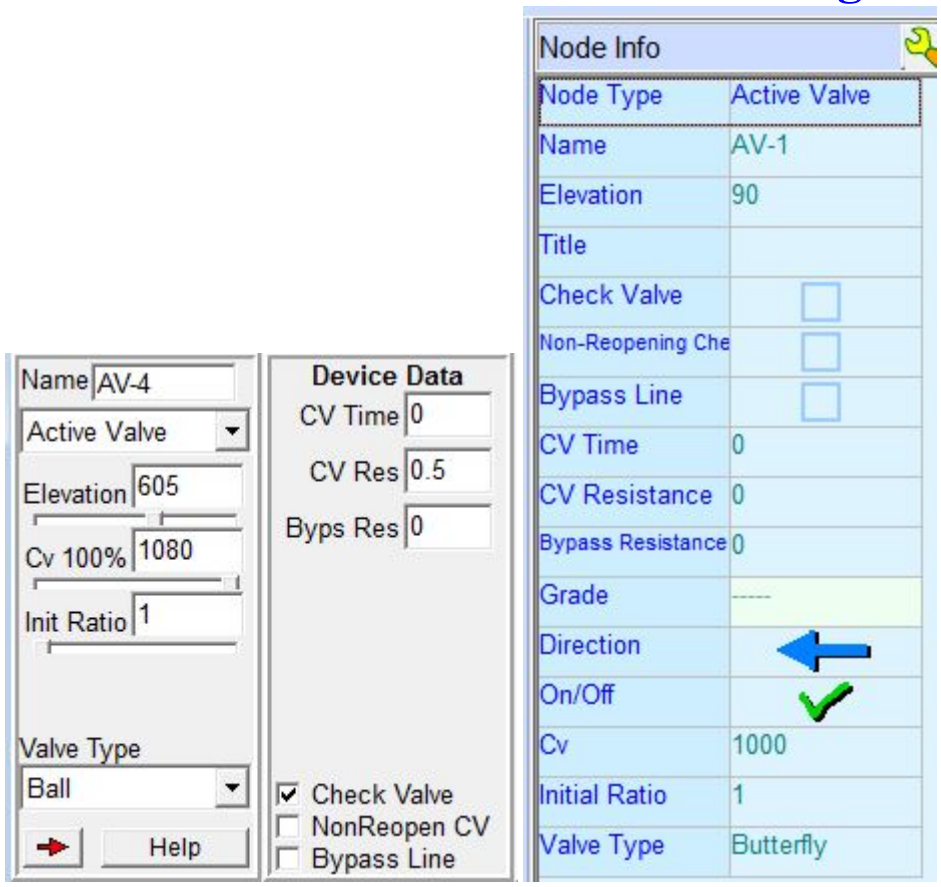
Residual Flow – specified flow units

Connection Restrictions: only one pipe connected to this device

This element models a connection to a supply where the available pressure depends on the supply flow. This applies to a

connection to an existing distribution system. The data required to describe the pressure/flow relation for this supply may be provided by a head (or pressure)/flow table or by hydrant test data (static pressure and residual pressure and flow) for a hydrant close to connection. For Surge analysis the same pressure/flow relation is maintained.

## SS & Surge – Active Valve



**Active Valve Data**

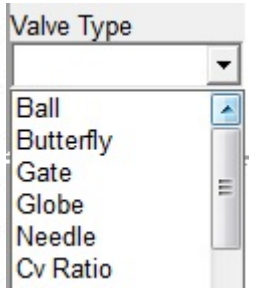
**On/off switch**

**Resistance (Wide Open) - H/Q²**

**Initial Ratio (stem) - % open/100%**

**Valve Type – select from drop down table**

| SRatio | 0 | .1 | .2 | .3 | .4 | .5 | .6 | .7 | .8 | .9 | 1 |
|---|---|---|---|---|---|---|---|---|---|---|---|
| Ball | 0 | .043 | .1136 | .2031 | .3062 | .4188 | .5371 | .6578 | .7773 | .8924 | 1 |
| Butterfly | 0 | .0123 | .0489 | .1090 | .191 | .2929 | .4122 | .546 | .691 | .8436 | 1 |
| Gate | 0 | .1271 | .2529 | .3762 | .4954 | .6090 | .7152 | .8119 | .8959 | .9626 | 1 |
| Globe | 0 | .1 | .2 | .3 | .4 | .5 | .6 | .7 | .8 | .9 | 1 |
| Needle | 0 | .19 | .36 | .51 | .64 | .75 | .84 | .91 | .96 | .99 | 1 |
| Cv Ratio | 0 | .1 | .2 | .3 | .4 | .5 | .6 | .7 | .8 | .9 | 1 |

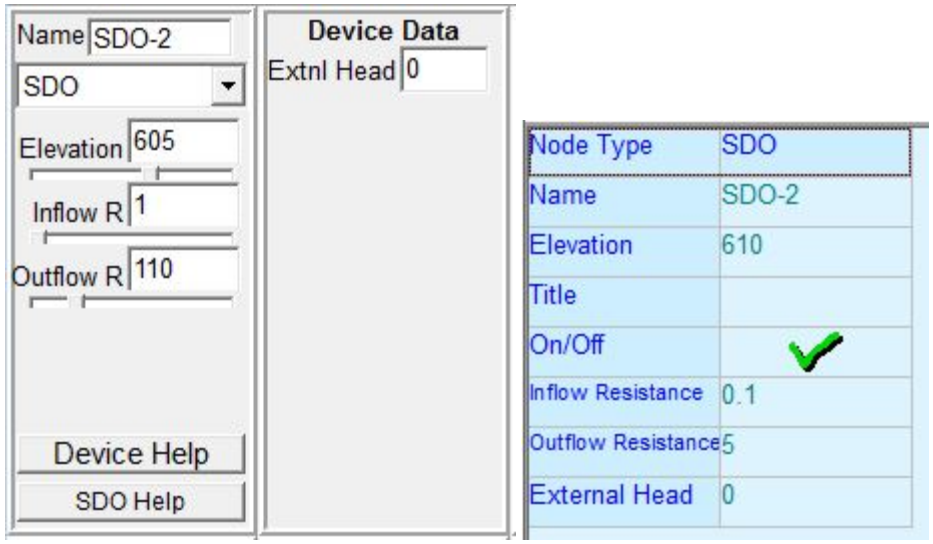



Connection Restrictions: one pipe connected to each side or one pipe connected to one side and reservoir on other side

This element models a variety of valves. A valve is described by the Flow Coefficient, Cv (or Resistance (head loss/flow$^2$) see Prefereces) and the data in the Active Valve Table shown above. This table gives the open area ratio (open area/fully open area) as a function of ratio of the stem movement to the full movement (S(tem) Ratio). This table shows, for example, a ball valve where the stem position is half way to full amount (0.5) the open area is 41.88% of the fully open area. Users can create their own valves. For Surge analysis valve closing and openings are modeled by providing the time history of the stem movement. For example, a Ball Valve can be linearly closed in 4 seconds, which means that the stem movement is linear.

Note: for Pipe2000 Version 2 users, the active valve element can be used to model a modulating pressure regulating valve. (See Tools | Modulating Regulating Valves). For other versions, a 'modulating' option is available for Regulators (see Regulators).

# SS & Surge – Side Discharge Orifice (SDO)




SDO Data

On/off switch

Inflow R(esistance) - $H/Q^2$

Outflow R(esistance) - $H/Q^2$

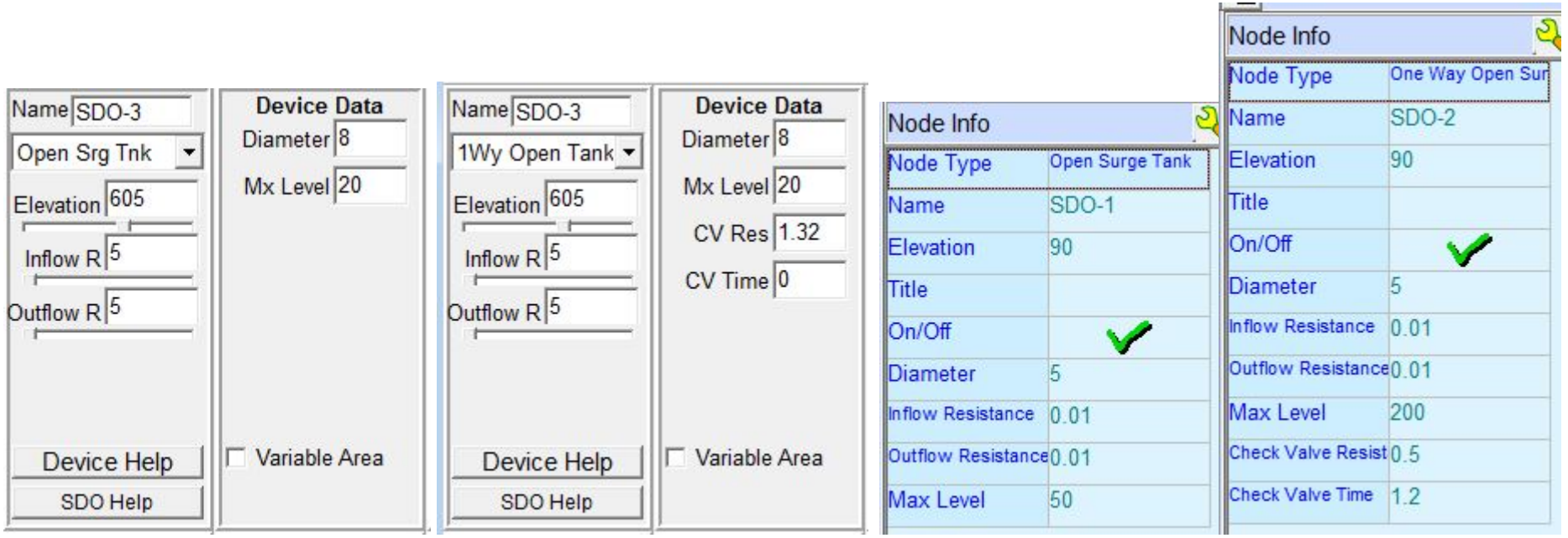
External Head – ft. or m

Connection Restrictions: one pipe connected to each side

This device represents an orifice in the pipeline where flow enters or exits the pipeline based on the orifice relation. The inflow and outflow resistances can be different (inflow refers to flow into the pipeline and outflow is out of the pipeline).

This device also is an integral part of a number of surge control devices including surge tanks, pressure relief valves and surge anticipation valves.

## Surge – Open Surge Tank (Spilling Tank and One Way (feed) Tank)




One Way Tank Open/Spilling Tank

Open Surge Tank Data

On/off switch

Diameter – ft. or m

Maximum Level – ft. or m

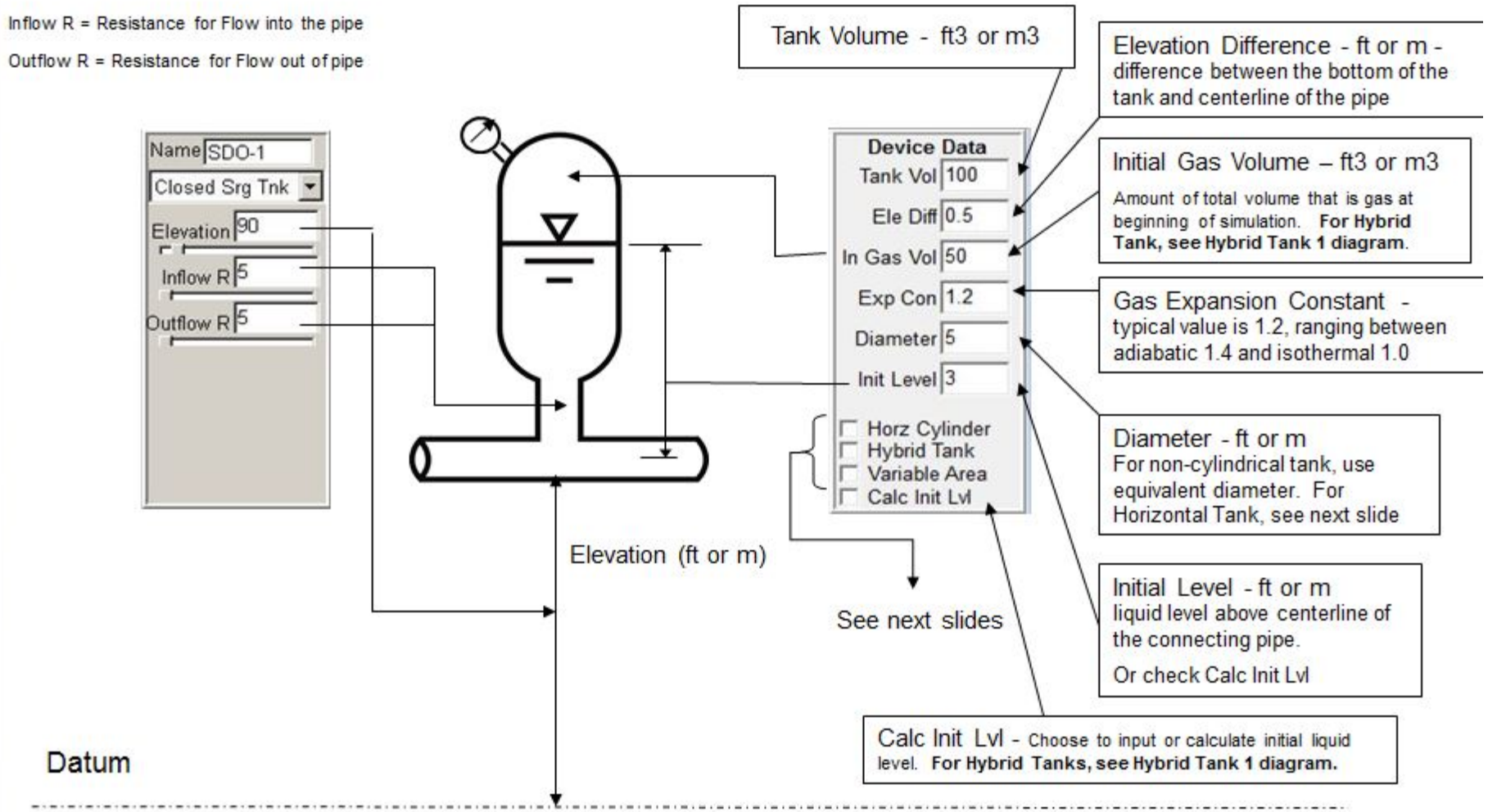
(spilling tank)

Connection Restrictions: one pipe connected to each side

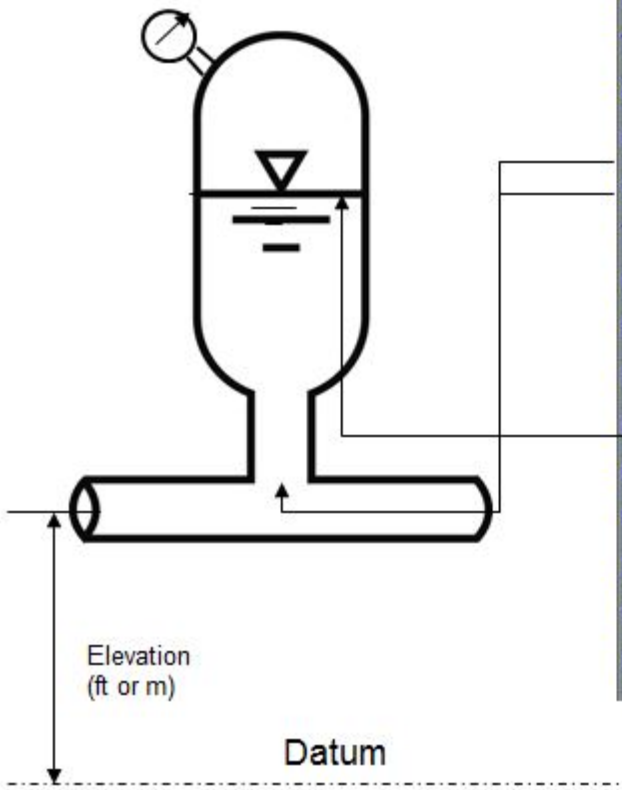
This device models an open surge tank including a spilling surge tank where the maximum level is specified and a one way open surge tank which allows flow only from the tank into the pipeline and requires a check valve to be defined.

# Surge – Closed Surge Tank

# Closed Surge Tank



# Closed Surge Tank




| Node Type | Vert Closed Surge |
|---|---|
| Name | SDO-2 |
| Elevation | 610 |
| Title | |
| On/Off | ✓ |
| Inflow Resistance | 0.1 |
| Outflow Resistance | 0.1 |
| Tank Volume | 4 |
| Initial Gas Volume | 62.8 |
| Expansion Constant | 1.2 |
| Diameter | 4 |
| Initial Level | 2 |
| Hybrid | ☐ |
| Elevation Difference | 0 |
| Device Type | Closed |
| Shape | Vertical |
| Calc Init Level | ☐ |

**Tank Volume (Vb) – ft3 or m3**

**Initial Gas Volume – ft3 or m3**

Amount of total volume that is gas at beginning of simulation.
**For Hybrid Tank, see Hybrid Tank 1 diagram.**

**Gas Expansion Constant**

typical value is 1.2, ranging between adiabatic 1.4 and isothermal 1.0

**Diameter - ft or m**

For non-cylindrical tank, use equivalent diameter.

See next slides for Horizontal and Variable Area Tanks.

**Initial Level - ft or m**

Level above centerline of connecting pipe. Also may choose to have Initial Level automatically calculated. See next slides

**Hybrid Tank** – See Hybrid Tank 1 and 2 Diagrams

**Elevation Difference - ft or m** - difference between the bottom of the tank and centerline of the pipe

**Choose tank shape**

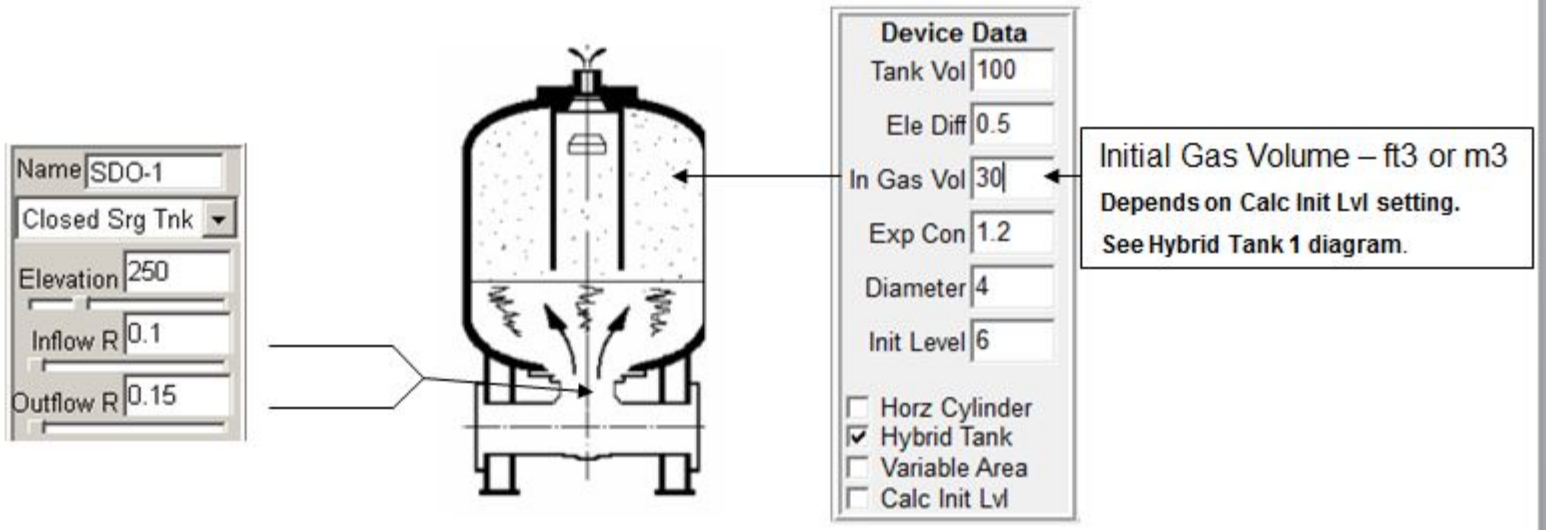
**Choose to input or calculate initial liquid level.**
**For Hybrid Tanks, see Hybrid Tank 1 diagram.**

Inflow R = Resistance for Flow into the pipe

Outflow R = Resistance for Flow out of pipe

# Closed Surge Tank: Hybrid

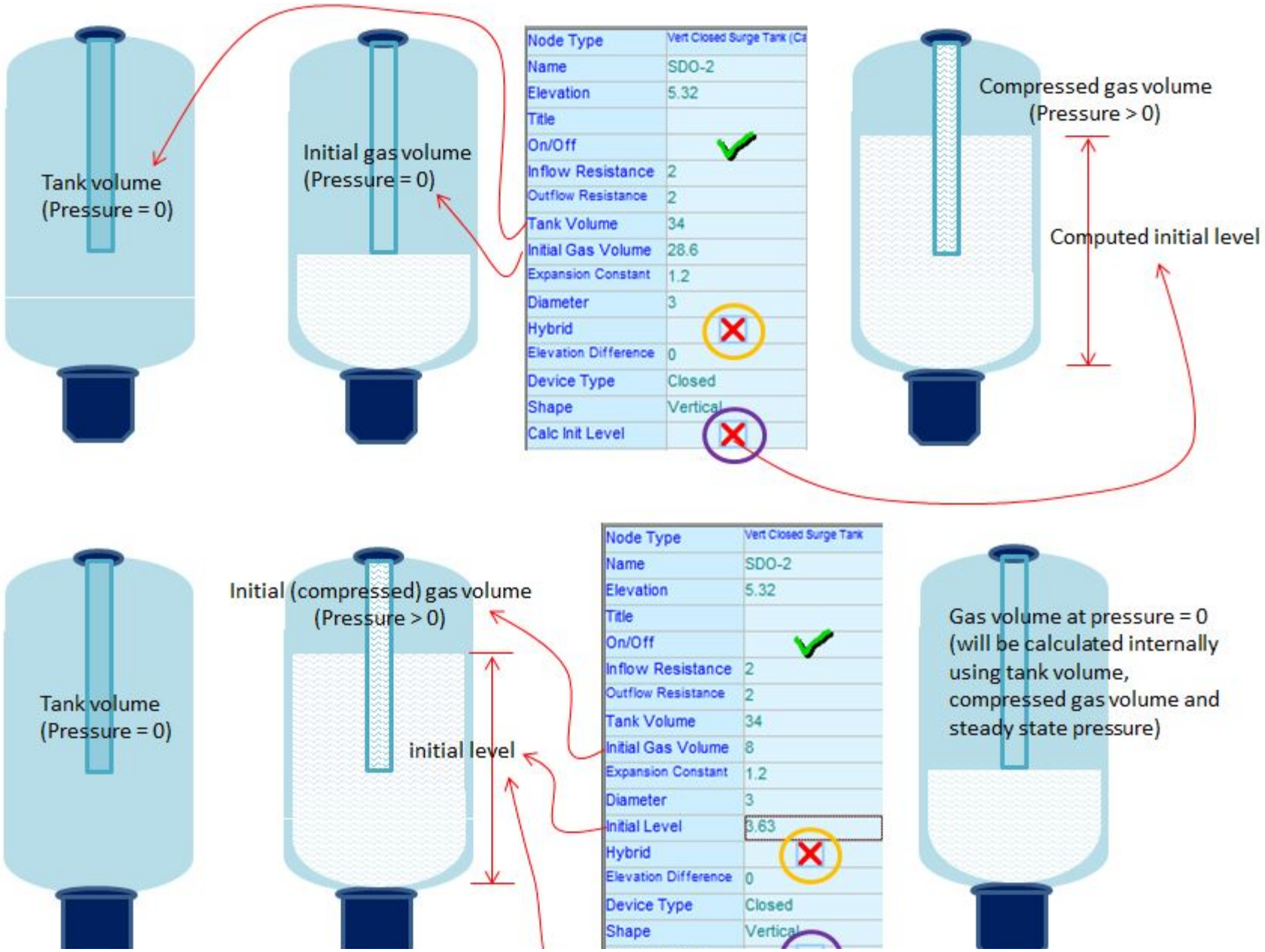
(closed surge tank such as a dipping tube)



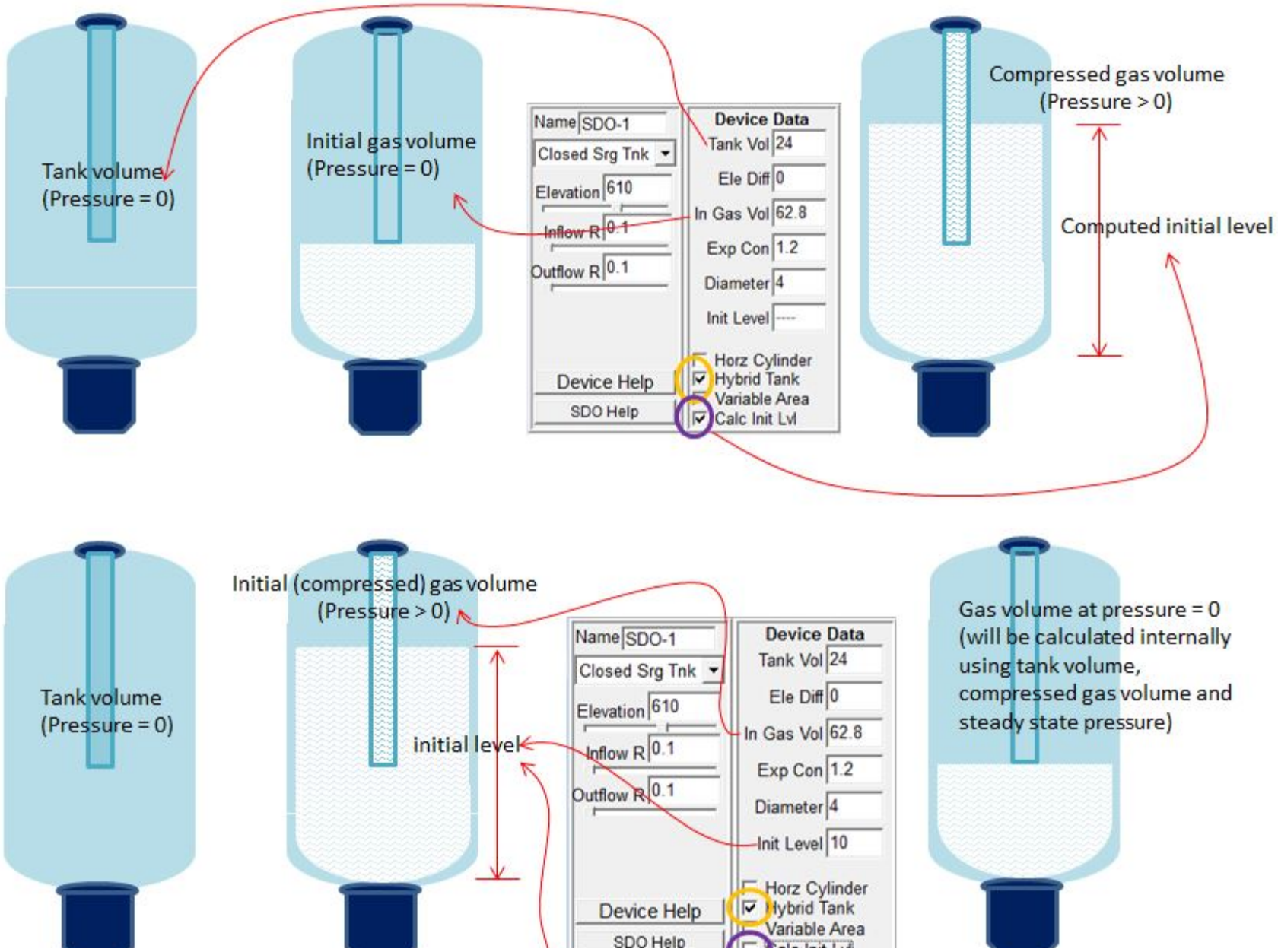

Inflow is flow INTO Pipe
Outflow is flow OUT of Pipe

Resistance = Headloss/Flow^2

**Hybrid Tank Diagram**


Tank volume
(Pressure = 0)
Initial gas volume
(Pressure = 0)
Node Type
Vert Closed Surge Tank (Cz
Name
SDO-2
Elevation
5.32
Title
On/Off
Inflow Resistance
2
Outflow Resistance
2
Tank Volume
34
Initial Gas Volume
28.6
Expansion Constant
1.2
Diameter
3
Hybrid
Elevation Difference
0
Device Type
Closed
Shape
Vertical
Calc Init Level
Compressed gas volume
(Pressure > 0)
Computed initial level
Tank volume
(Pressure = 0)
Initial (compressed) gas volume
(Pressure > 0)
initial level
Node Type
Vert Closed Surge Tank
Name
SDO-2
Elevation
5.32
Title
On/Off
Inflow Resistance
2
Outflow Resistance
2
Tank Volume
34
Initial Gas Volume
8
Expansion Constant
1.2
Diameter
3
Initial Level
3.63
Hybrid
Elevation Difference
0
Device Type
Closed
Shape
Vertical
Gas volume at pressure = 0
(will be calculated internally
using tank volume,
compressed gas volume and
steady state pressure)

Calc Init Level


Tank volume
(Pressure = 0)
Initial gas volume
(Pressure = 0)
Name SDO-1
Closed Srg Tnk
Elevation 610
Inflow R 0.1
Outflow R 0.1
Device Help
SDO Help
Device Data
Tank Vol 24
Ele Diff 0
In Gas Vol 62.8
Exp Con 1.2
Diameter 4
Init Level ----
Horz Cylinder
Hybrid Tank
Variable Area
Calc Init Lvl
Compressed gas volume
(Pressure > 0)
Computed initial level
Tank volume
(Pressure = 0)
Initial (compressed) gas volume
(Pressure > 0)
initial level
Name SDO-1
Closed Srg Tnk
Elevation 610
Inflow R 0.1
Outflow R 0.1
Device Help
SDO Help
Device Data
Tank Vol 24
Ele Diff 0
In Gas Vol 62.8
Exp Con 1.2
Diameter 4
Init Level 10
Horz Cylinder
Hybrid Tank
Variable Area
Gas volume at pressure = 0
(will be calculated internally
using tank volume,
compressed gas volume and
steady state pressure)

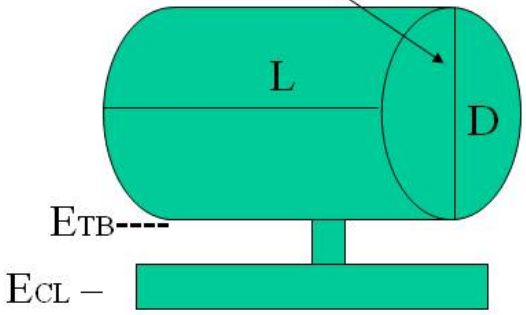
Calc Init Lvl

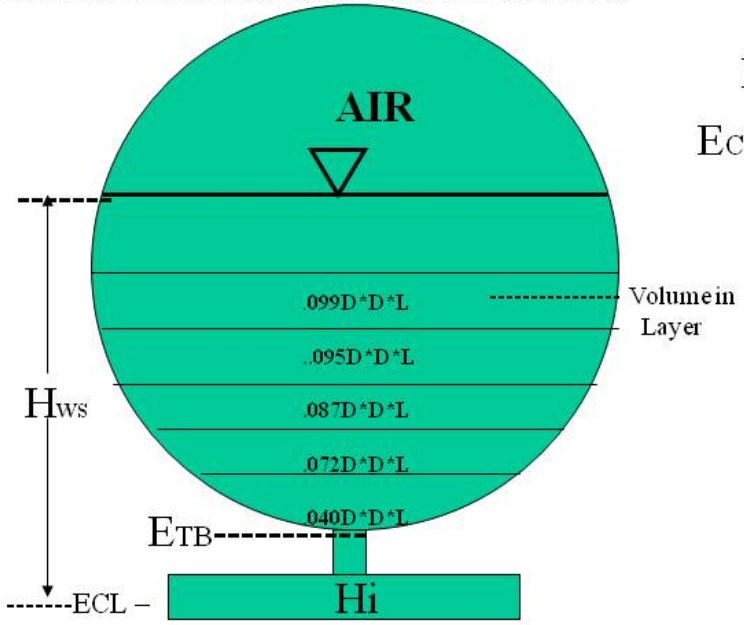
# Horizontal Tank

Input Data:
Vt – volume of tank (ft. or m)
D – Diameter of Tank (ft. or m)
ECL – Elevation of pipe CL
ETB – Elevation Tank Bottom
Vi – Initial Volume of Air (Compressor) or Precharge pressure (Bladder)
***Calculate - L = 4 Vt/πD*D***
***From initial steady state – Calculate initial line head (Hi)***

***Finally calculate the height water surface – Hws (isothermal expansion)***

Diameter is cylindrical diameter
Not equivalent diameter




| Depth | surface area | Volume | Ratio |
|---|---|---|---|
| 0 | 0 | 0*D*D*L | 0.0 |
| .1D | .60DL | .040 | 0.0508 |
| .2D | .80DL | .112 | 0.1423 |
| .3D | .90DL | .199 | 0.2529 |
| .4D | .975DL | .295 | 0.3748 |
| .5D | 1.00DL | .394 | 0.5000 |
| .6D | .975DL | .493 | 0.6264 |
| .7D | .90DL | .588 | 0.7471 |
| .8D | .80DL | .675 | 0.8577 |
| .9D | .60DL | .747 | 0.9492 |
| 1.0D | 0 | .787 | 1.0000 |

Closed Surge Tank Data

On/off switch

Tank Volume - $ft^3$ or $m^3$

Initial Gas volume – $ft^3$ or $m^3$

Expansion constant

Diameter - (required) ft or m. This value is used in the calculation.

Init(ial) Level - ft or m above the centerline of the connecting pipe.

Connection Restrictions: one pipe connected to each side

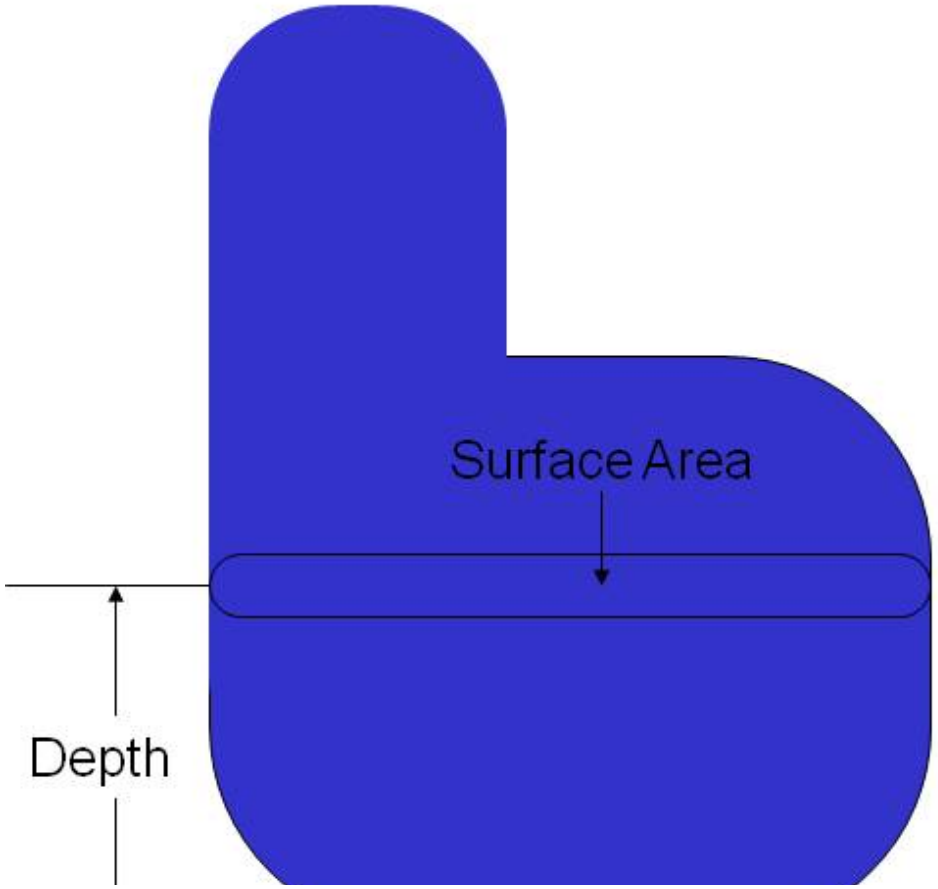
The closed surge tank will contain the Initial Gas Volume and will be initially pressurized to the starting pressure as determined by the initial steady state analysis.

The expansion constant varies between 1 (isothermal) and 1.4 (adiabatic) and a value between these limits should be used.

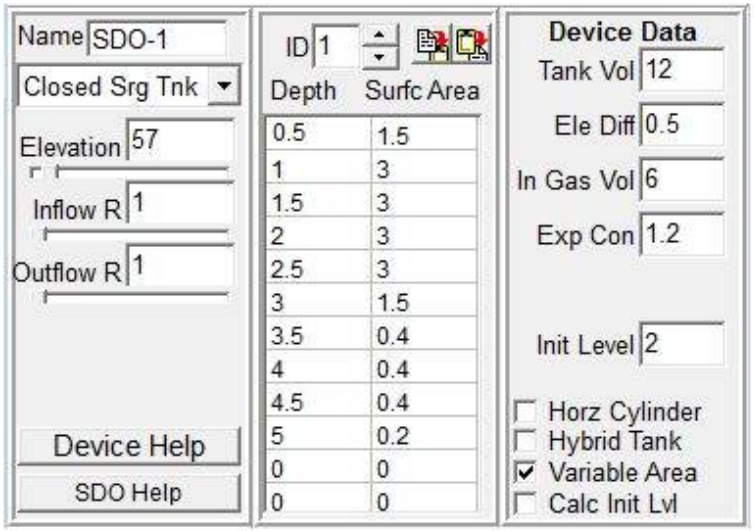
Hybrid Tanks

It is assumed that the vent is large enough that the pressure in the air is basically atmospheric when the vent is open (no significant pressure drop across the vent). This is how hybrid surge tanks are designed so this is considered to be an acceptable assumption. Therefore Vent Diameter is no longer required input for Hybrid Surge Tanks.

# Variable Area
# Closed Surge Tank

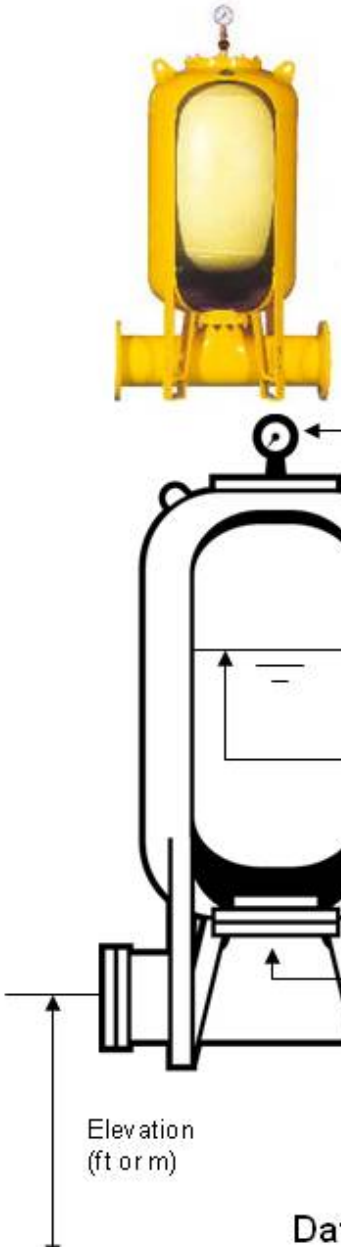



Input Surface Area vs. Depth data over the range of the whole tank height



## Surge – Bladder Surge Tank

# Bladder Surge Tank




| | |
|---|---|
| Node Type | Vert Bladder Tank ( |
| Name | SDO-1 |
| Elevation | 90 |
| Title | |
| On/Off | ✓ |
| Diameter | 4 |
| Inflow Resistance | 5 |
| Outflow Resistance | 5 |
| Expansion Constant | 1.2 |
| Initial Level | 4 |
| Tank Volume | 72 |
| Preset Head | 73 |
| Elevation Difference | 0.5 |

**Diameter - ft or m**
For non-cylindrical tank, use equivalent diameter.
See next slides for Horizontal and Variable Area Tanks.

**Gas Expansion Constant**
typical value is 1.2, ranging between adiabatic 1.4 and isothermal 1.0

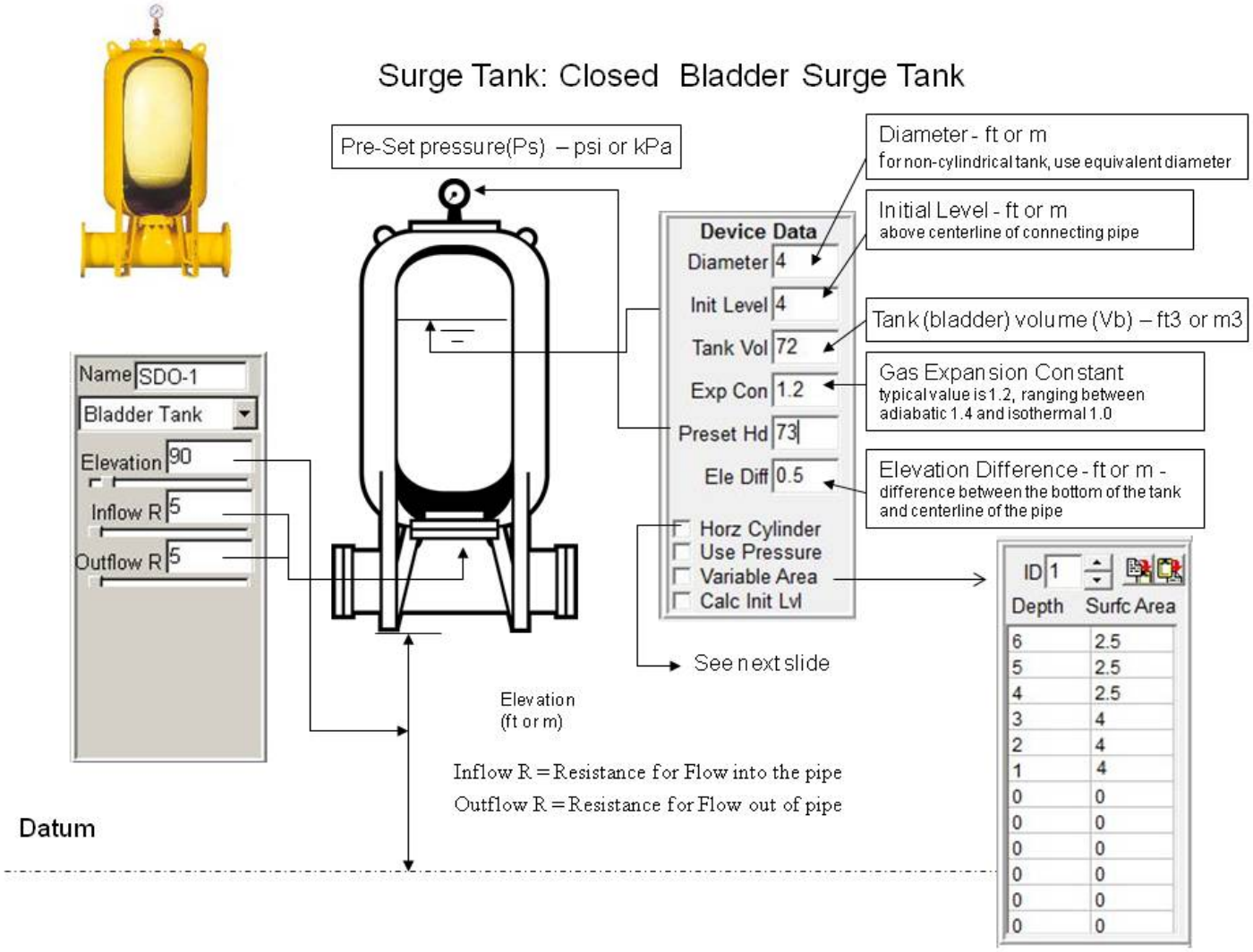
**Initial Level - ft or m**
Level above centerline of connecting pipe. Also may choose to have Initial Level automatically calculated. See next slides

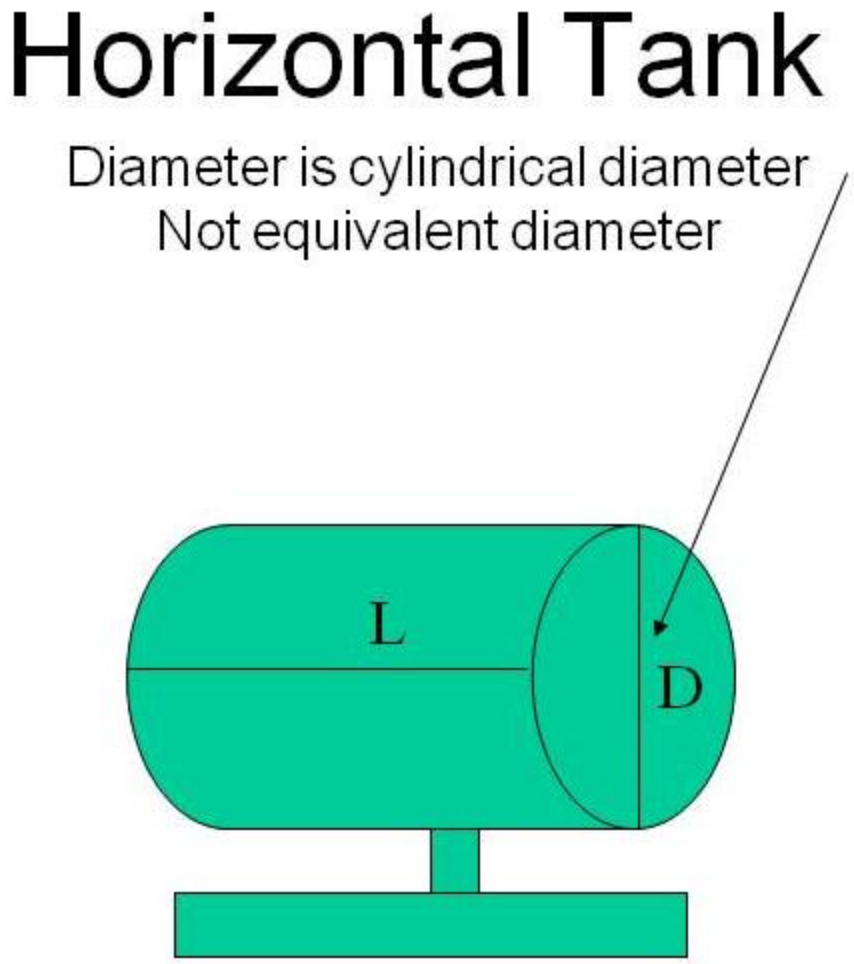
**Tank (bladder) volume (Vb) – ft3 or m3**

**Pre-Set Head/Pressure – ft or m / psi or kPa**
Select setting units along with tank type. See next slides.

**Elevation Difference - ft or m** - difference between the bottom of the tank and centerline of the pipe

Inflow R = Resistance for Flow into the pipe
Outflow R = Resistance for Flow out of pipe


Surge Tank: Closed Bladder Surge Tank
Pre-Set pressure(Ps) – psi or kPa
Diameter - ft or m
for non-cylindrical tank, use equivalent diameter
Initial Level - ft or m
above centerline of connecting pipe
Tank (bladder) volume (Vb) – ft3 or m3
Gas Expansion Constant
typical value is 1.2, ranging between
adiabatic 1.4 and isothermal 1.0
Elevation Difference - ft or m -
difference between the bottom of the tank
and centerline of the pipe
Device Data
Diameter 4
Init Level 4
Tank Vol 72
Exp Con 1.2
Preset Hd 73
Ele Diff 0.5
Horz Cylinder
Use Pressure
Variable Area
Calc Init Lvl
See next slide
Name SDO-1
Bladder Tank
Elevation 90
Inflow R 5
Outflow R 5
Elevation
(ft or m)
Inflow R = Resistance for Flow into the pipe
Outflow R = Resistance for Flow out of pipe
Datum
ID 1
Depth Surfc Area
6 2.5
5 2.5
4 2.5
3 4
2 4
1 4
0 0
0 0
0 0
0 0
0 0
0 0




Bladder Surge Tank Data

On/off switch

Diameter - ft or m

Init(ial) Level - ft or m above the centerline of the connecting pipe.

Tank (bladder) volume (Vb) – $ft^3$ or $m^3$

Expansion constant

Preset Prs (pressure) default is ft or m head,

if "Use Pressure" box is checked - psi or kPa.

Connection Restrictions: one pipe connected to each side

The pre-set pressure (Ps) is the pressure set in the gas in the bladder prior to opening the valve connecting the tank and the pipeline.. At any pressure lower than the Pre-set pressure the bladder fills the tank and there is no flow into the pipeline. This is an undesirable condition. The initial gas volume (Vi) is determined by the program using the initial line pressure (Pi) at the location of the bladder tank. This volume is calculated assuming isothermal compression (PiVi = PsVb) For example, for the data displayed above if the initial line pressure at the Bladder Tank is 80 psi, the initial volume is Vi = (14.7+Pa)*135/(50+Pa) = 41.9 cubic feet. Pa is atmospheric pressure (14.7 psi)

The expansion constant varies between 1 (isothermal) and 1.4 (adiabatic) and a value between these limits should be used.

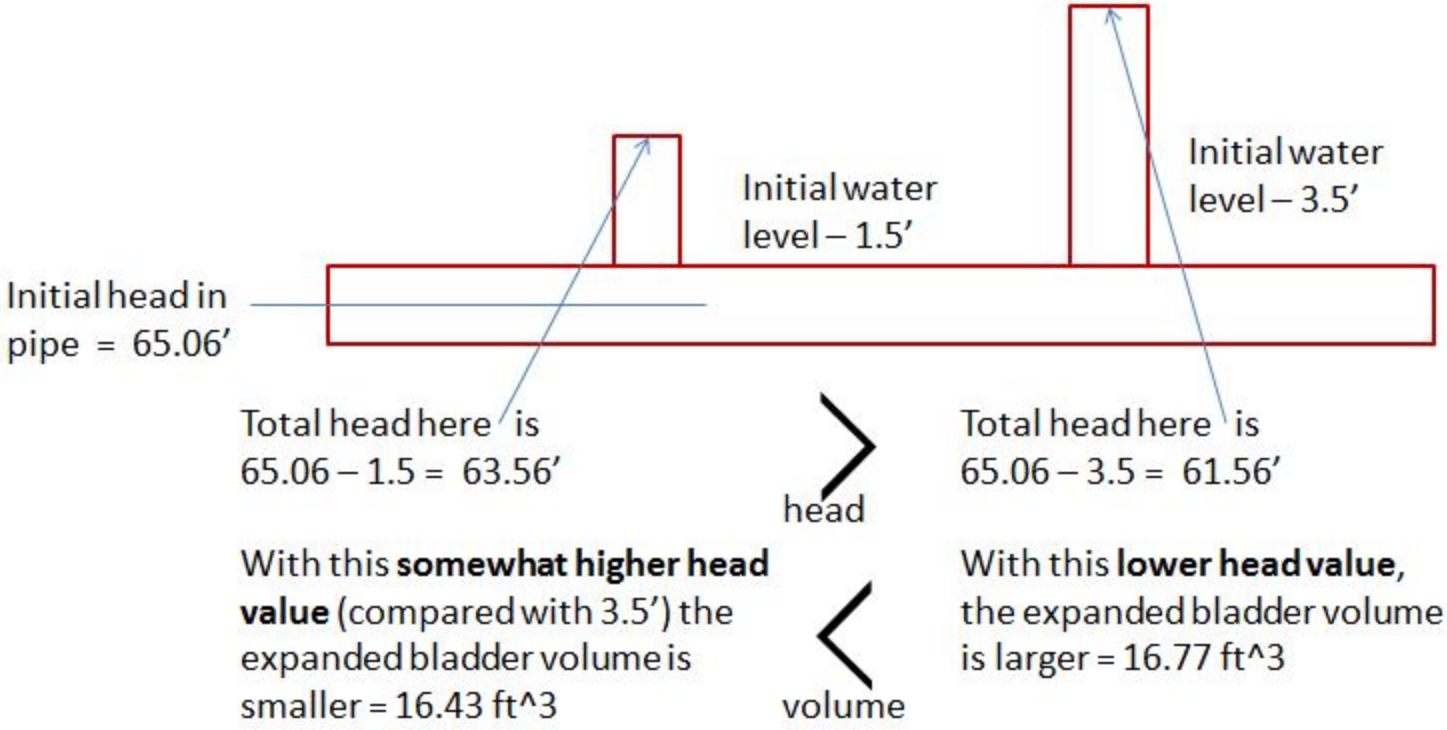
Notes: Initial air volume is typically 50% of the total volume. Ratios of 40% to 60% water, possibly 30/70 are ok but if initial gas volume is too low the precharge pressure might have to be an infeasible value (negative). See Bladder Precharge Tool.

If a bladder tank is positioned higher than the supply grade, it will "bottom out" and not be able to provide the intended surge protection.

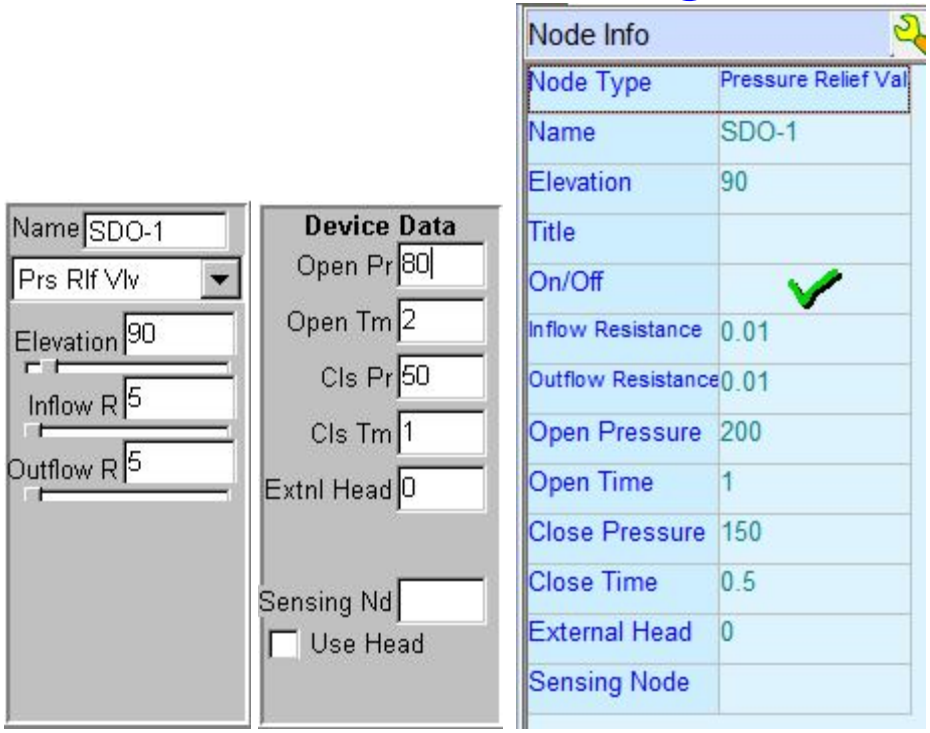
For "Bottomed out" tanks in low pressure applications, it is possible to make the Preset Head/Pressure equal to the initial line head/pressure IF the Initial Level is set to 0 and the "Calc Init Lvl" option is NOT used.

**Initial Level vs. Volume Example.**

The confusion is that with bladder tanks we do NOT specify the initial volume. This is entirely dependent on the pressure that the bladder comes into contact with. The Initial Level informs the model about what the exact pressure is at the bladder/liquid interface. Then the bladder volume is calculated using that pressure.



# Surge – Pressure Relief Valve




Pressure Relief Valve Data

On/off switch

Opening pressure – psi or kPa

or Head - ft. or m

opening time - sec

closing pressure – psi or kPa

or Head – ft. or m

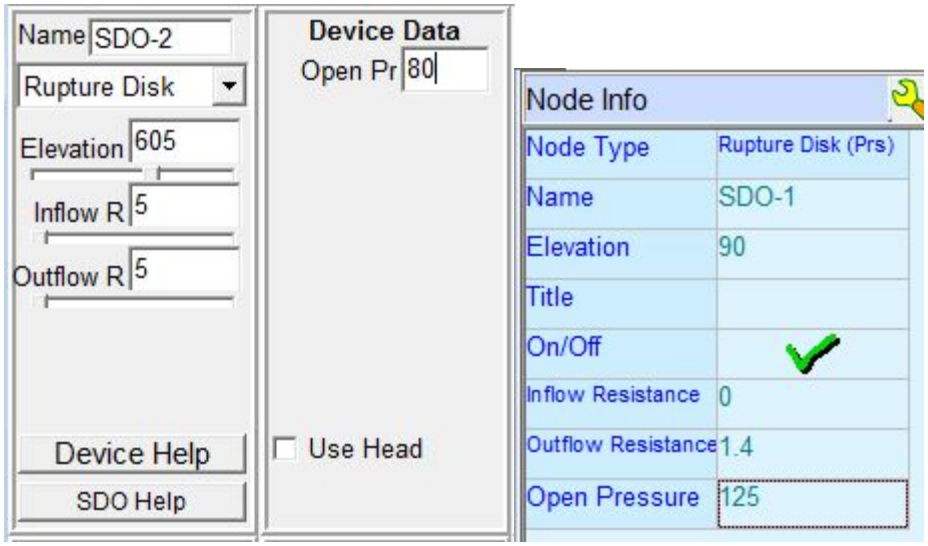
Closing Time - sec

External Head – exit head for relief valve – ft. or m

Connection Restrictions: one pipe connected to each side

A Pressure Relief Valves begins to open when the pressure at the sensing node exceeds the opening pressure. The opening time is the response time for the valve to go from the start to the fully open position. The valve closure is initiated when the pressure (head) drops below the closing pressure. Because of pressure fluctuation at the sensing node, this valve can remain open for longer periods. If the valve exits to a pressurized region (tank etc.) the External head should be input. The pressure to activate the valve is generally sensed at the valve but any node can be used for this purpose

## Surge – Rupture Disk




Rupture Disk Data

On/off switch

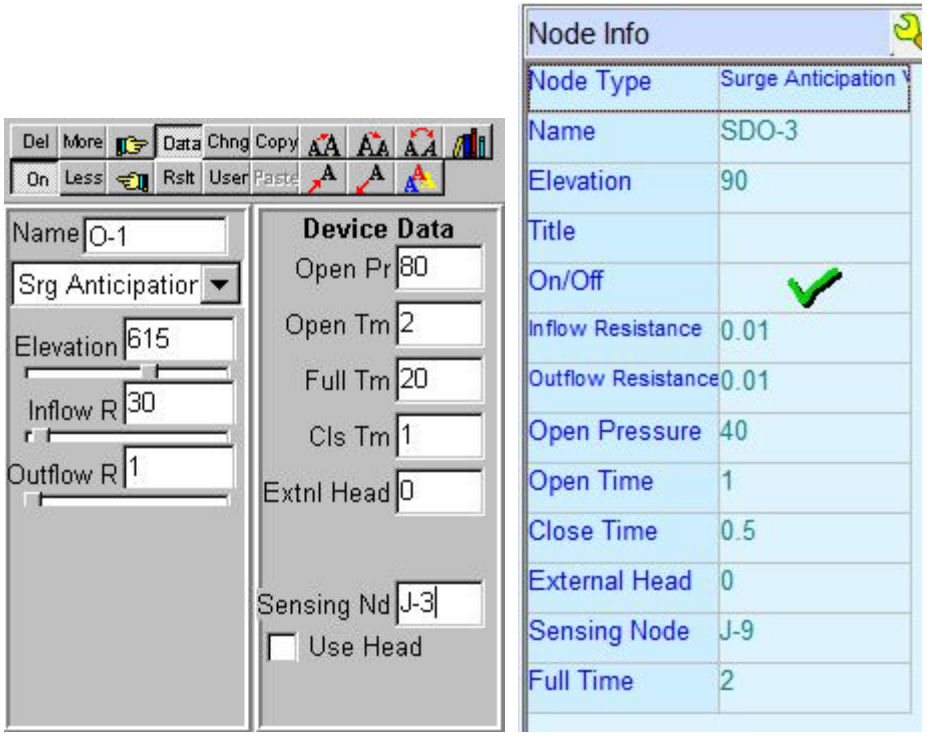
Opening pressure – psi or kPa

or Head – ft. or m

Connection Restrictions: one pipe connected to each side

This device opens very quickly when the pressure exceeds the opening pressure. Good for modeling a pipe break. Set the Opening Head at a value lower than the initial static head. This will result in a steady state with the Rupture Disk closed and it will open immediately. The Resistance of the RD will be based on the size of the opening caused by the pipe break

## Surge – Surge Anticipation Valve




Surge Anticipation Valve Data

On/off switch

Opening pressure – psi or kPa

or (head) – ft. or m

opening time – sec.

Fully open time – sec.

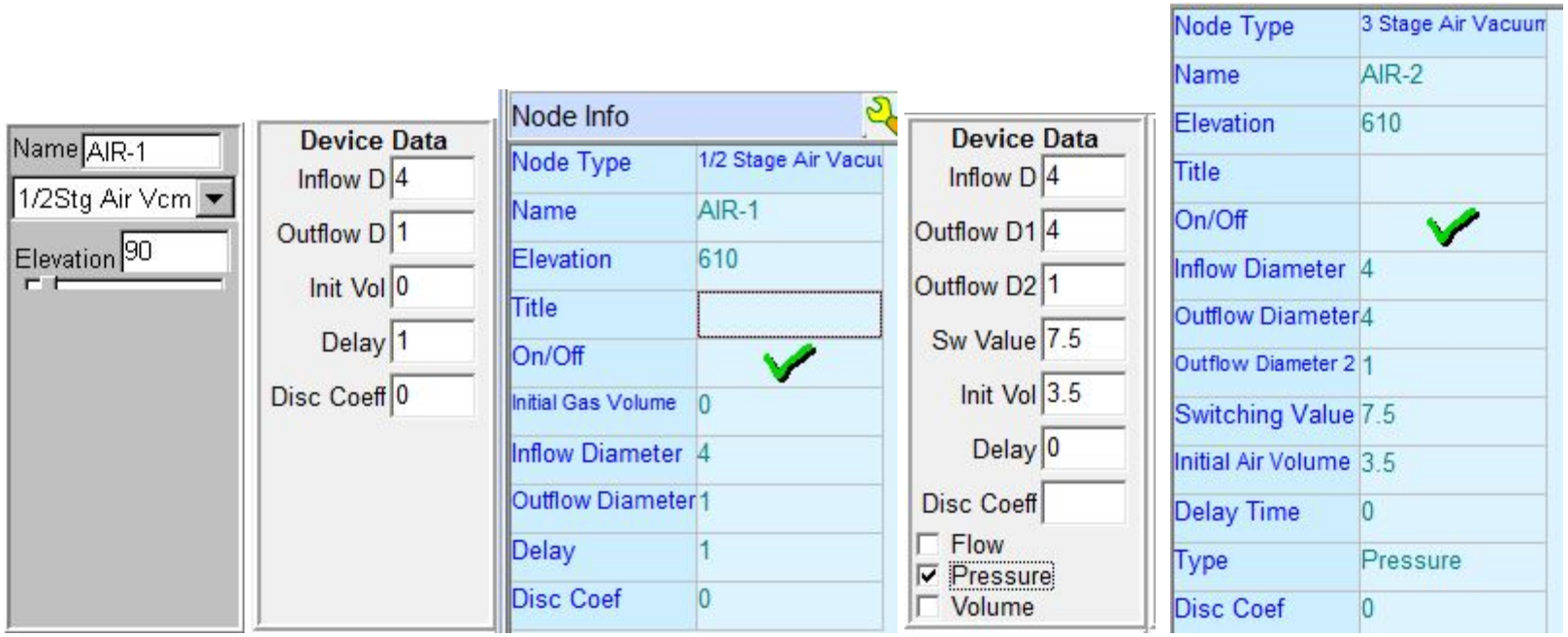
Closing time – sec.

External Head – ft. or m

Connection Restrictions: one pipe connected to each side

This device activates on a down surge when the pressure at the sensing node drops below the opening pressure. After opening is initiated the valve completes a cycle where it opens fully in the opening time, remains fully open for the fully open time and closes completely in the closing time. If the valve exits to a pressurized region (tank etc.) the External head should be input. The pressure to activate the valve is generally sensed at the valve but any node can be used for this purpose. For both low and high pressure pilots, either use the SAV in combination with a pressure relief valve, or use the HydSAV element.

## Surge – Air Vacuum Valve –1, 2 and 3 Stage




**1 and 2 Stage** **3 Stage**

Air/Vacuum Valve Data

On/off switch

Inflow diameter – in or mm

Outflow diameter (D1) – in or mm

Outflow diameter (D2) – in or mm

Switching value -

Initial Air Volume – $ft^3$ or $m^3$

Delay - seconds

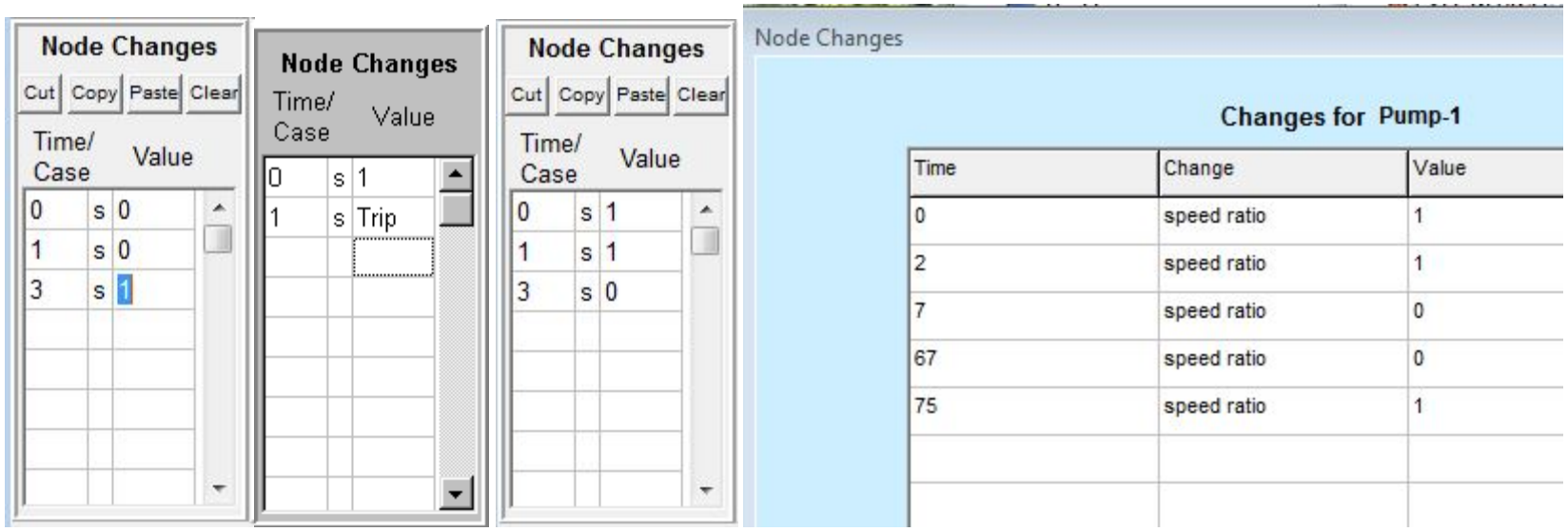
Switch type – flow, pressure or volume

Connection Restrictions: one pipe connected to each side

A one-stage (or nominal) air/vacuum valve has a single orifice for air inflow and outflow. A two-stage valve has a second (smaller) orifice for air outflow (to reduce “air slam”). A three-stage valve has a third orifice that will be utilized for outflow if the flow pressure or air volume exceeds the corresponding switching value. For this valve the same orifice can be used for inflow and outflow prior to the switch (D1=D2). For all three types an initial air volume can be designated at the valve. This feature allows the modeling of start-up with air in the pipeline. The solution for expulsion of air from the pipeline considers the compressibility of the air, the pressure variations in the pipeline, and the characteristics of the orifice. If no switch value is entered for a 3-Stage air valve, D2 will be used for outflow.

Inlet and outlet diameter may be entered as a negative (-) value to represent orifice area instead of diameter.

## Creating a Transient - Pumps




1) All Pumps (startup) 2) Pump File (trip) 3) All Pumps (speed changes)

Operating speed changes for pumps procedure transients. A time dependent change in the speed ratio(s) (operating speed/rated speed) may be defined for all types of pumps as shown above (left). Pumps described by a pump file also can be tripped (lose power) and the resulting rundown calculated based on the pump and motor inertia and dynamic characteristics described in the pump file. A tripped pump may be restarted, but it is necessary for the speed to be brought to zero before restarting.

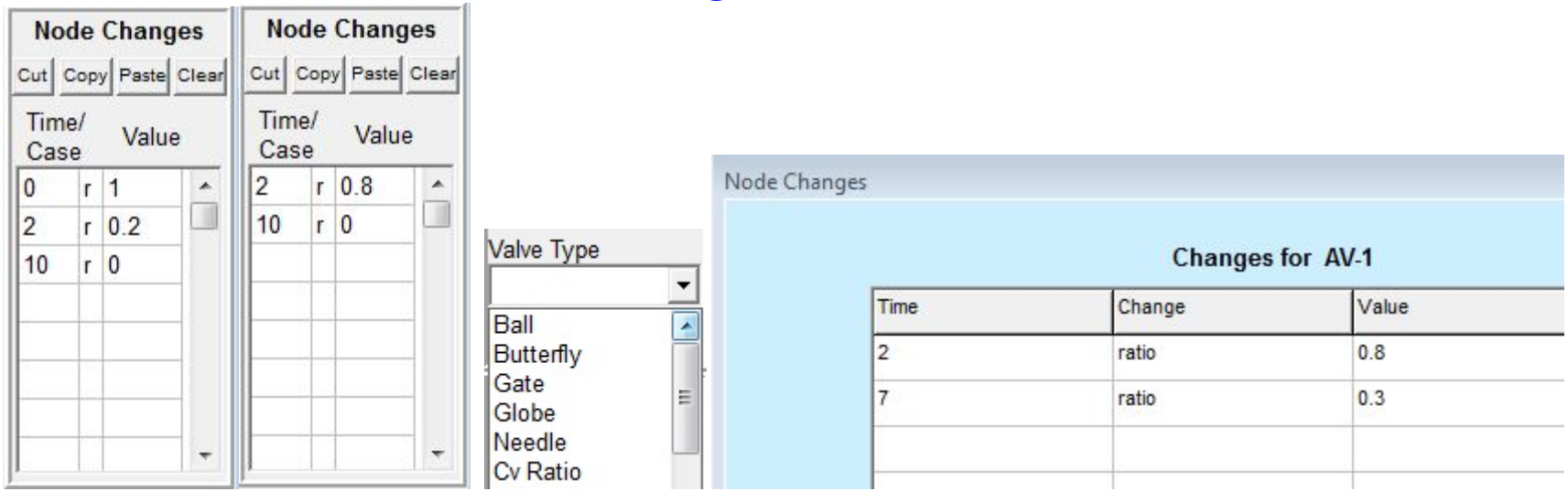
A trip can be specified with a ratchet, which will present reverse rotation. A pump with a ratchet will not allow the pump impeller to rotate in the opposite direction. Following a pump trip the flow can reverse but the impellers cannot undergo reverse rotation.

Several scenarios are shown above:

1) The pump is off (speed = 0) for 1 second and then the speed ramps up to full speed (1) over the next 2 seconds. Note: Pumps used for Startup must have internal check valves. Do not use non-reopen check valves.
2) The pump trips (loses power) at 1 second.
3) The pump speed ramps down from 1 to zero in 2 seconds.

4) The pump ramps down from 2 seconds to 7 seconds, then restarts beginning at 67 seconds ramping up to 100% at 75 seconds.

# Creating a Transient - Valves




1) User Valve 2) Standard Valves

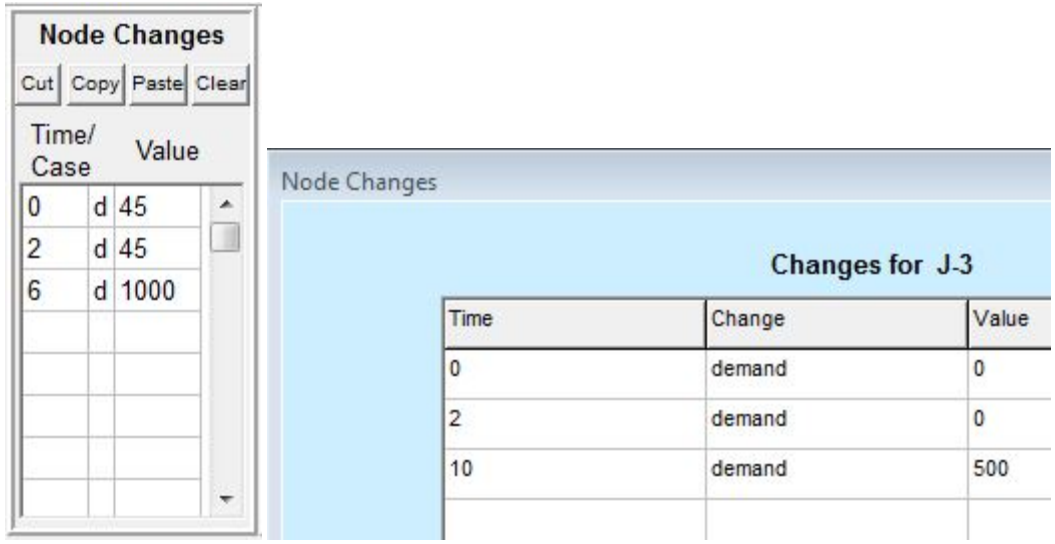
A change in the ratio of the stem position (r) for a valve will produce a transient. A ratio of 1 means the valve is fully open, 0.5 means the stem has turned 50% of fully closed and 0 means the valve is fully closed. For User Valves, any number of pairs of time/ratios can be specified to define a closure, opening a combination. For other types of valves, only two pairs of value are utilized. The first pair is the time and ratio at the beginning of the valve action and the second pair defines the values at the end. It is assumed that the valve stem moves linearly during the action period.

Two scenarios are shown above:

1) The User is fully open at time – 0 and 20% open at time = 2. The valve is fully closed at time = 10 seconds.
2) The Gate valve is 80% open at 2 seconds and fully closed at 10 seconds.

3) The Gate valve is 80% open at 2 seconds and ramps down to 30% open over the next 5 seconds.

## Creating a Transient – Junction Demand Changes

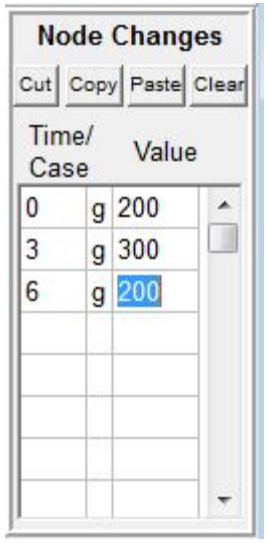


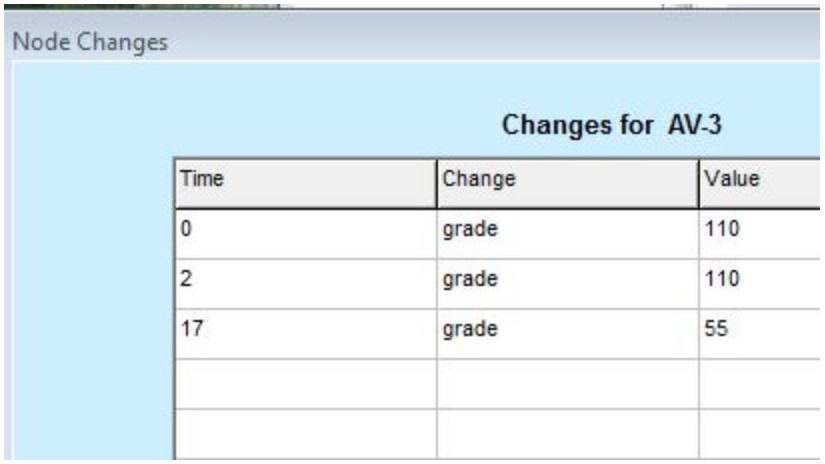

A change in the demand at a junction will produce a transient. This can be used to simulate a hydrant opening or any rapid change in demand. The setup for this as shown above is to provide time/demand pairs to define the demand variation. The demand is in user specified flow units.

One scenario is shown above

1) A demand of 45 is held for 2 seconds and then ramps up to 1000 over the next 4 seconds.
2) A demand begins a 2 seconds and ramps up to 500 over the next 8 seconds

## Creating a Transient – Reservoir Grade Changes




Reservoir of Value

A change in the grade (elevation + head) at a reservoir will produce a transient. This feature may be applied **only at a valve**, which connect a reservoir (one pipe connected to valve a pump – other side is a reservoir). The setup for this situation is shown above is to provide time/grade pairs to define the head variation.

Several scenarios are shown above:

1) At a valve the grade ramps from 200 to 300 in the first 3 seconds and then back to 200 in the next 3 seconds.
2) At a pump the grade ramps down from 110 at 2 second to 55 at 17 seconds.

# Creating a Transient – Periodic Variations

Type: 0=Node 1=Connecting Rerservoir

| NODE | TYPE | PERIOD | PHASE SHIFT | AMPLITUDE | AVERAGE |
|---|---|---|---|---|---|
| AV-2 | 0 | 0.02 | 0 | 0.25 | 0.5 |
| Pump-1 | 0 | 0.02 | 3.14159 | 0.2 | 1 |
| AV-2 | 1 | 0.02 | 0 | 50 | 100 |
| J-3 | 0 | 0.02 | 0 | 10 | 50 |

Periodic variations in valve stem positions ratios, pump speed ratios, grades at valves (a pumps) and junction demands may be specified. This is done as shown above in the Other Data/Periodic Input screen.

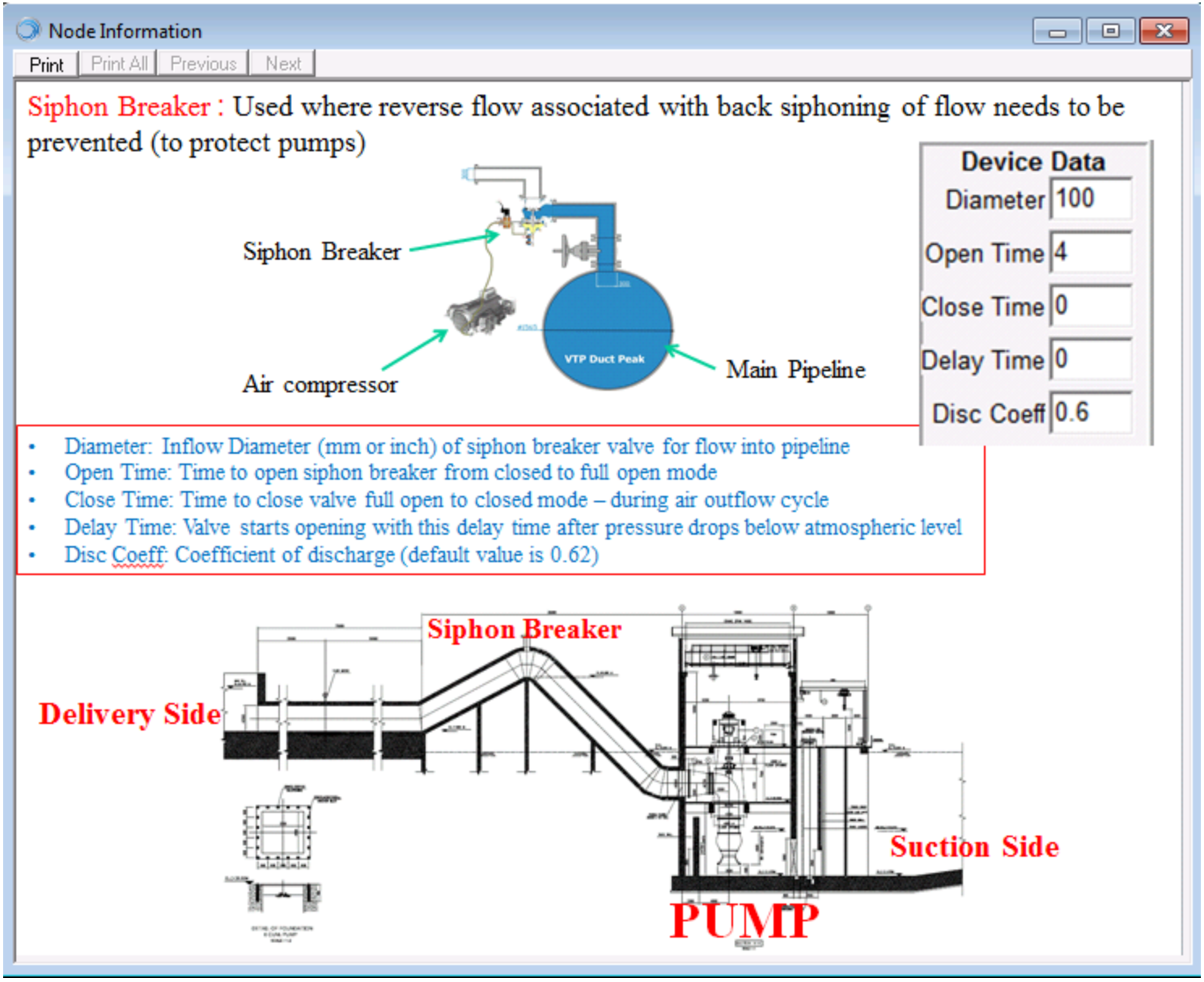
A common application of periodic variation is to model a positive displacement pumps, which is modeled as periodic inflow at a junction node.

Several scenarios are shown above:

1) A valve operates at 50 cps (period = 0.02 sec) with stem position ratio from 0.25 to 0.75 (average = 0.50).
2) A pump operates at 50 cps with a speed variation from 0.8 to 1.2. This oscillation is 180° (3.14159 radians) out of phase with the other shown.
3) The grade at a valve varies from 50 to 150 feet (m) at 50 cps.
4) The demand at junction J-3 varies from 40 to 60 at 50 cps.

Note: For grade variations at a valve or pump, use Type = 1. For all others, use Type =0.

## 1.5.5.1 Siphon Breaker


Node Information
Print
Print All
Previous
Next
Siphon Breaker : Used where reverse flow associated with back siphoning of flow needs to be prevented (to protect pumps)
Siphon Breaker
Air compressor
VTP Duct Peak
Main Pipeline
Device Data
Diameter 100
Open Time 4
Close Time 0
Delay Time 0
Disc Coeff 0.6
• Diameter: Inflow Diameter (mm or inch) of siphon breaker valve for flow into pipeline
• Open Time: Time to open siphon breaker from closed to full open mode
• Close Time: Time to close valve full open to closed mode – during air outflow cycle
• Delay Time: Valve starts opening with this delay time after pressure drops below atmospheric level
• Disc Coeff: Coefficient of discharge (default value is 0.62)
Siphon Breaker
Delivery Side
Suction Side
PUMP

Siphon Breaker (Surge) - Uses the concept that a submerged discharge located at a lower elevation than the required discharge creates a siphon which lowers the total head required by a pump. A siphon breaker valve is placed at the top of the siphoning section of pipe. This valve is open when the pump is not operating or is beginning to close, breaking the siphon. It allows air to discharge during pump start-up. When the pipeline is full, the valve closes and maintains the siphon.

Diameter - diameter of the siphon breaker valve (mm or in)

Open Time - if applicable, the time (in seconds) for the valve to open

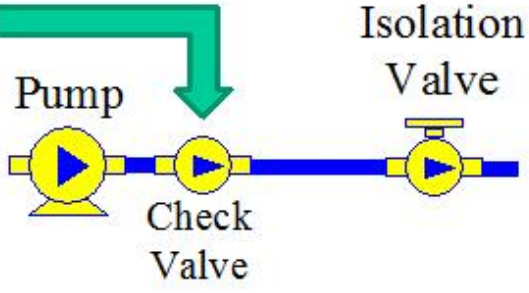
Close Time - if applicable, the time (in seconds) for the valve to close

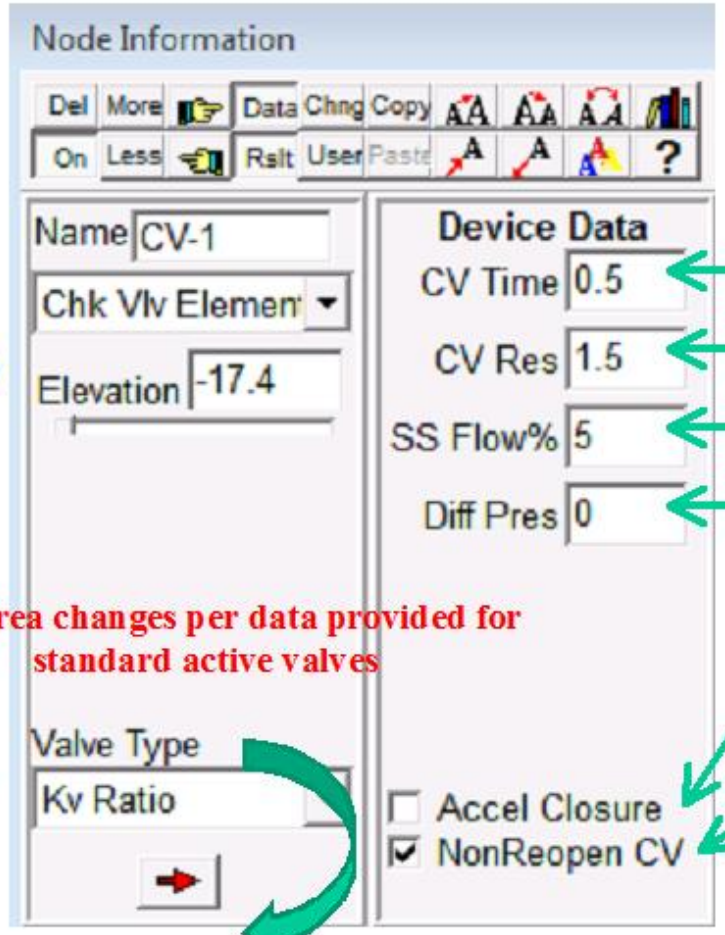
Delay Time - the amount of time (in seconds) actions are delayed.

Disc Coeff(icient) - (units) provided by manufacturer.

## 1.5.5.2 Check Valve Node

# Check Valve (CV) or Non Return Valve (NRV)




Time for 100% open to completely closed position

Valve resistance at 100% open position

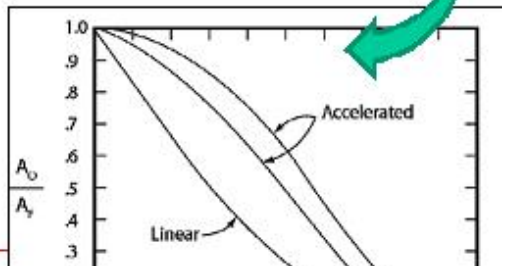
Valve starts closing when flow drops below this value

Valve starts opening when pressure head exceeds this value

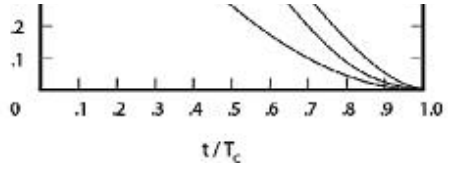
**Flow area changes per data provided for standard active valves**

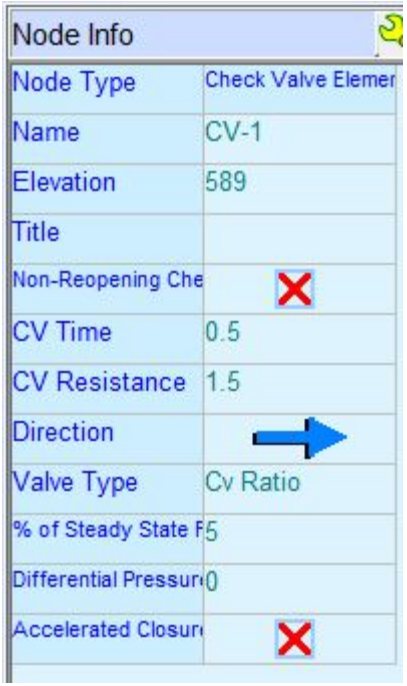
Flow area reduces at constant acceleration. Higher closing speed towards the end of closure

Once closed, the valve remains closed for rest of the simulation



If SS Flow% is 0, valve starts closing at flow reversal
If DiffPresHd is zero, valve starts opening at positive differential head
Resistance defined as headloss (m or ft) divided by square of flowrate in ($m^3/s$ or $ft^3/s$)



Check Valve Node implements constant-rate accelerated closure. The valve starts closing when flowrate drops below certain percentage of steady state flow, and starts opening only when difference in pressure head between u/s and d/s locations of CV exceeds a specified value.

The idea behind starting to close below certain flowrate is that some check valves need a minimum velocity to stay in fully open position and they start closing when flow velocity drops below that threshold. Similarly, check valves do need some differential pressure head to crack them open.

It does not work well to reopen a check valve when the valve is set to close below certain percent of steady state flow (ahead of flow reversal). The valve may stay in closed status at pump restart condition or may not develop enough forward flow for it to start opening. This led to utilizing certain threshold differential pressure to open the closed check valves. It is a process that may even help reduce check valve chattering. Basic Data:

Data for Check Valve Node:

Elevation
Valve Type (pull down menu)
CV Time
CV Resistance
% of SS Flow (Close CV when flow drop below X% of steady state flow)
Diff Pres (Open CV when differential head > X)
Accelerated Closure (check box)
Non-reopening CV(check box)

See also Dynamic Characteristics and Deceleration for check valves.

## 1.5.5.2.1 Dynamic Characteristic of Check Valves

**Dynamic Characteristic of Check Valves**

Improper sizing of check valves results in check valve slam, a phenomenon that produces undue increase in pressure head due rapid decrease in reverse flow velocity associated with check valve closure. Check valves are placed in pipe systems to prevent flow reversal. However, most check valves require a finite time to activate after sensing flow reversal and the flow could develop significant velocity in the reverse direction before the valve is completely closed. This reverse flow velocity is brought to rest when the valve is closed completely. The rapid change in reverse flow velocity produces a rapid increase in pressure head. The reverse flow velocity at the time of complete closure depends on several factors including the pump inertia, steady state head and flow values in the pipe system, length of liquid column to the source of major wave reflection such as a reservoir, friction losses in piping systems, slope of pipeline, and wave celerity.

Although it is difficult to model the exact closure characteristics of a check valve for lack of reliable manufacturer suggested closure characteristics data, one could verify the validity of check valve modeling assumptions with the help of manufacturer suggested deceleration vs. reverse velocity data.

The proposed check valves for this project are VAG Slanted Seat Swing valves. The manufacturer suggested deceleration vs. reverse velocity data for Slanted Seat Swing check valves is shown in Figure 22. The example calculations shown on this Figure pertain to deceleration calculations following a rapid closure of a control valve at the discharge end of a pipeline. ΔH is the magnitude of pressure wave generated by rapid closure of control valve and L is the pipe length. The deceleration equation was generated by replacing the celerity (c) with L/Δt and rearranging the terms in the Joukowsky's equation. This example is not applicable for the present modeling study which involves pump failure scenarios and therefore should be ignored. A different approach should be used to calculate deceleration and the associated reverse flow velocities.

Reverse velocity at the time of complete closure of check valve can be assessed based on the deceleration at the time of flow reversal. The deceleration at the time of flow reversal can be determined by analyzing Surge model of the pipeline system with NO check valves at pumps thereby allowing flow reversal. Figure 23 shows the flow velocity at pump discharge following pump trip along with the corresponding deceleration for 2030 flow conditions. From Figure 23, the deceleration at the time of flow reversal is roughly 1.3 $m/s^2$. This deceleration may then be used to determine the potential reverse flow velocity at the time of complete closure of check valve from manufacturer suggested data. Figure 24 shows an approximate reverse flow velocity of 0.05m/s for a deceleration value of 1.3 $m/s^2$. Knowing the celerity (c) of ductile iron pipe (1200m/s) and the velocity change (ΔV =0.05m/s), the change in pressure head (pressure spike ΔH) at the time of complete closure of check valve may be determined using Joukowsky's equation as

$$\Delta H = (c/g)\,\Delta V$$

$$\Delta H = (1200/9.81)\,0.05$$

$$\Delta H = 6.11m$$

This change in pressure head at the time of complete closure may be compared with change in pressure computed by the Surge model to determine if the assumed check valve closure characteristics are reasonable. Figure 25 shows the flow and head variation at pump discharge for the first few seconds of Surge simulation. It may be noted that all Surge models for the pipe system under consideration used a 1 second linear closure time for the check valves. From Figure 25, the change in pressure head at the time of complete check valve closure (zero flowrate though the pump) is around 25m. Since the change in pressure head computed by the Surge model is significantly higher than the value computed from the deceleration characteristics (6.11m), the assumed check valve closure characteristics provide conservative (worse than what's possible in reality) estimates for the transient pressures. Although deceleration at the time of flow reversal (determined from Figure 23) is an accurate representation for calculating the reversal velocity at the time of complete closure, it may be worthwhile computing reverse velocity based on an average deceleration for further assurance. The average deceleration may be computed based on initial steady state velocity and time for flow reversal as follows.

$$d = (V_o - 0) / \Delta t$$

where d is average deceleration, $V_o$ is steady state velocity, $\Delta t$ is time required for the velocity to change from $V_o$ to 0. The average deceleration for this model computed from Figure 23 is 2.7m/s$^2$. The corresponding reverse flow velocity may be computed from Figure 24 as 0.16 m/s. The resulting increase in pressure head determined using Joukowsky's equation is 19.6m. Since this value is also less than the increase in pressure head of 25m computed by the Surge program, the measures recommended in this report will provide more than adequate protection against pressure spikes resulting from check valve action.

This exercise was repeated for current flow conditions. Figure 25a shows the variation of velocity with time. Since the average deceleration results in higher reverse flow velocity, the pressure head change calculations were done with average deceleration only. The average deceleration for current flow conditions is 1.1 m/s$^2$ and the corresponding reverse velocity is 0.04 m/s (Figure 24). The resulting increase in pressure head is 4.9 m (Joukowsky's Equation) which is also less than the value (25m) predicted by the Surge model.

Figure 26, which shows generic closure characteristics for VAG Slanted Seat Swing Check Valves, also assures that the assumed linear closure characteristics for the check valves will results in conservative estimates for the transient pressures as the actual closure characteristics reduce the flow area very rapidly in the first half of the closure time.

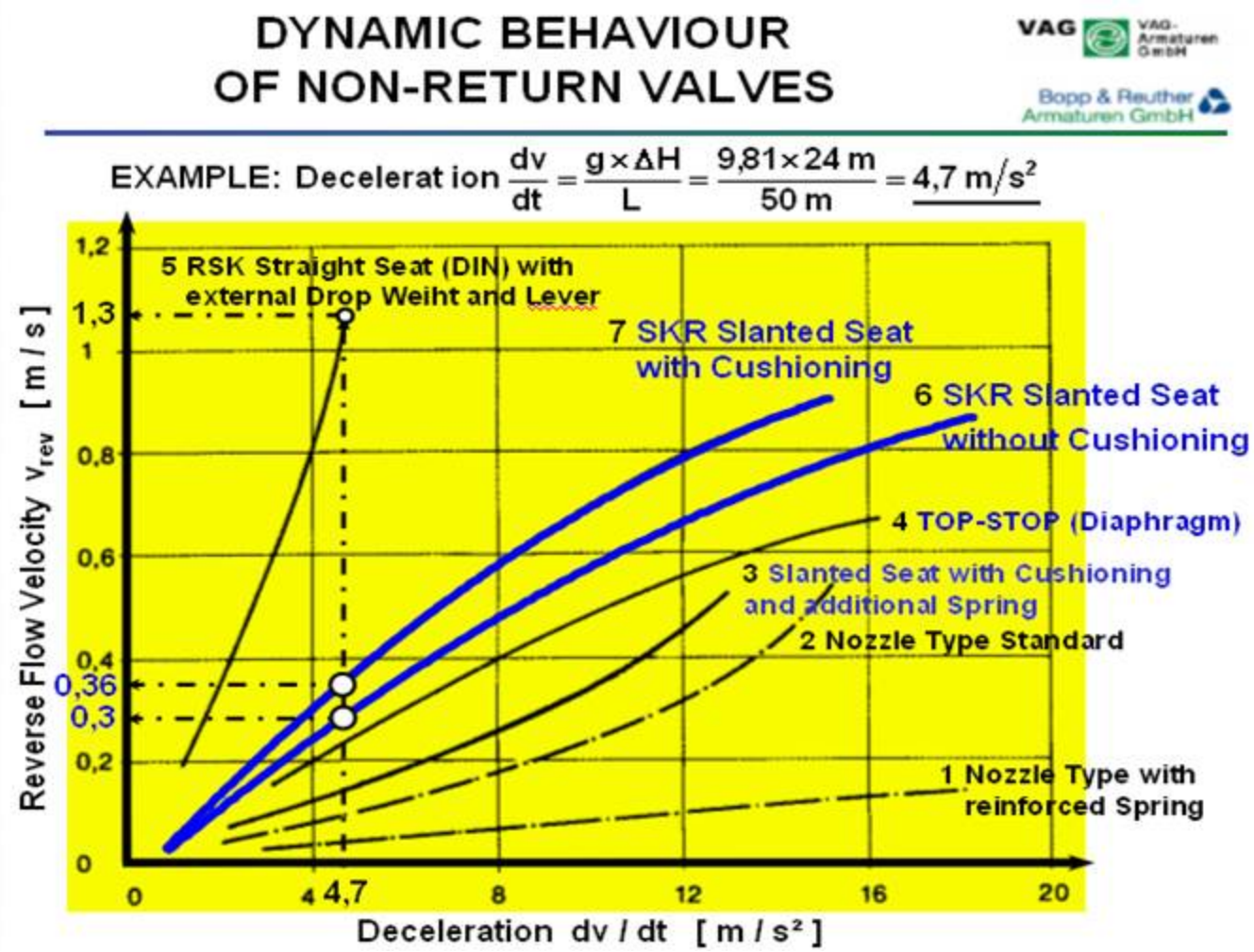



Figure 22. Dynamic characteristics of VAG SKR Slanted Seat (blue lines) non-return valves (courtesy: VAG)

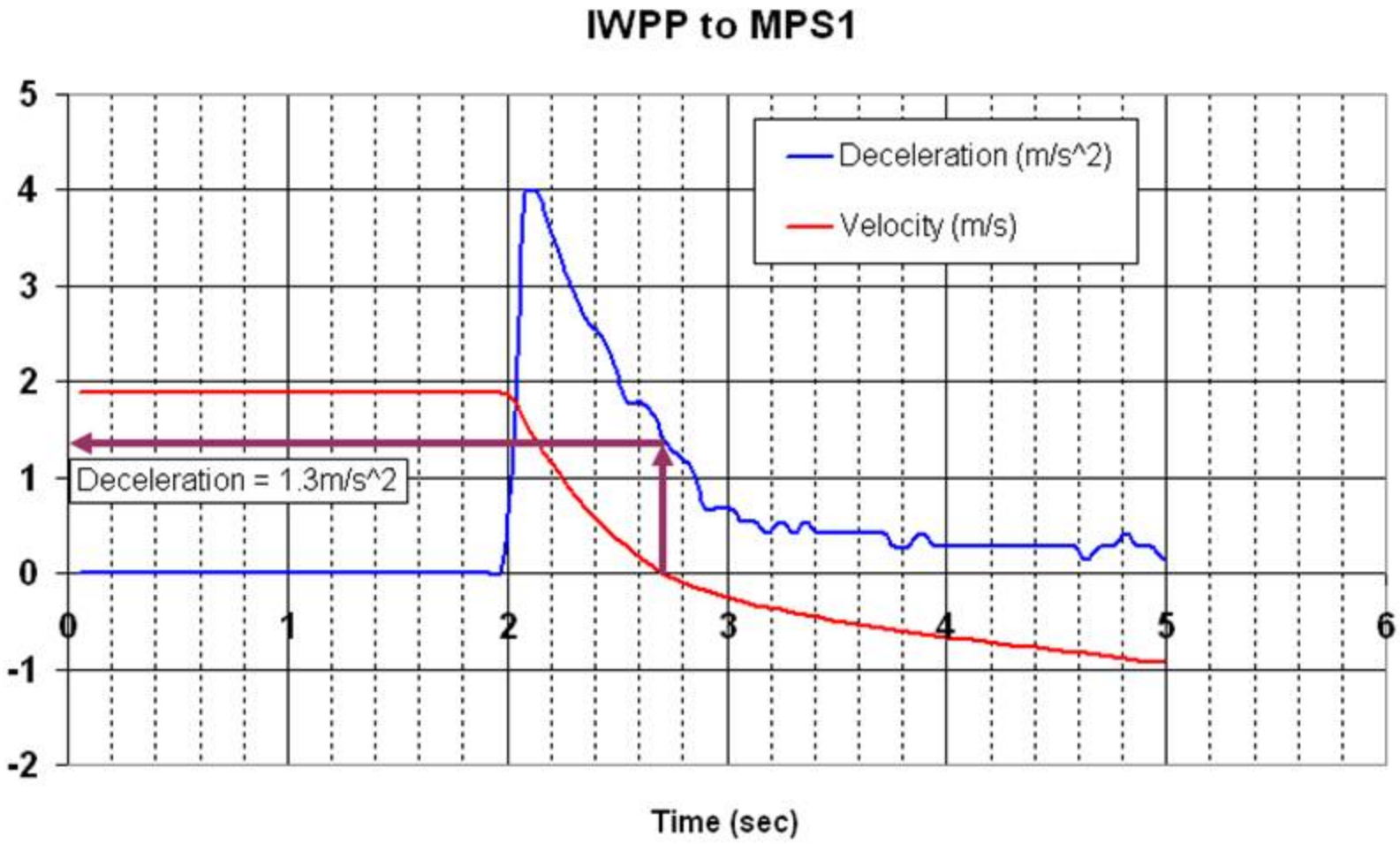
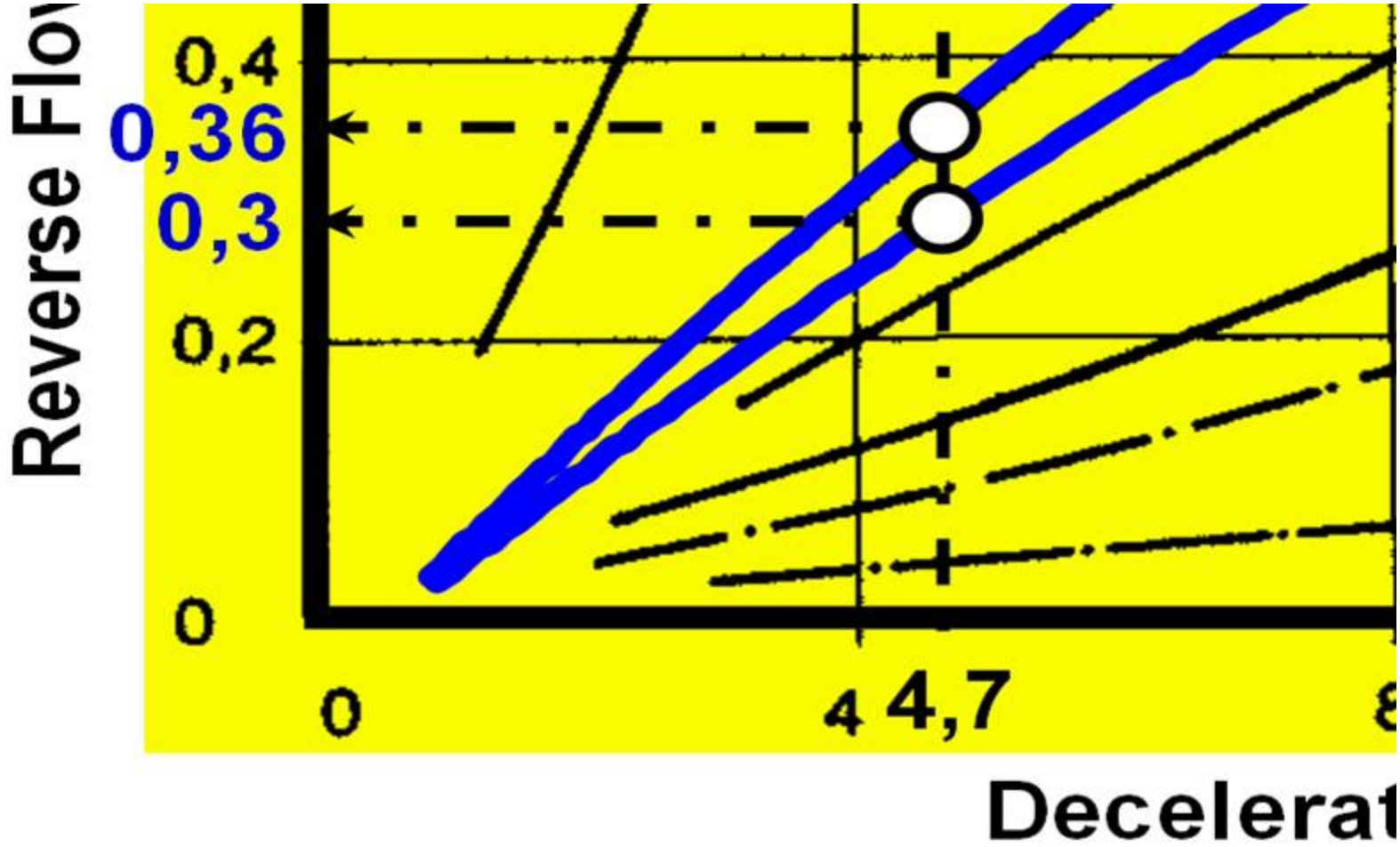



Figure 23. Flow deceleration at pump discharge following pump trip – pumps were modeled with NO check valves




Reverse Velocity = 0.05m/s

Deceleration = 1.3m/s^2

Figure 24. Enlarged view of Figure 24 showing expected reverse velocity based on calculated deceleration

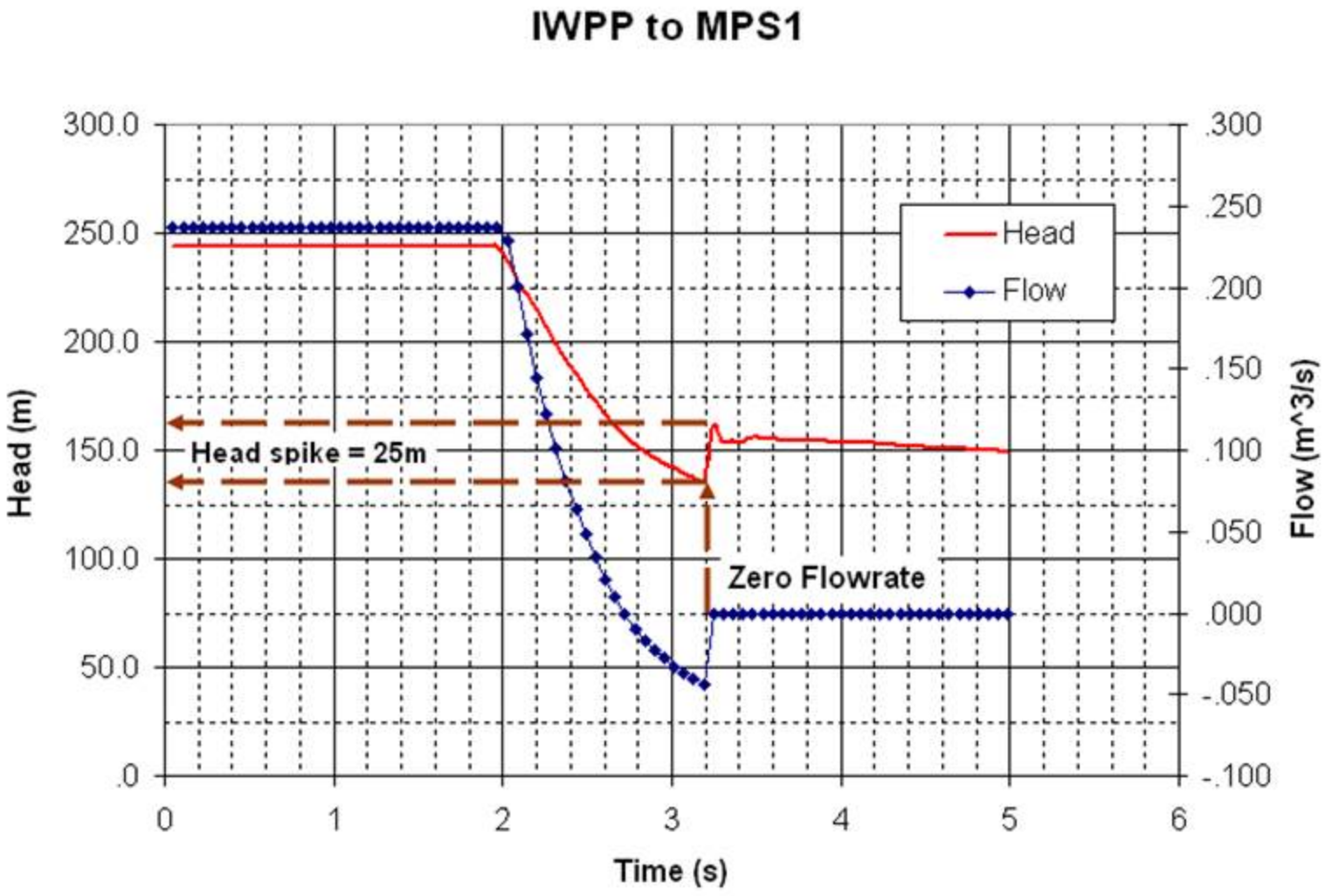



Figure 25. Pressure head variation from Surge model – pumps modeled with check valves (linear closure time of 1 s)

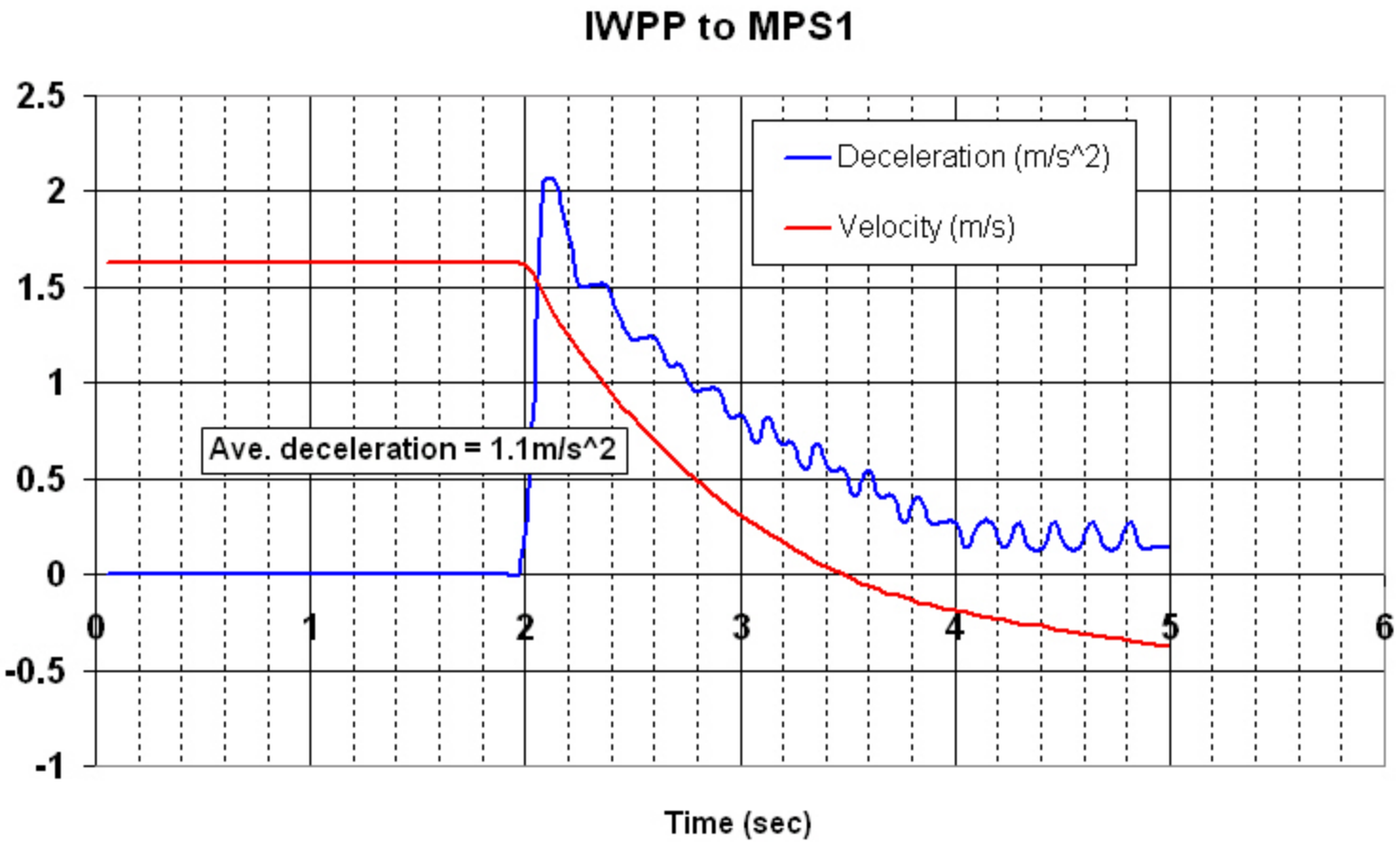



Figure 25a. Change in velocity plot for current flow conditions

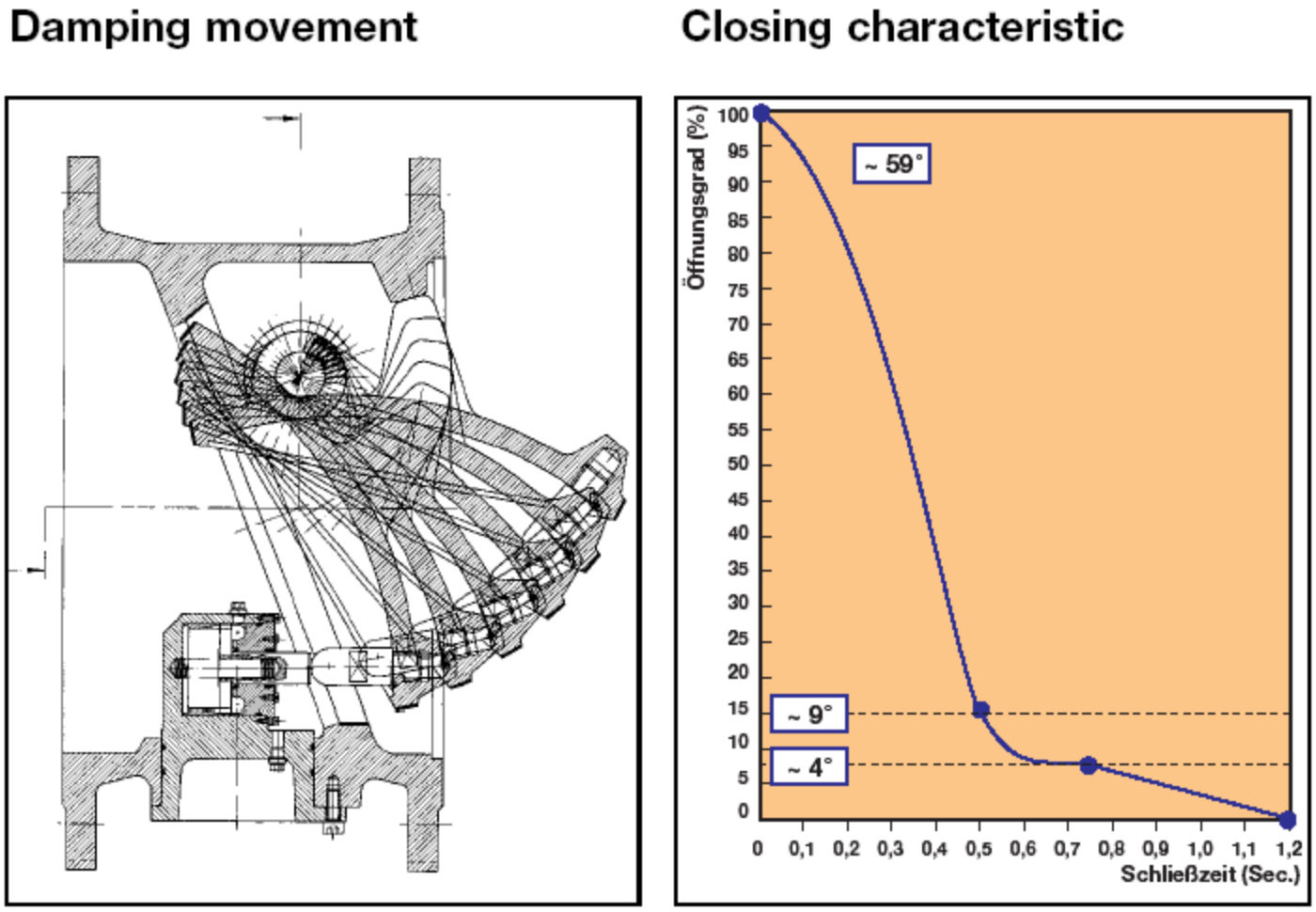



Figure 26. Generic closure characteristics of VAG Slanted Seat Swing Check Valves (Courtesy: VAG)

### 1.5.5.2.2 Deceleration in Check Valve Dynamics Example

# Validation of assumed CV closure time

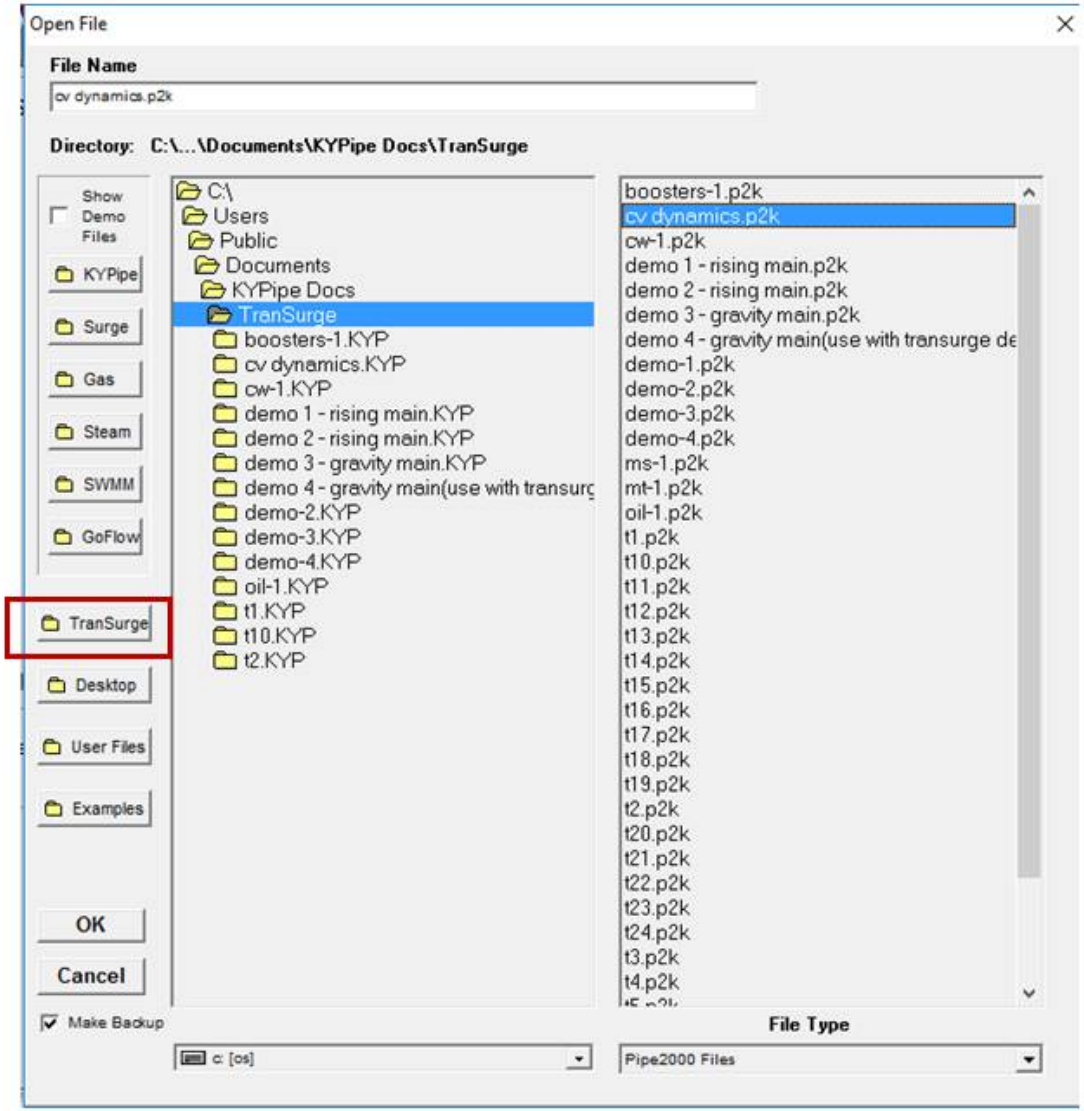
- Load the demo file “cv dynamics.p2k” from TranSurge folder.
- Note the diameter of the pipe connected to the discharge side of the pump (600 mm)
- Note the pump name (Pump-1).
- Run CV Dynamics from the tools menu

# Open “cv dynamics.p2k”

Go to File – Open

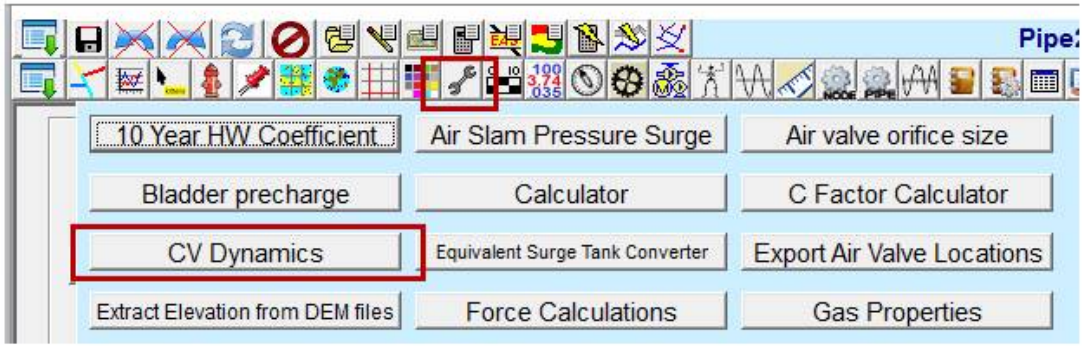
Click the TranSurge folder button.

Note: It is ok if Surge license is used with TranSurge demo model files.

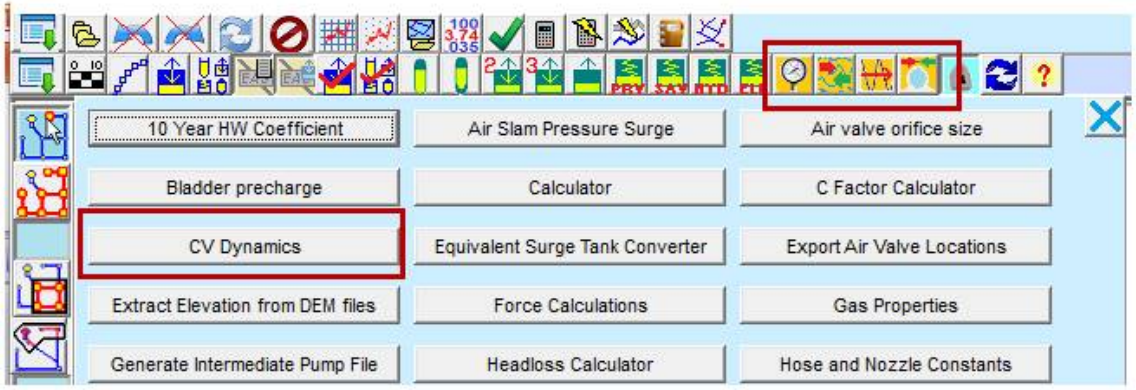


# Run CV Dynamics Tool

From Surge, click Tools icon



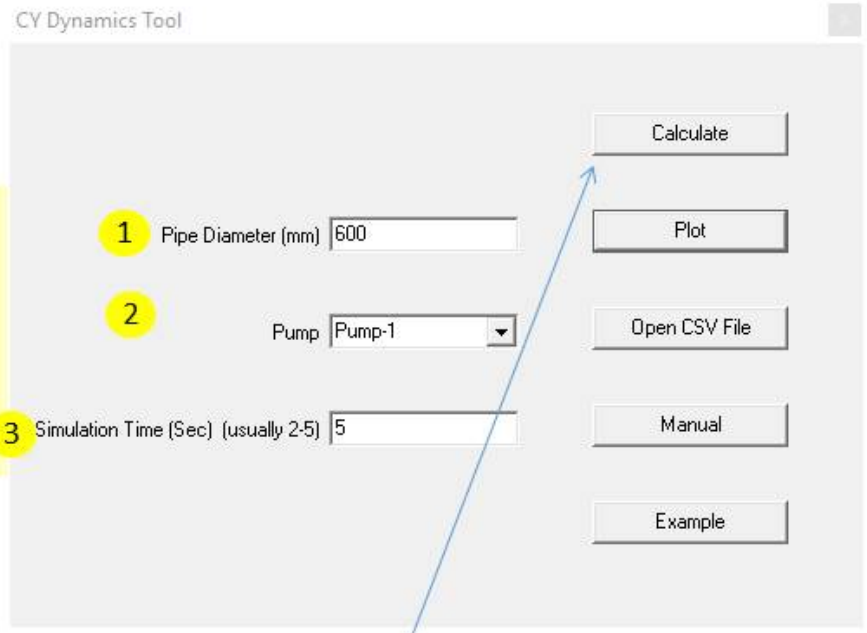

From TranSurge, **right** click one of the indicated icons.



Enter pump discharge diameter in mm (inches for English Units).

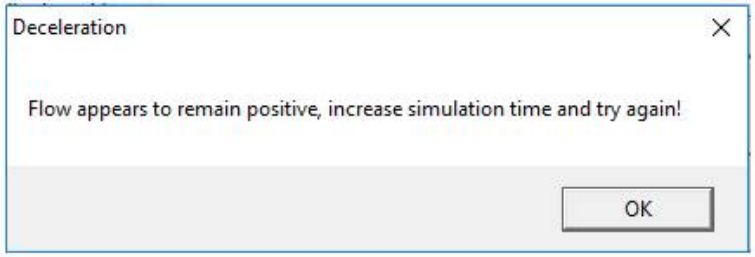
Select pump name.

Enter small simulation time value such as 5 seconds.

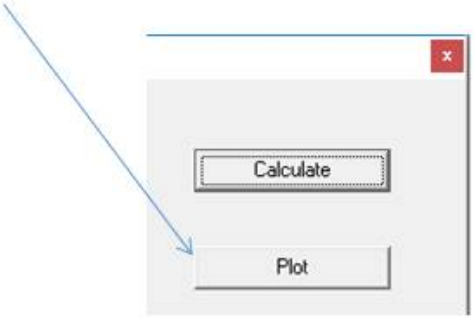


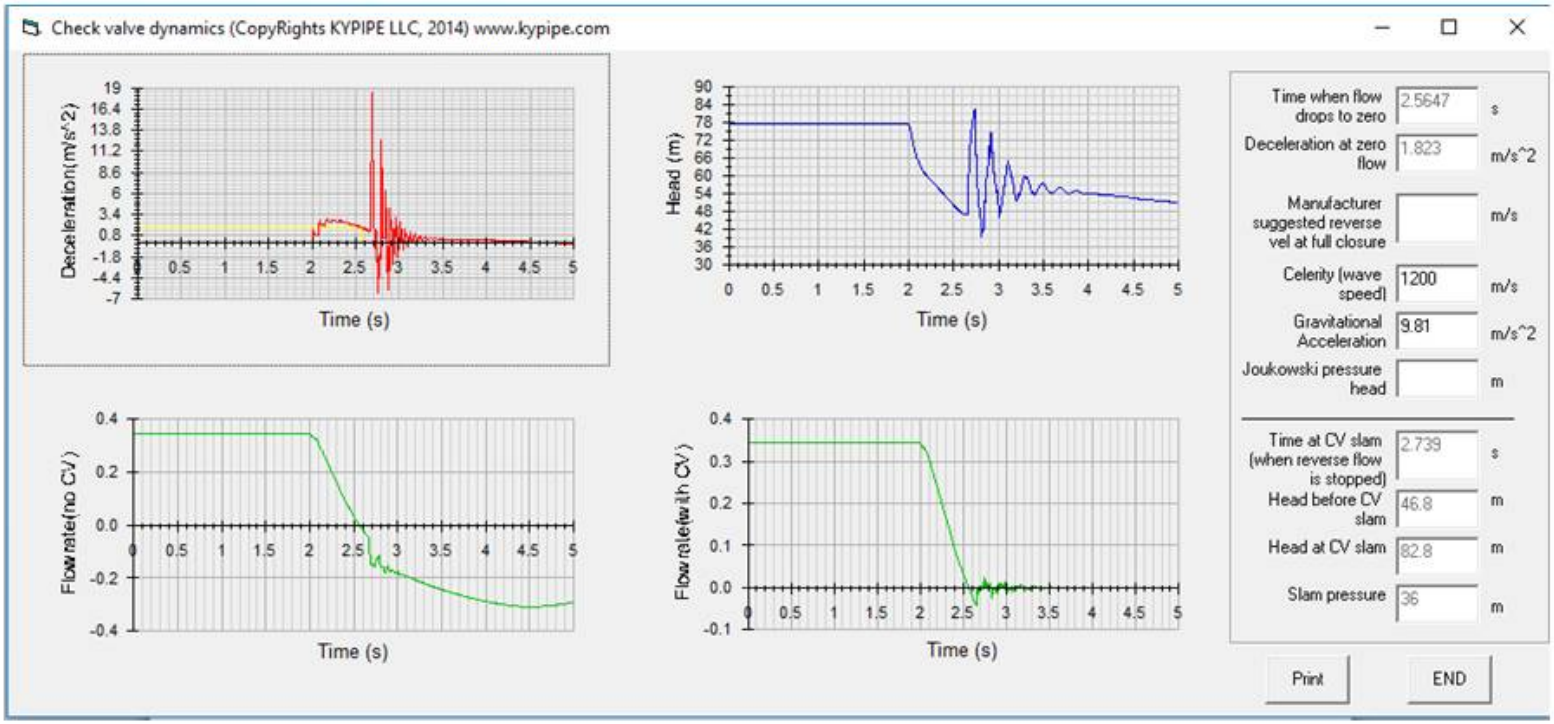

Click Calculate.

If check valve action is not complete (flow reversal) within the allotted simulation time, this message will appear. Increase Simulation Time and run again.

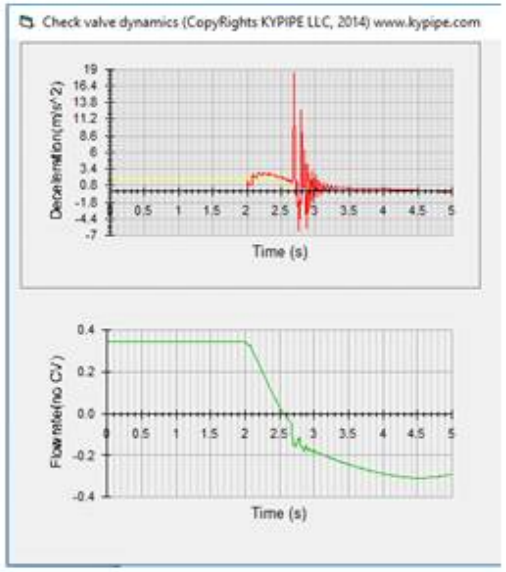


If calculation was successful, next click Plot.



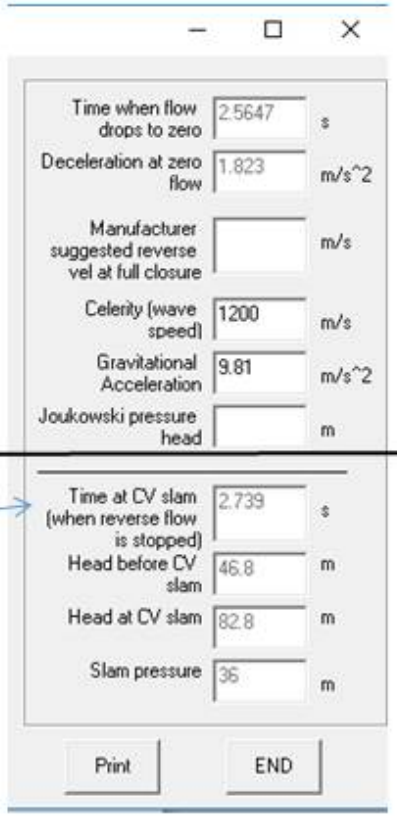



# Plot Overview

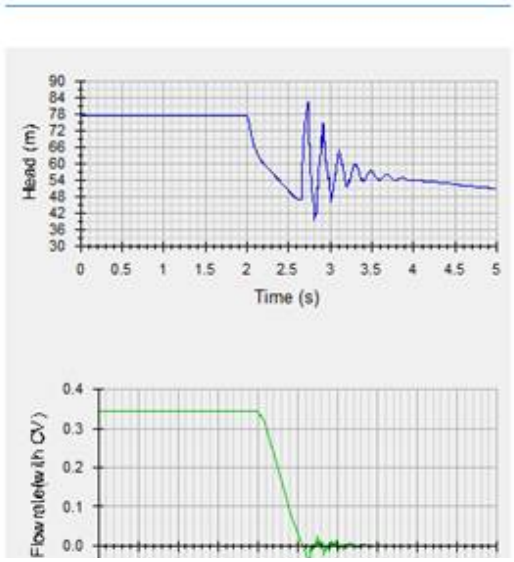



Deceleration from analysis with no CV

Flow from analysis with no CV

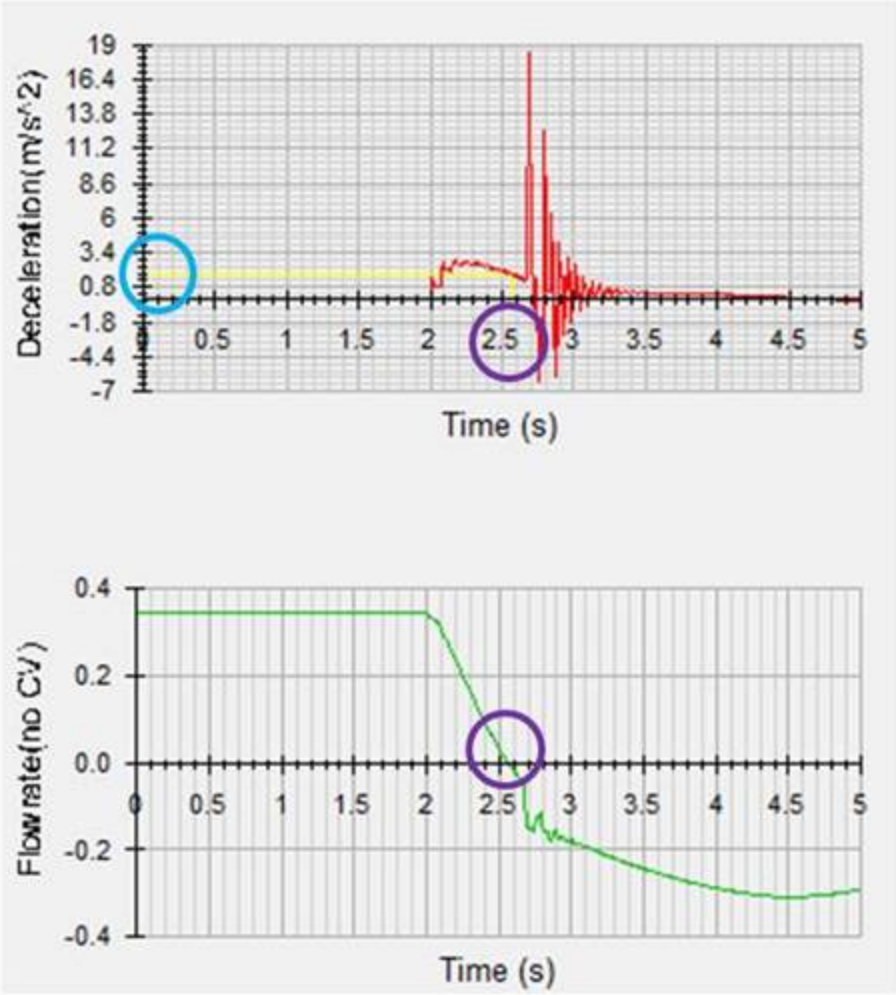



Time when flow in downstream pipe is no longer negative, also maximum pressure occurs



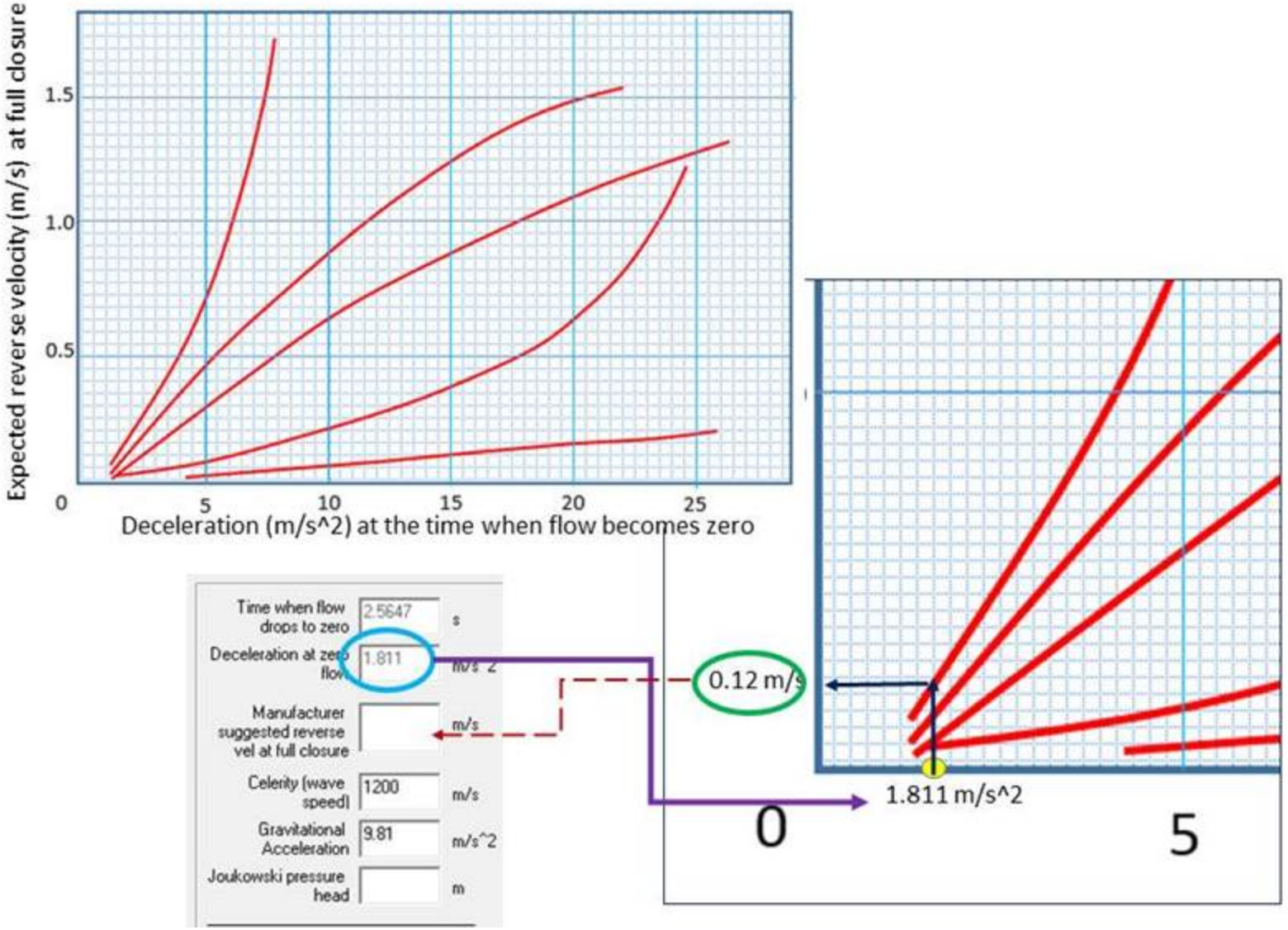

Results with CV in the pump using the CV Time specified in the model.

-0.1
0
0.5
1
1.5
2
2.5
3
3.5
4
4.5
5
Time (s)

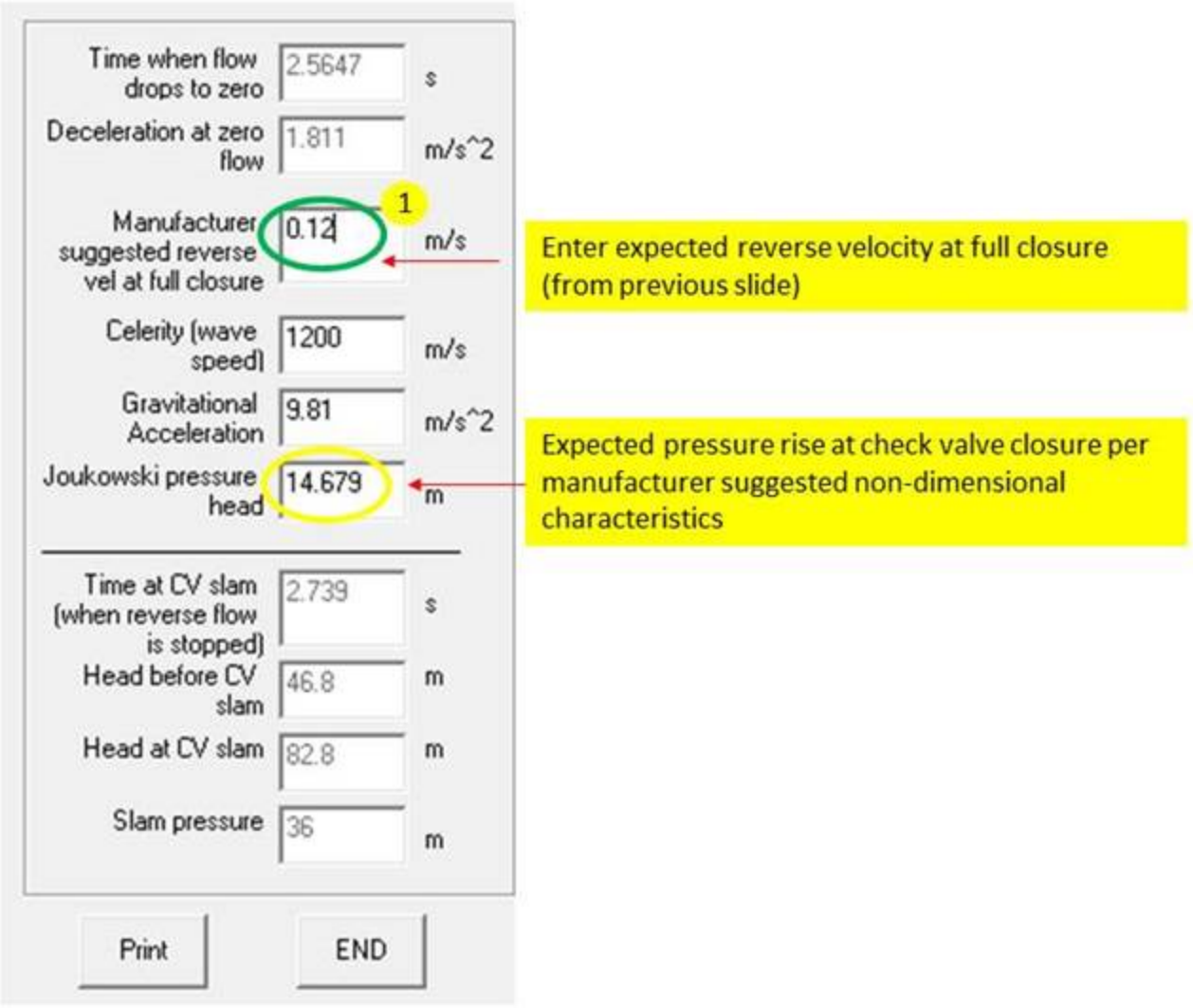



| | | |
|---|---|---|
| Time when flow drops to zero | 2.5647 | s |
| Deceleration at zero flow | 1.811 | m/s^2 |
| Manufacturer suggested reverse vel at full closure | | m/s |
| Celerity (wave speed) | 1200 | m/s |
| Gravitational Acceleration | 9.81 | m/s^2 |
| Joukowski pressure head | | m |

Typical manufacturer suggested non-dimensional check valve characteristics



CV Dynamics Tools




Enter expected reverse velocity at full closure (from previous slide)

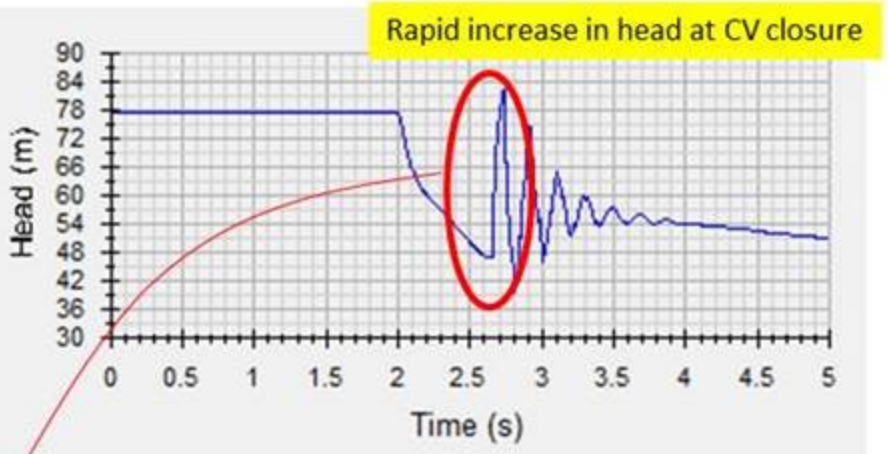
Expected pressure rise at check valve closure per manufacturer suggested non-dimensional characteristics

CV Dynamics Tools

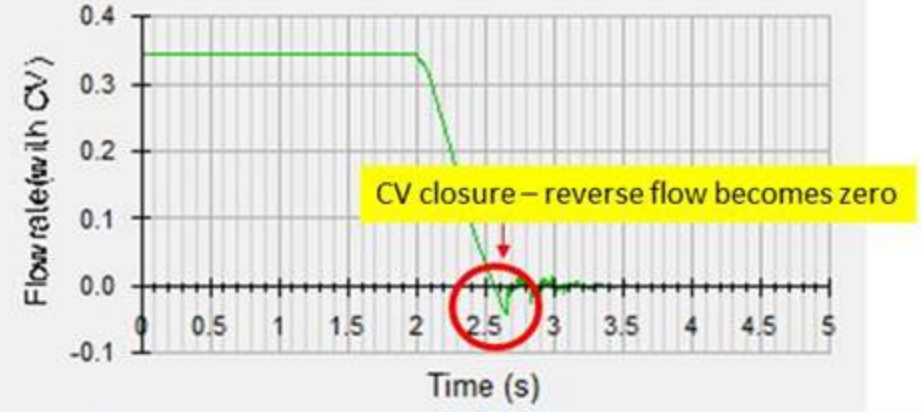
| | | |
|---|---|---|
| Time when flow drops to zero | 2.5647 | s |
| Deceleration at zero flow | 1.811 | m/s^2 |
| Manufacturer suggested reverse vel at full closure | 0.12 | m/s |
| Celerity (wave speed) | 1200 | m/s |
| Gravitational Acceleration | 9.81 | m/s^2 |
| Joukowski pressure head | 14.679 | m |
| Time at CV slam (when reverse flow is stopped) | 2.739 | s |
| Head before CV slam | 46.8 | m |
| Head at CV slam | 82.8 | m |
| Slam pressure | 36 | m |

Rapid increase in head at CV closure

Print END




Model results with assumed CV closure time




Slam pressure from model (Hm): 36m

Slam pressure from non-dimensional curves (Hd): 14.68m

Hm > Hd → assumed closure time is valid (in this case, could be reduced slightly)

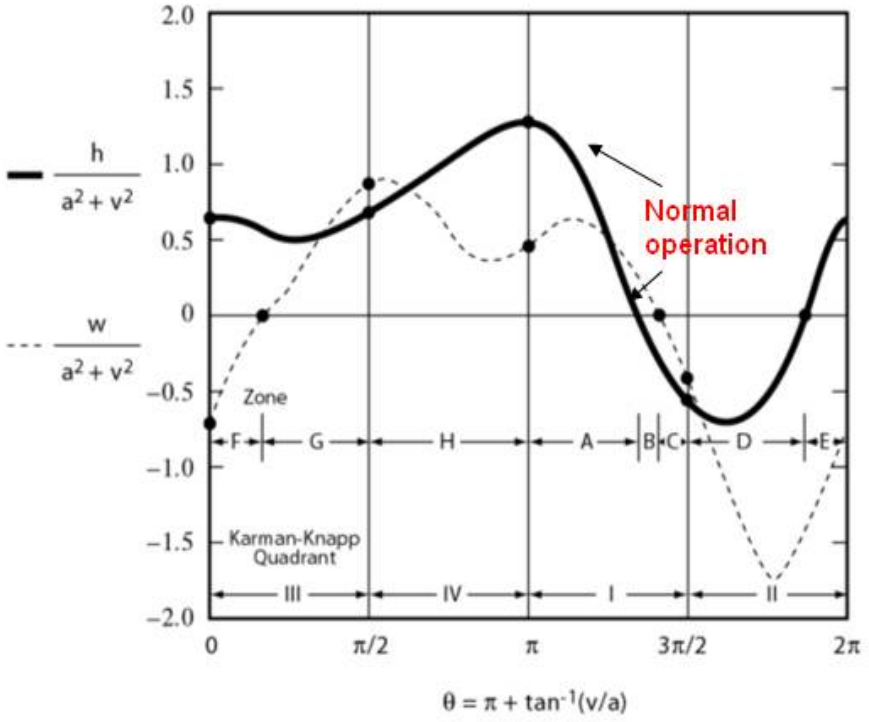
### 1.5.5.3 Creating Custom Pump Files

# Pump Files – Suter Diagrams

Suter Diagrams are used to model pump trips. The Suter Diagram is comprised of 2 plots as shown below. The solid one is a normalized head plot (**h/(a^2+v^2))** and the dashed one is a normalized torque plot (**w/(a^2+v^2)).** Separate Suter Diagrams are provided with Surge. The pump files are text files containing 89 points for the normalized head followed by 89 points of normalized torque data. The terms h, w, a, and v are defined below. To locate and view the text files, see instructions in Appendix A

Data is from the Streeter Wylie Fluid Transients book for files 1,2,3 and files 4-8 came from data published by Dr. David Thorley. It is assumed that pumps with similar Specific Speeds will have a similar Suter Diagram and the appropiate file is selected to use based on the Specific Speed. Data is almost never provided by pump manufacturers for a specific pump.

# Pump Modeling - Abnormal Operation




- Complete Head and Torque Characteristics of a Radial-Flow Pump in Suter Diagram
- Use pump file (four quadrant). Fit A-B-C curve to small region. Calculate speed (torque-inertia).

83

**h = H/$H_R$ (head ratio), w = T/$T_R$ (torque ratio), a = N/$N_R$ (speed ratio**
**v = Q/$Q_R$ (flow ratio)**
**H = pump head, $H_R$ = rated pump head**
**T = pump torque, $T_R$ = rated pump head**
**N = pump speed (rpm), $N_R$ = rated pump speed**
**Q = pump flow, $Q_R$ = rated pump flow**

Figure 1 below shows the text file for Suter File 9991 and Figure 2 shows an Excel plot of this data.

| | | | |
|---|---|---|---|
| .634 | 1.225 | -.684 | .552 |
| .643 | 1.172 | -.547 | .579 |
| .646 | 1.107 | -.414 | .603 |
| .64 | 1.031 | -.292 | .616 |
| .629 | .942 | -.187 | .617 |
| .613 | .842 | -.105 | .606 |
| .595 | .733 | -.053 | .582 |
| .575 | .617 | -.012 | .546 |
| .552 | .5 | .042 | .5 |
| .533 | .368 | .097 | .432 |
| .516 | .24 | .156 | .36 |
| .505 | .125 | .227 | .288 |
| .504 | .011 | .3 | .214 |
| .51 | -.102 | .371 | .123 |
| .512 | -.168 | .444 | .037 |
| .522 | -.255 | .522 | -.053 |
| .539 | -.342 | .596 | -.161 |
| .559 | -.423 | .672 | -.248 |
| .58 | -.494 | .738 | -.314 |
| .601 | -.556 | .763 | -.372 |
| .63 | -.62 | .797 | -.58 |
| .662 | -.655 | .837 | -.74 |
| .692 | -.67 | .865 | -.88 |
| .722 | -.67 | .883 | -1 |
| .753 | -.66 | .886 | -1.12 |
| .782 | -.655 | .877 | -1.25 |
| .808 | -.64 | .859 | -1.37 |
| .832 | -.6 | .838 | -1.49 |
| .857 | -.57 | .804 | -1.59 |
| .879 | -.52 | .758 | -1.66 |
| .904 | -.47 | .703 | -1.69 |
| .93 | -.43 | .645 | -1.72 |
| .959 | -.36 | .583 | -1.65 |
| .996 | -.275 | .52 | -1.59 |
| 1.027 | -.16 | .454 | -1.52 |
| 1.06 | -.04 | .408 | -1.42 |
| 1.09 | .13 | .37 | -1.32 |
| 1.124 | .295 | .343 | -1.23 |
| 1.165 | .43 | .331 | -1.1 |
| 1.204 | .55 | .329 | -.98 |
| 1.238 | .62 | .338 | -.82 |
| 1.258 | .634 | .354 | -.684 |
| 1.271 | | .372 | |
| 1.282 | | .405 | |
| 1.288 | | .45 | |
| 1.281 | | .486 | |
| 1.26 | | .52 | |

**Figure 1 File 9991 (89 values of h/(a^2+v^2) and w/(a^2+v^2))**

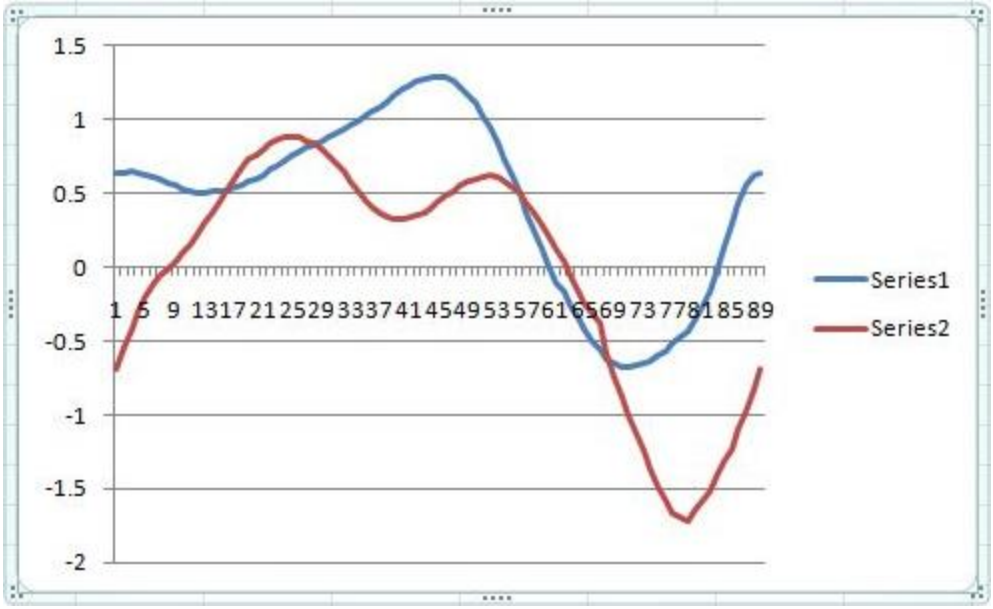



**Figure 2 Excel Plot of 9991**

Users may create their own Suter file if they have suitable dynamic pump data. They need to create a text file with exactly 89 points of normalized head data followed by 89 points of normalized torque data and name the file 9999.txt. Then it must be copied into the folder located by the batch file, as described in Appendix A.

Appendix A: Locating the text files containing Suter Pump File Data.

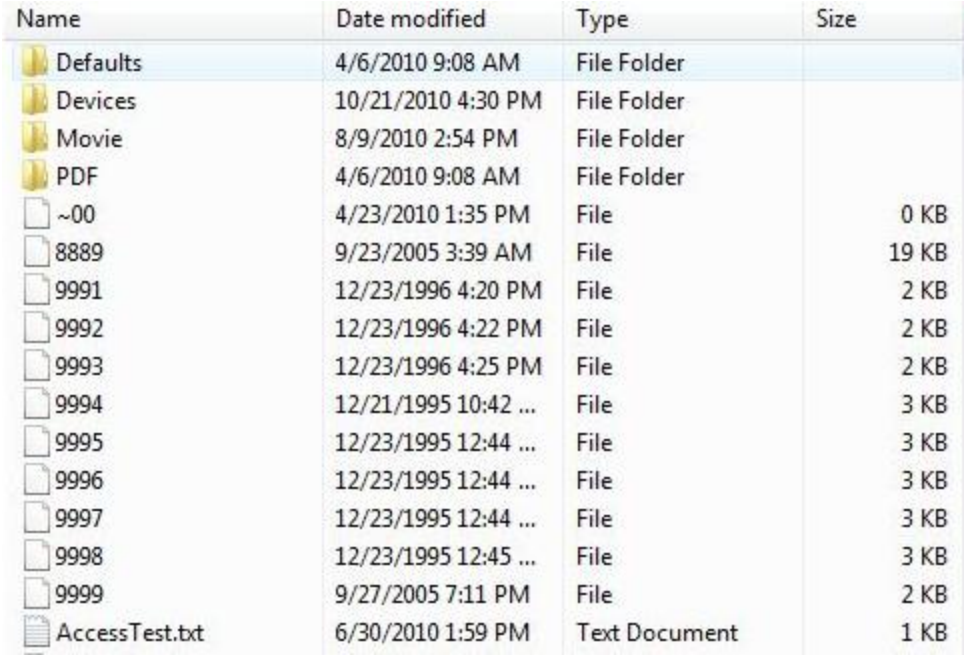
There are two ways to locate the folder where Suter Pump File Data is located

1) Go to the Start menu and find the Pipe2018 folder. Use the shortcut called "Program Data Folder" (in Win 7 it is under Utility).

2) or, Browse to the Pipe2018 folder. This is located in C:\Program files (x86)\KYPipe\Pipe2018 (or similar pathname). Find the file called “Open KYPipe Application Data.bat” (batch file). You may sort the contents of this folder by Date

Modified, most recent at the top, and this file should be one of the most recent files as it gets regularly updated. Double click on this batch file.

Both options will open up a Windows Explorer browser window to the location of the text files with Suter Pump File data. You will see folder contents similar to the following:

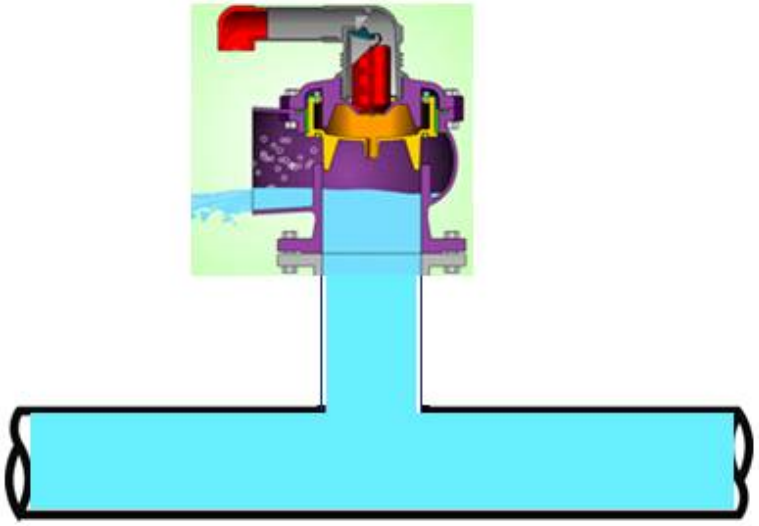


| Name | Date modified | Type | Size |
|---|---|---|---|
| Defaults | 4/6/2010 9:08 AM | File Folder | |
| Devices | 10/21/2010 4:30 PM | File Folder | |
| Movie | 8/9/2010 2:54 PM | File Folder | |
| PDF | 4/6/2010 9:08 AM | File Folder | |
| ~00 | 4/23/2010 1:35 PM | File | 0 KB |
| 8889 | 9/23/2005 3:39 AM | File | 19 KB |
| 9991 | 12/23/1996 4:20 PM | File | 2 KB |
| 9992 | 12/23/1996 4:22 PM | File | 2 KB |
| 9993 | 12/23/1996 4:25 PM | File | 2 KB |
| 9994 | 12/21/1995 10:42 ... | File | 3 KB |
| 9995 | 12/23/1995 12:44 ... | File | 3 KB |
| 9996 | 12/23/1995 12:44 ... | File | 3 KB |
| 9997 | 12/23/1995 12:44 ... | File | 3 KB |
| 9998 | 12/23/1995 12:45 ... | File | 3 KB |
| 9999 | 9/27/2005 7:11 PM | File | 2 KB |
| AccessTest.txt | 6/30/2010 1:59 PM | Text Document | 1 KB |

The text files 9991, 9992, etc. are the Suter Pump File Data files. They may be viewed in Notepad, WordPad, Word, etc. Only edit file 9999. To use this file in Pipe2018, Select Pump- File K for the Node Type. Then in in File # (1-20) enter "9".

### 1.5.5.4 Dynamic Air Valves

## Dynamic Air Valve: Air valve that opens and closes in a controlled manner allowing water flow after all air is out of pipeline



| Device Data | |
|---|---|
| Inflow D | 100 |
| Outflow D | 100 |
| Close Time | 2 |
| Init Vol | 0 |
| Open Time | 2 |
| Disc Coeff | 0.6 |

- Inflow D: Inflow Diameter (mm or inch) for flow into the pipeline
- Outflow D: Outflow Diameter (mm or inch) for flow out of pipeline
- Close Time: Time to close dynamic air valve from full open to closed mode
- Init Vol: Volume of air at time zero of transient simulation, usually zero.
- Open Time: Time to open dynamic air valve from closed to full open mode
- Disc Coeff: Coefficient of discharge (default value is 0.62)

Dynamic air valve (DAV) starts opening when pressure at valve location drops below atmospheric levels allowing intake of air. Air is let out of the valve when pressure at air valve goes above atmospheric levels. DAV doesn't close after all air is out of pipeline, continues to let water out of pipeline thus preventing air slam conditions. DAV starts closing after all air is out of pipeline and closes in a few seconds.

## Modeling of Dynamic Air Valves

Dynamic Air Valves are designed to eliminate secondary surge pressures due air slam during air venting cycle. Air slam occurs as the moving water column (behind the air column during air venting cycle) is brought to rest abruptly when the float of an ordinary air valve gets lifted up after all air is evacuated. Typical 2- or 3- stage air valves only discharge air from the pipeline. Dynamic Air Valves continue to allow water to flow out of air valve after all air is vented out (instead of abruptly stopping this flow by lifting the float as in ordinary air valves) and gradually (in about 1 to 2 seconds) closing the valve to reduce the water column velocity in a highly controlled manner completely eliminating the air slam pressures.

Though dynamic air valves offer perfect solution for air slam problems, they come with their own set of problems. One main problem is the

need for discharging water to atmosphere. The second problem is the complexities associated with modeling of dynamic air valves in surge analysis programs. The following section describes these complexities and how they are handled in Pipe2018: Surge.

**Dynamic Air Valve Modeling in Pipe2018: Surge**

Variables:

- Full open area of air valve (FA)
- Opening time (TO)
- Closing time (TC)

Operating Scenarios:

- **Valve fully closed and a negative pressure wave (down surge) passes by**: Valve starts opening at the instant it senses the negative pressure (pressure below atmospheric level). Valve opens fully in user-defined time period (TO). Ordinary air valves are expected to go from fully closed (zero area) to fully open (FA) area instantaneously (assuming no delay time) or in one computational time step (which is roughly 0.001 seconds for most pipeline systems). Dynamic air valves start opening on sensing down surge and are fully open only after TO seconds (which might be roughly 1 second). For example, if the inflow orifice of a dynamic air valve is 100mm, and TO is 1 second, the open area of the air valve is only about 0.1mm equivalent in a time step of 0.001 second. Depending on the magnitude of the down surge, the pressure at the air valve location might drop considerably below atmospheric levels if the valve opens only to an equivalent area of 0.1mm. Smaller opening time (TO) will help reduce the initial negative pressure problems at dynamic air valves. However, smaller TO valves seem to cause another big problem as illustrated in subsequent sections. Pipe2018: Surge computes actual open area of the dynamic valve based on the full open area (FA), opening time (TO) and computational time period (Δt) at the end of each time step and computes the amount of air that gets into the pipeline.

  **Down Surge continues even after the valve is fully opened**: Air valve size is set to fully open area and air continues to flow into the air valve.

  **Upsurge after the valve is fully opened**: Air valve size remains at fully open level and air venting cycle starts. Water column behind the air column continues to accelerate as air gets vented out. Ordinary air valves stop water column abruptly by lifting the float after all air is out of the pipeline at the location of air valve. Dynamic air valves continue to release water through the fully open area after all air is out of pipeline. Pipe2018: Surge models this stage of the air valve as an orifice discharging freely into atmosphere and the orifice size is equivalent to fully open area (FA). The orifice area gets reduced in a controlled manner from fully open (FA) to zero in a user defined valve closing time (TC). Valve closes more rapidly in the first 50% of the closing time and slowly in the later 50% of the closing time. 75% of the flow area is reduced in first 50% of TC and the remaining 25% flow area in the second half of TC. Once the valve is fully closed, it

gets opened only on next down surge.

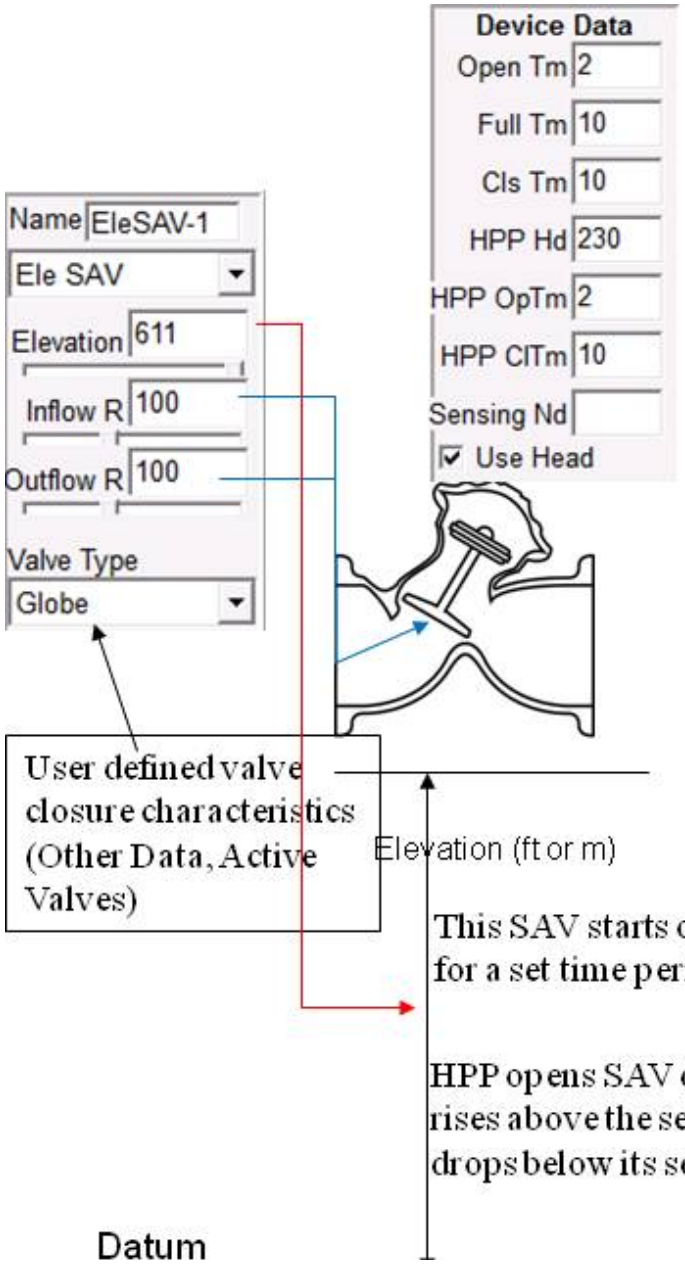
- **Down surge during water outflow cycle (before the valve is fully closed)**: In general, release of water from pipelines is expected to reduce the pressures at that location. Larger outflow generally translate to larger drop in pressures. This phenomenon offers added advantage for dynamic air valves. First, dynamic air valves continue to release water after all air is out of pipelines thereby reducing the air slam pressures. Next, release of water through the outflow orifice reduces the pressure at that location thereby reducing the overall magnitude of pressure resulting from the upsurge. However, it is also common knowledge that release of large quantities of water (more than a desirable value) can drop the pressures at that location below atmospheric levels. This is possible when the size of dynamic air valve is larger than a desirable (optimal) value resulting large outflow of water and consequent drop in pressure. It is also possible that a reflected pressure wave due to large outflow of water from dynamic air valve could result in negative pressure condition at the air valve during water outflow cycle. This negative pressure condition can now reverse the water flow mode to air flow mode and offers greater complexity for modeling. Pipe2018: Surge keeps track of the orifice size during water flow cycle and switches to air flow cycle at the same orifice size and starts opening the air valve until it is fully opened.

  Occasionally, a sharp negative pressure wave might be passing by a dynamic air valve which is discharging water to atmosphere. For example, suppose the dynamic valve is closed and the local pressure is 50m while a sharp negative pressure wave is passing by that valve. The sharp negative pressure wave might drop the pressure at dynamic valve momentarily to a negative value but bounces back to 50m after the wave passes by. Because the dynamic valves are set to open on sensing negative pressures, the valve opens to allow air into the pipeline. If the air valve is set to open instantaneously (TO = 0) the valve opens and allows the pressures to stay at atmospheric level, otherwise the pressure at this location might drop below atmospheric levels. However, opening the valve instantaneously or rapidly (within a very short time such as 0.1 seconds) can lead to serious problems after the negative pressure wave has passed. Suppose the valve opens completely to take care of the negative pressure wave, it stays open and allow water to flow out of the valve and closes slowly in TC seconds. This is not a realistic scenario as the valve takes a finite time to open and close. Opening the valve in a controlled manner and keeping track of the open area when the switch from air flow mode to water flow mode allows eliminating unrealistic pressure waves. In addition, the current version of Pipe2018: Surge does not allow water flow mode if the air valve opens less than 5% of full area during air flow mode before switching to water flow mode. This approach is more realistic as it takes into account the inertia associated with opening and closing of valves and eliminates the numerical instabilities.

### 1.5.5.5 Electrically Actuated Surge Anticipation Valve

This is a combination of SAV and HydSAV and requires one additional data item called Delay Time (compared to Hyd SAV).

# Electrically Actuated Surge Anticipation Valve (Ele SAV)




HPP: High Pressure Pilot

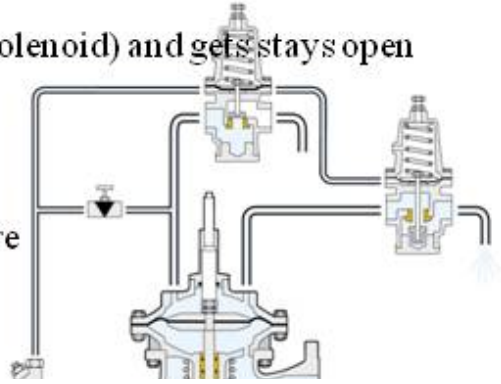
Open Tm: Time for SAV to go from closed to full open mode on down surge
Full Tm: Time for SAV to stay in full open mode during down surge
Cls Tm: Time for SAV to go from full open to closed mode on down surge

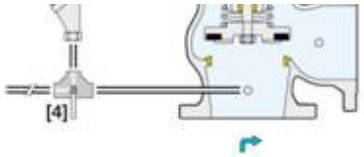
HPP Hd: Pressure head setting for HPP
HPP OpTm: Time for HPP to open SAV fully closed to fully open mode
HPP ClTm: Time for HPP to close SAV from fully open to fully closed mode

Sensing Nd defaults to Ele SAV location if left blank
Inflow R = Resistance for flow into the pipe (use a large number such as 99999 if no flow is allowed into pipe)
Outflow R = Resistance for flow out of pipe
Resistance ($s^2/ft^5$ or $s^2/m^5$) = headloss/flow$^2$

This SAV starts opening on pump trip condition (actuated by a solenoid) and gets stays open for a set time period (Full Tm) and closes thereafter.

HPP opens SAV during upsurge when pressure at sensing pos. rises above the setting and keeps the valve open until the pressure drops below its setting.






| Node Info | |
|---|---|
| Node Type | EleSAV (Hd) |
| Name | EleSAV-1 |
| Elevation | 589 |
| Title | |
| On/Off | ✓ |
| Inflow Resistance | 100 |
| Outflow Resistance | 100 |
| Sensing Node | |
| Valve Type | Ball |
| High Prs Pilot Hd | 230 |
| HPP Open Time | 2 |
| HPP Close Time | 10 |
| External Head | 0 |
| Close Time | 2 |
| Open Time | 10 |
| Full Time | 10 |
| Delay Time | 0 |

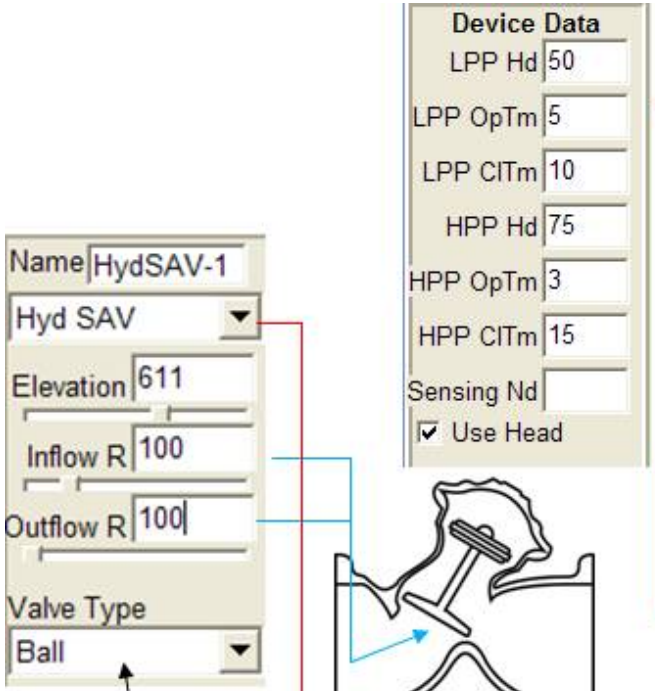
Electrically actuated SAV is triggered to open at the beginning of simulation irrespective of the change conditions. In reality, an SAV is wired to open on loss of power irrespective of pressure conditions. In the software, it would be difficult to identify the SAV with any particular pump (there could be multiple pumps and pump stations) and whether a trip condition is specified. Therefore, the SAV gets activated right at the beginning of simulation. If a start up condition is being modeled, the best way to avoid

activation of the SAV is to turn the EleSAV off for pump start up conditions.

### 1.5.5.6 Hydraullically Actuated Surge Anticipation Valve (HydSAV)

HydSAV is a surge anticipating valve that is controlled by pressures rather than time. It can model both low and high pressure pilots

## Hydraulically Actuated Surge Anticipation Valve (Hyd SAV)




LPP: Low Pressure Pilot
HPP: High Pressure Pilot

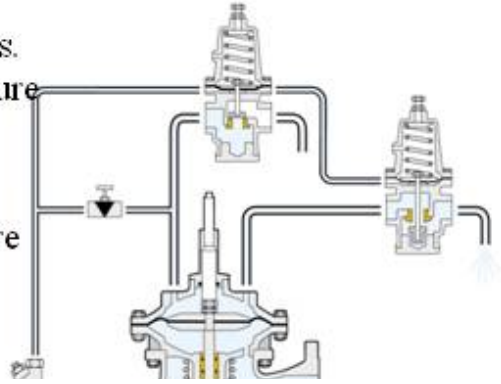
LPP Hd: Pressure head setting for LPP
LPP OpTm: Time for LPP to open SAV from closed to full open mode
LPP ClTm: Time for LPP to close SAV from full open to closed mode

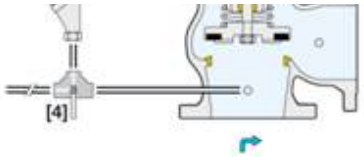
HPP Hd: Pressure head setting for HPP
HPP OpTm: Time for HPP to open SAV from closed to full open mode
HPP ClTm: Time for HPP to close SAV from full open to closed mode

Sensing Nd default to Hyd SAV location if left blank
Inflow R = Resistance for Flow into the pipe (use a large number such as 99999 if no flow is allowed into pipe)
Outflow R = Resistance for Flow out of pipe
Resistance ($s^2/ft^5$ or $s^2/m^5$) = headloss/flow$^2$

LPP opens SAV during down surge when pressure at sensing pos. drops below the setting and keeps the valve open until the pressure rises above its setting.

HPP opens SAV during up surge when pressure at sensing pos. rises above the setting and keeps the valve open until the pressure drops below its setting.

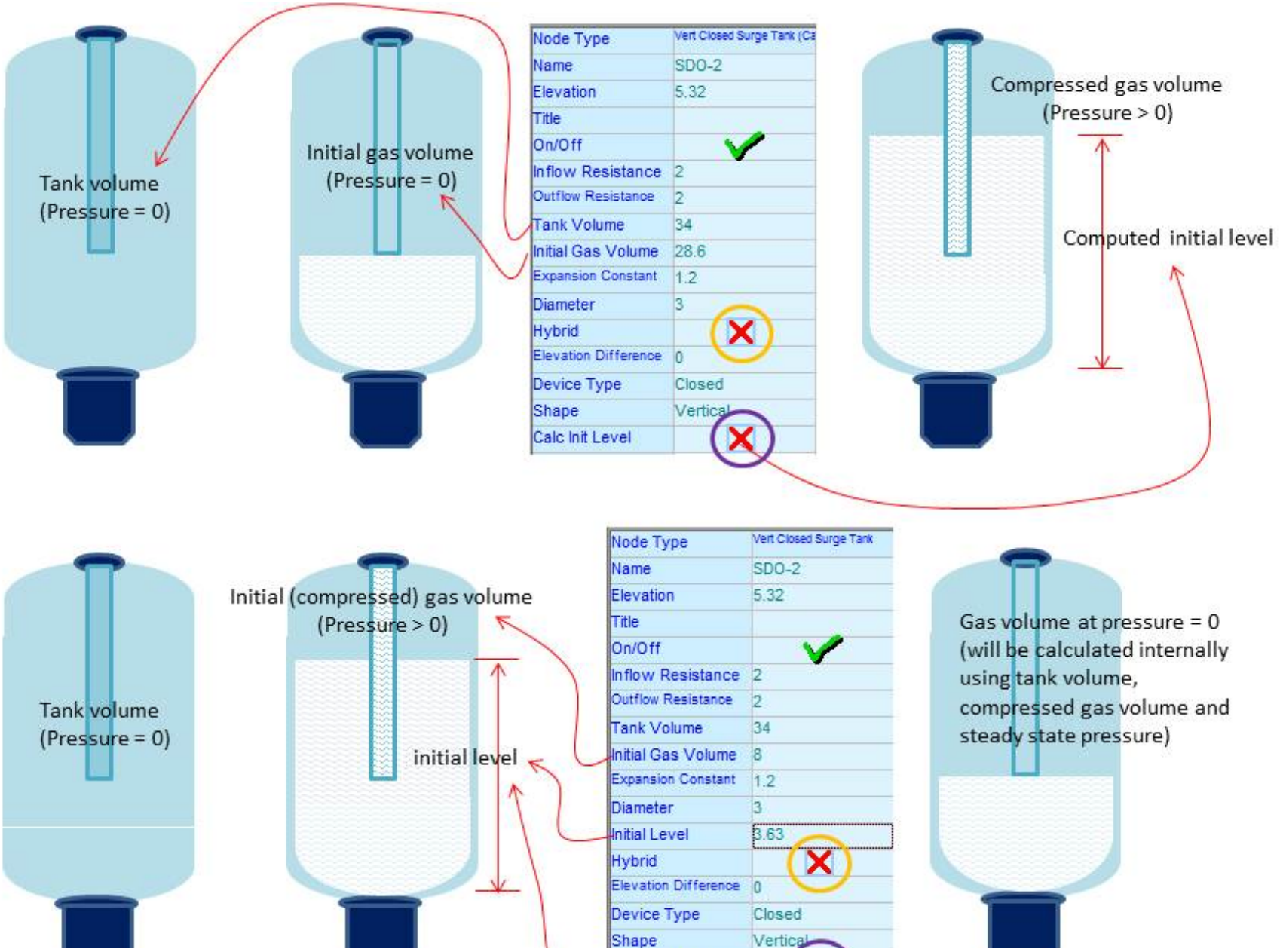


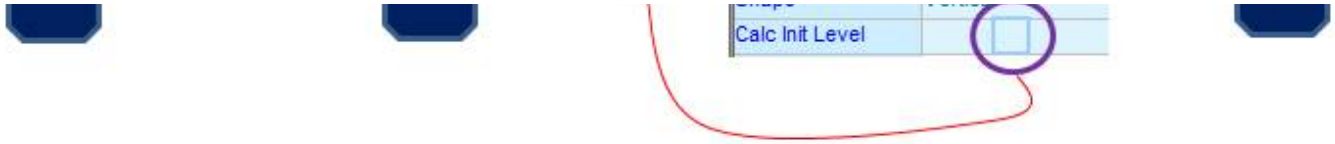


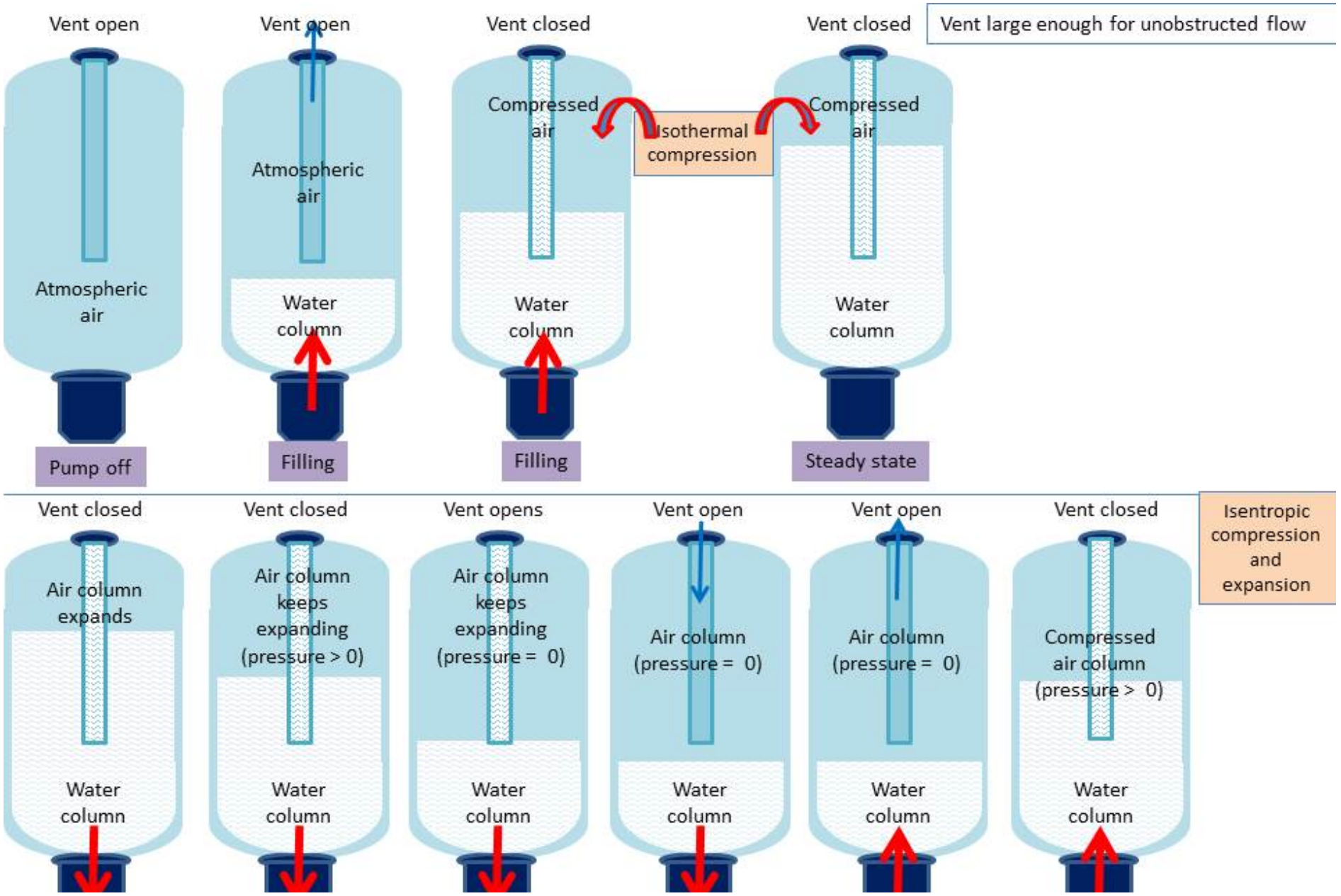

| Node Info | |
|---|---|
| Node Type | Hydraulically Actua |
| Name | HydSAV-2 |
| Elevation | 589 |
| Title | |
| On/Off | ✓ |
| Inflow Resistance | 100 |
| Outflow Resistance | 100 |
| Sensing Node | |
| Valve Type | Ball |
| Low Prs Pilot Hd | 50 |
| LPP Open Time | 5 |
| LPP Close Time | 10 |
| High Prs Pilot Hd | 75 |
| HPP Open Time | 3 |
| HPP Close Time | 15 |
| External Head | 0 |

### 1.5.5.7 Hybrid Surge Tanks

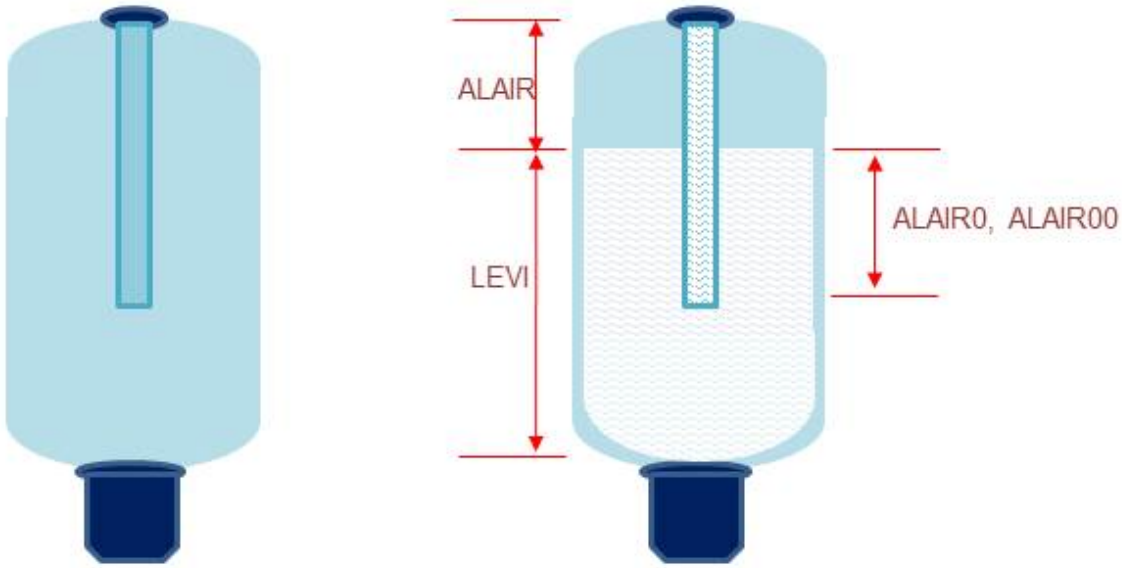
See also Surge Elements


Tank volume
(Pressure = 0)
Initial gas volume
(Pressure = 0)
Node Type
Vert Closed Surge Tank (Ca
Name
SDO-2
Elevation
5.32
Title
On/Off
Inflow Resistance
2
Outflow Resistance
2
Tank Volume
34
Initial Gas Volume
28.6
Expansion Constant
1.2
Diameter
3
Hybrid
Elevation Difference
0
Device Type
Closed
Shape
Vertical
Calc Init Level
Compressed gas volume
(Pressure > 0)
Computed initial level
Tank volume
(Pressure = 0)
Initial (compressed) gas volume
(Pressure > 0)
initial level
Node Type
Vert Closed Surge Tank
Name
SDO-2
Elevation
5.32
Title
On/Off
Inflow Resistance
2
Outflow Resistance
2
Tank Volume
34
Initial Gas Volume
8
Expansion Constant
1.2
Diameter
3
Initial Level
3.63
Hybrid
Elevation Difference
0
Device Type
Closed
Shape
Vertical
Gas volume at pressure = 0
(will be calculated internally
using tank volume,
compressed gas volume and
steady state pressure)


Calc Init Level


Vent open
Vent open
Vent closed
Vent closed
Vent large enough for unobstructed flow
Atmospheric air
Atmospheric air
Water column
Compressed air
Isothermal compression
Compressed air
Water column
Water column
Pump off
Filling
Filling
Steady state
Vent closed
Vent closed
Vent opens
Vent open
Vent open
Vent closed
Isentropic compression and expansion
Air column expands
Air column keeps expanding (pressure > 0)
Air column keeps expanding (pressure = 0)
Air column (pressure = 0)
Air column (pressure = 0)
Compressed air column (pressure > 0)
Water column
Water column
Water column
Water column
Water column
Water column

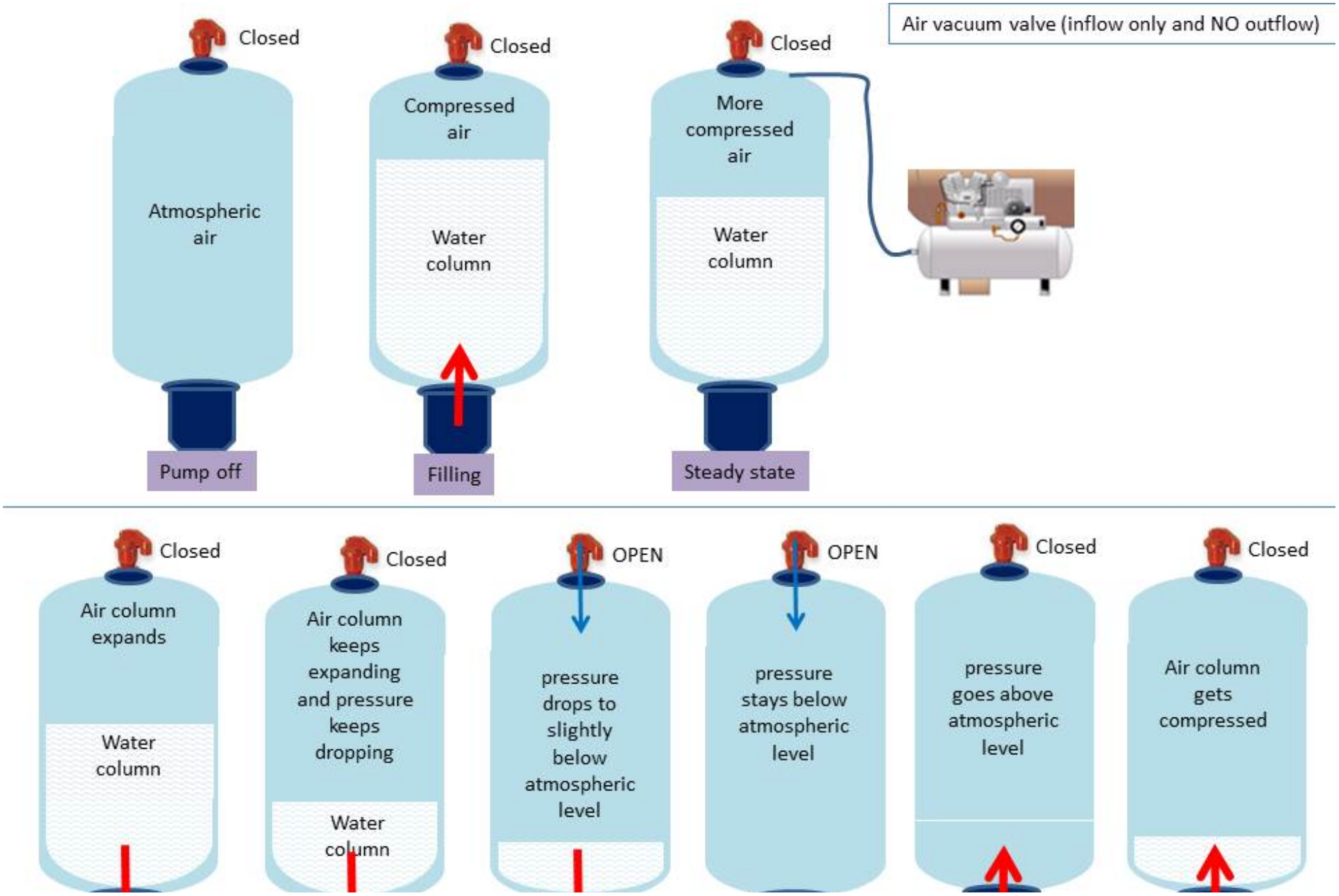
Pump trip
Closed surge tank
Down surge
Closed surge tank
Down surge
Open surge tank
Down surge
Open surge tank
Up surge
Open surge tank
Up surge
Closed surge tank




```
ALAIR0 = ALAIR * ((L5+HATM)/HATM)

ALAIR00= ALAIR * ((L5+HATM)/HATM)

LSWITCH1(I) = LEVI(I)+ALAIR-ALAIR0

LSWITCH2(I) = LEVI(I)+ALAIR-ALAIR00

LSWITCH(I) = LSWITCH2(I)
```

```
IF(LEVI(I).LT.LSWITCH1(I)) LSWITCH(I) = LSWITCH1(I)
….
….
….

IF(LEVI(I).LT.LSWITCH(I)) THEN
      IF(LSWITCHED(I).EQ.0) then
            LSWITCHED(I) = 1
      End if
      GOTO 6260

ELSE

      IF(LSWITCHED(I).EQ.1) then
```

```
            LSWITCHED(I) = 0
      End if
ENDIF
```


Air vacuum valve (inflow only and NO outflow)
Closed
Atmospheric air
Pump off
Closed
Compressed air
Water column
Filling
Closed
More compressed air
Water column
Steady state
Closed
Air column expands
Water column
Closed
Air column keeps expanding and pressure keeps dropping
Water column
OPEN
pressure drops to slightly below atmospheric level
OPEN
pressure stays below atmospheric level
Closed
pressure goes above atmospheric level
Closed
Air column gets compressed

## 1.5.5.8 Turbines

Turbines

---

This section provides help on modeling reaction turbines (Francis and Kaplan Turbines) in Pipe2018. Impulse turbines (Pelton Wheels) may be modeled using the active valve feature of the Pipe2018 program as the turbine itself will not have any influence on the transient pressures in the penstock.

Turbine characteristics are very site specific. Therefore, turbine data is not built-in. Pipe2018 allows for modeling turbine elements with controlled variation in speed as well load rejection modeling along with wicket element. The turbine characteristics data must be provided in Suter 4 quadrant format. A tool is provided (see below) that converts data in Hill chart format into the required format.

Suter representation of non-dimensional pump characteristics has been widely practiced to simulate transient pressures associated with pump elements in piping systems. To facilitate modeling of pumps working as turbines with the same set of non-dimensional characteristic curves, Pipe2018 employs the same Suter representation to model turbines, pumps and pumps as turbines (PATs) elements. If the turbine data is available in a format other than the Suter representation, then the available turbine data must be converted into Suter format. One popular format for collecting/representing turbine data is in the form of a Hill chart. The basic Hill charts comprise two plots, one relating unit discharge ($Q_{11} = Q/(D^2H^{0.5})$ with unit speed ($N_{11} = ND/H^{0.5}$) and the other relating unit torque ($T_{11} = T/(D^3H^{0.5})$ with unit speed. Suter representation of pump characteristics relates $h/(a^2 + v^2)$ with ð + $\tan^{-1}$ v/a and relates $w/(a^2 + v^2)$ with ð + $\tan^{-1}$ v/a, where $h = H/H_R$, $w = T/T_R$, $a = N/N_R$ and $v = Q/Q_R$ and subscript R represents the machine rated conditions. Pipe2018 comes with a tool (Turbine Data Converter under Tools in the main menu) that converts turbine data in Hill charts format into a Pipe2018 Suter file format. Turbine data may get further complicated when the turbine operation is controlled by a wicket gate. Turbine characteristic data at different wicket gate openings should also be converted into Suter file format for modeling wicket gate operation in the Pipe2018 program. Once again the Hill charts conversion (Turbine Data Converter) tool may be used to convert the data at different gate openings into Suter File format.

**Turbine Data Converter tool:**

The available turbine characteristics data (Hill charts) must first be written to an Excel Spreadsheet file in the following format and the file must be saved as a CSV (comma separated variable) file. Table 1 shows the sample file format for the raw data file. An example file, Example-1.CSV, is available to demonstrate the complete file format. The conversion tool comes with a plotting routine to visualize the raw data (data in Hill charts format) as well as the processed data (data

generated in Suter files format). Figure 1 shows a screen shot from the Hill chart conversion tool along with a plot of $h/(a^2 + v^2)$ vs ð + $\tan^{-1}$ v/a for several different wicket gate openings. Two separate files are generated by this tool, one file (999X) at full wicket gate opening in the standard Suter file format used by Pipe2018 program and other file (888X) at full and all other wicket gate openings. The X in 999X and 888X represents the file number specified by the user in the Hill Chart conversion tool as well as the file number associated with the pump/turbine element (****under device data of Pipe2018). Table 2 shows the format for 888X file.

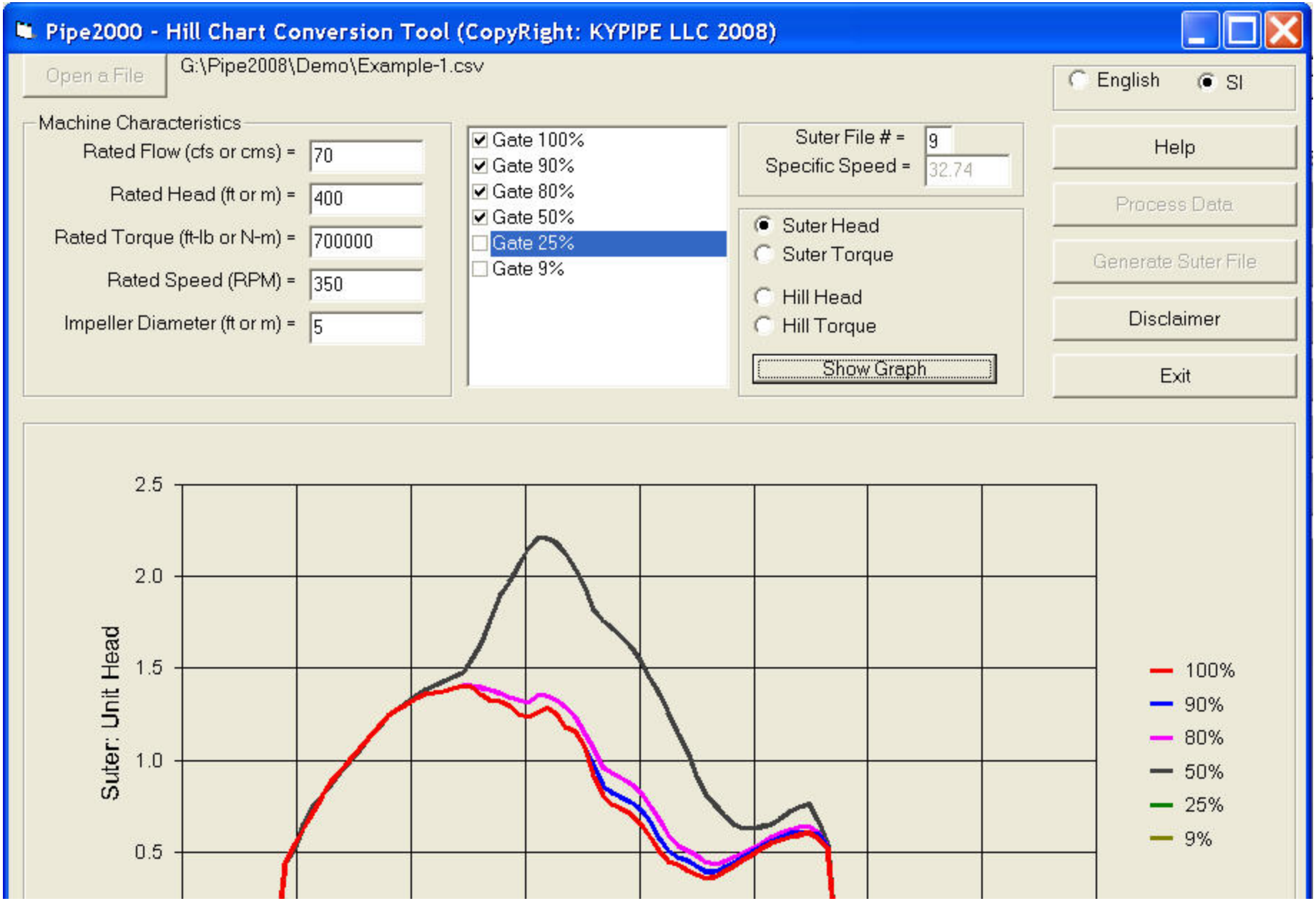
Table 1. Turbine data in Hill charts format

| | | |
|---|---|---|
| Diameter (ft or m) | | |
| 5.000 | | |
| Rated Head (ft or m) | | |
| 400.000 | | |
| Rated Flow (cfs or cms) | | |
| 70.000 | | |
| Rated Torque (ft-lb or N-m) | | |
| 700000.000 | | |
| Rated Speed (RPM) | | |
| 350.000 | | |
| Guide Vane Opening (%) | | |
| 9.000 | | |
| N11, Q11, T11 | | |
| -102.718 | -0.041 | 107.383 |
| -96.675 | -0.036 | 93.312 |
| -90.633 | -0.031 | 78.501 |
| ….. | …. | …. |
| ….. | ….. | ….. |
| 114.940 | -0.022 | -105.158 |

| | | |
|---|---|---|
| 120.990 | -0.025 | -120.181 |
| 127.039 | -0.029 | -139.710 |
| 133.089 | -0.033 | -165.249 |
| **-99999** | | |
| 25.000 | | |
| -102.717 | -0.103 | 177.737 |
| …. | …. | …. |
| …. | …. | ….. |
| 108.891 | -0.027 | -103.656 |
| 114.941 | -0.031 | -123.936 |
| 120.990 | -0.034 | -146.470 |
| 127.040 | -0.039 | -172.760 |
| 133.089 | -0.045 | -206.561 |
| -99999.000 | | |
| -99999.000 | | |

In Suter representation, the value of è (where è = ð + $\tan^{-1}$ v/a) ranges from 0 to 2ð and it covers all possible modes of operation of a hydraulic machine during a transient event. While pumps may go through almost all modes of operation during a severe transient event, the modes of operation of turbines generally do not span more than 3 different zones. From this point of view, it may not be necessary for the Hill charts to cover all 4 quadrants completely. In most cases the data converted into Suter representation from Hill charts (of turbines) will not cover entire possible range of operation, that is è = 0 to ð. However, Pipe2018 expects the Suter files to cover the entire range of è = 0 to ð. Instead of ignoring (or setting to 0) the data outside the ranges covered by Hill charts are replaced by the data from standard Suter Pump Files available with Pipe2018 program. The Hill charts conversion tool will interpolate the available Suter Pump Files if the machine specific speed does not match with one of the existing pump files.

The non-dimensional Suter curves tend to be much steeper at lower wicket gate openings. Small measurement errors in raw data (Hill charts) may produce significant distortions in the Suter files data. The raw data may be adjusted to minimize the distortions in the Suter data before using the Suter files with the Pipe2018 program.


Pipe2000 - Hill Chart Conversion Tool (CopyRight: KYPIPE LLC 2008)
Open a File
G:\Pipe2008\Demo\Example-1.csv
English
SI
Machine Characteristics
Rated Flow (cfs or cms) = 70
Rated Head (ft or m) = 400
Rated Torque (ft-lb or N-m) = 700000
Rated Speed (RPM) = 350
Impeller Diameter (ft or m) = 5
Gate 100%
Gate 90%
Gate 80%
Gate 50%
Gate 25%
Gate 9%
Suter File # = 9
Specific Speed = 32.74
Suter Head
Suter Torque
Hill Head
Hill Torque
Show Graph
Help
Process Data
Generate Suter File
Disclaimer
Exit
Suter: Unit Head
2.5
2.0
1.5
1.0
0.5
100%
90%
80%
50%
25%
9%

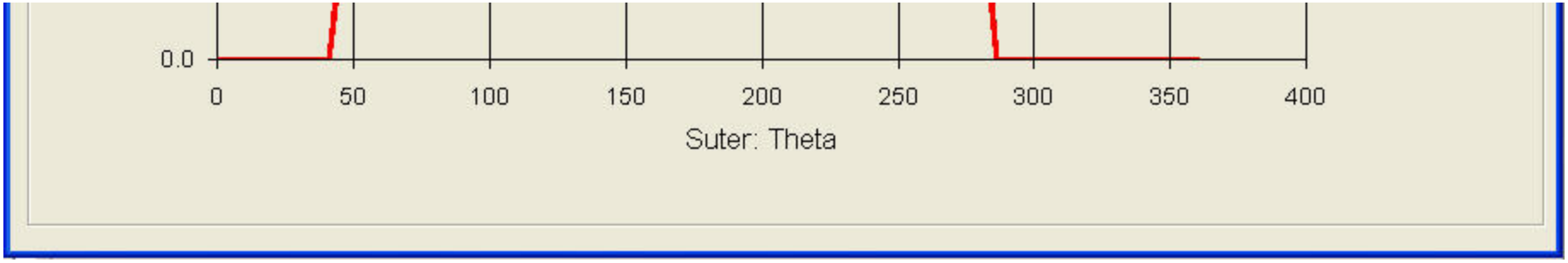



Figure 1. Screen shot from Hill Charts Conversion Tool

Table 2. Format for 888X file.

Line 1: Wicket gate opening (%)
Lines 2 – 90: 89 data points representing $h/(a^2 + v^2)$ at the above wicket gate opening for each è (where è = ð + $\tan^{-1}$ v/a ) at ð/44 interval.
Lines 91 – 180: 89 data points representing $w/(a^2 + v^2)$ at the above wicket gate opening for each è at ð/44 interval.
Line 181: -999
Repeat lines 1-181 for additional wicket gate openings
Last line: -999 indicating end of data file.

Example 1 (Turbine Ex1.p2k). This example illustrates the modeling of a simple pipeline system with a turbine element in Pipe2018 program. The Pipe2018 model (Turbine Ex1.p2k) for this example is in the in the Turbines Data folder. Figure 2 shows the schematic for the pipeline system from the upstream reservoir to tailrace on the downstream end.

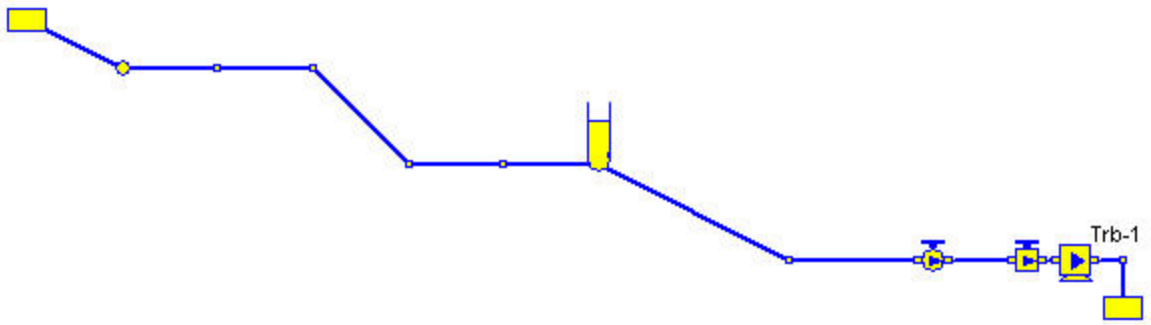
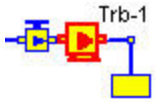



Figure 2. Schematic of the pipeline system for Example-1

The penstock is a roughly 2.5 km long 2.5m diameter steel pipeline. The turbine is rated to operate at 64$m^3$/s of flow and 460m of head. Rated speed is 425 RPM. The combined Generator and Turbine inertia for this system is 19620000N-m. A 9999 file has been derived from the available Hill charts for this turbine and are stored in the Pipe2018 folder. Figure 3 shows the device data for the Turbine element. The turbine full load rejection event has been simulated in this example. Figure 3 also shows the change data associated with the turbine element illustrating the data needed to simulate the load rejection or trip event under full load operation. Note the negative sign for the initial speed ratio as well as for the speed ratio value under change data at time 0. Negative speed ratio for the turbine element indicates that the device is operating in the turbine (generating) mode. This ratio should be positive if the turbine element starts as a pump in a pumped storage plant. The data under device data refers to the machine characteristics while it is operating as a turbine. Figures 4, 5 and 6 show the pressure head, speed and flow variations respectively, following full load rejection at and near turbine element.




Name Trb-1
Turbine
Elevation 1240
Sp Ratio -1
Effcny% 89

**Device Data**

File (1-20) 9
Rated Hd 460
Rated Flw 64
Rated Spd 425
Inertia 19.62e

**Node Changes**

Cut Copy Paste Clear

| Time/ Case | | Value |
|---|---|---|
| 0 | s | 1 |
| 2 | s | 1 |
| 2 | t | Trip |

**Node Info**

| Node Type | Turbine |
|---|---|
| Name | Trb-1 |
| Elevation | 1240 |
| Title | |
| Efficiency % | 89 |
| Speed Ratio | -1 |
| Grade | ----- |
| File# (1-20) | 9 |
| Rated Head | 460 |
| Rated Flow | 64 |
| Rated Speed | 425 |
| Inertia | 19.62e6 |
| Direction | |
| On/Off | |

**Node Changes**

| Time | Change | Value |
|---|---|---|
| 0 | speed ratio | 1 |
| 2 | speed ratio | 1 |
| 2 | trip | Trip |
| | | |

Figure 3. Device data and change data for the turbine element – Example 1.

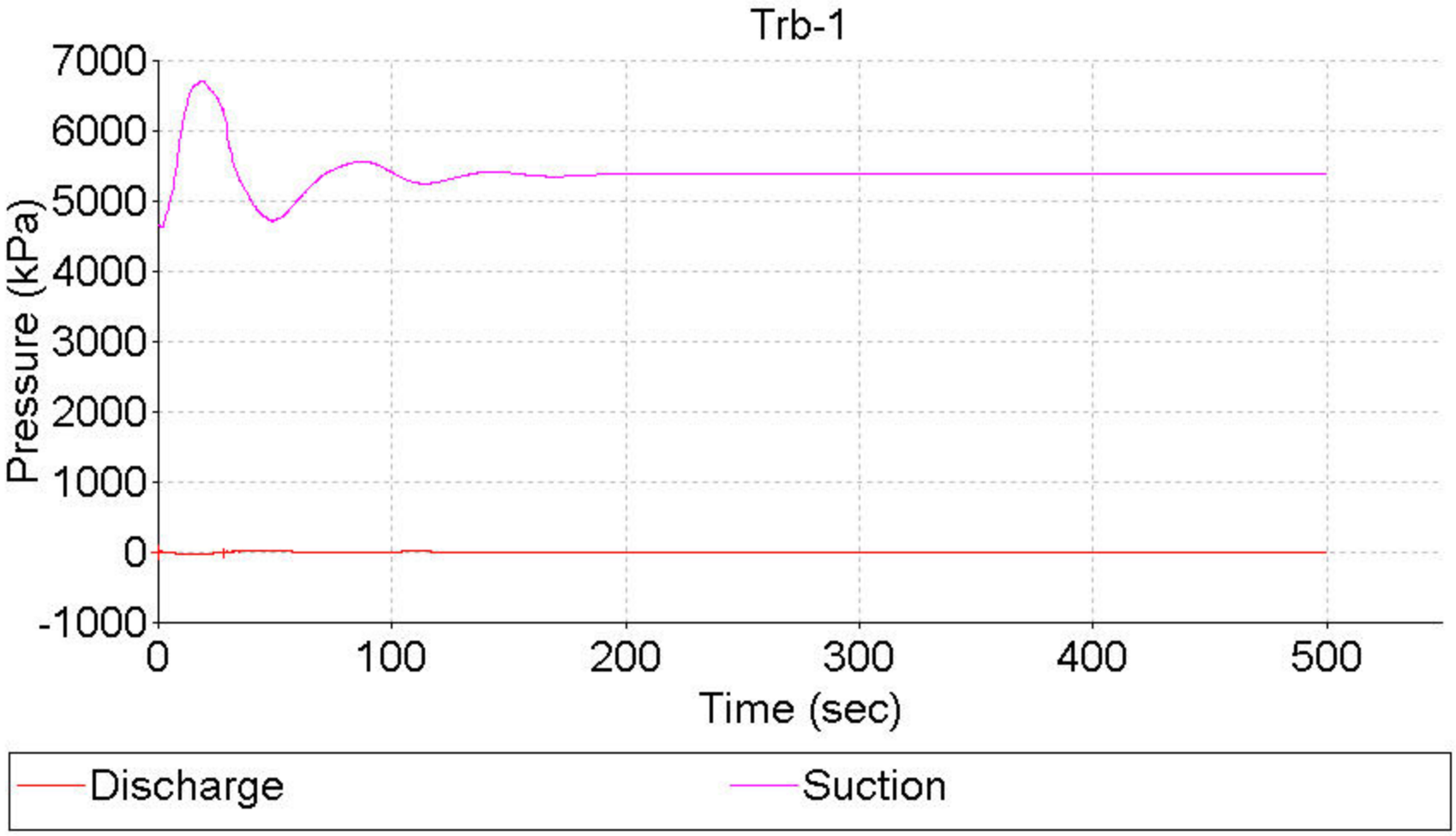



Figure 4. Pressure head trace at turbine element (purple line - downstream side)

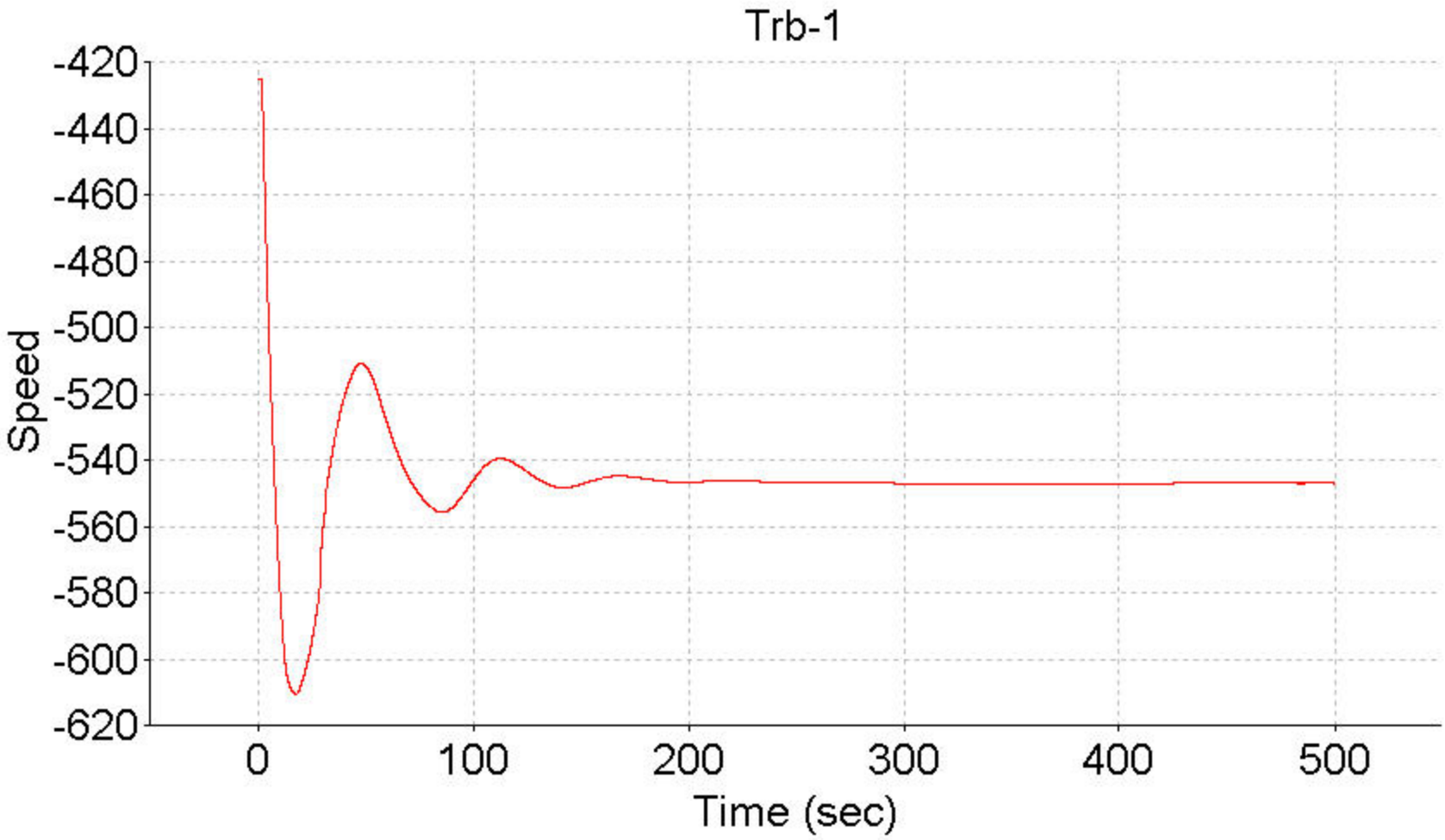



Figure 5. Variation of Turbine Speed following full load rejection

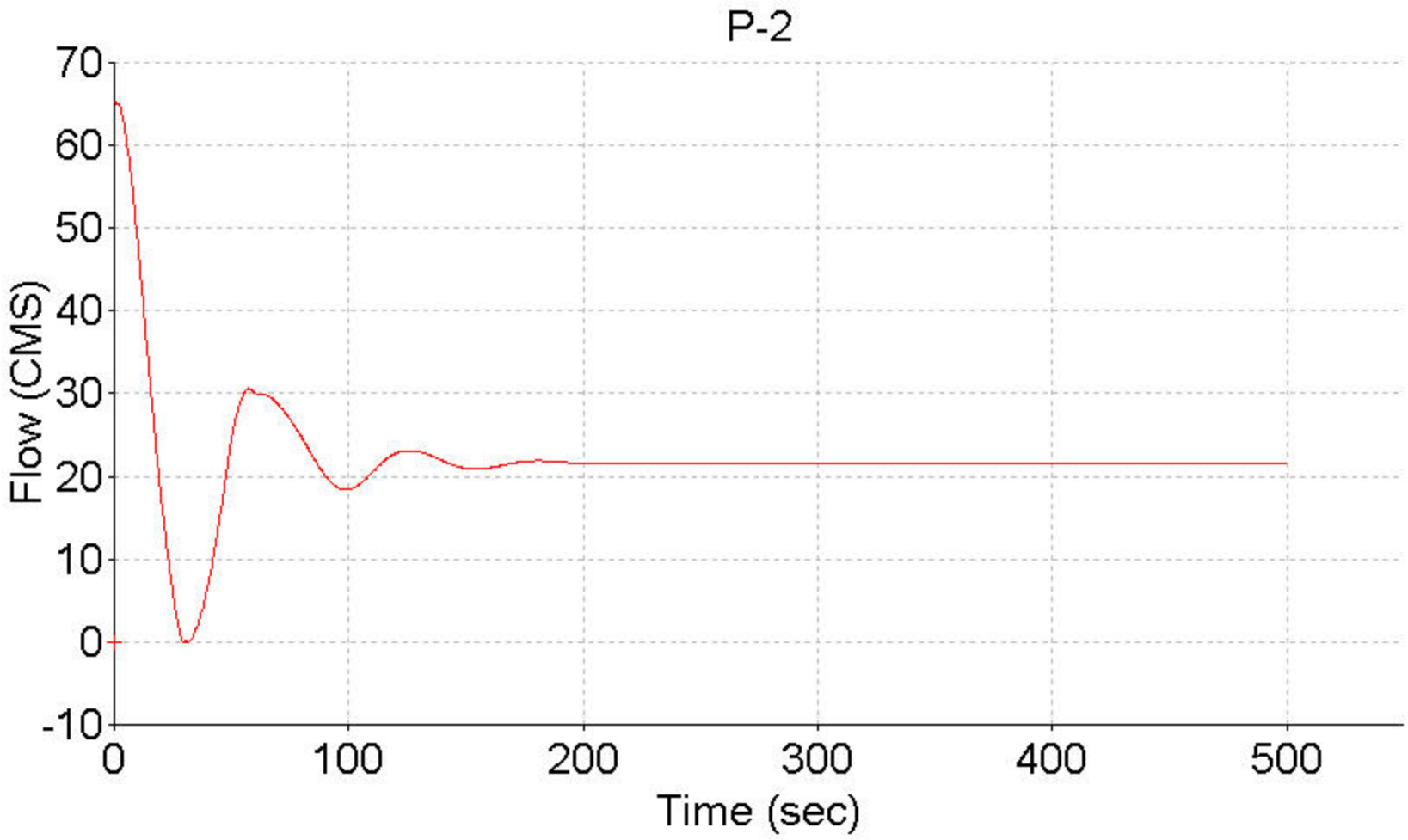



Figure 6. Variation of Flowrate at Tailrace following Load Rejection

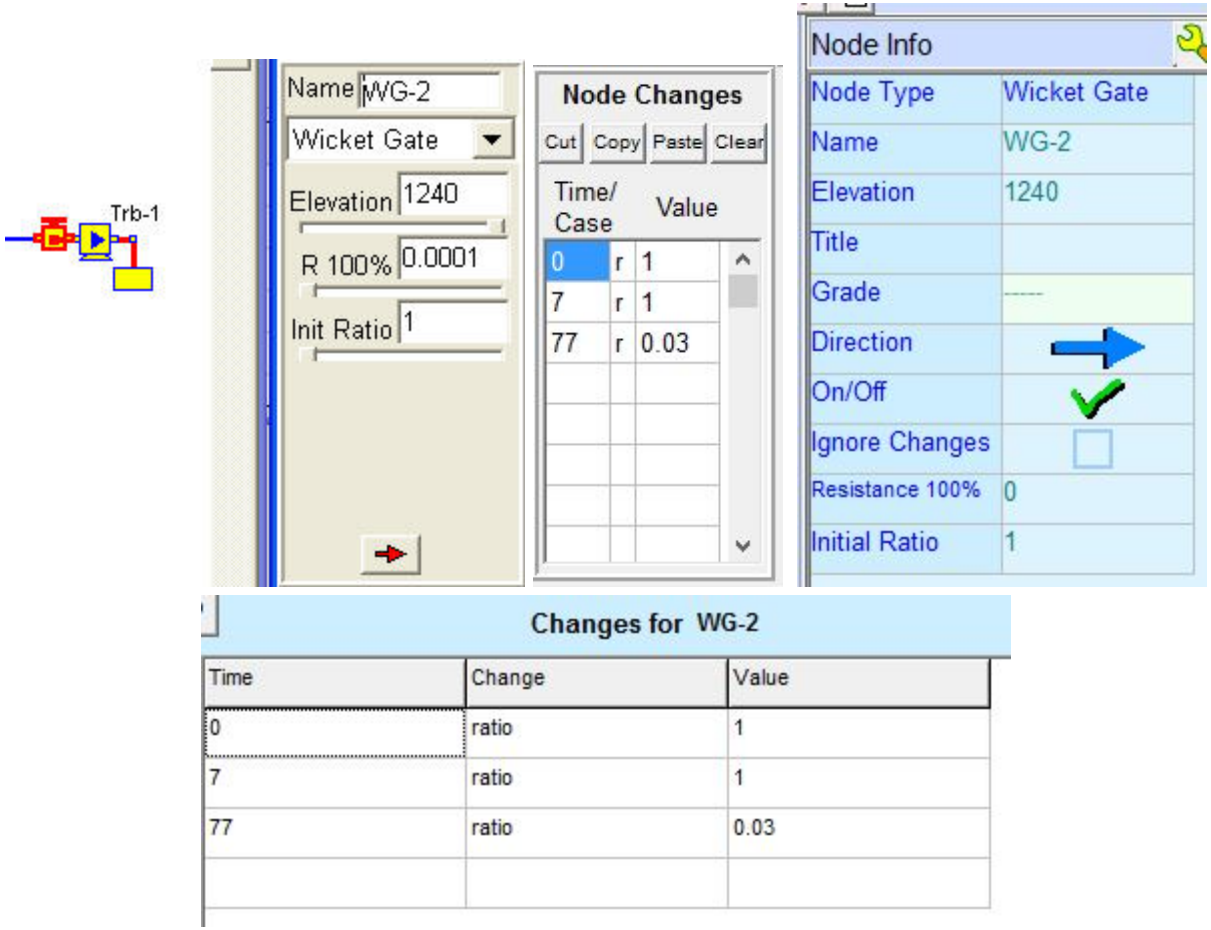



Figure 7. Wicket Gate Characteristics and Change Data

Example – 2 (Turbine Ex2.p2k): This is a continuation of the previous example wherein a wicket gate immediately upstream from the turbine element starts closing 5 seconds after load rejection and completely closes in 70 seconds. Though the wicket gate element was created for the previous example, no change data was provided for the wicket gate and therefore it would act as a simple active valve with a small resistance. It is important to note that a wicket gate element must be located immediately upstream from the turbine element separated by a very short length pipeline.

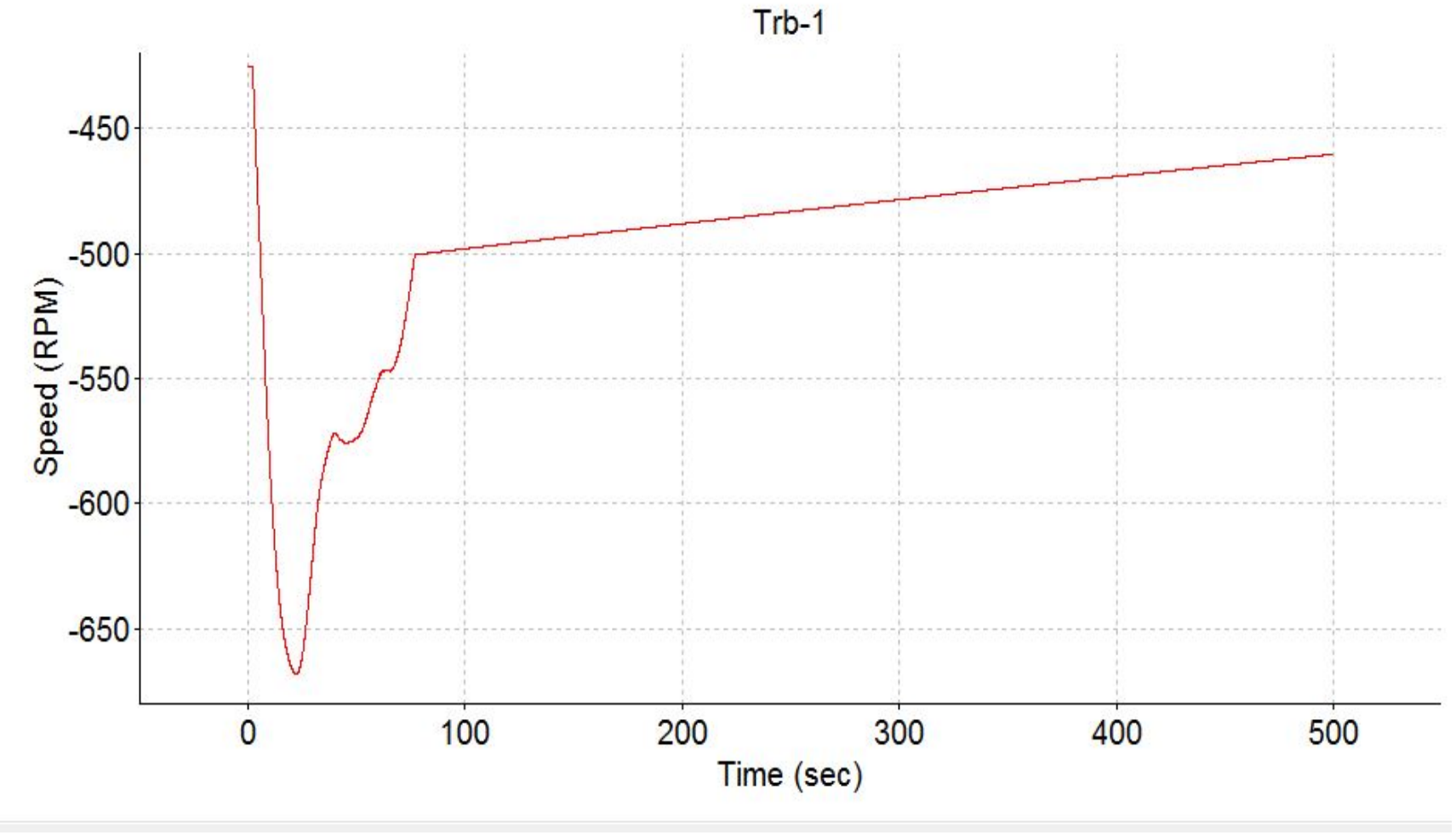



Figure 8. Variation of Turbine Speed following full load rejection with wicket gate closure

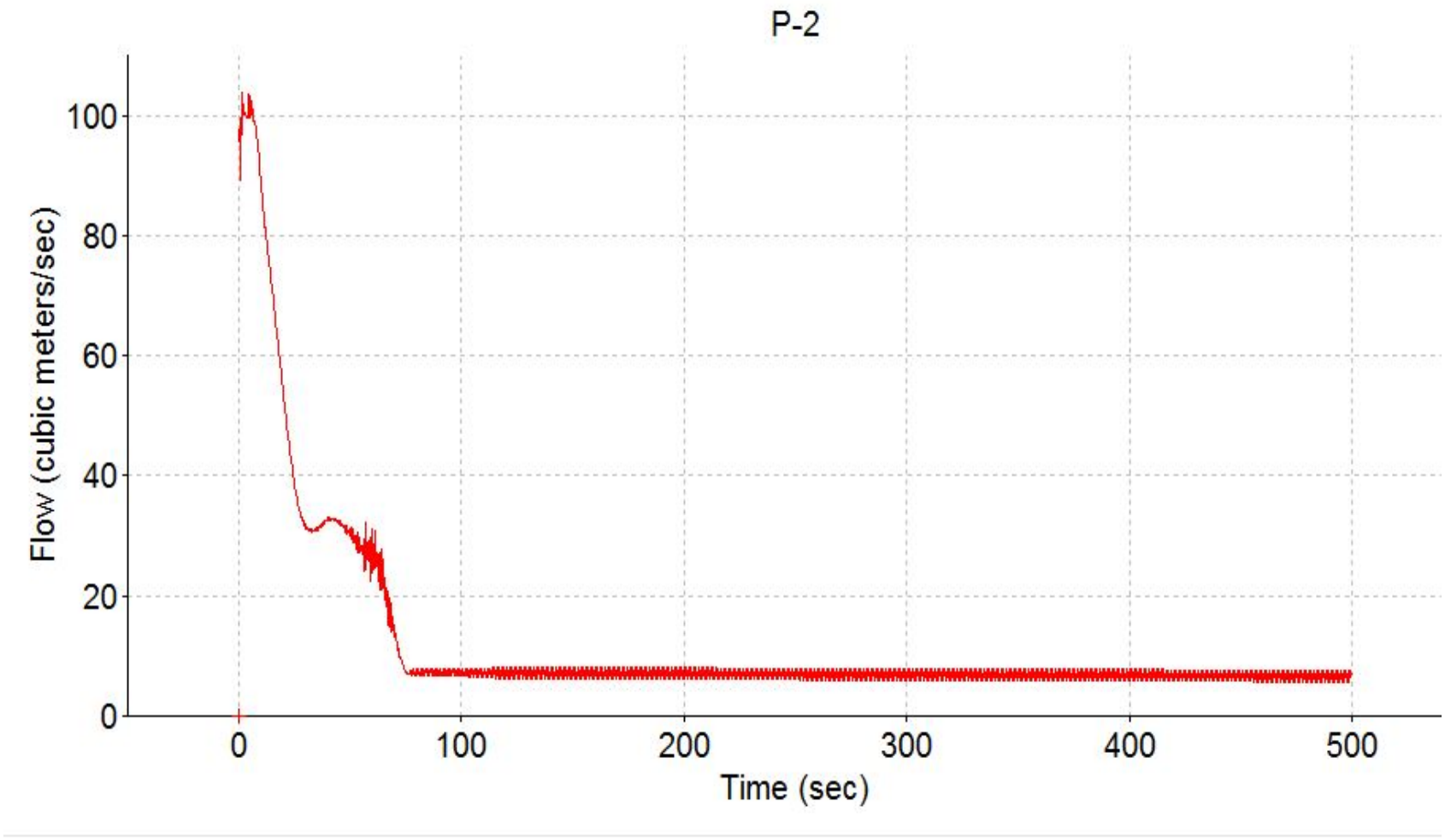



Figure 9. Variation of Flowrate at Tailrace following Load Rejection – wicket gate closes after load rejection

Figures 8 and 9 show the variation in turbine speed and flowrate following load rejection and closing of wicket gate. Because of the difficulties in getting the accurate turbine characteristic data at low wicket gate openings, the lowest gate opening with available turbine characteristic data was only 3.5% and not 0%. This was the reason the flow rate staying above zero despite completely closing the wicket gate – no data was available that correctly represents wicket gates in fully closed position.

Example 3 (Turbine Ex3.p2k): Turbine Trip Followed by Restart. Figure 10 illustrates the change data to simulate turbine startup event following a load rejection event. The first part of the change data is similar to Example – 1. Following the trip event, the wicket gate closes by 75% in 70 seconds before the turbine was restarted. The 200 in the change data is a dummy event where in the speed at that time period is captured from the transient run and is used as a starting speed at that time period. The corresponding zero under the value column of change data is ignored – instead the program calculates the speed ratio internally at that time period. Starting at the internally calculated speed ratio, the turbine speed will be ramped up to the rated speed in 20 seconds as illustrated in the change data (Figure 10). Figure 11 shows the speed variation graph and Figure 11 shows the pressure head variation graph for this example.

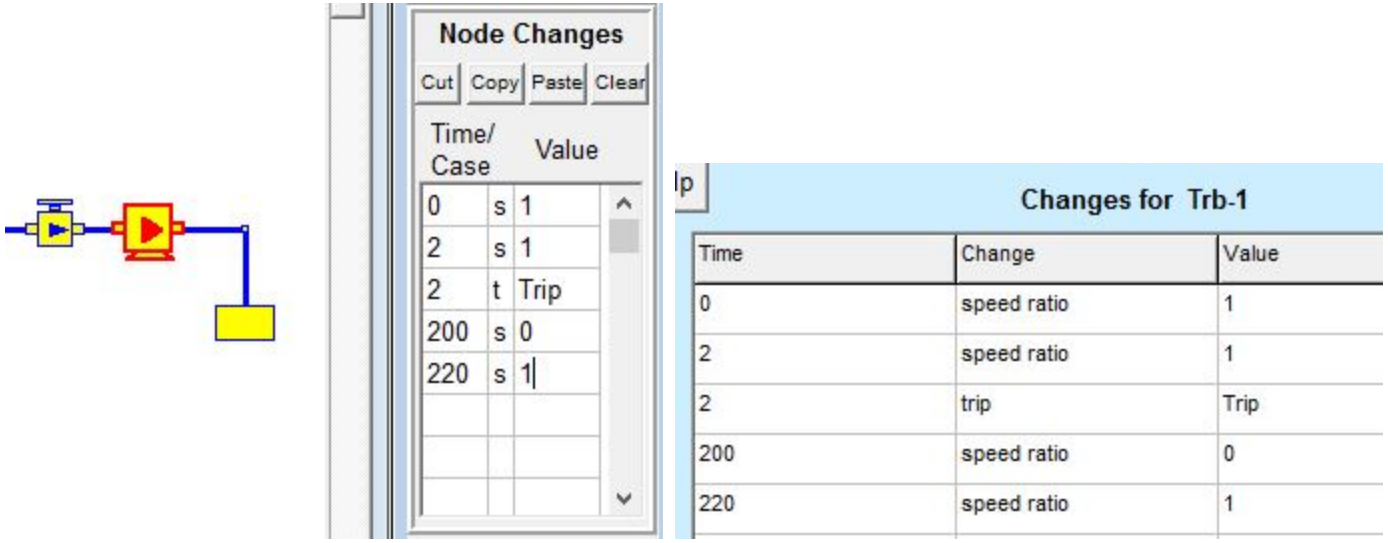



Figure 10. Change data for turbine element

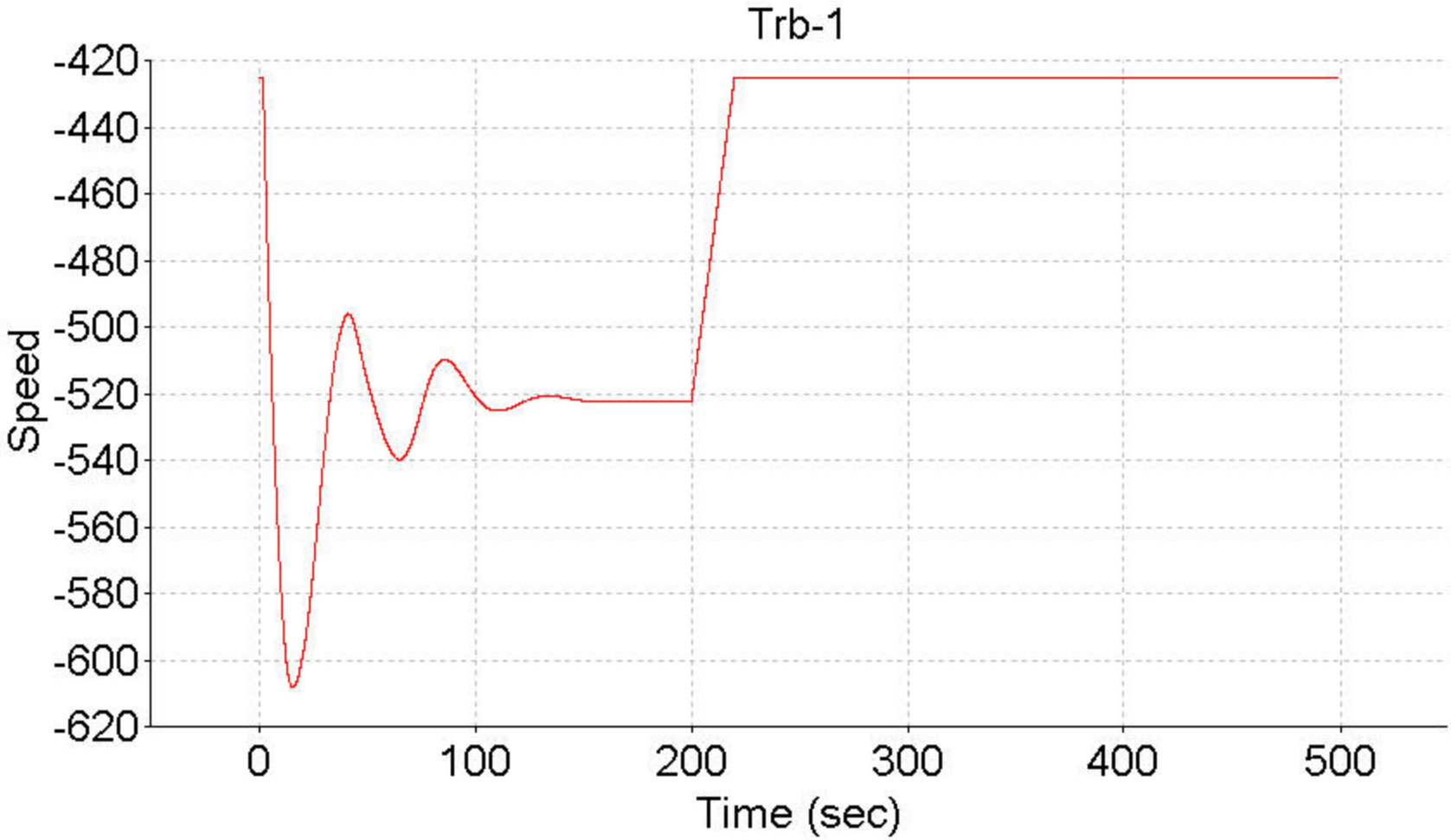



Figure 10. Speed Variation after a turbine trip event followed by a restart event

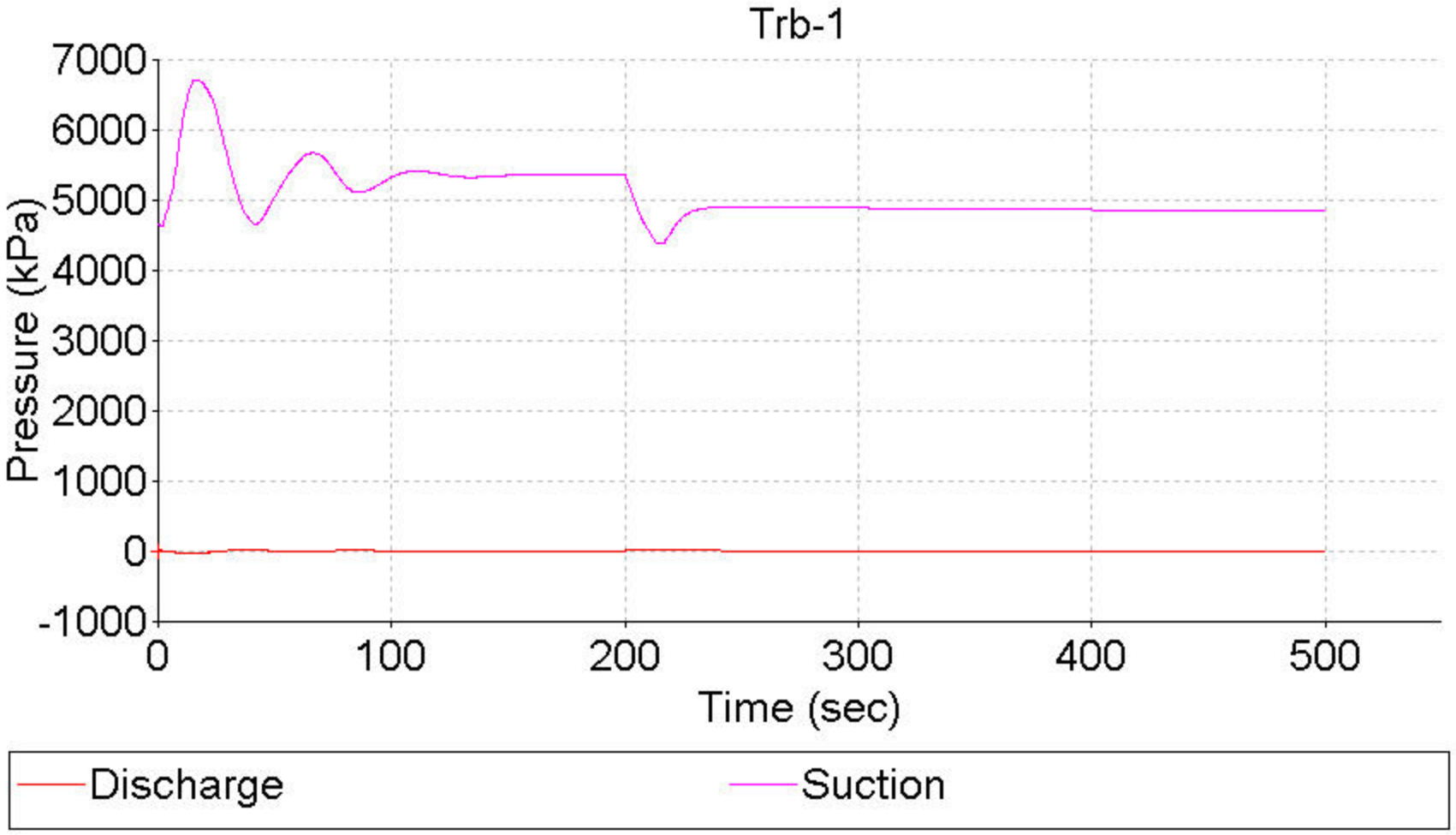



Figure 11. Head Variation after a turbine trip event followed by a restart event

## 1.5.5.9 Zero Velocity Valves

**Zero velocity valves (ZVV):**

**Definition:** A valve that closes exactly when the flow velocity in the pipeline becomes zero.

**Purported Uses:** To be installed at strategic locations on rising mains to prevent reversal of flow after a pump trip event thereby protecting the pipeline from high surge pressures.

**Discussion:** ZVV is a non return valve placed at some strategically selected locations along the pipeline while traditional NRVs are located just near the pump station. The main purpose of both ZVVs and NRVs is same: to prevent reversal of flow through pipeline, except that one is to mainly protect the pump station (NVR) and the other is to protect the pipeline from high positive surge pressures (ZVV). Though the main purpose of NRV is to protect the pump station from flow reversal, an ideal NRV must also close when the flow velocity is exactly at zero to prevent secondary pressure surges.

The need for closing ZVV (or NRV) exactly when the flow velocity is zero arises from the fact that any rapid changes in velocity can result in rapid changes in pressures in accordance with Joukowski's equation:

$$\Delta H = (c/g)\, \Delta V$$

Where $\Delta H$ is change in pressure (m), c is celerity or wave speed (m/s), g is gravitational acceleration ($m/s^2$) and $\Delta V$ is change in velocity (m). For steel pipes, c/g is roughly 100 implying, a $\Delta V$ of 1.0m/s can produce a $\Delta H$ of 100m or 10bars.

In cases where the change in velocity ($\Delta V$) was generated by a valve closure event, there will be an increase in pressure by $\Delta H$ on one side of the valve where the velocity decreases by $\Delta V$ and there will be a corresponding decrease in pressure on the other side of the valve because of the separation of water column. Therefore, if the ZVV is not set to close exactly when the velocity is zero, there is a potential for a sharp pressure rise in proportion to the change in velocity (at the time of complete closure) on one side of the ZVV and there would be a corresponding sharp decrease in pressure on the other side.

Suppose the residual velocity in the pipeline at ZVV is only 0.1m/s just before the ZVV is closed completely, the complete closure of the ZVV generates about 10m of positive surge pressure on one side of ZVV (downstream side of ZVV on a typical rising main) and a negative surge pressure of -10m on the other side of ZVV. Even if we ignore a 10m of positive surge pressure we cannot ignore the negative surge pressures as they can potentially lead to cavitation conditions with subsequent collapse of vapor cavities leading to potentially unsafe positive surge pressures. If the ZVV closes completely when the pressure is already negative at upstream end of pipeline, cavitation conditions can occur even at smaller residual velocities ($\Delta V<0.1$ m/s). Such small thresholds on residual velocities make the ZVVs extremely sensitive elements of the surge protection system. Their real-life performance must be very close to how they were simulated in transient modeling of the rising main. Any deviations in its performance at the initial stages of operation or during the lifetime of pipeline project may result in extreme surge pressures capable of inflicting widespread damage to the pipeline system.

Theoretically, it is easy to assume that the valve (ZVV) closes just when the flow velocity in the rising main near ZVV becomes zero. Most computer modeling tools for surge analysis assume that the valve (ZVV) closes just when the flow velocity becomes zero. But one must examine if it is physically possible to have the ZVV close exactly when the flow velocity becomes zero or close to zero. The important issue is how a ZVV operates in real-life and responds to changing flow conditions in a rising main.

Two important questions to be addressed for proper functioning of a ZVV on a rising main are: 1. What triggers the closing of ZVV? and 2. How much time it takes to close the ZVV from fully open position to fully closed position?

In case of single or multi-door ZVV that open and close based on fluid pressure (and no external energy), the flaps start closing as soon as the pressure force drops below the force required to hold the flap in open position. Depending on the type of flaps used and orientation of ZVV on the rising main, the flaps may start closing 1.exactly when the flow velocity becomes zero, 2. slightly before the flow velocity becomes zero, or 3. slightly after the flow velocity becomes zero. Once the ZVV starts closing, it will take some finite time to go from fully open mode to fully closed mode. Depending on the actual closure characteristics of the ZVV and the flow characteristics of the rising main, the exact time for closure may vary and it could be as high as 1 second (or more). Flow continues to accelerate in the reverse direction during this period (while the ZVV is going from fully open to fully closed mode) implying the potential for unacceptable levels of residual flow velocity just when the ZVV closes completely. As discussed earlier, even a residual flow velocity of 0.1m/s can potentially generate cavitation pressures on the upstream side of the ZVV which in turn can result in significantly large positive surge pressures on collapse of vapor cavities. The reverse velocities can be much higher than 0.1m/s in pipeline systems where the water column accelerates or decelerates rapidly. Another critical aspect of single or multi-door ZVVs is that the speed with which the flaps close increases as the valve approaches completely closed state. This is due to increase in fluid pressure force on the flaps in the direction of reverse flow as the flaps approach closed state. Such an accelerated closing of ZVV further enhances the chances of higher velocity changes ($\Delta V$) at the time of complete closure of ZVV resulting in more severe transient pressures.

In case the ZVVs are fitted with external source of energy (hydraulic power pack) to control the closing of flaps, the actual closing time of ZVV

increases compared to those that work purely based on fluid pressure. This defeats the purpose for using the ZVV (i.e. prevent flow reversal in the pipeline to protect high pressure problems near pump station) and can also generate unwanted pressure spikes within the pipeline due to ZVV closure at higher reverse velocities. It is difficult to capture the actual closure characteristics of a ZVV fitted with external energy source in order to simulate those characteristics in surge modeling study. The behavior of hydraulic power pack can be affected by the dynamic pressure forces during the flow reversal cycle. The characteristics of a power pack tested under static conditions (on rising main or in factory) could be completely different from those under dynamic conditions and get much more complex when there is significant wave action within the rising main (for example, air slam conditions etc). Dependency on fine tuning the closure characteristics under dynamic conditions is not acceptable as the damage to the pipeline or pump station (during the trial and error adjustment process) can be severe and irreversible. An ideal NRV is one that closes right when the flow becomes zero (just before flow reversal) or the one that closes very slowly to prevent secondary pressure surges associated with reverse flow velocities. However, an ideal ZVV is one that closes only when the flow becomes zero. The one that closes very slowly is NOT an option as it defeats the purpose of having a ZVV on the pipeline. The delayed closing of ZVV may ensure that the local secondary pressure surges are minimized but it lets the reverse flow velocities to develop and create enough damage to the pump station and other parts of the pipeline. Besides, this is with the assumption that one can model the actual closing characteristics of hydraulic ZVV more accurately which could be a difficult task. One other critical issue with hydraulic ZVV is its triggering mechanism to initiate the closure of flaps, in case the flaps are directly connected to the hydraulic power pack. There should be some elaborate and reliable sensor network to sense the flow reversal and initiate the closing operation. Wrong flow reversal signals can create more damage to the pipeline.

Other issues that affect ZVV functionality are, multiple pump operation, controlled shutdown of pump stations, pump startup related transients. Assuming that the designers were able to arrive at an ideal closing characteristics for the ZVV based on a full pump trip event (power failure resulting in tripping all pumps), the water deceleration characteristics may be different during other transient events such as single pump trip during a multiple pump operation etc. The deceleration will be different during a controlled shutdown of a pump station. There should be a mechanism to adjust the closure characteristics for each of the operating scenarios which is almost impossible.

In view of these observations, though the ZVVs offer theoretically a good solution to protect rising mains from returning pressure waves, the practical implementation is a highly complex process and any deviations can lead to catastrophic damages to the pipelines and at times to people.

## 1.5.6 Surge Pipe Data (Wave Speed)

Surge Pipe Data (Wave Speed)
See also Surge Elements - Pipes

What is Wave Speed?
Wave Speed Calculation Tool
Displaying Wave Speed Data.

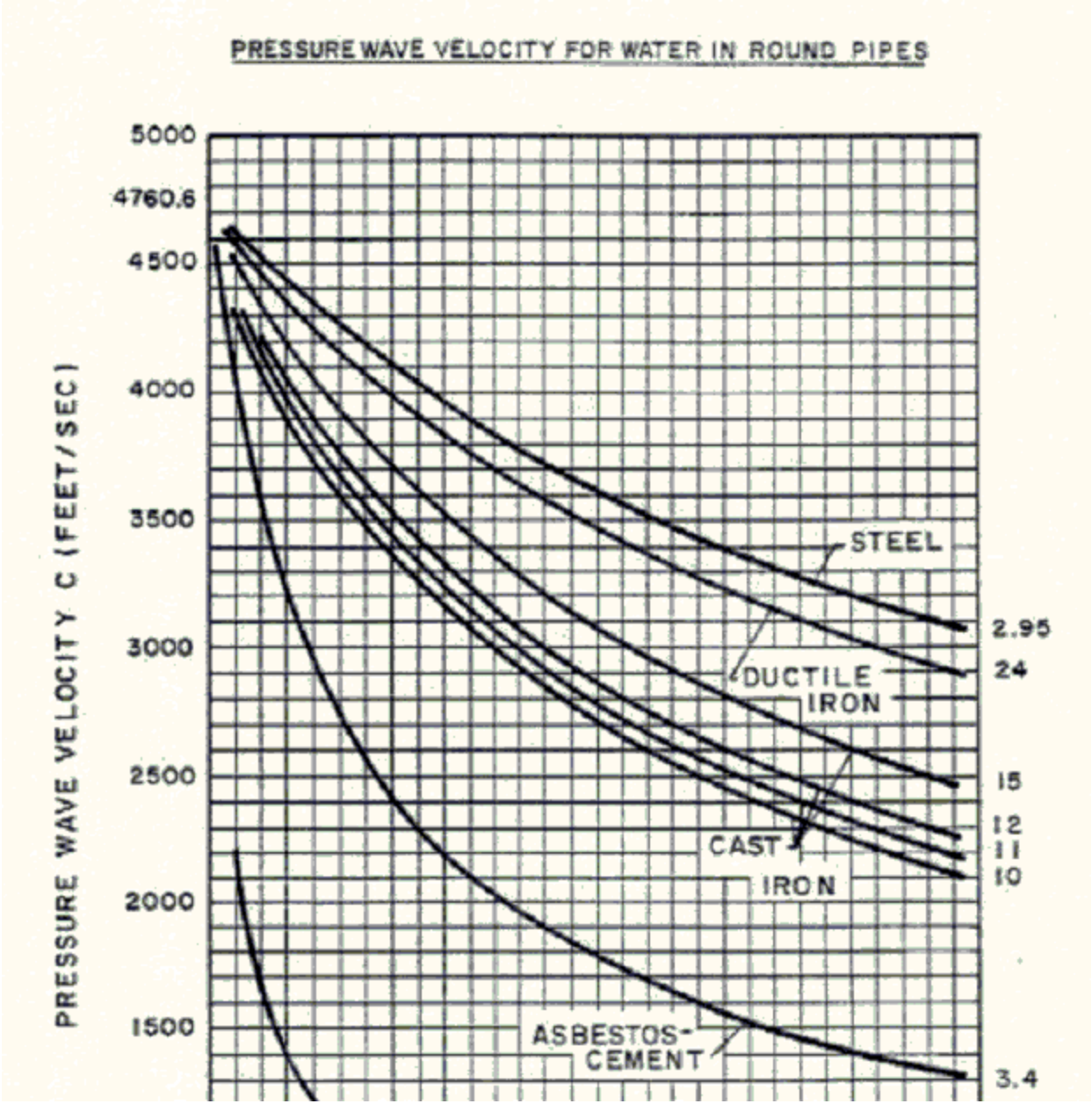
Defining Wave Speed

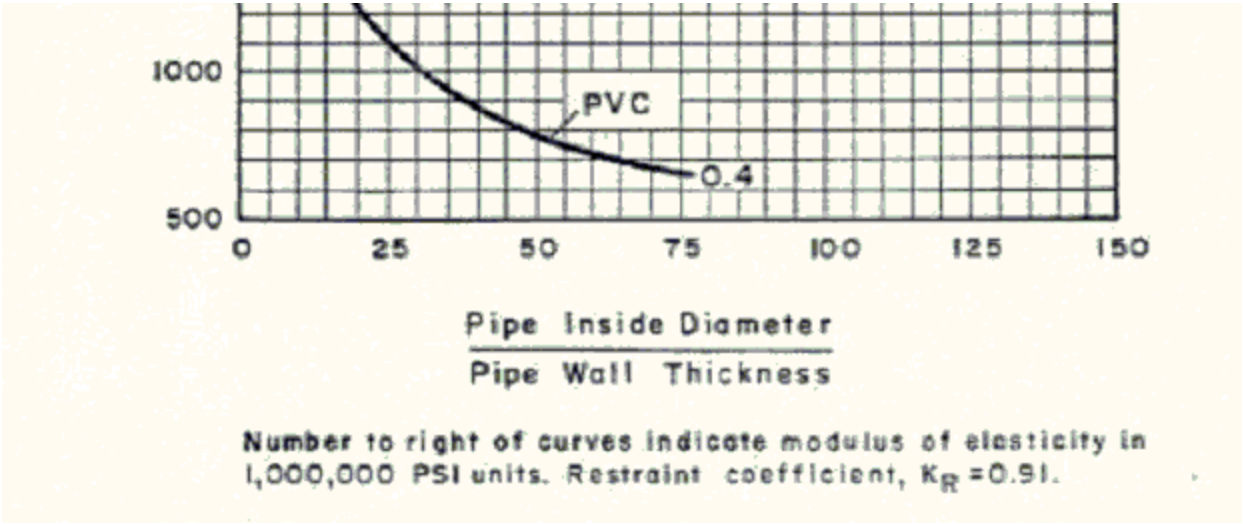
---

With the exception of Wave Speed, the Pipe Data is same as required for steady state model.

**What is Wave Speed?**

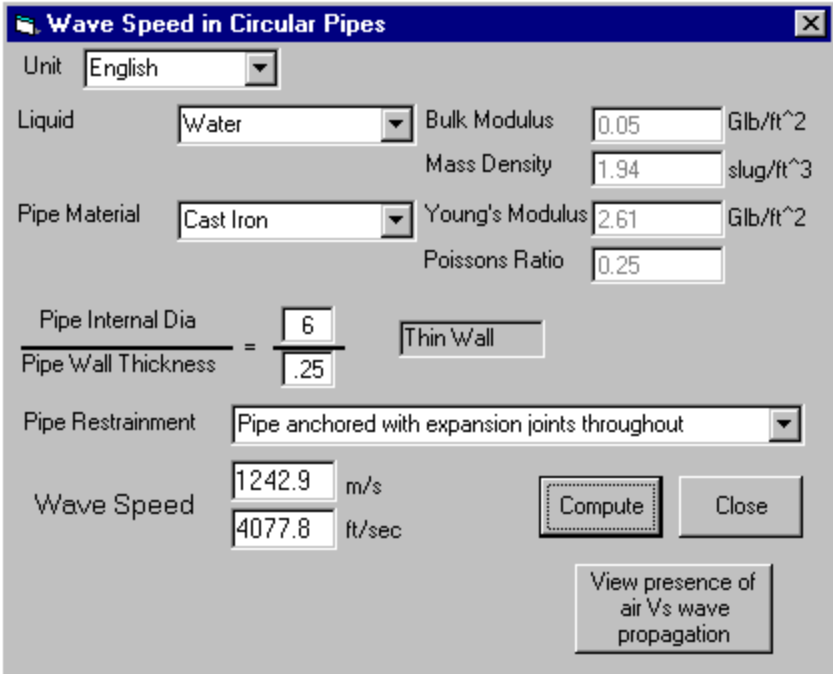
The wave speed is the speed for pressure wave propagation in a pipe and is an essential data item for transient analysis. The wave speed has a very significant effect on the magnitude of the pressure surge generated by a transient event. The wave speed depends on the properties of the pipe and liquid. The diagram below shows values for some standard pipes.


PRESSURE WAVE VELOCITY FOR WATER IN ROUND PIPES
PRESSURE WAVE VELOCITY C (FEET/SEC)
5000
4760.6
4500
4000
3500
3000
2500
2000
1500
STEEL
DUCTILE IRON
CAST IRON
ASBESTOS-CEMENT
2.95
24
15
12
11
10
3.4




**Wave Speed Calculation Tool**

Surge provides a tool for calculating the wave speed. This is found under Tools in the main menu. You can select the liquid, pipe material and type of restraint and provide the diameter and wall thickness and calculate the wave speed as shown below.

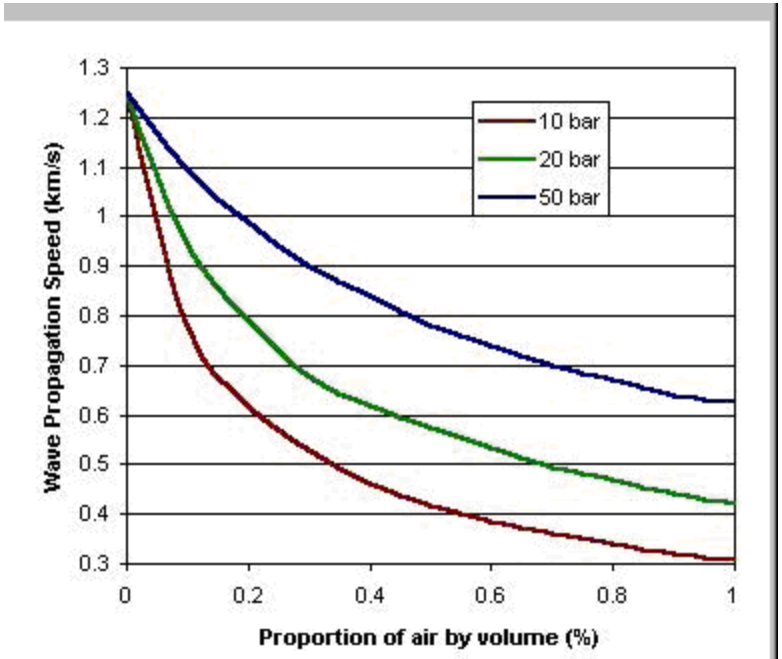



For composite pipes, Composite YoungsMod = (YoungMod1 * Thickness1 + YoungMod2 * Thickness2) / Total Thickness

The Bulk Modulus values given in this table are XE9 Pascals. Liquefied Natural Gas (LNG) may be considered as Methane Hydrate and its Bulk Modulus is 5.6E9 Pascals.

| Property | Ice | Hydrate |
|---|---|---|
| Dielectric constant at 273 °K | 94 | ~58 |
| NMR rigid lattice 2nd moment of H2O protons(G2) | 32 | 33 ± 2 |
| Water molecule reorientation time at 273 °K (μsec) | 21 | ~10 |
| Diffusional jump time of water molecules at 273 °K (μsec) | 2.7 | >200 |
| Isothermal Young's modulus at 268 °K (109Pa) | 9.5 | ~8.4 |
| Speed of longitudinal sound at 273 °K | | |
| Velocity (km/sec) | 3.8 | 3.25-3.6 |
| Transit time (μsec/ft) | 3.3 | 92 |
| Velocity ratio Vp/Vs at 272 °K | 1.88 | 1.95 |
| Poisson's ratio | 0.33 | ~0.33 |
| Bulk modulus (272 °K) | 8.8 | 5.6 |
| Shear modulus (272 °K) | 3.9 | 2.4 |
| Bulk density (gm/cm3) | 0.916 | 0.912 |
| Adiabatic bulk compressibility at 273 °K 10-11Pa | 12 | =14 |

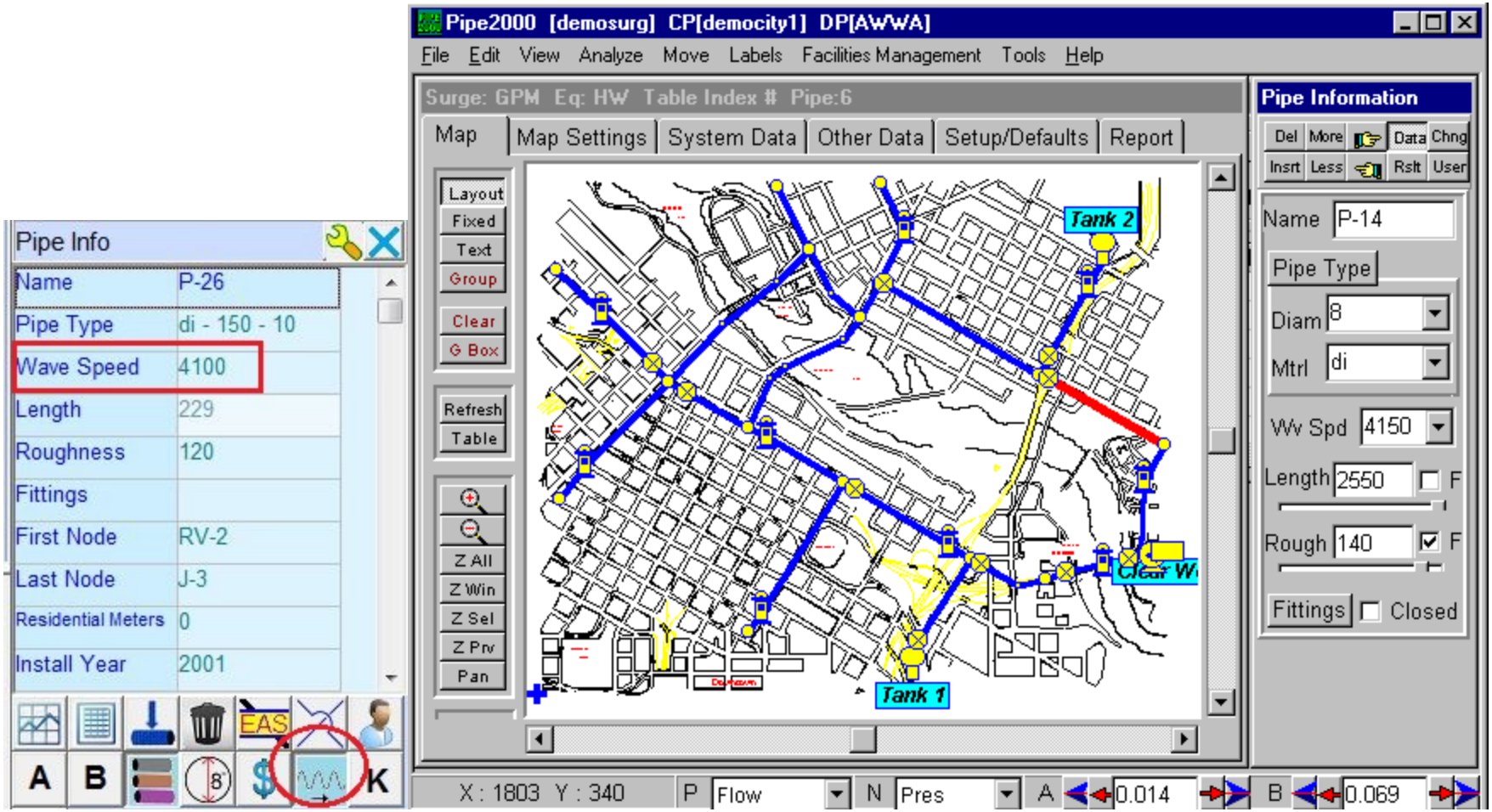
Small amounts of air entrapped in the liquid can greatly affect the wave speed as shown in the diagram below.




The Presence of even small quantities of air can significantly reduce the wave propagation speeds in water in pipes and tunnels. The above graph is for water in a steel pipe having a Diameter/thickness ratio of 20. Ref: Thorley

**Displaying Wave Speed Data**.

The value of the wave speed is displayed on the main Pipe Data box and the appropriate value can be entered in that location as shown below. All wave speed data defined as described below in item 1) may be displayed as map labels.



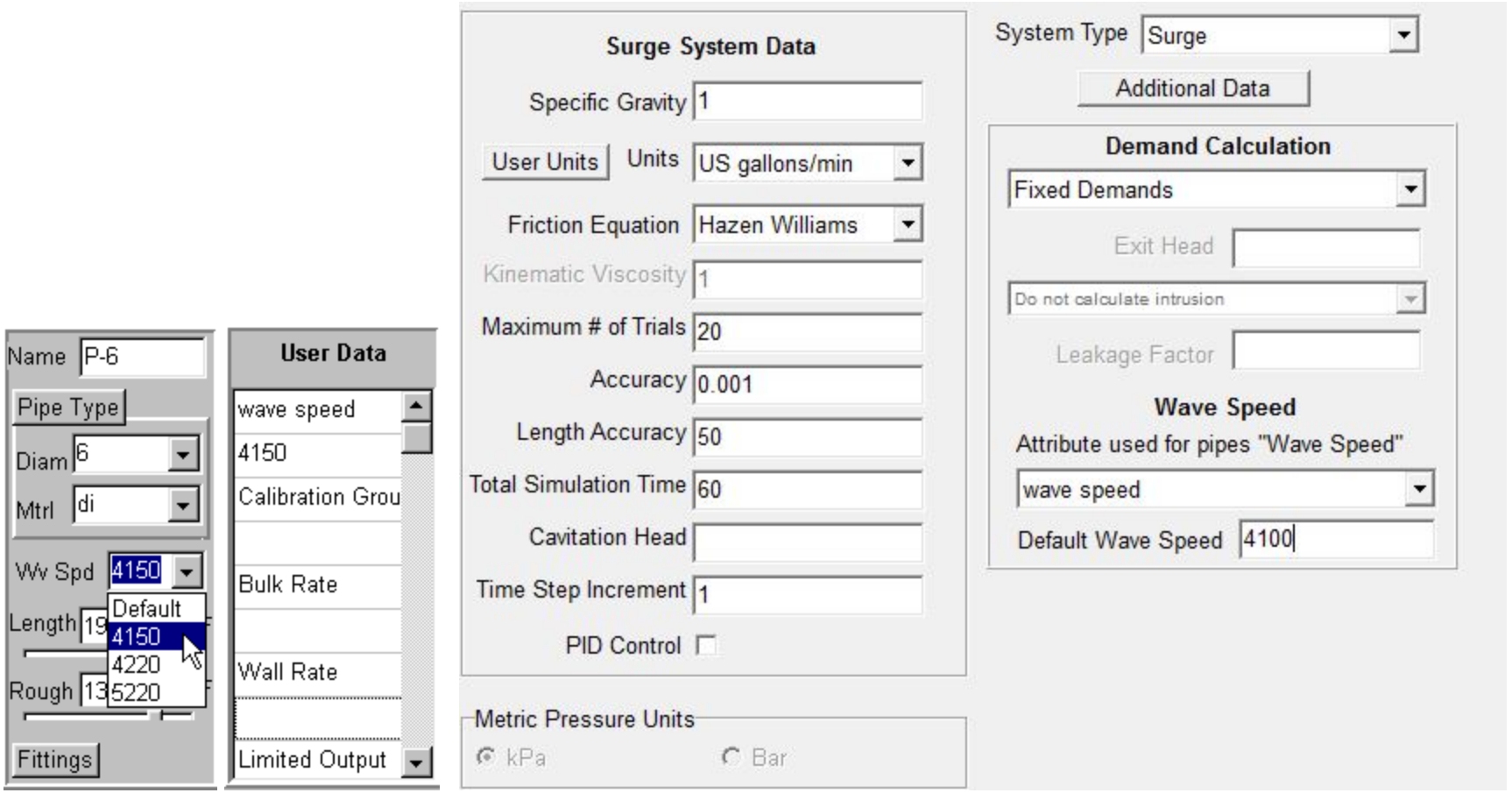

## Defining Wave Speed

There are several ways to define the wave speed. They are, in order of precedence:

### 1. Pipe Data Box / Pipe Attribute

You can define wave speed by entering its value in the location labeled Wv Spd provided on the Pipe Data box as shown. If a value is keyed in here it will be added to the drop down list which can be used to select the appropriate wave speed entry for other pipes. After it is keyed in that value will also appear in the User Data box under the wave speed user data item. By selecting the word "Default" in the drop down list, any keyed-in wave speed value is erased and the wave speed is defined by one of the other methods described.

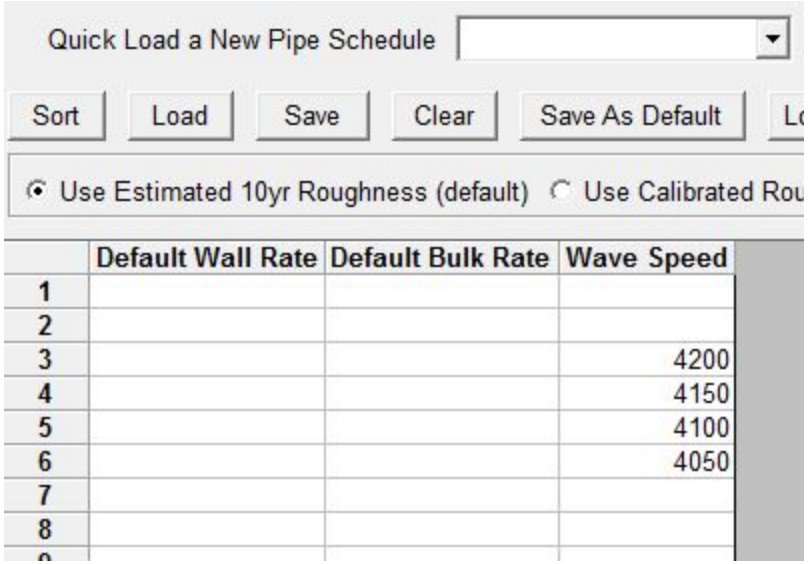
For new files, this data item is automatically created and called "wave speed". It is specified under System Data icon or System | Simulation Specs (Classic) in the Attribute used for pipes "Wave Speed" box. However, if you import an older KYPipe file, this attribute may not have been defined. If the Attribute used for pipes "Wave Speed" box is blank, you will need to create a User Data item called "wave speed" (see User Data / Adding User Data Items).




**2. Pipe Type**

This is a very efficient way to provide wave speed data as a system is being created. Under Components icon or Setups/Defaults | Pipe Type (Classic) the wave speed may be defined in the Pipe Type Data table as shown below. It will then be part of the data included as pipes are created using

the Pipe Type button. Values assigned this way are over-ridden by wave speed data that is keyed directly into the Pipe Data box or in the User Data box under "wave speed".




**3. Overall Default Wave Speed**

Under System Data | Simulation Specs, a box for Default Wave Speed is provided. If there is no data in the Pipe Data box under Wv Spd or in the User Data box under "wave speed", or no wave speed has been assigned using the Pipe Type table then the Default Wave Speed defined here will be used.

**4. Pipe2018 internal default**

If no wave speed is defined by methods 1, 2 or 3, then Pipe2018 assigns a default wave speed internally.

## 1.5.7 Surge Components

Surge Components

---

See also Surge Elements and Data Requirements

The following describes special data requirements and capabilities for Surge components. Surge Control Devices are not included (see Surge Control Devices). These components are:

Check Valves

Bypass lines

Pumps- head/flow table

- constant power
- rated conditions
- pump files (Suter diagrams)

Active Valves

Regulating Valves (PRVs, PSVs, and FCVs)

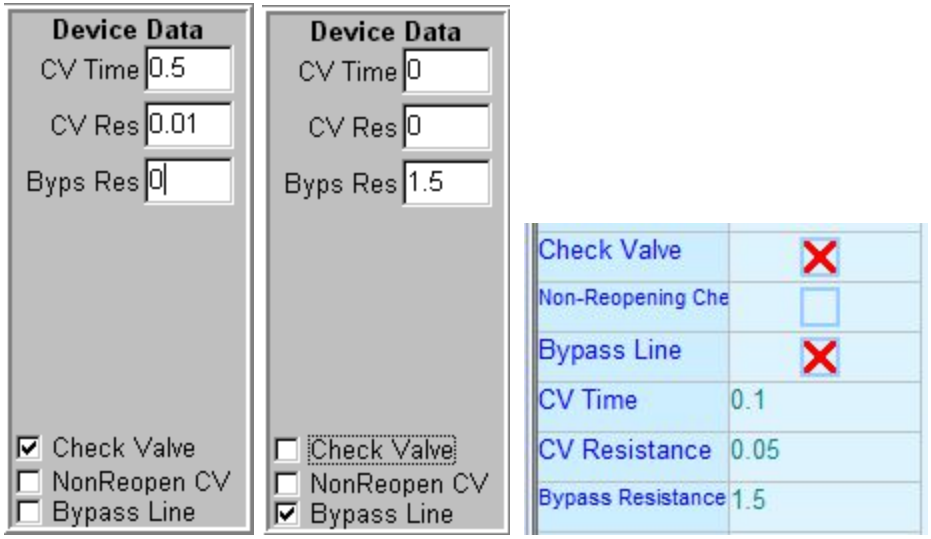
### Check Valves

Pumps, active valves, and loss elements all can be equipped with a check (non-return) valve to prevent backflow. Either a normal check valve which will close or open depending on conditions or a non-reopening check valve which will close only once and remain closed can be designated. The closing time (CV Time) is the time it takes for the check valve to close once closing is initiated. The check valve resistance (CV Res) is the resistance (head/flow^2) when the valve is fully open.

### Bypass Lines

A pump bypass line is one which will open when the suction (upstream) head exceeds the discharge (downstream) head. The bypass line resistance (Byps Res) is the resistance of this pipe.

A valve bypass line is used to define a different resistance when flow reverses and is assumed to flow in the bypass line.




**What is a Pump File?** A pump file is a table of values defining head and torque as a function of flow and speed. These are based on experimental data and 8 files are available to use. The file selection is based on specific speeds and the following files are available.

| File number | specific speed | |
|---|---|---|
| | English | Metric |
| 1 | 1270 | 25 |
| 4* | 3725 | 72 |
| 5* | 4409 | 85 |
| 6* | 5203 | 101 |
| 7* | 6792 | 131 |
| 2 | 7600 | 147 |
| 8* | 8764 | 169 |
| 3 | 13500 | 261 |

These files were selected to cover the range of specific speeds for which data is available. This specific speed is defined as

specific speed = (Nr (Qr^0.5))/(Hr^0.75)

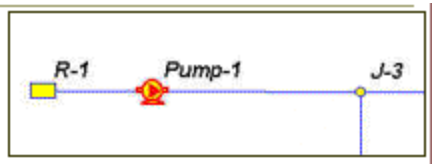
*files 1, 2, and 3 were available for versions of SURGE prior to SURGE 5 and files 4 - 8 were added to version SURGE 5.1.

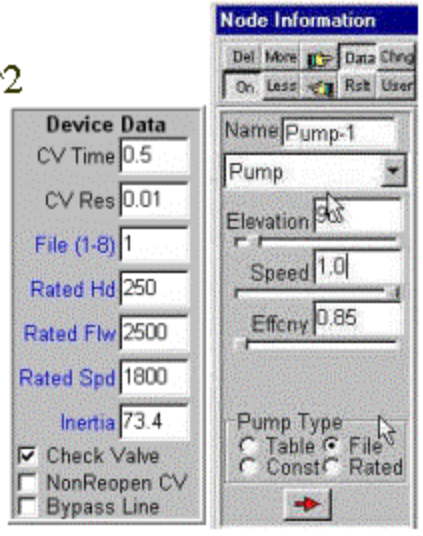
The pump file is customized to your pump by providing the Rated Head, Rated Flow, Rated Speed and Inertia. See the Surge Tools for help in calculating the specific speed and inertia. The use of this tool is illustrated below for this example.

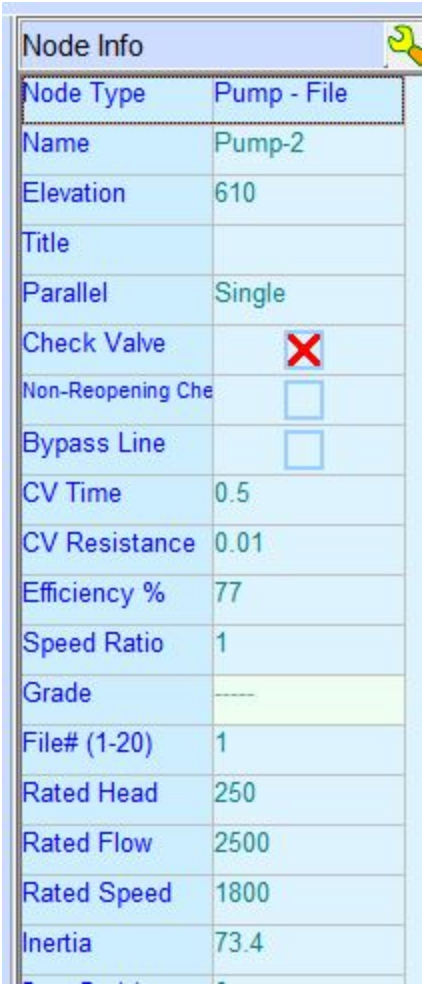
Transients for pumps are initialized by changing the pump speed ratio (pump speed/rated speed) or designating a pump trip where the rundown speed is calculated (see Variable Input Data).

# Pumps: (Four Quadrant) Files

- File number = 1 (for 9991)
- Rated Head = 250ft
- Rated Flow = 2500gpm
- Rated Speed = 1800rpm
- Pump+Motor Inertia=73.4 lb-ft^2
- Pump Efficiency = 85%
- Initial Pump Speed Ratio = 1.0
- Elevation = 90feet
- Check valve resistance=0.01
- CV open time = 0.5sec



**Pumps: Head Flow Table**

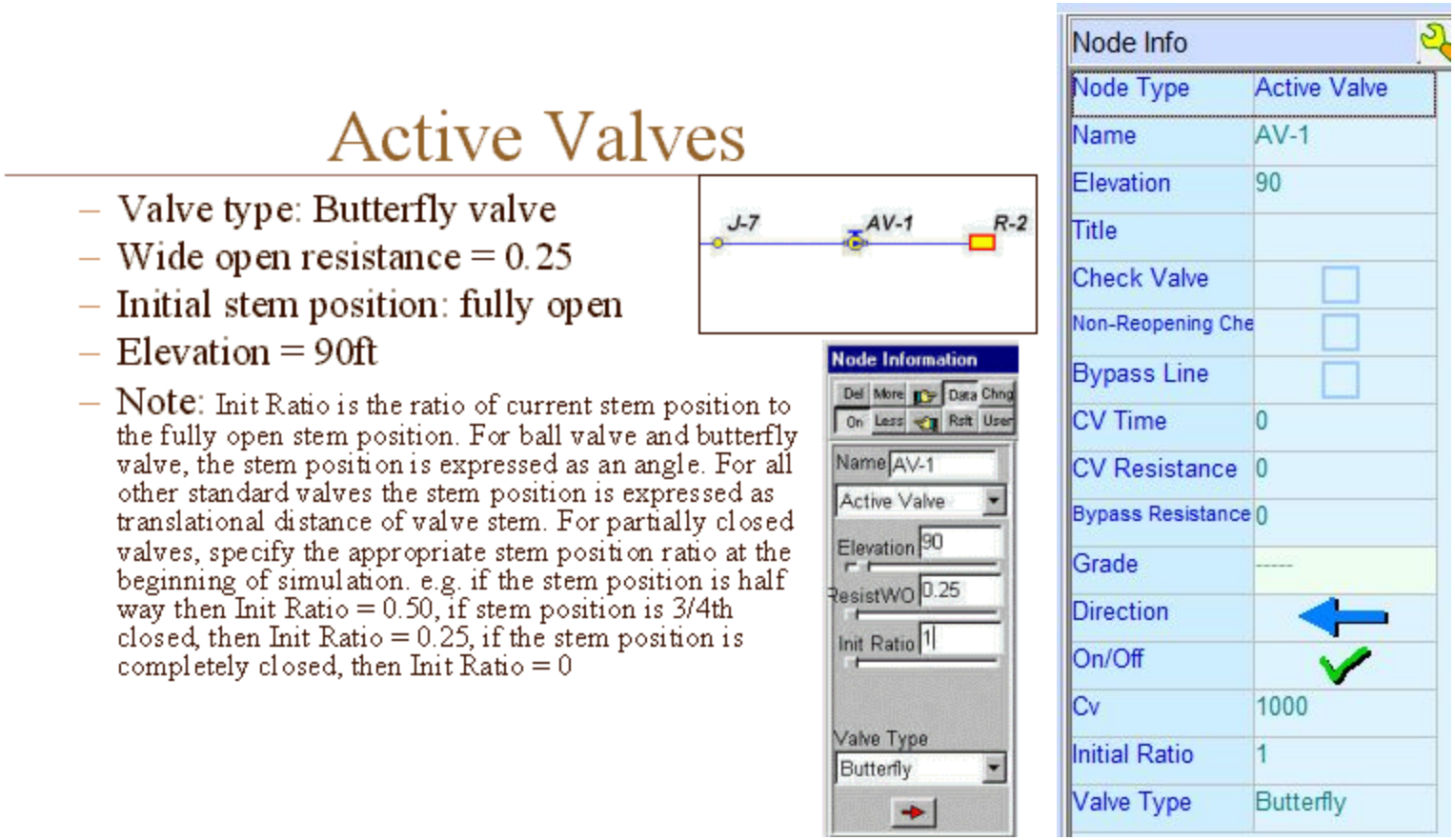
These pumps should be described by just three data points. If a Kypipe file is imported with additional data points only the first three will be used in the Surge analysis. Since only normal conditions (positive head, positive flow) are described by the data, transient analysis should be limited to situations where the pumps operate under normal conditions. If abnormal conitions occur (reverse flow, turbining, etc.) a pump file should be used.

**Pumps: Constant Power**

This pump description should not be used for Surge applications.

**Pumps: Rated Conditions**

These pumps are described by three points of head-flow data based on the rated conditions and the previous comments apply (see Pumps: Head Flow Table)

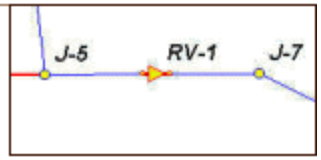


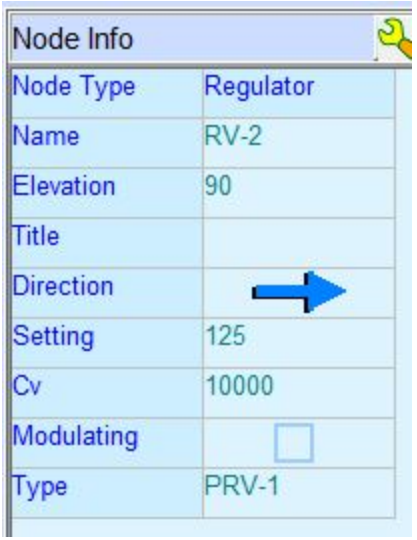
### Regulating Valves

Regulating valves which control pressure or flow for steady state calculations are modeled as a normal valve for Surge analysis with the initial resistance calculated based in the steady state conditions. For example, if a PRV reduces the pressure from 100 to 50 psi (pressure change = 40 psi = 92.3 ft) with a flow of 600 gpm (1.337 cfs), the intial resistance can remain constant or be changed using Variable Input (Change) data.

## Pressure Reducing Valve

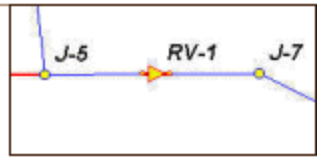
- For steady state, reduces pressure downstream of the valve to a set value (switches to wide open condition if pressure upstream of the valve is less than the set value under steady state conditions)
- Elevation = 90ft
- Pressure Setting = 125psi
- Note:For transient analysis, a component with initial resistance based on steady state conditions is used

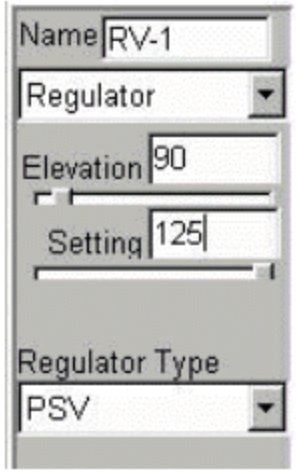




# Pressure Sustaining Valve

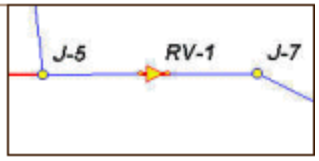
- For steady state, maintains pressure upstream of the valve to a set value (switches to wide open condition if pressure downstream of the valve is greater than the set value under steady state conditions)
- Elevation = 90ft
- Pressure Setting = 125psi
- Note:For transient analysis, a component with initial resistance based on steady state conditions is used

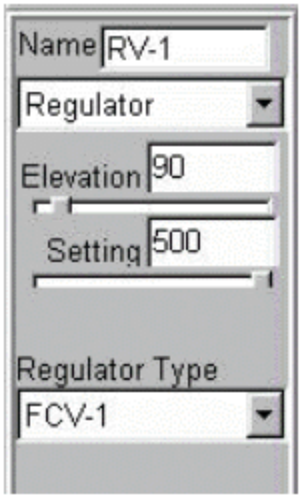


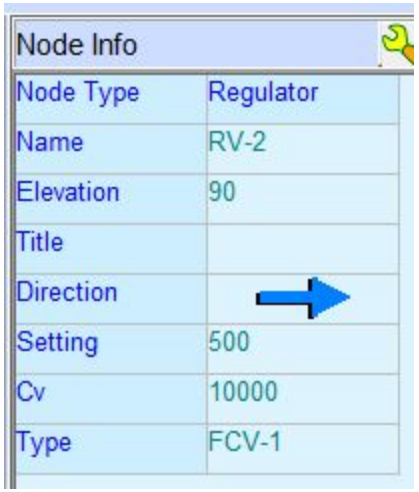



# Flow Control Valve

- For steady state, maintains a set flow rate through the pipe on which the flow control valve (FCV) is located (switches to wide open condition if the hydraulics doesn't permit minimum flow rate under steady state conditions)
- Elevation = 90ft
- Flow rate Setting =500gpm
- Note:For transient analysis, a component with initial resistance based on steady state conditions is used



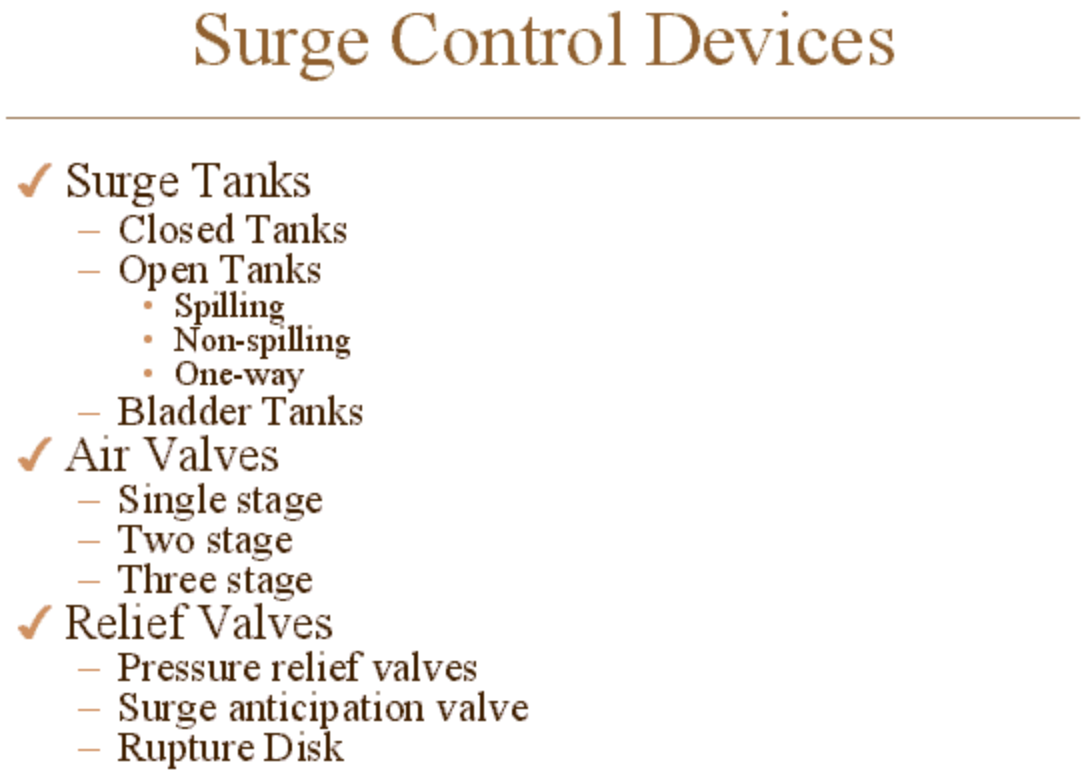




1.5.8 Surge Control and Control Devices

See also Surge Elements and Data Requirements

The following is an explanation of the data requirements for various surge control devices used with Surge.

Click on the device names below to view the data requirements.



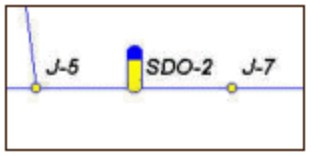

**Note:** For surge tanks and relief valves the input data includes the inflow and outflow resistance. The resistance is defined as the head drop (ft or m) divided by the flow squared (cfs or cms). The resistance depends on the type of connection (pipe section or orifice) and can be calculated from the characteristics of the connection. See the Surge Tools for help with the calculations.

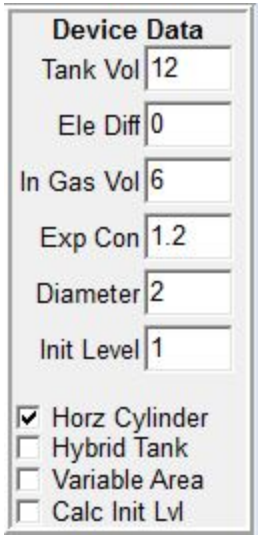
**What is a closed surge tank?** A closed surge tank has a volume of gas, usually air, in the space above the liquid. The gas expands and compresses as flow leaves and enters the tank.

# Closed Surge Tank

Data:

- Diameter = 2ft
- Initial fluid level = 1ft
- Initial gas volume = 0.5 cubic feet
- Gas expansion constant = 1.2
- Inflow resistance = 0.01
- Outflow resistance = 0.01
- Elevation = 90feet





Init(ial) Level - ft or m above the centerline of the connecting pipe.

Diameter of the tank (required) ft. or m. This value is used in the calculation.

Tank volume - ft3 or m3 *NOTE: This is not used in the calculation, only Initial Gas Volume affects the performance of the device.

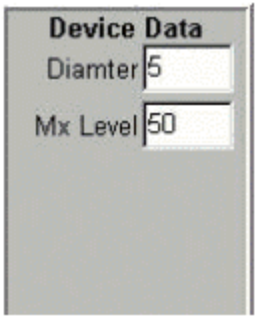
The expansion constant varies between 1 (isothermal) and 1.4 (adiabatic) and a value between these limits should be used.

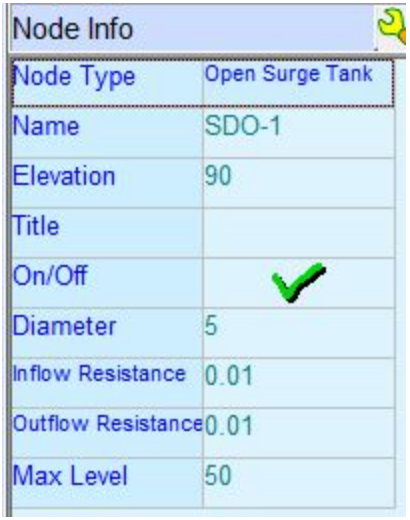
**What is an Open Surge Tank?** This is a tank connecting to the pipeline which is open to the atmosphere. If the tank becomes full where water spills over the maximum (Mx) level should be input and the tank is modeled as a spilling tank. See previous note (Closed Surge Tank) if tank is not a vertical cylinder.

# Open Surge Tank - Spilling

- Data:
  - Diameter = 5ft
  - *Maximum fluid level = 50ft
  - Inflow resistance = 0.01
  - Outflow resistance = 0.01
  - Elevation = 90feet

*Maximum fluid level is specified ONLY for a *spilling tank*. Zero defaults to a *NON spilling tank*

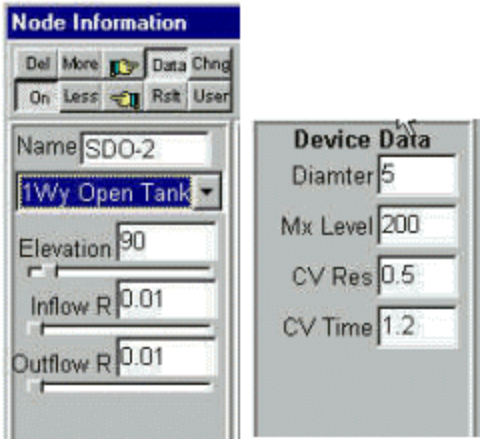


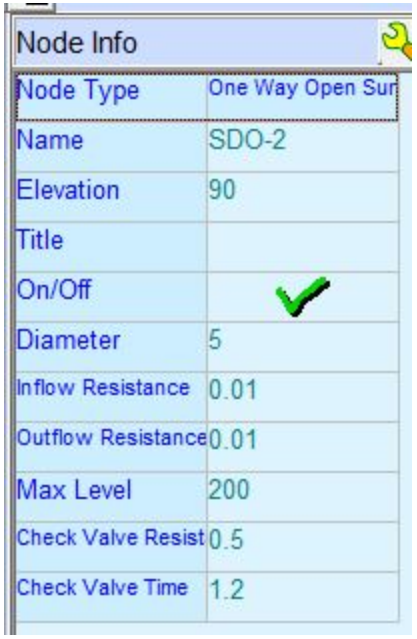




**What is a One-Way Open Surge Tank (Feed Tank)?** This open surge tank is equipped with a check valve at the entrance which prevents flow from entering the tank. This is useful for controlling down-surges at a point where the static head is large and a normal open surge tank would not be practical.

# One-way Open Surge Tank

- Data:
  - Diameter = 5ft
  - Maximum fluid level = 200ft
  - Inflow resistance = 0.01
  - Outflow resistance = 0.01
  - Elevation = 90feet
  - Check valve resistance=0.5
  - Check valve time = 1.2 sec
  - Note: The check valve data distinguishes the one-way tanks (flow only out of tank).



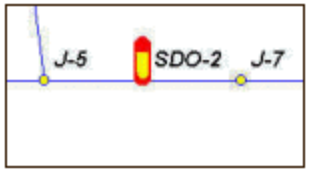



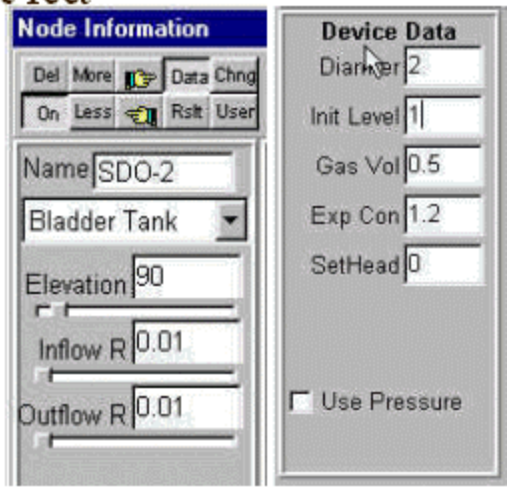

**What is a Bladder Surge Tank?** This is a closed surge tank with a gas-filled bladder. This tank will activate at the set pressure for the bladder and will act as a closed surge tank for pressures above the set pressure. For pressures below the set pressure the tank has no effect.

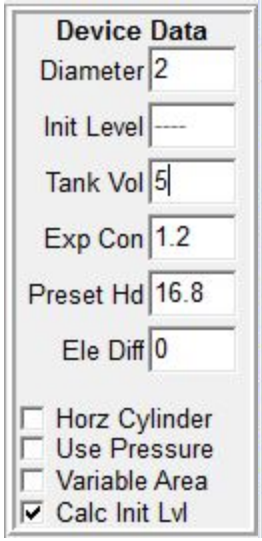
# Bladder Surge Tank

- Data:
  - Diameter = 2ft
  - Initial fluid level = 1ft
  - Initial gas volume = 0.5 cubic feet
  - Expansion constant = 1.2
  - *Set Head = 80ft
  - Inflow resistance = 0.01
  - Outflow resistance = 0.01
  - Elevation = 90feet

*The bladder tank will activate above this setting



Init(ial) Level - ft or m above the centerline of the connecting pipe.

Tank volume - ft3 or m3

Preset Prs (pressure) default is ft or m head, if "Use Pressure" box is checked - psi or kPa.

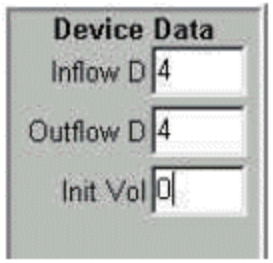
**What is a Single Stage Air Vacuum Valve?** This device takes air in when the pipeline pressure drops below atmospheric and releases air when the pipeline pressure exceeds atmospheric. The single stage air valve has the same size orifice for inflow and outflow (usually the same orifice). Also an initial volume of air can be specified to model startup with air at the air valve location. Delay is in seconds. A discharge coefficient of zero defaults internally to the standard value of 0.62.

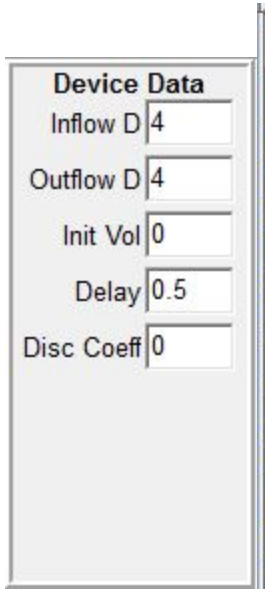
Note: Inlet and outlet diameter may be entered as a negative (-) value to represent orifice area instead of diameter.

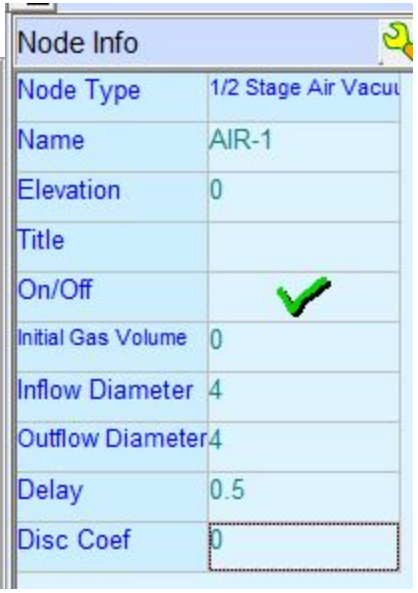
# Single Stage Air Vacuum Valves

- Data:
  - Inflow diameter = 4 inches
  - Outflow diameter = 4 inches
  - *Initial air volume = 0 cu. ft.
  - Elevation = 90feet

*This is used to model startup with air at this location



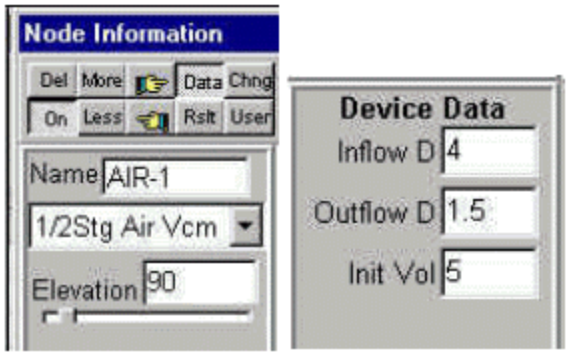



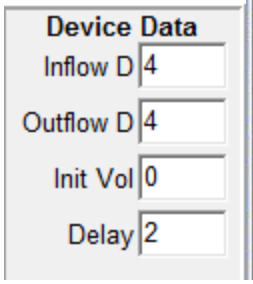



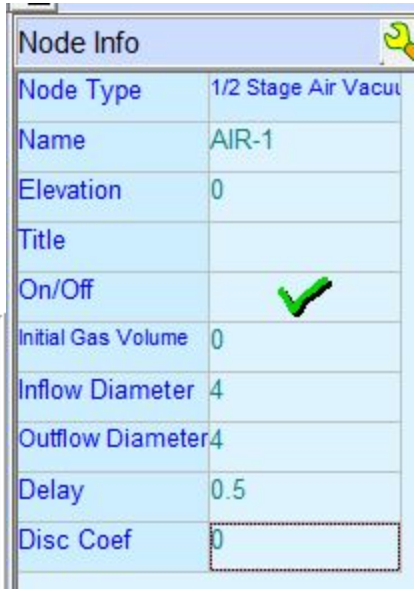

**What is a Two-Stage Air Vacuum Valve?** A two-stage air valve has a different size orifice for inflow and outflow. The outflow orifice is smaller to reduce the rate of air expulsion and the "air slam" which occurs when all the air is expelled and the liquid columns rejoin. See Single-Stage Air Vacuum Valves for additional details.

# Two Stage Air Vacuum Valves

- Data:
  - Inflow diameter = 4 inches
  - Outflow diameter = 4 inches
  - Initial air volume = 5 cu. ft.
  - Elevation = 90feet



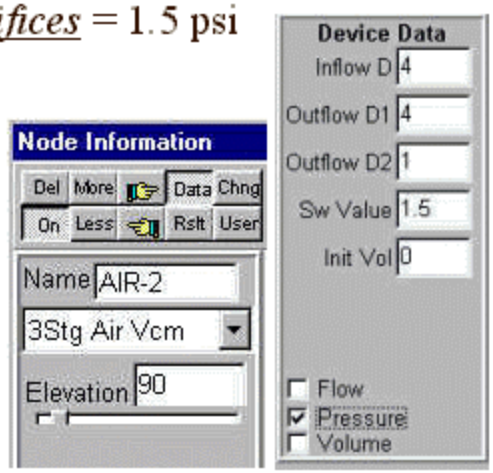



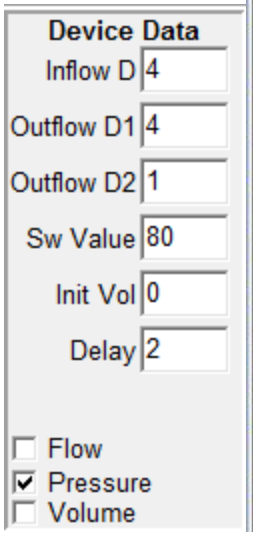




**What is a Three-Stage Air Vacuum Valve?** A three-stage air valve has a second outflow orifice (smaller) to release the air more slowly and reduces "air slam". The transition from the primary outflow orifice (diameter D1) to the second orifice (D2) usually occurs at a specified pressure (psi or kpa) but can be modeled to switch based on flow (cfs or cms) or volume (ft^3 or m^3).

# Three Stage Air Vacuum Valves

- Data:
  - Inflow diameter = 4 inches
  - First Outflow Orifice diameter =4inches
  - Second Outflow Orifice diameter =1inches
  - *Pressure that switches the orifices* = 1.5 psi
  - Initial air volume = 0 cu. ft.
  - Elevation = 90feet



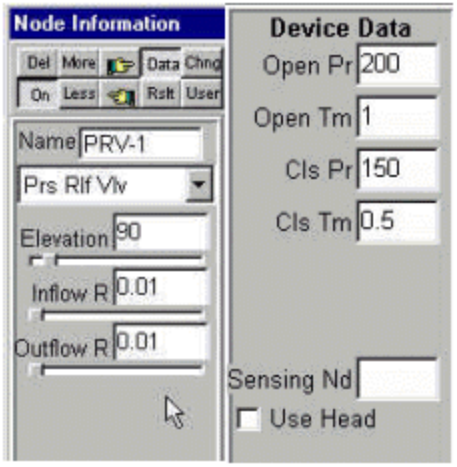



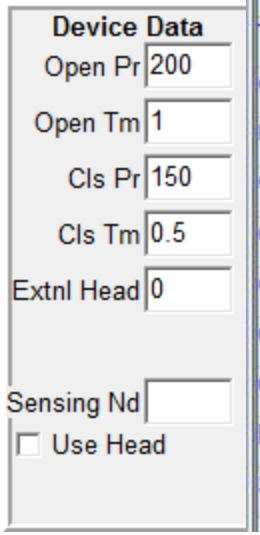
**What is a Pressure Relief Valve?** A pressure relief valve is one which opens at a designated pressure (psi or kpa) or head (ft or m) and closes at a second (lower) pressure or head. The opening and closing times are specified.

# Pressure Relief Valves

- Data:
  - Opening Pressure = 200psi
  - Opening Time = 1 sec
  - Closing Pressure = 150psi
  - Closing Time = 0.5 sec
  - *Sensing node: --
  - Inflow resistance = 0.01
  - Outflow resistance = 0.01
  - Elevation = 90feet

*This defaults to the location of the relief valve, if sensing node is NOT specified



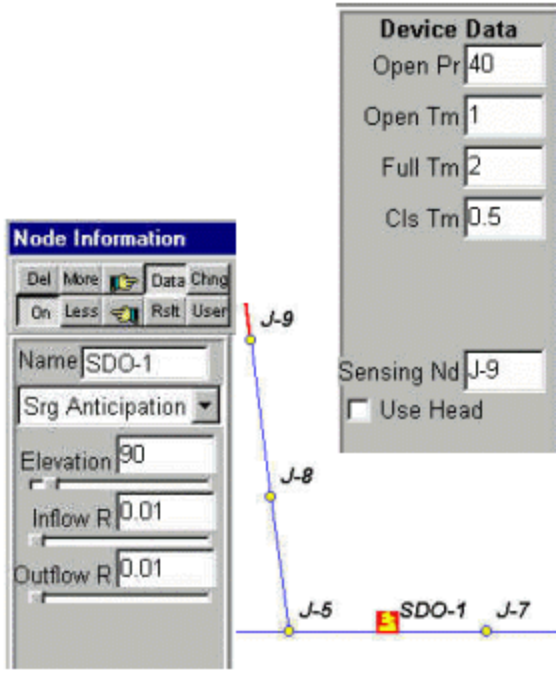
**Note:** A sensing node other than the location of the relief valve can be specified. If this specification is not made the sensing is assumed to occur at the valve location.

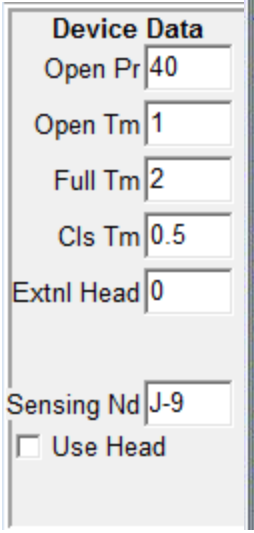
**What is a Surge Anticipation Valve?** A surge anticipation valve is a device normally located at a pump discharge which opens on a down-surge at a specified pressure (or head) to provide protection for a subsequent upsurge. Once activated this device works on a timed cycle. The valve opens fully (Open Tm) and stays fully open for a designated period (Full Tm) and then closes (Cls Tm). A sensing node can be designated if the opening is to be activated by the pressure at a different location.

# Surge Anticipation Valves

- ✓ Data:
  - Opening Pressure = 40psi
  - Opening Time = 1 sec
  - Full Open Time = 2 sec
  - Closing Time = 0.5 sec
  - *Sensing node J-9
  - Inflow resistance = 0.01
  - Outflow resistance = 0.01
  - Elevation = 90feet

*This defaults to SDO-1 (location of surge anticipation valve) if sensing node is NOT designated

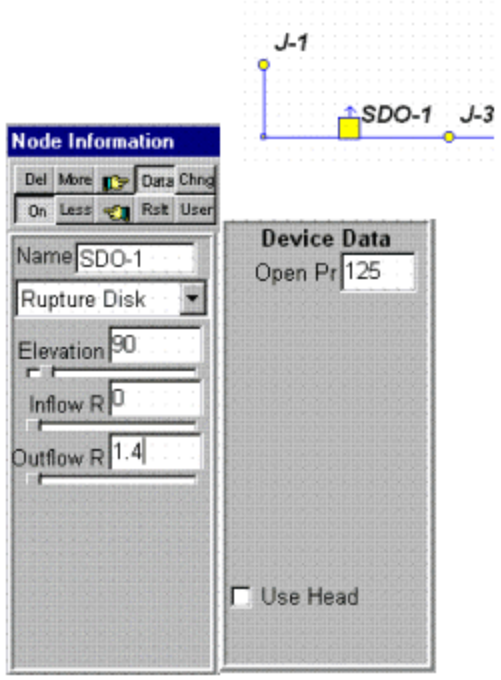


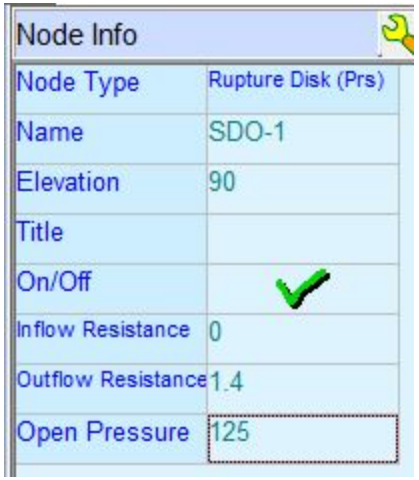



**What is a Rupture Disk?** A rupture disk is a device which opens quickly (ruptures) at the specified pressure (or head).

# Rupture Disk

- Data:
  - Opening Pressure = 125psi
  - Inflow resistance = 0
  - Outflow resistance = 1.4
  - Elevation = 90 feet

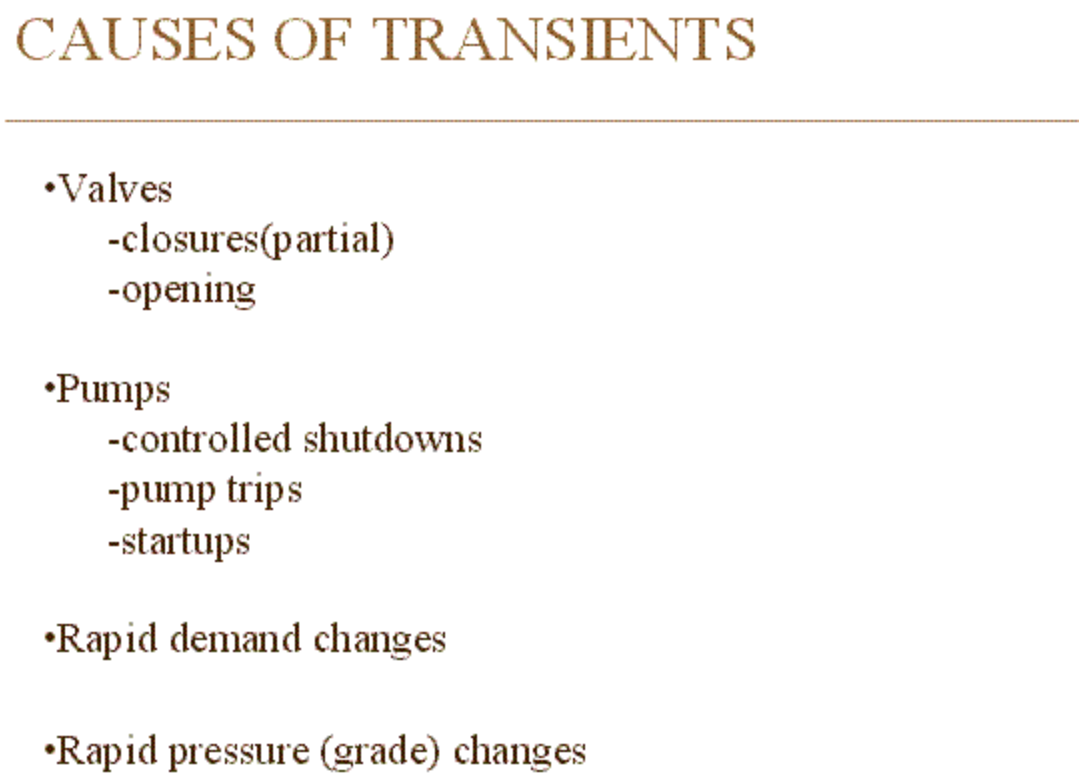



1.5.9 Surge Transient Setup Examples

See also Surge Elements and Data Requirements

Below are examples of the data used to describe the cause of a transient. The transient event is described using a series of pairs of data points for time and the value of a parameter which is the cause of the transient. Also periodic changes can be specified to describe the cause of the transient.



# Providing Variable Input Data

**•Use Change Date to specify changed parameter vs. time**

•Pumps
-pump speed ratio (speed/rated speed) vs. time

•Valves
-valve stem position ratio vs. time
-effective open area ratio vs. time

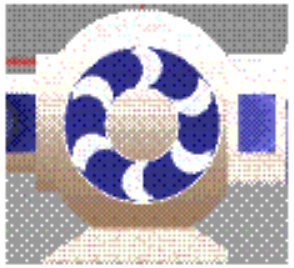
•Rapid demand changes
- demand vs. time

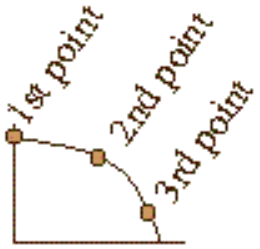
•Rapid pressure changes
- hydraulic grade vs. time

# Variable Input Data - Pumps

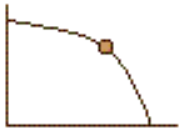
(transient initiating event)

- ✓ Pump defined by Table option (3-point curve) or Rated option (reference point)
- ✓ Pump startup or shutdown
- ✓ Pump operates in a normal range



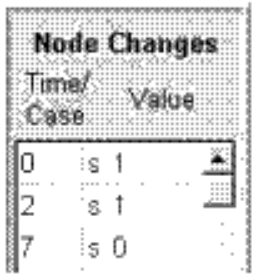



3-point curve
(user defines all three points)



Reference point
(program computes
1st and 3rd points)

- ✓ Pump (Table or Rated) shutdown
- ✓ Example 1:
  - Pump speed reduced from 1800rpm to 0 rpm in 5 seconds (speed changes linearly from 1800 to 0 in 5 seconds)
  - At time=0sec, pump speed ratio = 1
  - At time=2sec, pump speed ratio = 1 (hold steady state for the first 2 seconds)
  - At time=7sec, pump speed ratio = 0

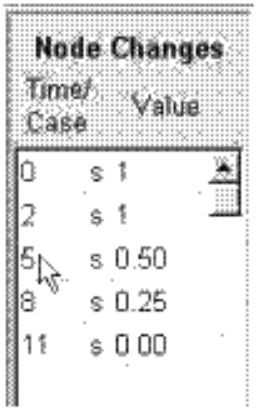


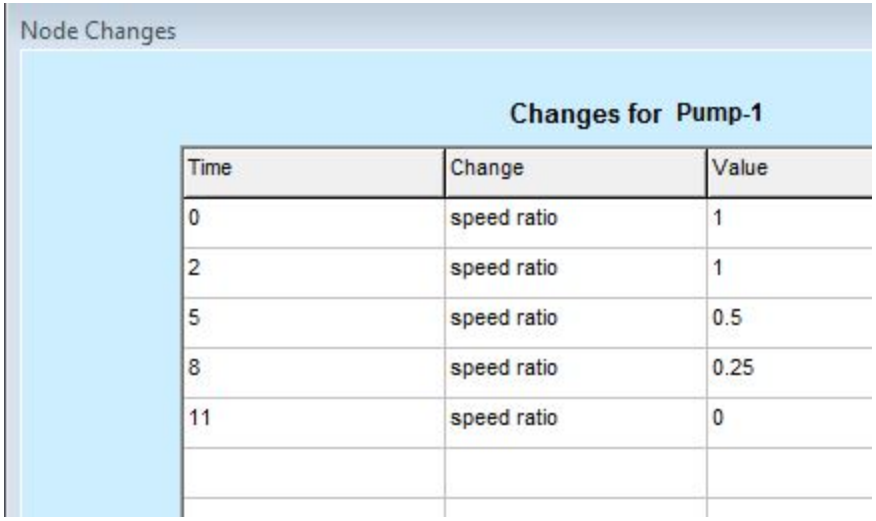

Node Changes

Changes for Pump-1

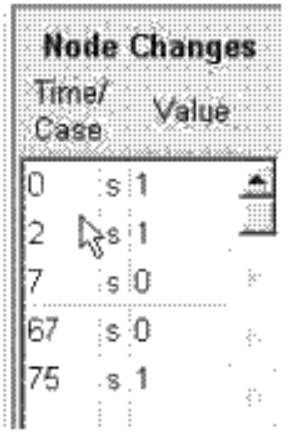
| Time | Change | Value |
|---|---|---|
| 0 | speed ratio | 1 |
| 2 | speed ratio | 1 |
| 7 | speed ratio | 0 |
| | | |

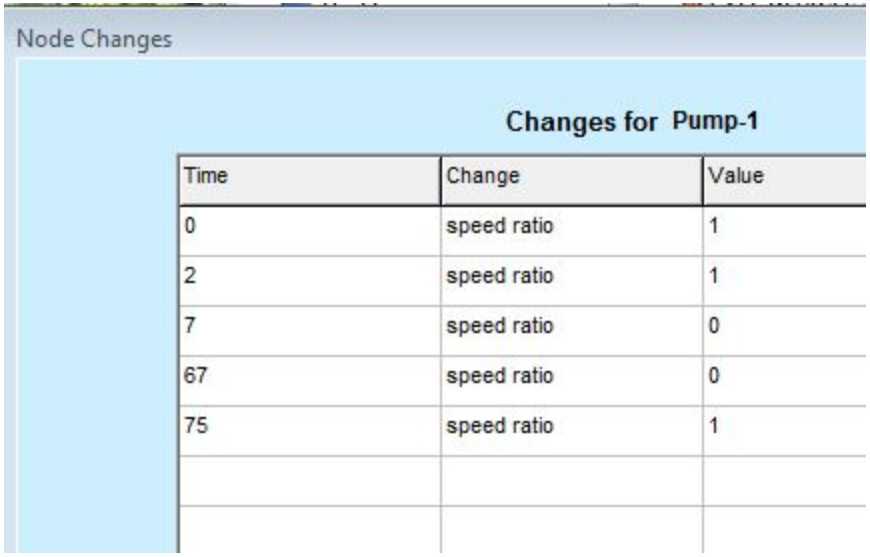
- ✓ Pump (Table or Rated) shutdown
- ✓ Example 2:
  - Pump speed reduced from 1800rpm to 0 rpm in 9 seconds (speed changes in a controlled manner from 1800 to 0 in 9 seconds)
  - At time=0sec, pump speed ratio = 1
  - At time=2sec, pump speed ratio = 1 (hold steady state for the first 2 seconds)
  - At time=5sec, pump speed ratio = 0.50
  - At time=8sec, pump speed ratio = 0.25
  - At time=11sec, pump speed ratio = 0.00




Node Changes
Changes for Pump-1
Time
Change
Value
0
speed ratio
1
2
speed ratio
1
5
speed ratio
0.5
8
speed ratio
0.25
11
speed ratio
0

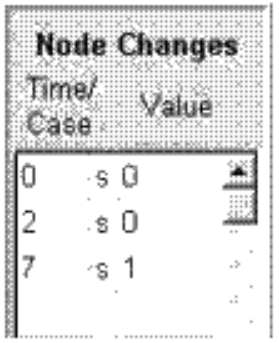
- ✓ Pump (Table or Rated) shutdown followed by startup
- ✓ Example 3:
  - Pump speed reduced from 1800rpm to 0 rpm in 5 seconds, kept shutdown for the next 60 seconds before starting up in 8 seconds
  - At time=0sec, pump speed ratio = 1
  - At time=2sec, pump speed ratio = 1 (hold steady state for the first 2 seconds)
  - At time=7sec, pump speed ratio = 0
  - At time=67sec, pump speed ratio = 0
  - At time=75sec, pump speed ratio = 1

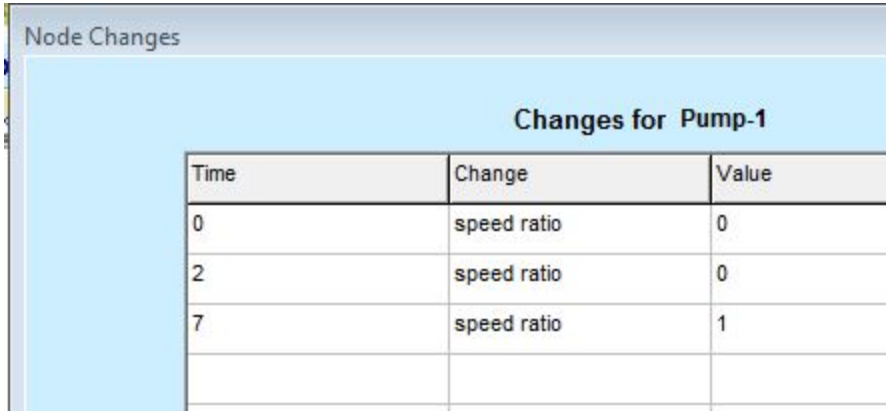




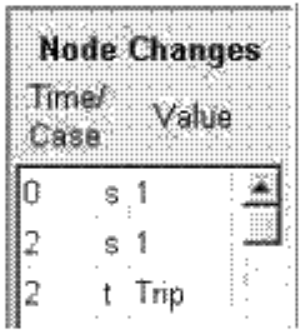

Pumps used for Startup must have internal check valves. Do not use non-reopen check valves.

- ✓ Pump (Table or Rated) startup
- ✓ Example 4:
  - Pump speed increased from 0 rpm to 1800 rpm in 5 seconds (linear variation)
  - At time=0sec, pump speed ratio = 0
  - At time=2sec, pump speed ratio = 0 (hold steady state for the first 2 seconds)
  - At time=7sec, pump speed ratio = 1

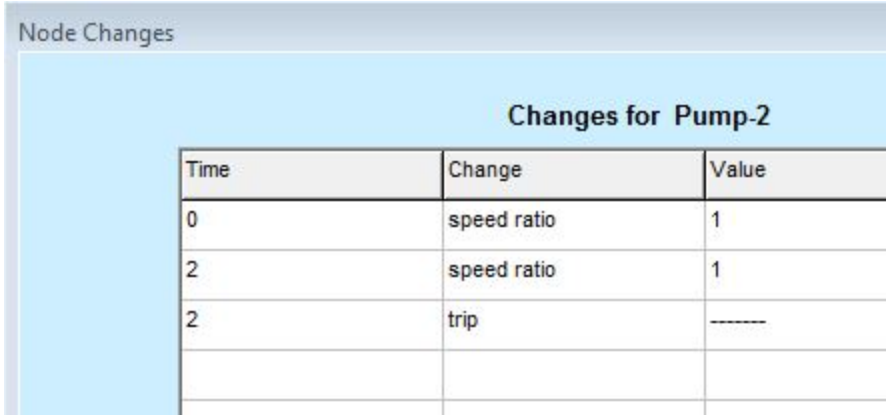


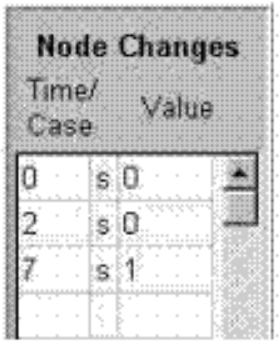
- ✓ Pump trip (pump file definition)
- ✓ Includes abnormal operation
- ✓ Example:
  - Pump trip after holding steady state for the first 2 secs
  - At time=0sec, pump speed ratio = 1
  - At time=2sec, pump speed ratio = 1
  - At time=2sec, pump trip*

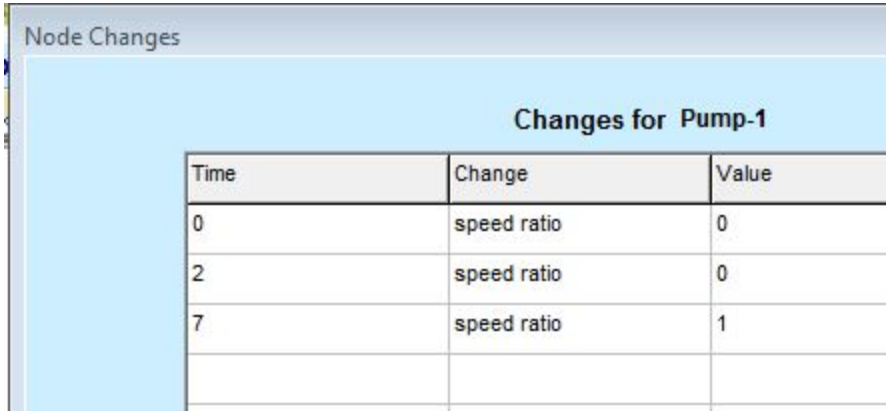



*Note: The pump speed following loss of power is calculated using dynamic analysis including torque, inertia, etc.



- Pump startup (pump file definition)
- Example:
  - Pump speed increased from 0 rpm to 1800 rpm in 5 seconds
  - At time=0sec, pump speed ratio = 0
  - At time=2sec, pump speed ratio = 0 (hold steady state for the first 2 seconds)
  - At time=7sec, pump speed ratio = 1

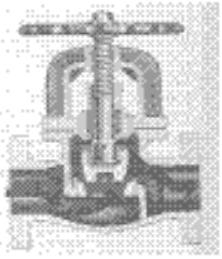


# Variable Input Data - Valves

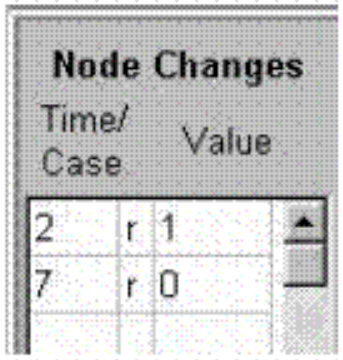
(transient initiating event)

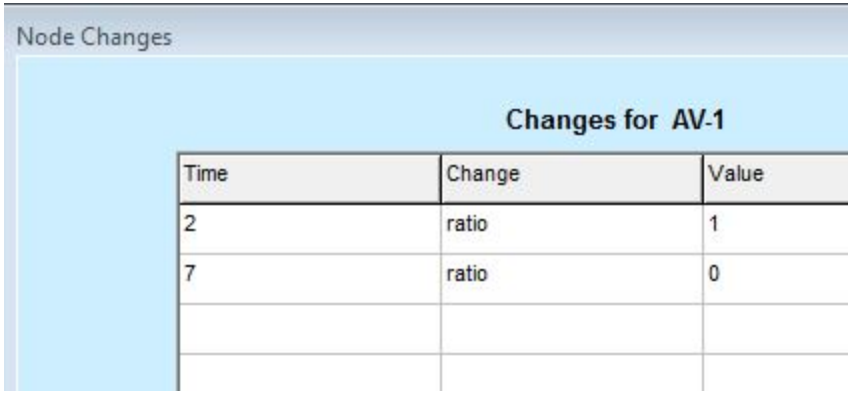
- Valve Closure – Standard Valves*



- Example 1:
  - Valve closed from full open to full close in 5 seconds
    - At time=0sec, Stem position ratio = 1
    - At time=2sec, Stem position ratio = 1 (hold steady state for 2 seconds)
    - At time=7sec, Stem position ratio = 0

*Standard valves include ball, butterfly, gate, globe, and needle.

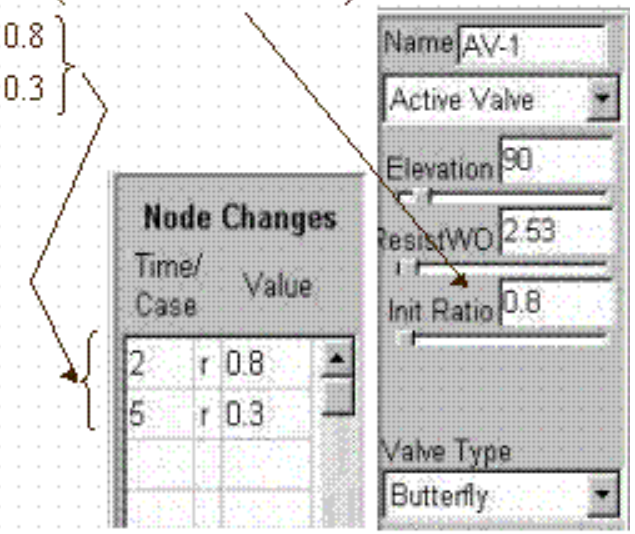


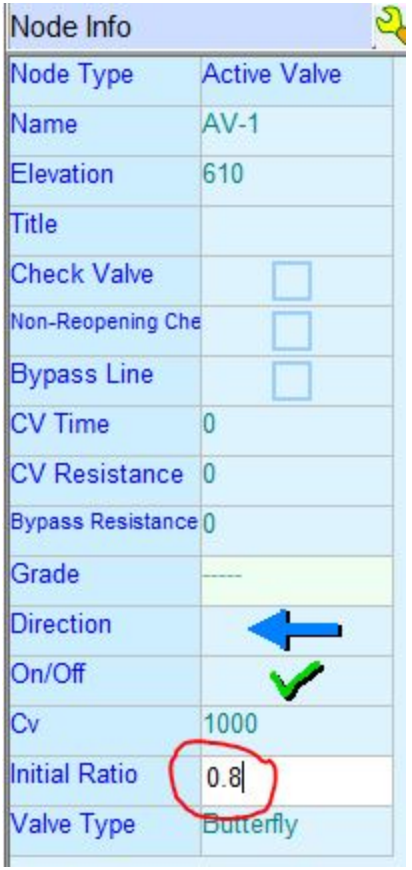

Node Changes
Changes for AV-1
Time
Change
Value
2
ratio
1
7
ratio
0

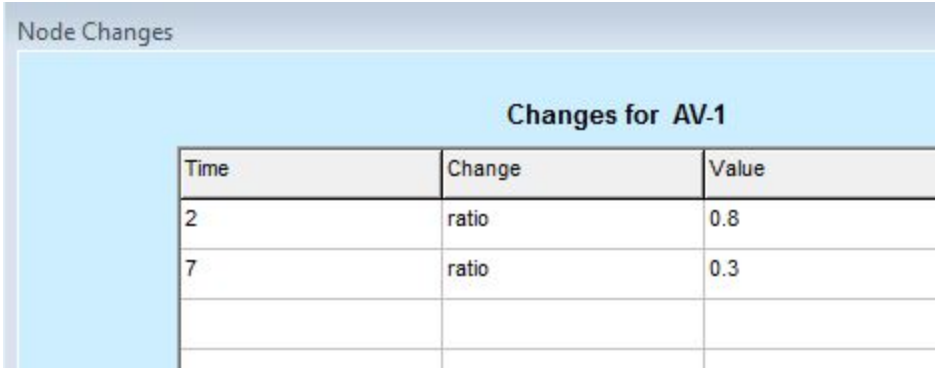
## ✓ Valve Closure – Standard Valves

## ✓ Example 2: Partial Open to Partial Closure

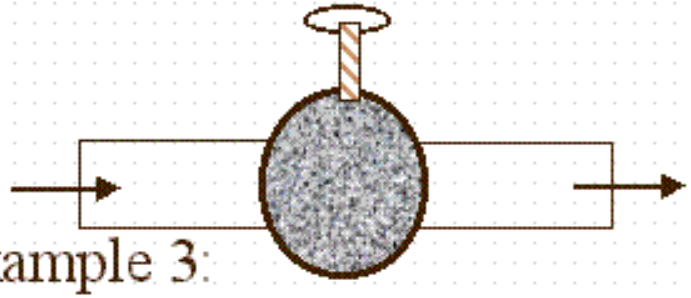
- Valve closed from 80% open to 30% open in 3 seconds
  - At time=0sec, Stem position ratio = 0.8 (note the initial ratio)
  - At time=2sec, Stem position ratio = 0.8
  - At time=5sec, Stem position ratio = 0.3



## ✓ Valve Closure – Non-standard Valves*



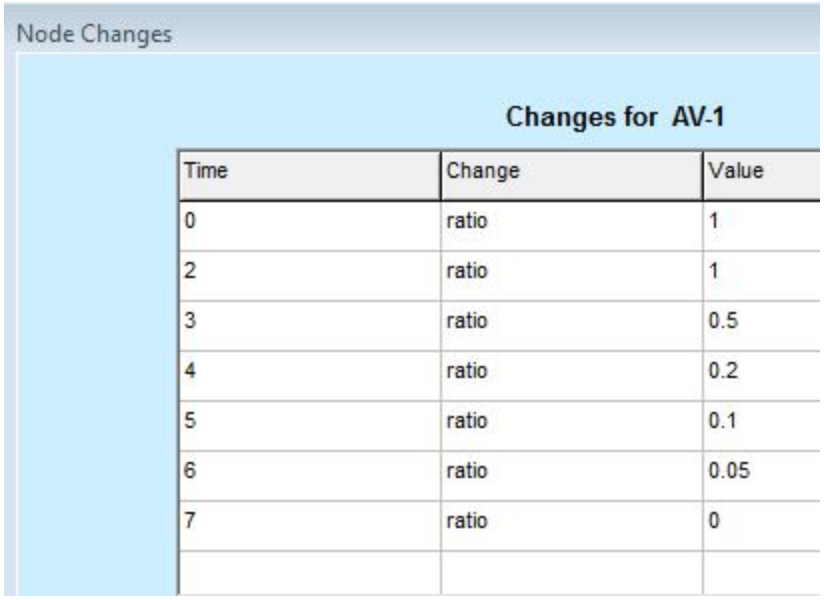
## ✓ Example 3:

- Valve closed from full open to full close in 5 seconds (user prescribed change of flow area with respect to time)
  - At time=0sec, Effective area Ratio = 1
  - At time=2sec, Effective area Ratio = 1 (hold steady state for 2 seconds)
  - At time=3sec, Effective area Ratio = 0.50
  - At time=4sec, Effective area Ratio = 0.20
  - At time=5sec, Effective area Ratio = 0.10
  - At time=6sec, Effective area Ratio = 0.05
  - At time=7sec, Effective area Ratio = 0.00

**Node Changes**

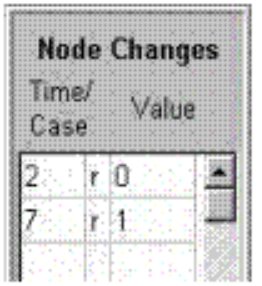
| Time/ Case | | Value |
|---|---|---|
| 0 | r | 1 |
| 2 | r | 1 |
| 3 | r | 0.50 |
| 4 | r | 0.20 |
| 5 | r | 0.10 |
| 6 | r | 0.05 |
| 7 | r | 0.00 |

*Note: select Other for valve type.

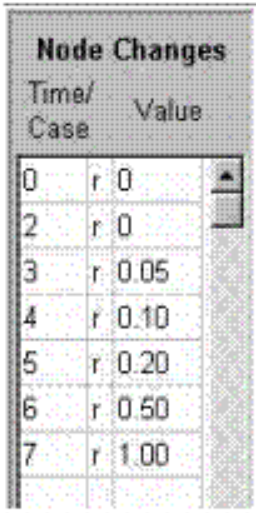


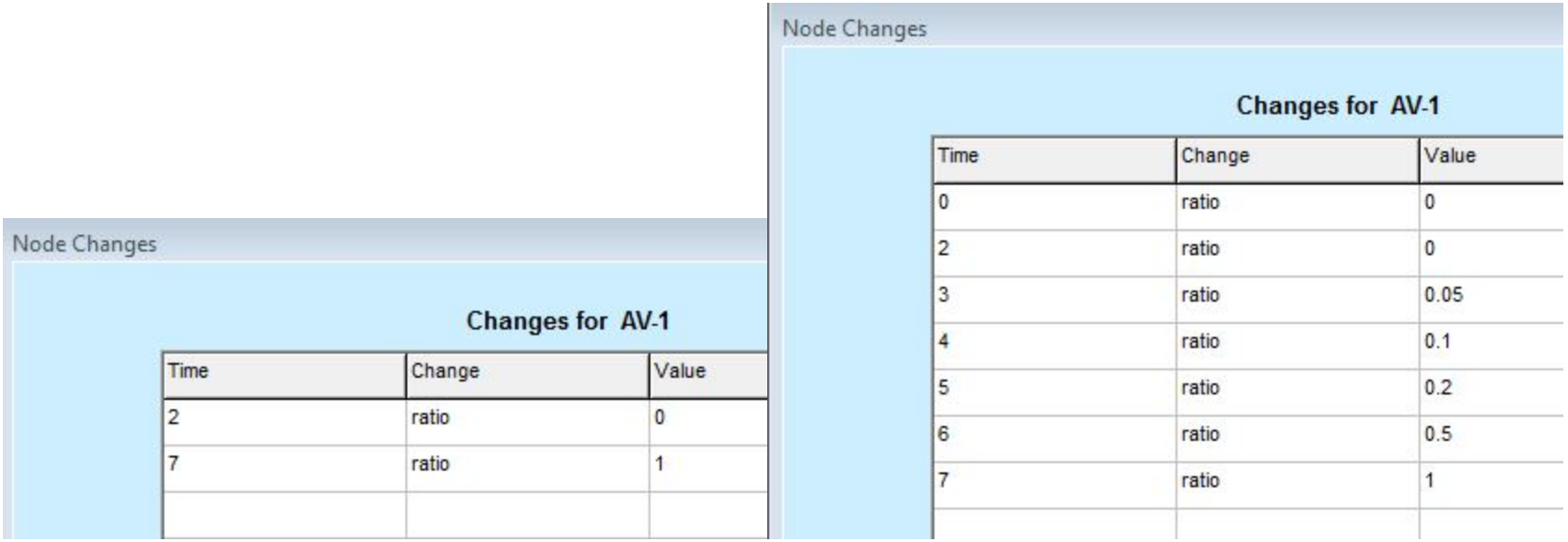

Note on setting up a transient with Active Valve. If more than two settings are required, use the "Cv Ratio" description, for the valve type.

- Valve Opening – Standard Valves
- Example 4a:
  - Valve closed from full close to full open in 5 seconds
    - At time=0sec, Stem Position Ratio = 0
    - At time=2sec, Stem Position Ratio = 0
    - At time=7sec, Stem Position Ratio = 1




- Valve Opening – Non-standard Valves
- Example 4b:
  - Valve closed from full close to full open in 5 seconds (user prescribed change of flow area with respect to time)
    - At time=0sec, Area Ratio = 0
    - At time=2sec, Area Ratio = 0
    - At time=3sec, Area Ratio = 0.05
    - At time=4sec, Area Ratio = 0.10
    - At time=5sec, Area Ratio = 0.20
    - At time=6sec, Area Ratio = 0.50
    - At time=7sec, Area Ratio = 1.00

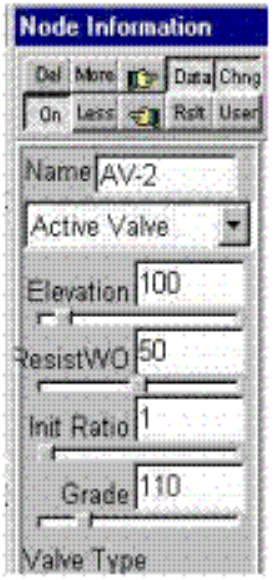


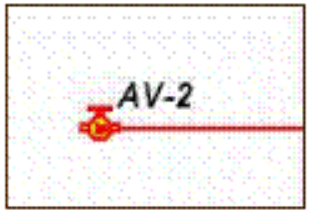

Node Changes
Changes for AV-1
Time
Change
Value
2
ratio
0
7
ratio
1
Node Changes
Changes for AV-1
Time
Change
Value
0
ratio
0
2
ratio
0
3
ratio
0.05
4
ratio
0.1
5
ratio
0.2
6
ratio
0.5
7
ratio
1

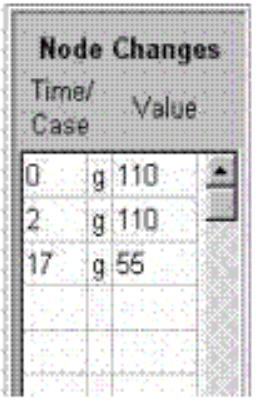
# Variable Input Data - Reservoir Grade

- ✓ Reservoir – Hydraulic grade change
- ✓ Example:
  - Hydraulic grade (water surface elevation) changes from 110ft to 55ft in 15seconds
    - At time=0sec, Hyd. Grade= 110ft
    - At time=2sec, Hyd. Grade= 110ft
    - At time=17sec, Hyd. Grade= 55ft

**Note:** To specify a hydraulic grade change at a reservoir, create an active valve or pump attached to a reservoir, and provide change data as shown for the valve or pump node.



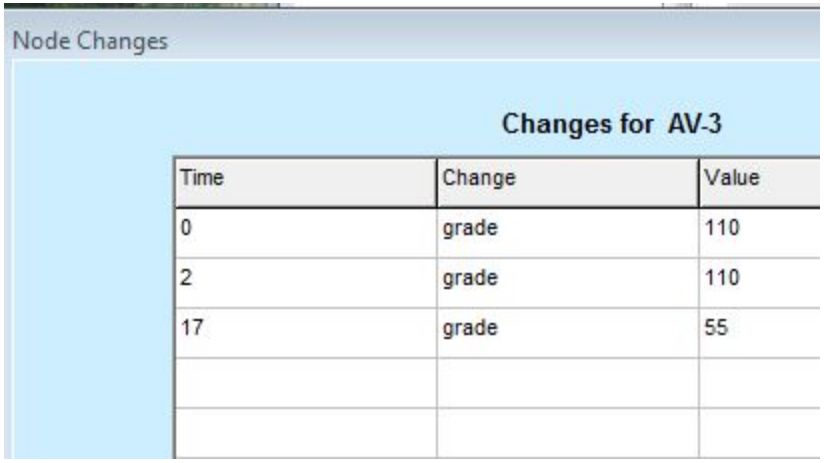




| Node Info | |
|---|---|
| Node Type | Active Valve |
| Name | AV-3 |
| Elevation | 100 |
| Title | |
| Check Valve | ☐ |
| Non-Reopening Che | ☐ |
| Bypass Line | ☐ |
| CV Time | 0 |
| CV Resistance | 0 |
| Bypass Resistance | 0 |
| Grade | 110 |
| Direction | → |
| On/Off | ✓ |
| Resistance 100% | 50 |
| Initial Ratio | 1 |
| Valve Type | Ball |

Note: wide open resistance of the Active Valve used for the grade change must be negligible. So use a very small Resistance, R value or large Flow Coefficient, Cv (Kv) value.

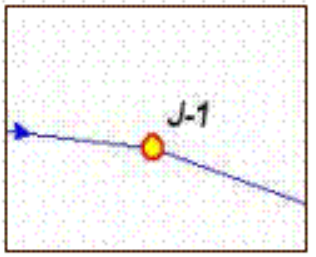
If the Active Valve is also needed to control flow, use a second Active Valve in series (upstream) of the valve with the grade change.

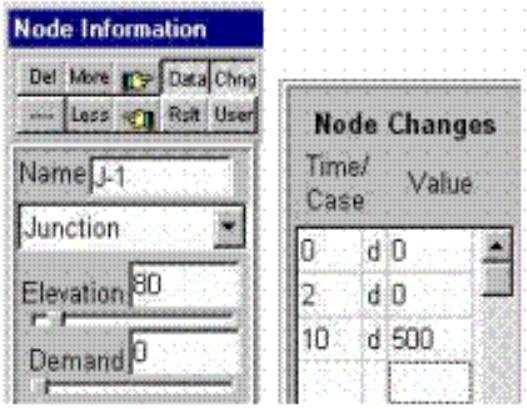


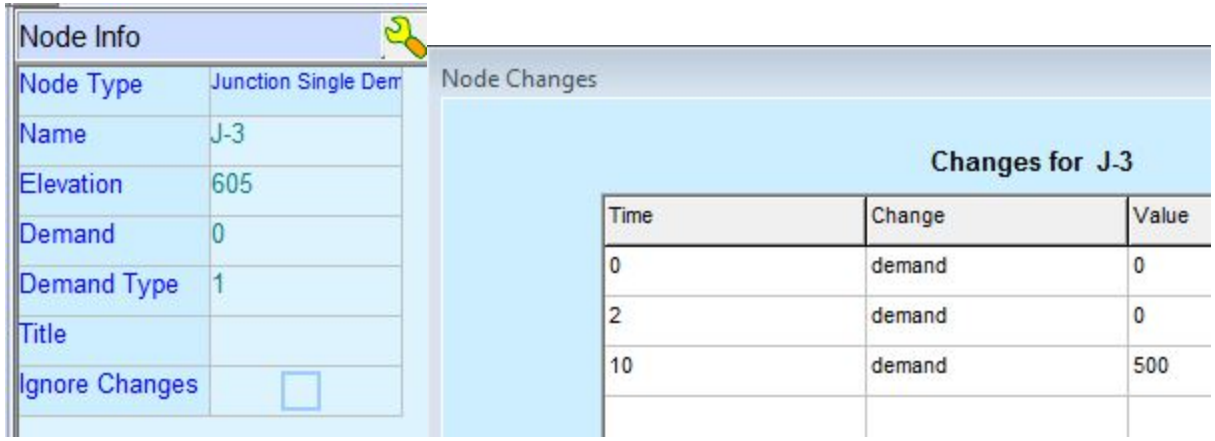
# Variable Input Data - Demand

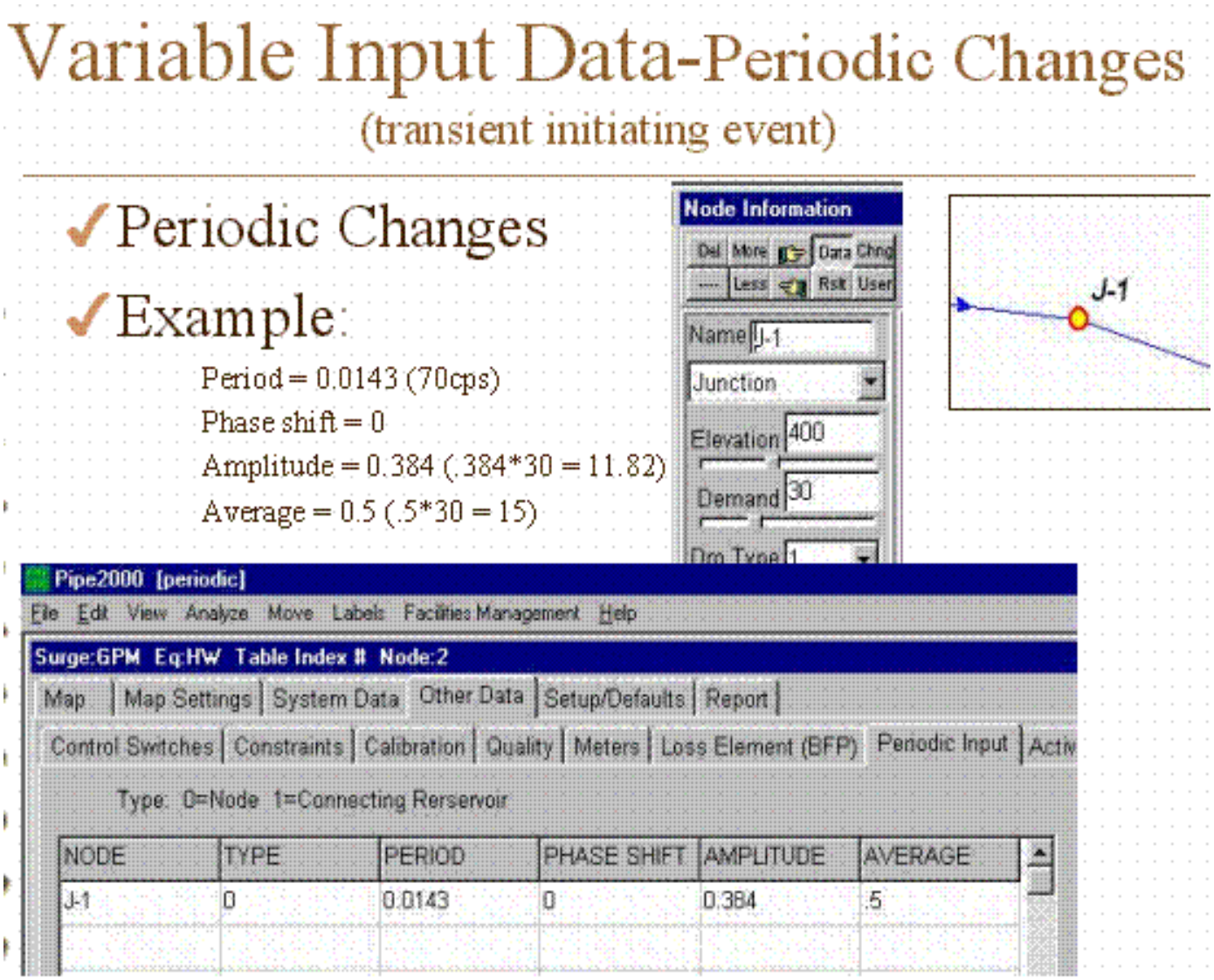
(transient initiating event)

- ✓ Junction Node – Demand Change
- ✓ Example:
  - Increase demand at node J-1 from 0 to 500gpm in 8 seconds
    - At time=0sec, Demand = 0
    - At time=2sec, Demand = 0
    - At time=10sec, Demand = 500gpm




Node Info
Node Type
Junction Single Dem
Name
J-3
Elevation
605
Demand
0
Demand Type
1
Title
Ignore Changes
Node Changes
Changes for J-3
Time
Change
Value
0
demand
0
2
demand
0
10
demand
500




**What is a Periodic Change?** A Periodic Change specifies a periodic variation for a changed parameter. The following data must be specified.

KYnetic Interface - Periodic Menu

**Node** - the node where the parameter change occurs

**Type** - this only applies to a change in the hydraulic grade at a reservoir attached to a valve or pump. For this application the Type = 1.

**Period** - the period in seconds (1/frequency). For the data shown the frequency is 70 cps and the period is 0.0143 = (1/70).

**Phase Shift** - the phase shift in radians is required only if there are 2 or more periodic inputs and there is a phase difference between the different inputs.

**Amplitude and Average -** These are ratios for each of the Variable Inputs

pump: amplitude (average) / rated speed

valve: amplitude (average) / 1.0

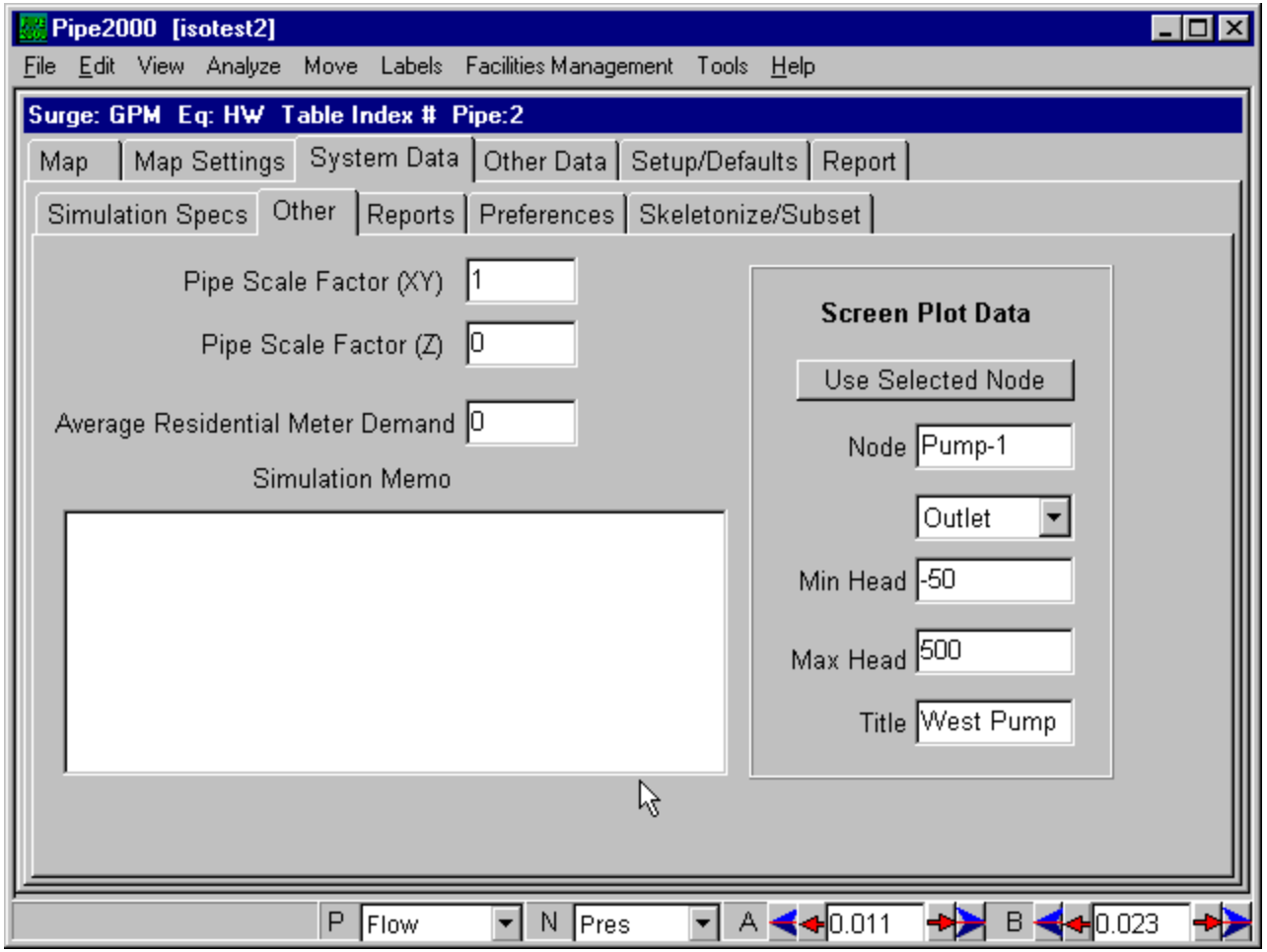
demand: amplitude (average) / initial value

grade: amplitude (average) / initial value

For the example shown, which models a sinusoidal demand variation, the amplitude is 0.834(30) = 11.52 and the average = 0.5(30) = 15

## 1.5.10 Surge - System Data / Other

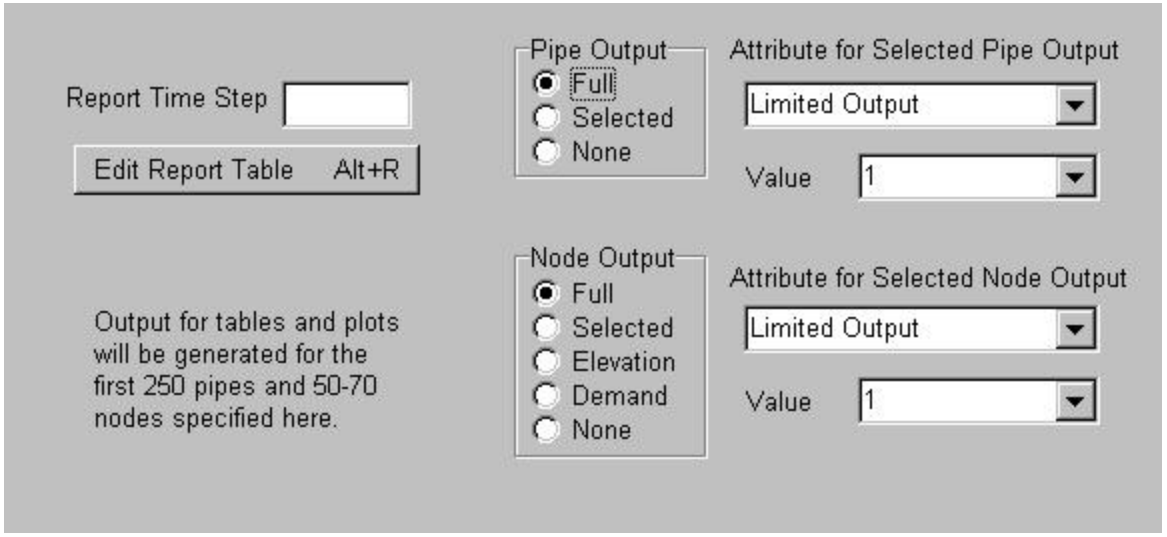
**KYnetic Interface icon:**




**Screen Plot Data** - It is necessary to designate a node for the screen plot which appears as Surge is executing. The best way is to select one node in Layout Mode and then click on **Use Selected Node**. Then fill in the rest of the data (Title is optional).

## 1.5.11 Surge - System Data / Reports

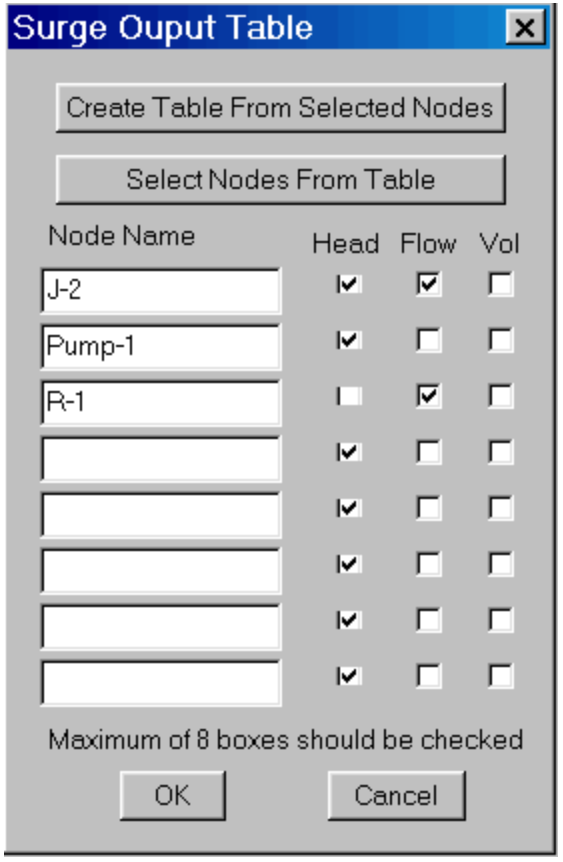
Surge - System Data / Reports




The Analysis Report Setup menu or System Data / Reports (Classic) allows the user to designate desired outputs. The comprehensive Tabulated Report table appears in the Surge Report and is limited to 8 items.

**Report Time Step** - this is the time step used in the Tabulated Output. This defaults to the computational time step which is usually very small. Using this default will result in a very long report. Units are in seconds.

**Edit Report Table** - Check on this to bring up the following screen. It is best to first go into Group Mode and select up to eight nodes for this report. Then click on **Create Table from Selected Nodes**. The nodes will appear and you can select the desired output parameters.




The Node Output options are used to select the nodes which can be accessed for customized plots and tables. Because of the very large amount of results data this output is limited to 70 nodes. You should designate one of the following:

**Full** - all nodes are selected. If there are more than 70, the first 70 will be used.

**Selected** - only nodes selected using the designated User Data attribute will have output. Normally the attribute "Limited Output" is selected and the value 1 used to designate the included nodes.

# 1.5.12 Surge - Reviewing and Presenting Surge Results

Surge - Reviewing and Presenting Results

Reducing the Number of Stored Results
Max/Min Heads (Pressures) at Nodes
Graphs and Tables
Surge Node Results
Group Node Results
Tabulated Reports
Profiles
Selecting Nodes for Limited Output
Surge Model Results – Excessive Pressure Spiking

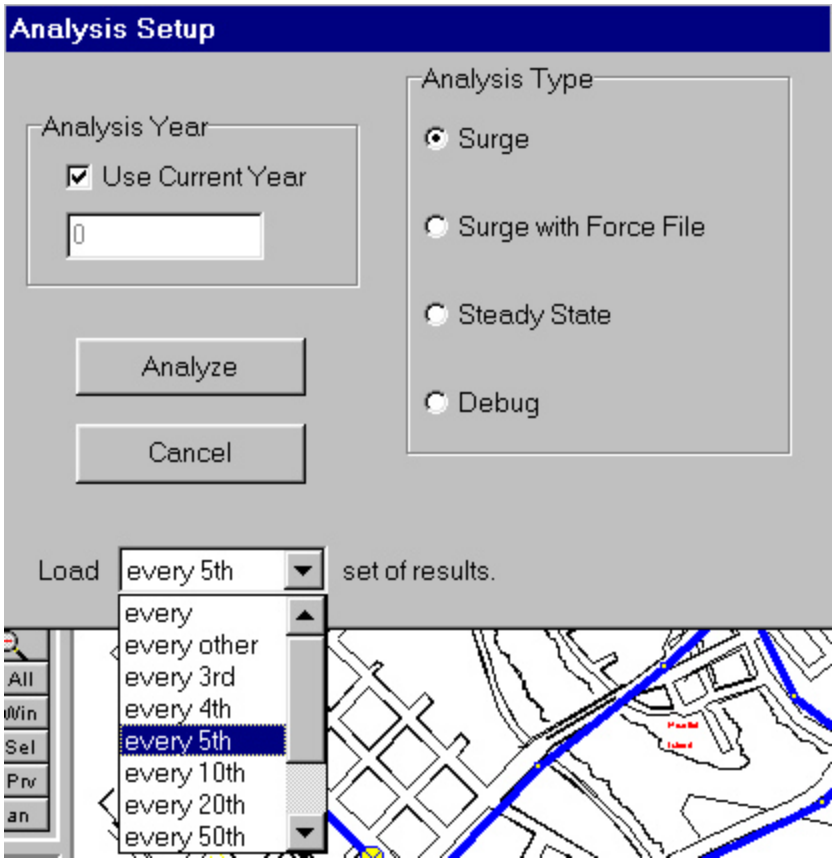
---

All the Pipe2018 capabilities for showing results are available for Surge (Tabulated Reports, Labels, Contours, Profiles, etc.). This section covers some special considerations for reviewing and presenting Surge results.

For licenses prior to Pipe2006 and when the "Use Former Analysis Method" box is checked, the full number of results is not stored. This is because a surge analysis produces so many results, it was not practical with this early version of the calculation engine to store every result for review. To do so may occasionally caused memory problems and excessive computational delays. In addition, every result is not normally required to evaluate the overall results. The following discussion details how large numbers of results are handled in Surge.

### Reducing the Number of Stored Results (Not Required for Pipe2018 though Pipe2008)

When the analysis is carried out with the Former Analysis Method (or earlier licenses), we can reduce the amount of results stored, hence increasing the speed of the analysis by not storing every result, as shown below. Because the computational time increment is often very small, storing every 5th or even 10th result may still produce very detailed graphs and tables.

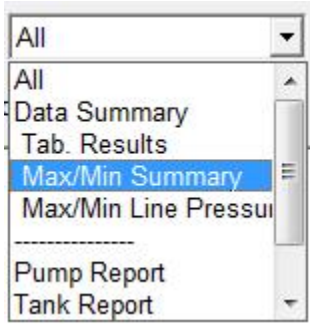



To further reduce the amount of results stored, the number of nodes for which results are stored is limited for larger systems as discussed in a subsequent section (see Selecting Nodes for Limited Output).

**Max/Min Heads (Pressures) at Nodes**

This is very useful information which is always available following the Surge analysis. You can display the max/min heads (pressure or HGL) at all nodes by selecting the Max/Min results label option. This will produce a display such as the one shown below.

Also the tabulated Report for Surge always includes a Max/Min Head Summary which can be selected from the Report Section Selector as shown.




This produces the report shown below.

SUMMARY OF MAXIMUM AND MINIMUM HEADS:

| Position no. | MaxHead (ft) | MinHead (ft) | Time Reverse Grad. | MaxPressure (psi) | MinPressure (psi) | MaxTime (sec) | MinTime (sec) |
|---|---|---|---|---|---|---|---|
| I-AV-1 | 177.83 | -13.29 | 1.587 | 77.058 | -5.760 | 11.00840 | 3.09960 |
| I-AV-2 | 135.44 | 102.01 | 0.000 | 58.693 | 44.204 | 9.80304 | 2.65680 |
| J-1 | 179.08 | 38.79 | 0.000 | 77.600 | 16.811 | 9.81534 | 3.41940 |
| J-10 | 226.34 | 6.10 | 0.000 | 98.081 | 2.643 | 11.64797 | 5.90401 |
| J-11 | 277.15 | -33.21 | 1.132 | 120.098 | -14.391 | 12.26295 | 5.97781 |
| J-12 | 182.20 | 33.82 | 0.000 | 78.952 | 14.656 | 10.06133 | 3.16110 |
| J-14 | 188.40 | 38.00 | 0.000 | 81.639 | 16.468 | 12.04156 | 5.86711 |
| J-15 | 180.27 | -15.58 | 0.738 | 78.118 | -6.751 | 8.91747 | 3.00120 |
| J-16 | 180.86 | 7.89 | 0.000 | 78.371 | 3.418 | 10.46722 | 3.38250 |
| J-2 | 257.30 | -33.21 | 1.501 | 111.496 | -14.391 | 11.95546 | 5.89171 |
| J-3 | 181.36 | -10.02 | 1.218 | 78.588 | -4.340 | 11.02070 | 3.11190 |
| J-4 | 181.80 | 32.70 | 0.000 | 78.779 | 14.170 | 12.86563 | 7.66291 |
| J-5 | 218.86 | 5.31 | 0.000 | 94.838 | 2.301 | 13.18541 | 4.56330 |

If negative pressures occur this report also presents the fraction of the simulation time at each node for which pressures are negative (and reverse leaks (intrusion) possible).

The Max/Min Line Pressure Report is as follows:

SUMMARY OF MAX/MIN LINE PRESSURES:

| START NODE | END NODE | MAX PRESS. psi or kPa | MIN PRESS. psi or kPa | Steady Press. at end nodes | | MaxPrDrop at end nodes | | LowestPress at end nodes | | |
|---|---|---|---|---|---|---|---|---|---|---|
| J-2 | J-10 | 111.50 | -14.39 | 51.93 | 55.29 | 66.32 | 52.64 | -14.39 | -11.04 | Cavitation Potential |
| J-10 | J-14 | 98.08 | 2.64 | 55.29 | 56.81 | 52.64 | 40.34 | 2.64 | 4.16 | |
| J-10 | J-11 | 120.10 | -14.39 | 55.29 | 55.43 | 52.64 | 69.82 | -14.54 | -14.39 | Cavitation Potential |
| J-12 | J-16 | 78.95 | 3.42 | 61.92 | 68.14 | 47.26 | 64.72 | -2.81 | 3.42 | |
| J-12 | I-AV-2 | 78.95 | 14.66 | 61.92 | 52.33 | 47.26 | 8.13 | 14.66 | 5.07 | |
| J-15 | J-1 | 78.12 | -6.75 | 68.35 | 62.05 | 75.10 | 45.24 | -6.75 | -13.05 | |
| J-16 | J-3 | 78.59 | -4.34 | 68.14 | 70.32 | 64.72 | 74.66 | -6.52 | -4.34 | |
| J-4 | J-7 | 78.78 | 14.17 | 60.15 | 53.74 | 45.98 | 38.55 | 14.17 | 7.76 | |
| J-4 | J-14 | 81.64 | 14.17 | 60.15 | 56.81 | 45.98 | 40.34 | 14.17 | 10.83 | |

Calculations are done to alert the user when there is a possibility that cavitation pressure could occur at some interior location within the pipeline. The largest of

the pressure drops at the end of the pipe are used to make this assessment of the worst possible situation. If the largest pressure drop could result in a pressure below cavitation the "Cavitation Potential" warning, as shown above, is printed. The user can insert one of more interior nodes to determine if interior cavitation actually might occur. The expectation is that it won't, but the user is alerted to this possibility.

### Graphs and Tables

For surge analysis, graphs and tables of pressures and flows at selected nodes are results which are commonly reviewed and presented. For each node up to six sets of results are generated and of these the following are available for review if the node is selected for output. Because of the excessive amount of results generated for even moderately sized systems, it is desirable to designate the nodes to be selected for output (up to 30 nodes can be selected - see Selecting Nodes for Limited Output). If you do not designate nodes, the first 30 or so (in the node table) will be used. For small systems all nodes will be automatically selected. For a selected node the following values are displayed.

**junction nodes**

pressure (head or HGL) (1)

(or) flow in first connecting pipe (1)

**component node (pump, valve etc.)**

pressure on each side (2)

(or) flow on each side (2)

**SDO node**

pressure internal and external (2)

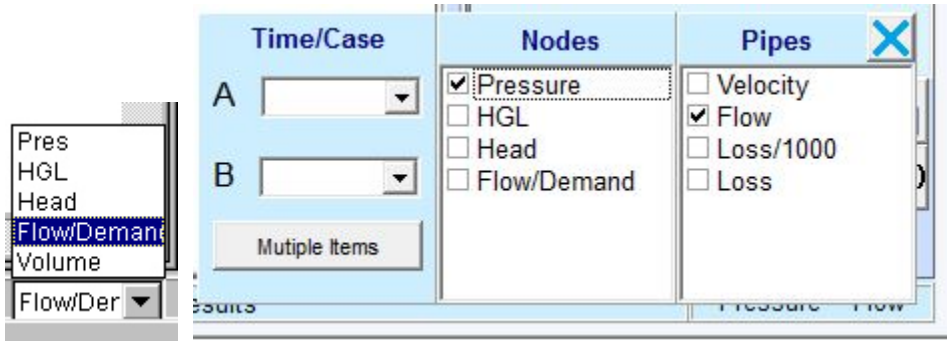
(or) flow side1, side2 and external (3)

(or) volume (air valves and air vessels) (1)

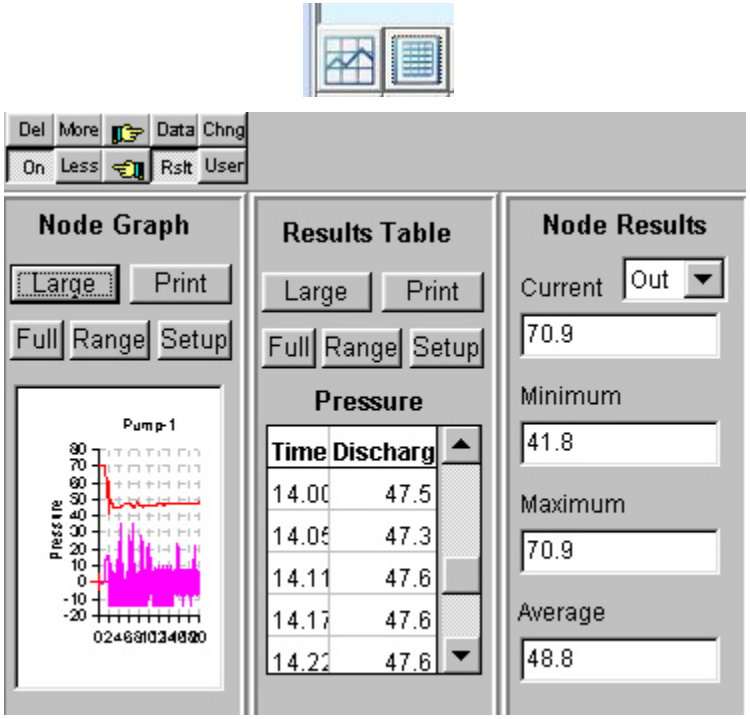
You can generate graphs and plots for one node at a time (layout or fixed mode) or for groups of nodes (group mode).

### Surge Node Results

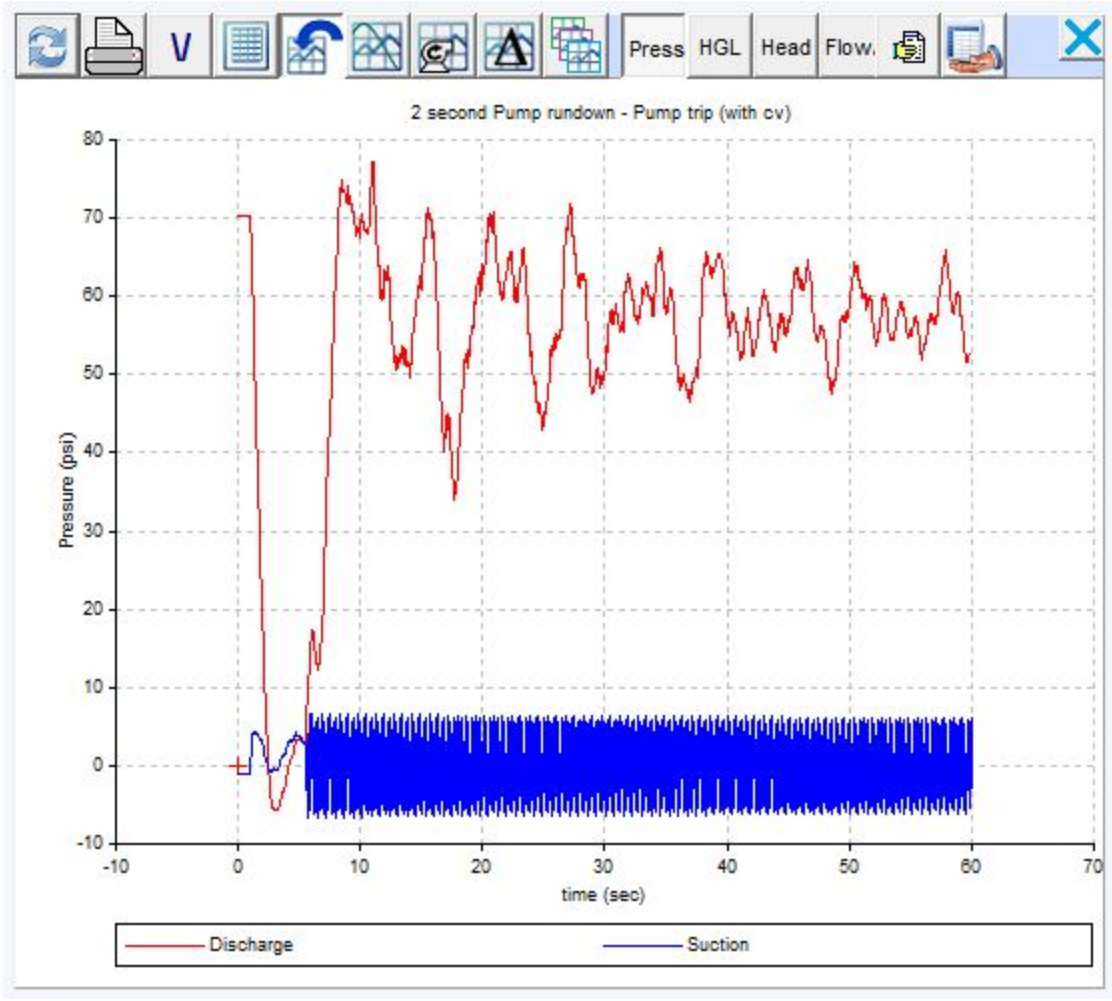
To see the results for a particular node, turn on the Results (Rslt) button and select that node and select the parameter to display from the Results Selector menu at the bottom of the program window as shown below.

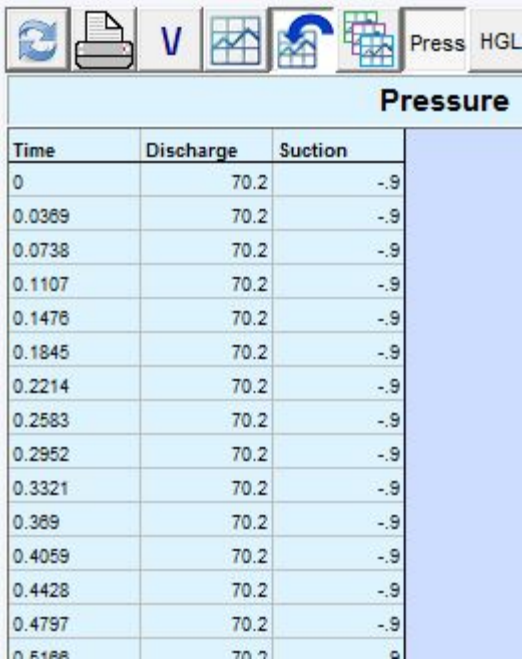



This will produce the Node Information Results boxes shown below which will include a graph or table (icons for KYnetic shown) which will have 1, 2, or 3 sets of results displayed as noted above. The picture below is for pump pressures which has 2 sets of results (inlet and outlet sides)




Classic

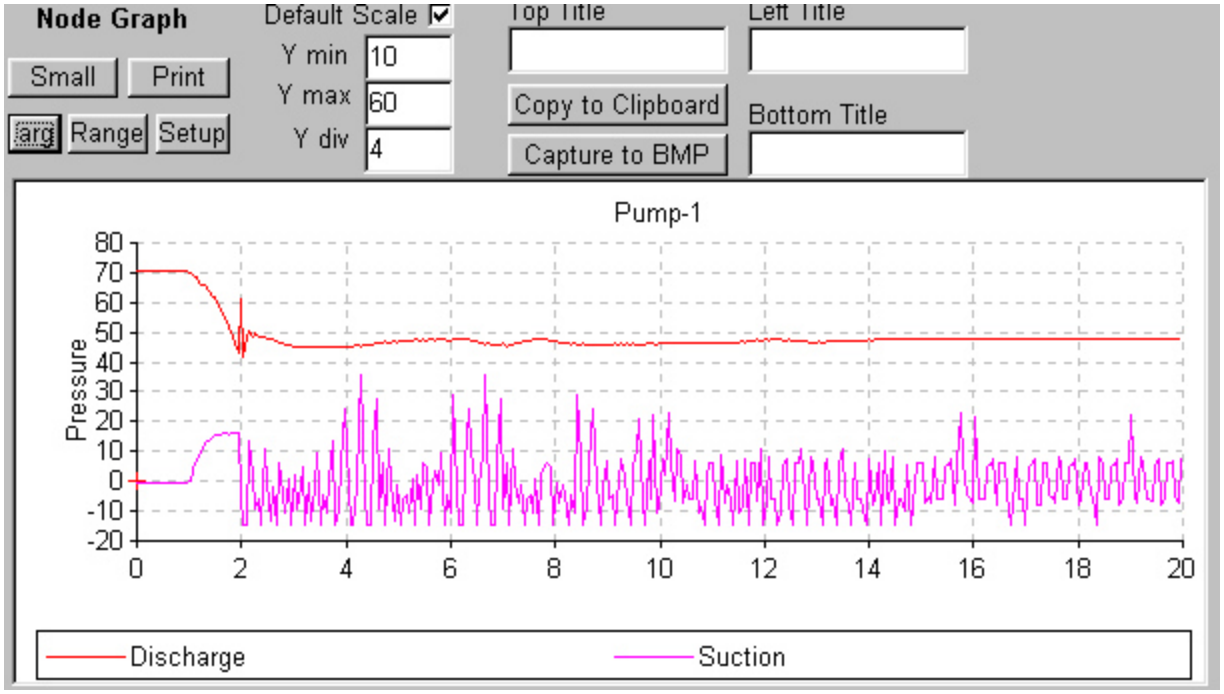


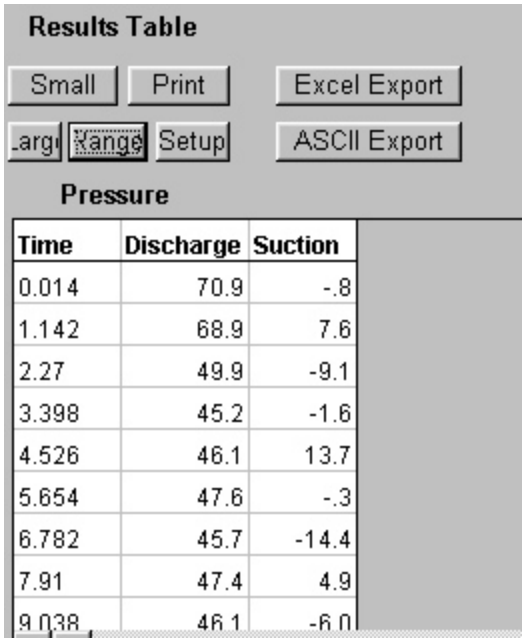

| Time | Discharge | Suction |
|---|---|---|
| 0 | 70.2 | -.9 |
| 0.0369 | 70.2 | -.9 |
| 0.0738 | 70.2 | -.9 |
| 0.1107 | 70.2 | -.9 |
| 0.1476 | 70.2 | -.9 |
| 0.1845 | 70.2 | -.9 |
| 0.2214 | 70.2 | -.9 |
| 0.2583 | 70.2 | -.9 |
| 0.2952 | 70.2 | -.9 |
| 0.3321 | 70.2 | -.9 |
| 0.369 | 70.2 | -.9 |
| 0.4059 | 70.2 | -.9 |
| 0.4428 | 70.2 | -.9 |
| 0.4797 | 70.2 | -.9 |

KYnetic

The graphs can be expanded in the Classic version as shown below and printed.

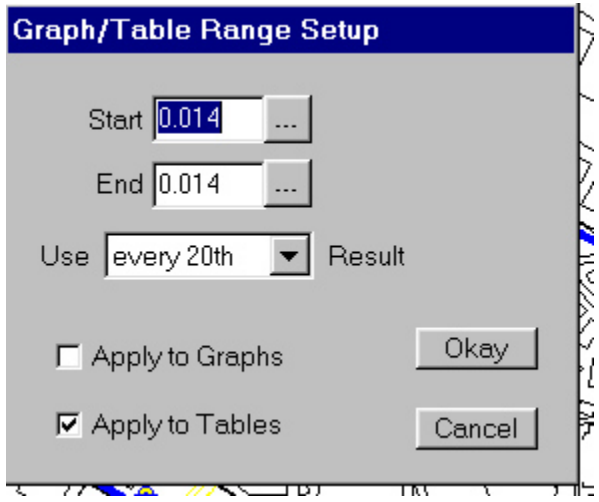



The Classic tables can also be expanded as shown below



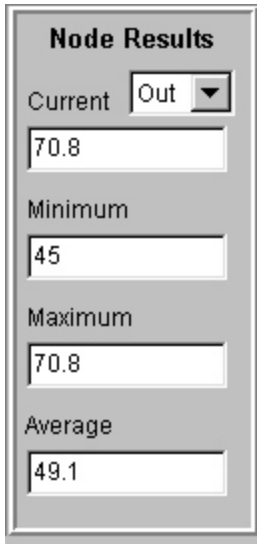

| Time | Discharge | Suction |
|---|---|---|
| 0.014 | 70.9 | -.8 |
| 1.142 | 68.9 | 7.6 |
| 2.27 | 49.9 | -9.1 |
| 3.398 | 45.2 | -1.6 |
| 4.526 | 46.1 | 13.7 |
| 5.654 | 47.6 | -.3 |
| 6.782 | 45.7 | -14.4 |
| 7.91 | 47.4 | 4.9 |
| 9.038 | 46.1 | -6.0 |

For both graphs and tables, the range and frequency of values displayed can be controlled by clicking on the Range button and providing data in the locations indicated as shown below. This is particularly useful for tables since the small time increments used in the Surge analysis will result in very long tables.



### Group Node Results (Classic)

Graphs and Tables showing multiple Node Results can be produced in Group Mode. Just select the multiple nodes desired in Group Mode and a graph or table showing results for the selected nodes will be generated. For components and SDO results, only one pressure or flow per node will be displayed for Group displays and this selection is controlled by the drop-down selector "Current" which appears in the Node Results box as shown below (must be in Layout mode to view).




## Other Considerations

### Tabulated Reports

Surge performs both a steady state (to determine initial conditions) and a transient analysis when a surge analysis is selected. A tabulated report is generated for each analysis and can be accessed using the Report button and the Load/Swap button to go between the steady state and surge output reports. If the transient analysis is selected the additional results (labels, graphs, tables, profiles, etc.) are for the transient results. If you wish to display steady state results you should select the Analysis and then select Steady State and perform the steady state analysis.

### Profiles

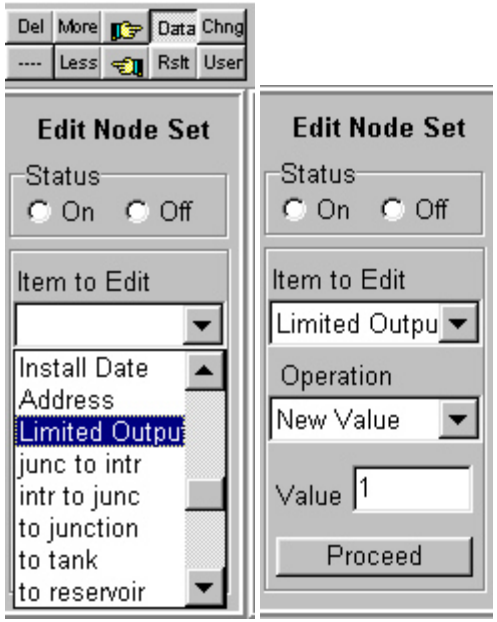
The Pipe2018 profiles which show the Max/Min head envelope are a particularly useful tool for reviewing results. This envelope can be generated for any path within the piping system. The maximum/minimum envelope is developed using all results at all nodes. However, the profile for a selected time will use only results for selected nodes which are included in the profile.

### Selecting Nodes for Limited Output (Older versions)

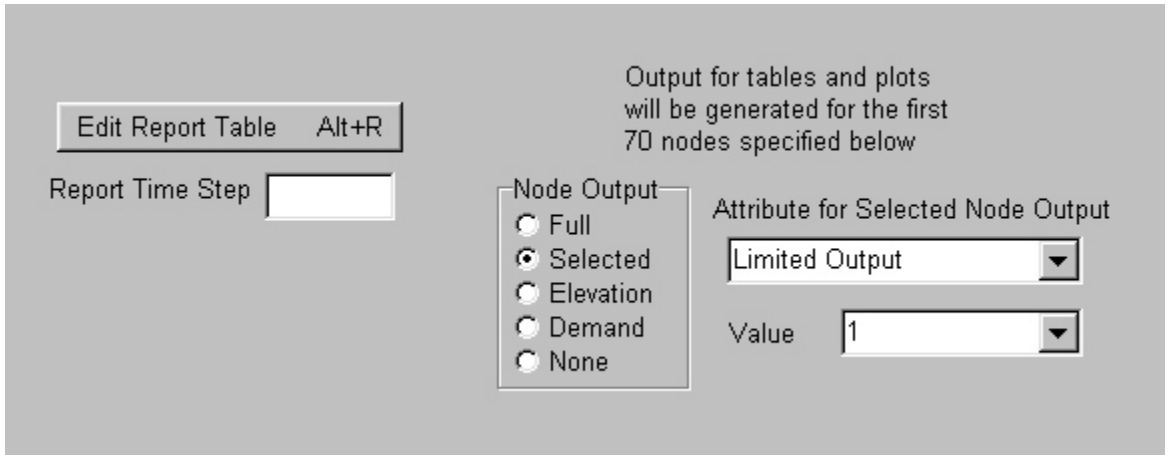
For system with 25 -30 or more pipes, selecting the nodes for which results are available is an important step to assure that you have access to the results

required to evaluate the transient. The normal procedure for doing this is as follows.

1. Select Group Mode
2. Select up to 25-30 nodes for detailed results
3. Turn on the Node Information Data box and select Limited Output as shown below. Then provide a New Value to identify the group (usually 1) for the Limited Output attribute for the group of nodes. Make sure you have first set this attribute to zero or a different value for all the other nodes.



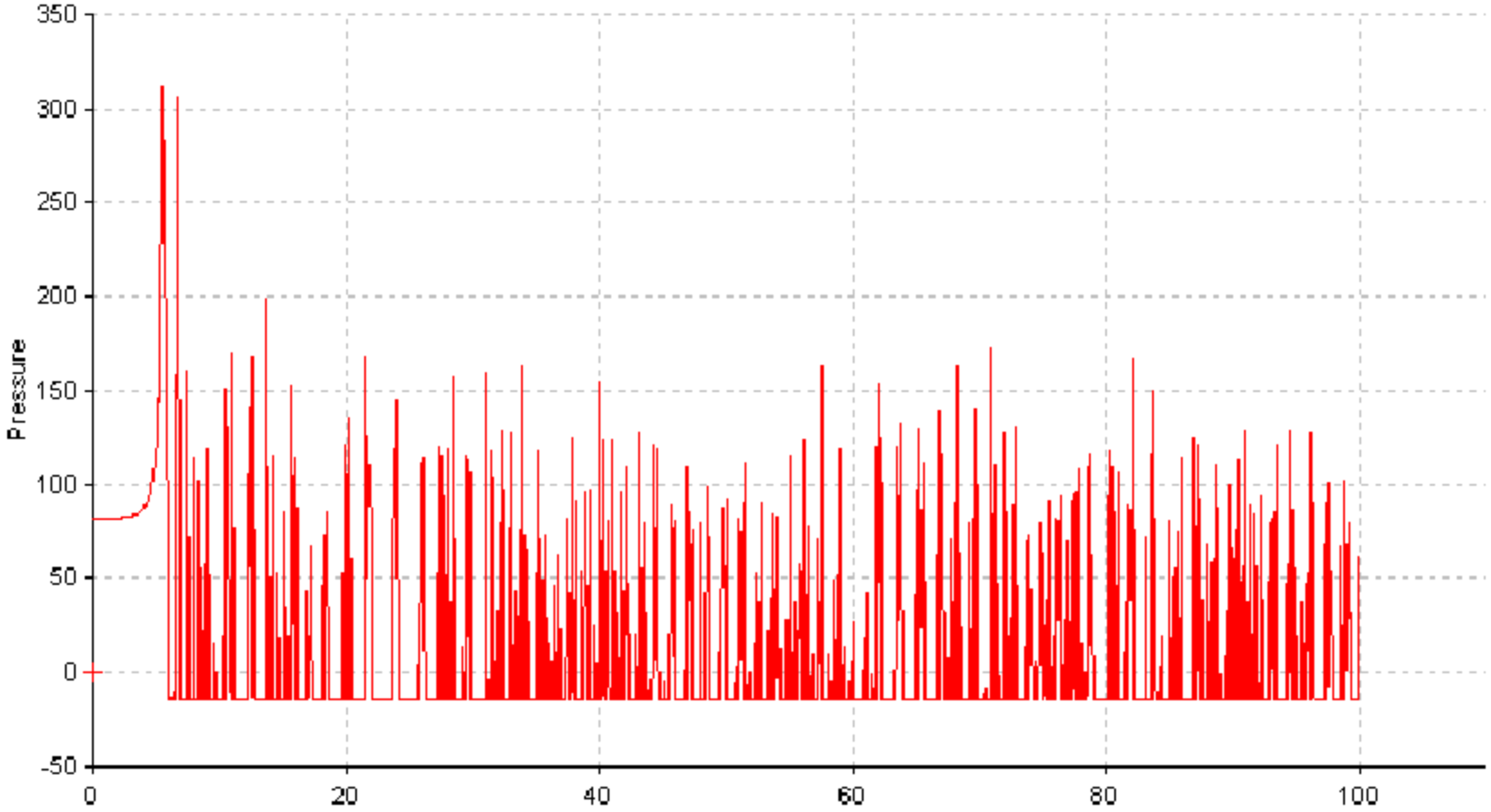

Finally under Report Setup menu or System Data/Reports (Classic) you need to designate Selected Output, the attribute (Limited Output) and value (1) used to designate the selected nodes as shown below.




Of course, you can use this procedure to select a number of groups with different Limited Output values (say 1, 2, and 3) and designate the desired value to use for a particular run.

**Surge Model Results – Excessive Pressure Spiking**

Sometimes the results of a transient analysis show excessive spiking of the pressure as shown




The solution continues as pressure spikes and no final steady state result will be reached. The spikes may even grow and reach very high values. This occurrence is almost always due to:

1) Cavitation – spikes due to cavity collapse
2) Check Valve action – opening and closing of CV's

Either one or a combination of these situations can produce an unstable result.

If this type of solution occurs due to check valve action at a pump which has been shut down then the pump is operating in an abnormal fashion (flow reversals, etc.). Therefore, it is essential that a pump file be used in the analysis and the pump trip option used for the pump shutdown. In this manner the behavior of the pump can be calculated. Also the effects of inertia and check valve properties can be evaluated.

When these results are obtained it is important to view the results more in a qualitative than quantitative manner. The actual calculated magnitude of the spikes are very sensitive to the system data and small changes can significantly affect the magnitude of the pressure spikes. The important result is that the response is very unstable. Because of the sensitivity of actual spike magnitudes to the timing of the events and data it is not reasonable to compare solutions based on the highest calculated pressure spikes. The solutions are just too sensitive. What can be concluded is that the transients can be unstable and excessive pressure spikes are possible.

If you want to further evaluate the cause of an unstable result you can:

1) Set the default Cavitation Head to a very low value (such as -1000 ft. (m)). When this is done cavitation and the resulting unstable solution will not occur.
2) Either remove check valves or set them to non-reopening.

These actions should allow for the calculation of a stable response and will allow you to evaluate the cause of the instability for your system.

## 1.5.13 Surge QuickStart Example

Surge QuickStart Example

Step 1 - Initial Preparation

Step 2 - System Layout

Step 3 - Provide pipe and node data

Step 4 - Save data file

Step 5 - Analyze System and Review Results

See the Demonstration Examples videos on the KYPipe web site.

---

This will guide you through the complete layout development, data entry and hydraulic analysis of a simple pipe network. We recommend that you run Surge 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, system data selections, background preparation and pipe type preparation.

**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 screen appears. Select Surge. Click OK and go to System Data icon or System Data / Simulation Specs (Classic). Specify the flow units (CFS), head loss equation to use (Hazen-Williams), and total simulation time (10 seconds).

Return to the Map.

**c. background preparation**
You can import a background 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 Surge calculate pipe lengths.

Click on Grids Menu or Map Settings / Grids (Classic) - 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.

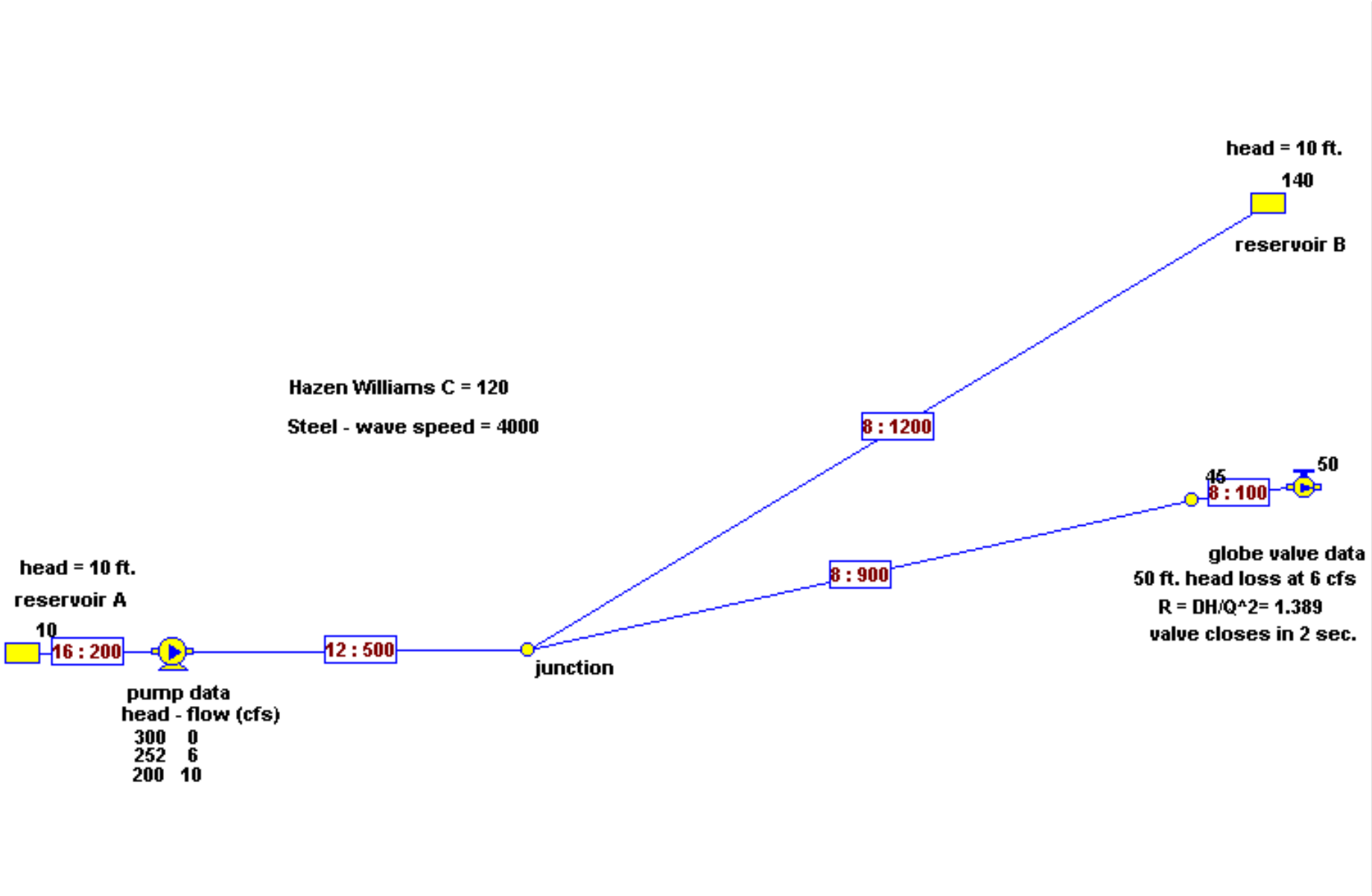
Return to the Map

**d. pipe type list**
Click on Components menu icon or Setup/Defaults (Classic) and Pipe Type and make sure that 16, 12, and 8 inch steel pipe, HWC = 120 and a wave

speed of 4000 ft/s are included in the Pipe Schedule you select. This will simplify your data entry requiring only a single selection to define all the pipe data.

### 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.


head = 10 ft.
140
reservoir B
Hazen Williams C = 120
Steel - wave speed = 4000
8 : 1200
45
8 : 100
50
8 : 900
globe valve data
50 ft. head loss at 6 cfs
R = DH/Q^2= 1.389
valve closes in 2 sec.
head = 10 ft.
reservoir A
10
16 : 200
12 : 500
junction
pump data
head - flow (cfs)
300 0
252 6
200 10

The system we wish to lay out is shown above. The node elevations are noted. The diameter and roughness are noted in a box for each pipe. The development of the pipe system model is accomplished in four steps. You should be in layout mode (button - top left) to proceed.

**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:

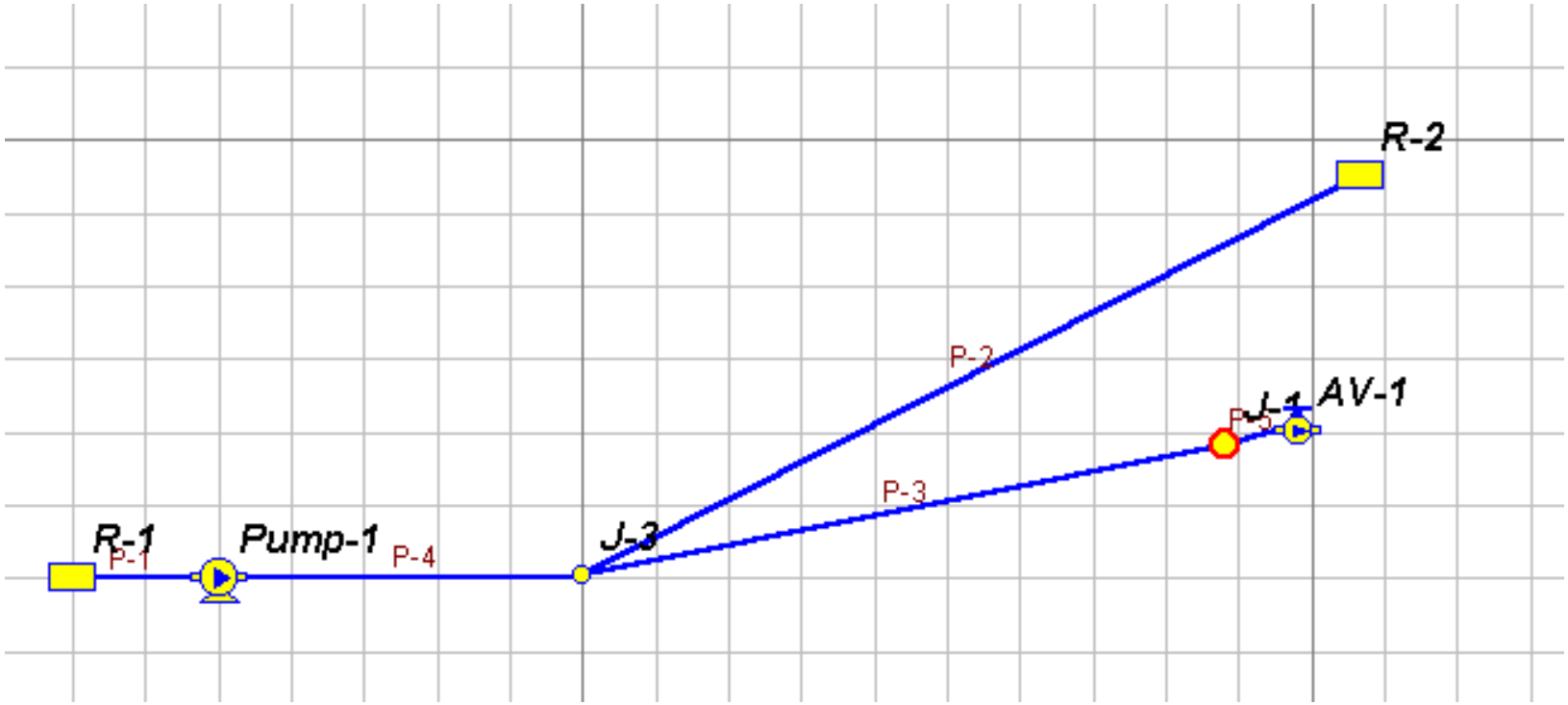
1) RC on grid-line intersection near the left edge to make first node
2) move mouse 200 feet (2 blocks) to right and RC
3) move mouse 500 feet to right and RC
4) move mouse 1200 feet right and up and RC (note the distance can be observed at the bottom of the screen next to "D")
5) select node at junction and move 900 feet right and up and RC (this node is provided to add a surge tank, if required).
6) move mouse an additional 100 feet and RC

Now all the pipes and nodes are laid out. Note all nodes are either junction or intermediate nodes and Surge 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 LC to 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
3) Select node at valve and change node type to Active Valve.
4) Select node at pump location and change node type to Pump.

The system should now look as shown below.




## Step 3 - Provide pipe and node data

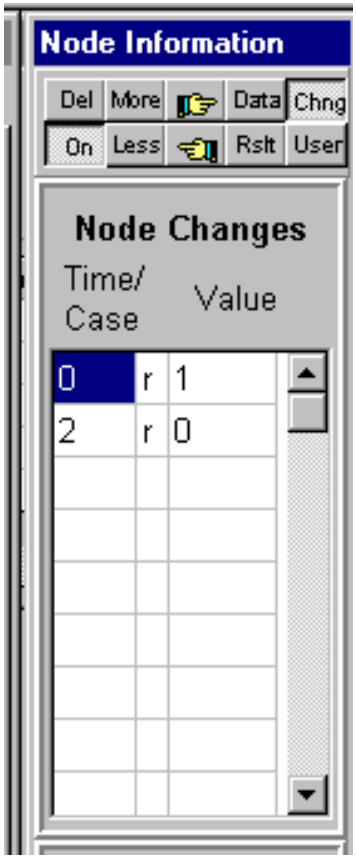
**a) Select each pipe and node and provide data**

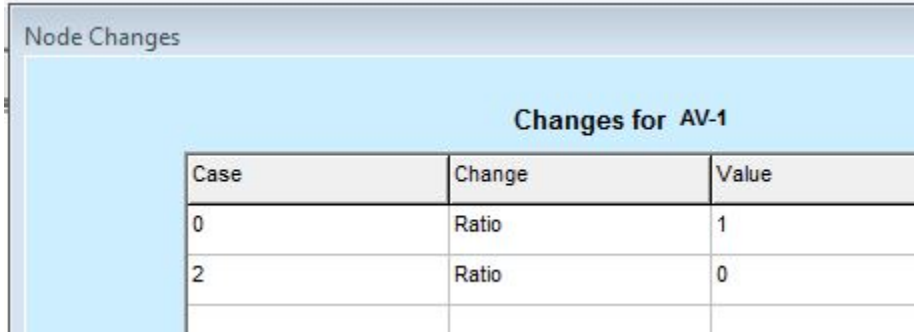
1) Select each pipe and click Pipe Type (Pipe Information Window) and select choice from drop down list. Select Steel - 16 for pipe from Reservoir A and the appropriate selections for the rest of the pipes. Note that default roughness and wave speed values are provided.
2) Select each Reservoir and provide values shown for Grade (HGL) and Elevation
3) Select each junction and intermediate node and provide Elevation
4) Select the active valve and provide type (globe), elevation, and wide open (WO) resistance.
5) Select the pump and check Table for the pump type and enter the head flow data in the table for ID =1.

**b) Provide Variable Input Data (Changes)**

1) Select the Active Valve and open just the Change box (Node Information) and provide the data shown below to specify a 2-second

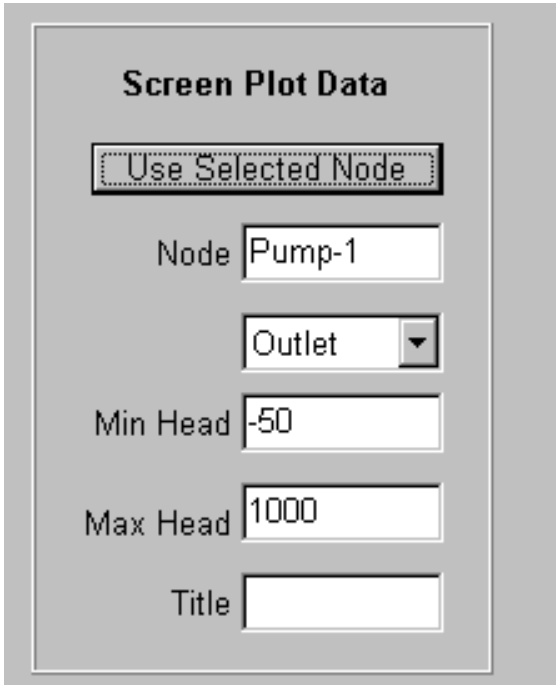
closure. Note the r denotes the ratio for the stem position and is entered by clicking on this location and selecting ration from the pop-up list.




**d) Select Node for Screen Plot Display**

1) We want to observe the pump outlet pressure during the analysis. Select the pump (layout mode). Click on Other Data icon or System Data/Other (Classic) and Use Selected Node on the menu which appears (shown below). Select Outlet and provide the limits for the plot as shown.




## Step 4 - 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 pop-up menu. Such as QSSurge (for Quick Start Surge).

### Step 5 - Analyze System and Review Results

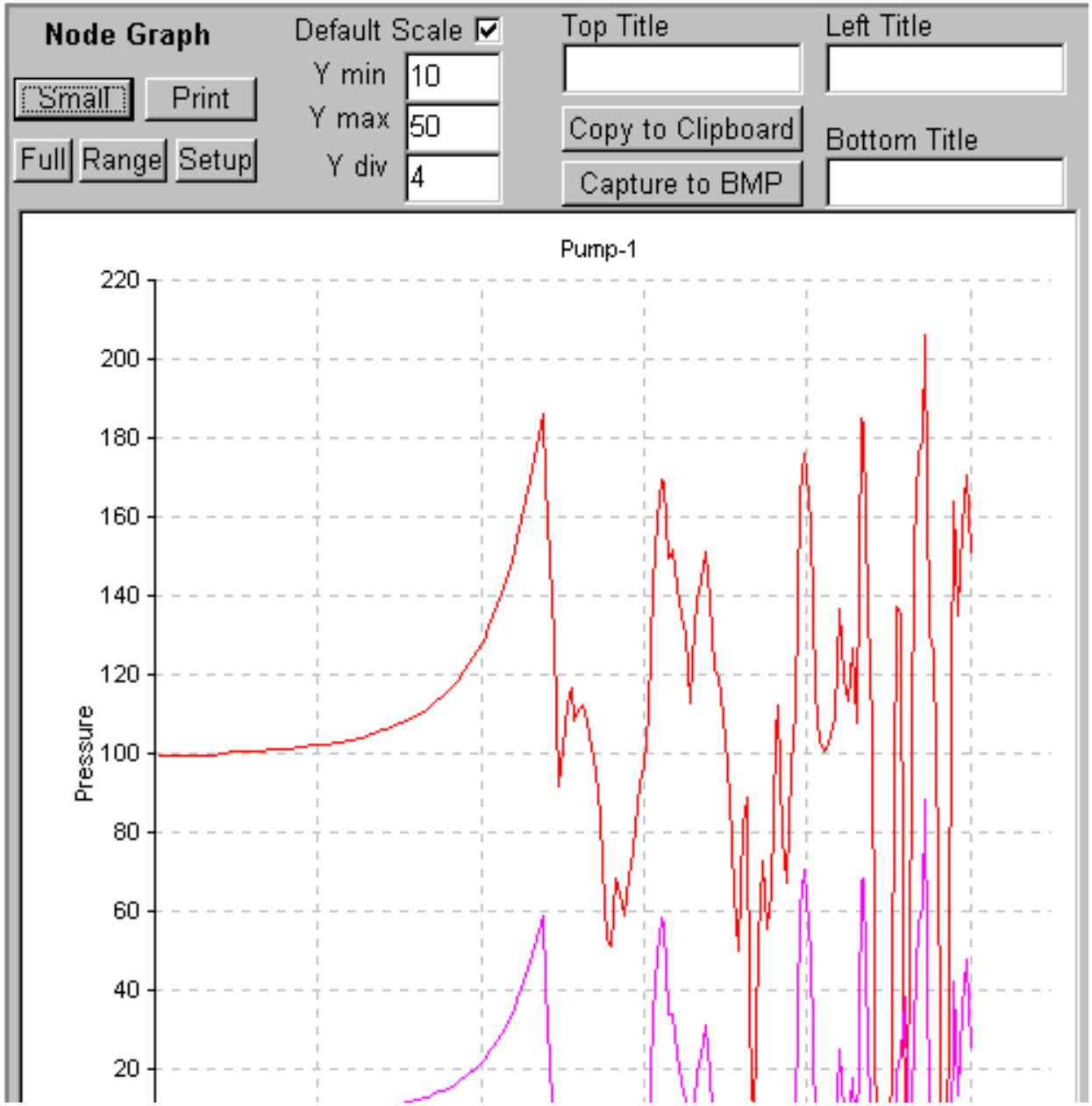
**a. check data and run analysis**

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 Surge and click Analyze on the pop-up menu to accept the defaults (Analyze with Surge, Use Current Year, and Load Every Result). A plot of the pump discharge pressure will appear. Click OK when finished to load results for review.

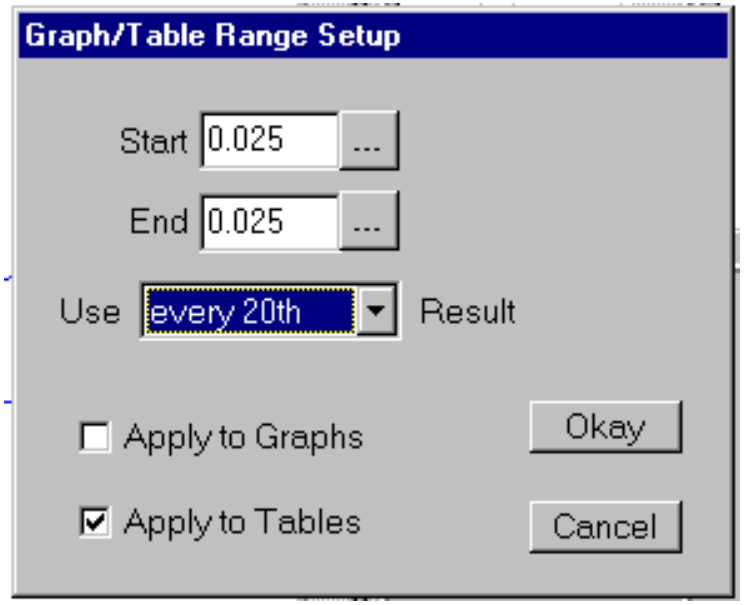
**b. review results**

The results can be reviewed on the schematic using Results Labels, by looking at the tabulated output or by generating customized tables and plots.

1) Click on Report. Any warnings or error messages will appear. Click Load/Swap to load the tabulated results for the transient analysis. These are presented in three sections. You can select a section or all sections and scroll through the tabulated summary of data and results. Click on Load/Swap to go between the results for the initial steady state results (KYPIPE) and transient results (Surge). Note the table of Max/Min heads at the end of the Surge report provides a very useful overview.
2) Click on Labels (Main menu) and select 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 Results Selector at the bottom of the program window to change the node results parameter to head or HGL.
3) Plots and Tables - You can produce plots for heads at all nodes and flows at nodes other than junctions. Select the Pump (layout mode) and under Node Information click the Table or Graph icons/boxes. (Classic) Click on Full and you will see the transient pump suction and discharge pressure as shown below.


Node Graph
Small
Print
Full
Range
Setup
Default Scale
Y min
10
Y max
50
Y div
4
Top Title
Copy to Clipboard
Capture to BMP
Left Title
Bottom Title
Pump-1
Pressure
220
200
180
160
140
120
100
80
60
40
20

(Classic) Click on Small to display map. For the Results Table showing every result may result in many values (at each 0.025 seconds). To limit this click on Range and select Every 20th Result and Apply to Tables as shown below.



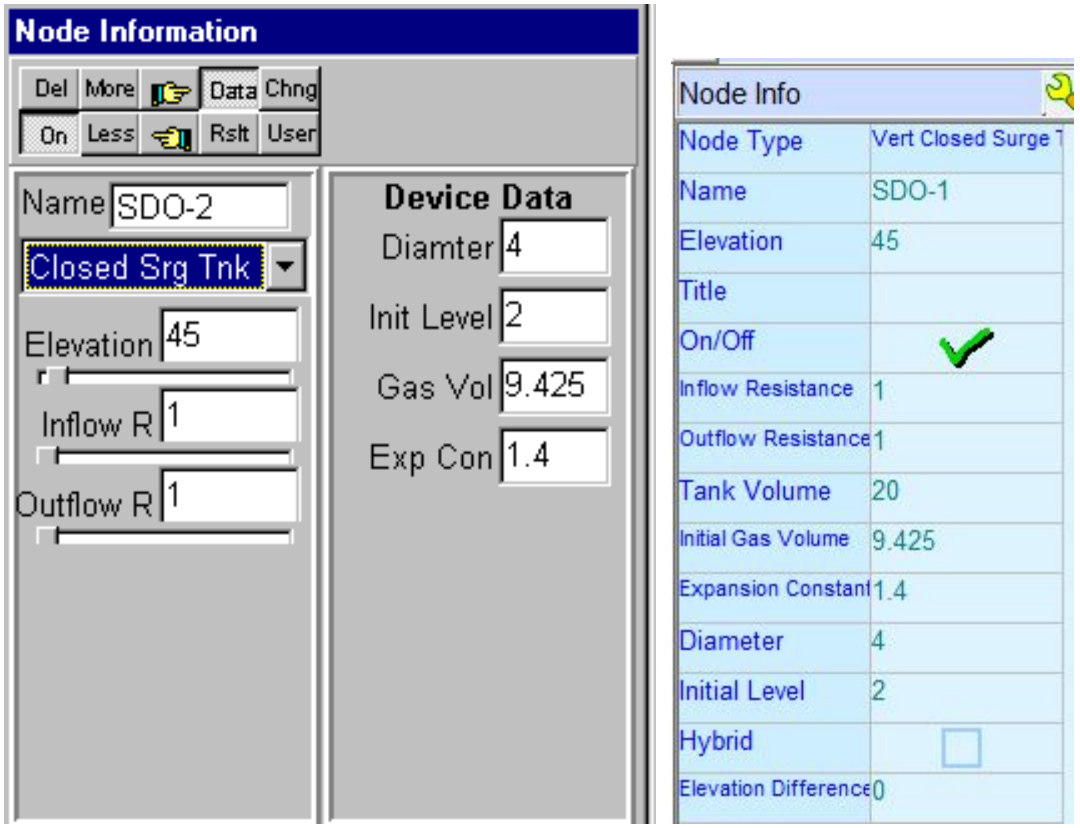

Click on Okay and then Full on the Results Table to show this table

| Pressure | | |
|---|---|---|
| Time | Discharge | Suction |
| 0.025 | 99.7 | 7.6 |
| 0.525 | 100.4 | 8.1 |
| 1.025 | 102.3 | 8.8 |
| 1.525 | 108.0 | 11.4 |
| 2.025 | 130.2 | 23.0 |
| 2.525 | 112.9 | -14.4 |
| 3.025 | 110.6 | 3.3 |
| 3.525 | 89.0 | -14.4 |
| 4.025 | 158.1 | 48.7 |
| 4.525 | 40.3 | -14.4 |
| 5 | 150.7 | 24.8 |
| Avg | 108.9 | 10.8 |
| Min | -14.4 | -14.4 |
| Max | 205.9 | 88.2 |

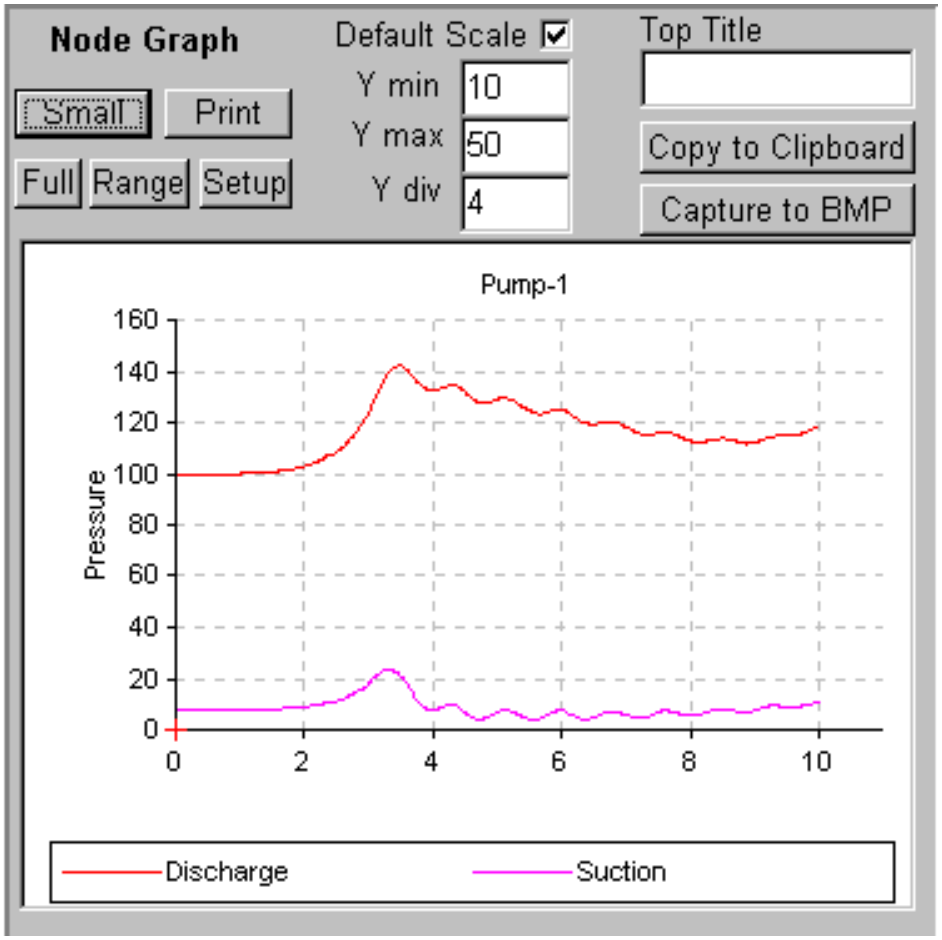
In Group Mode you can select groups of nodes to prepare customized plots and table.

### Step 6: Add a Closed Surge Tank

Add a closed surge tank at the node downstream from the pump. This tank is a 5 foot high cylinder initially with two feet of water. Assume a gas constant = 1.4. Click on the node and select Closed Srg Tnk from the drop down list. The provide the required data as shown below. The resistance of the connection to the storage tank can be obtained by clicking on Tools/Resistance Calculation and Connection to Tank and specifying a 6" diameter connection. A resistance of 1.0 is calculated and used in the data. The gas Volume is based on a two foot diameter - three foot high cylindrical space.




Click on Analyze and run the Surge Analysis. After completion click on the Pump and the Rslt button to get a plot of the pump discharge and suction pressures. The results show that the surge tank greatly reduced the surge pressures.

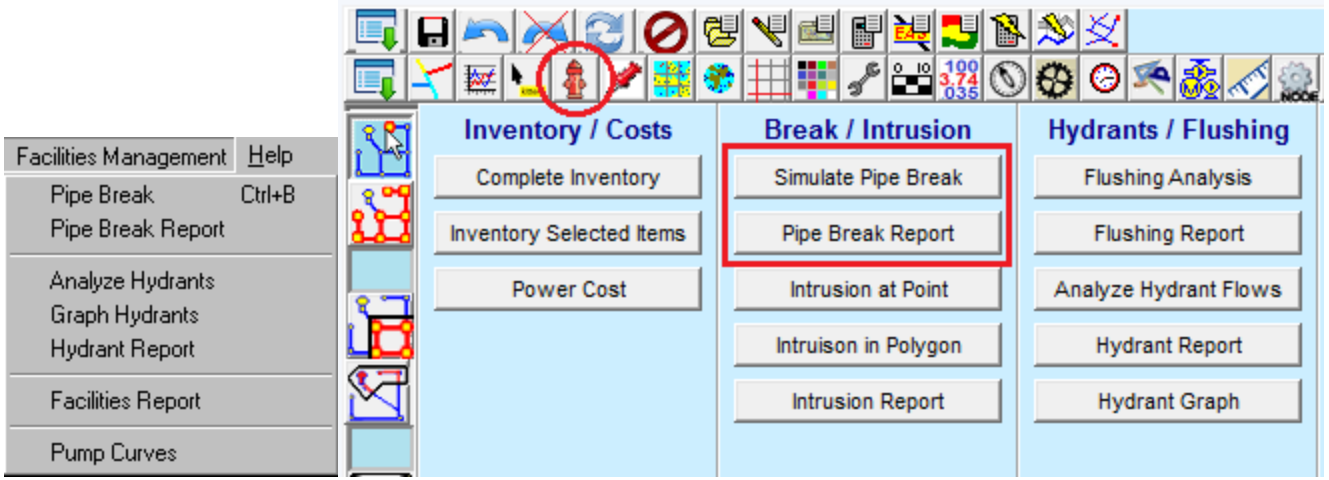

Node Graph
Small
Print
Full
Range
Setup
Default Scale
Y min
10
Y max
50
Y div
4
Top Title
Copy to Clipboard
Capture to BMP
Pump-1
Pressure
160
140
120
100
80
60
40
20
0
0
2
4
6
8
10
Discharge
Suction

## 1.5.14 Surge Pipe Break Analysis

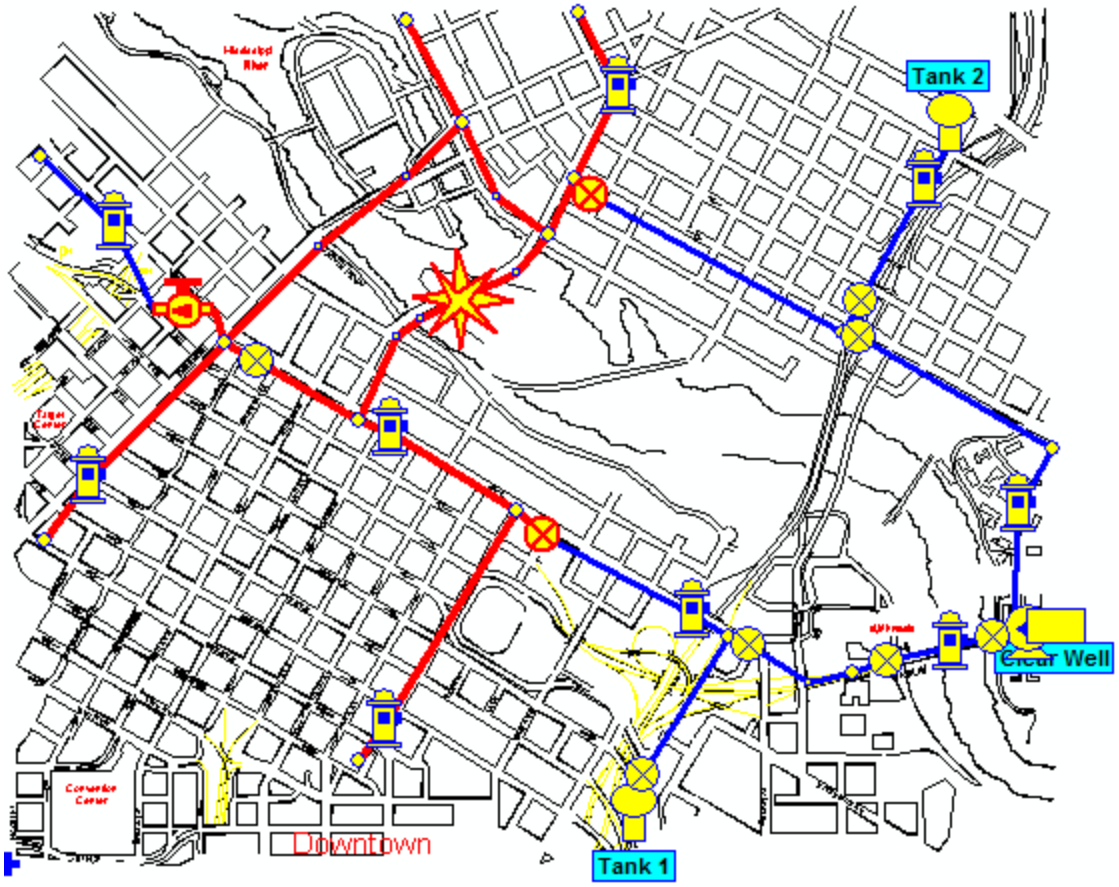
Pipe2014 and later versions.

Use Pipe Break Simulation to set up a pipe break Surge simulation. Once the Pipe Break has been applied and actively appears in the map view, if a Surge analysis is run, the option to use this pipe break as the transient event is given.

Select Pipe Break under Facilities Management menu

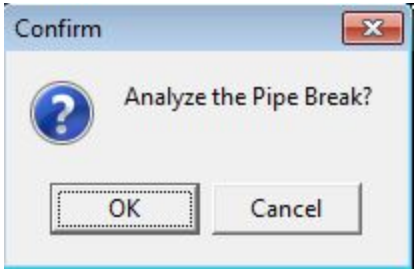



Click on a pipe. The break will be shown along with valves used to isolate it.

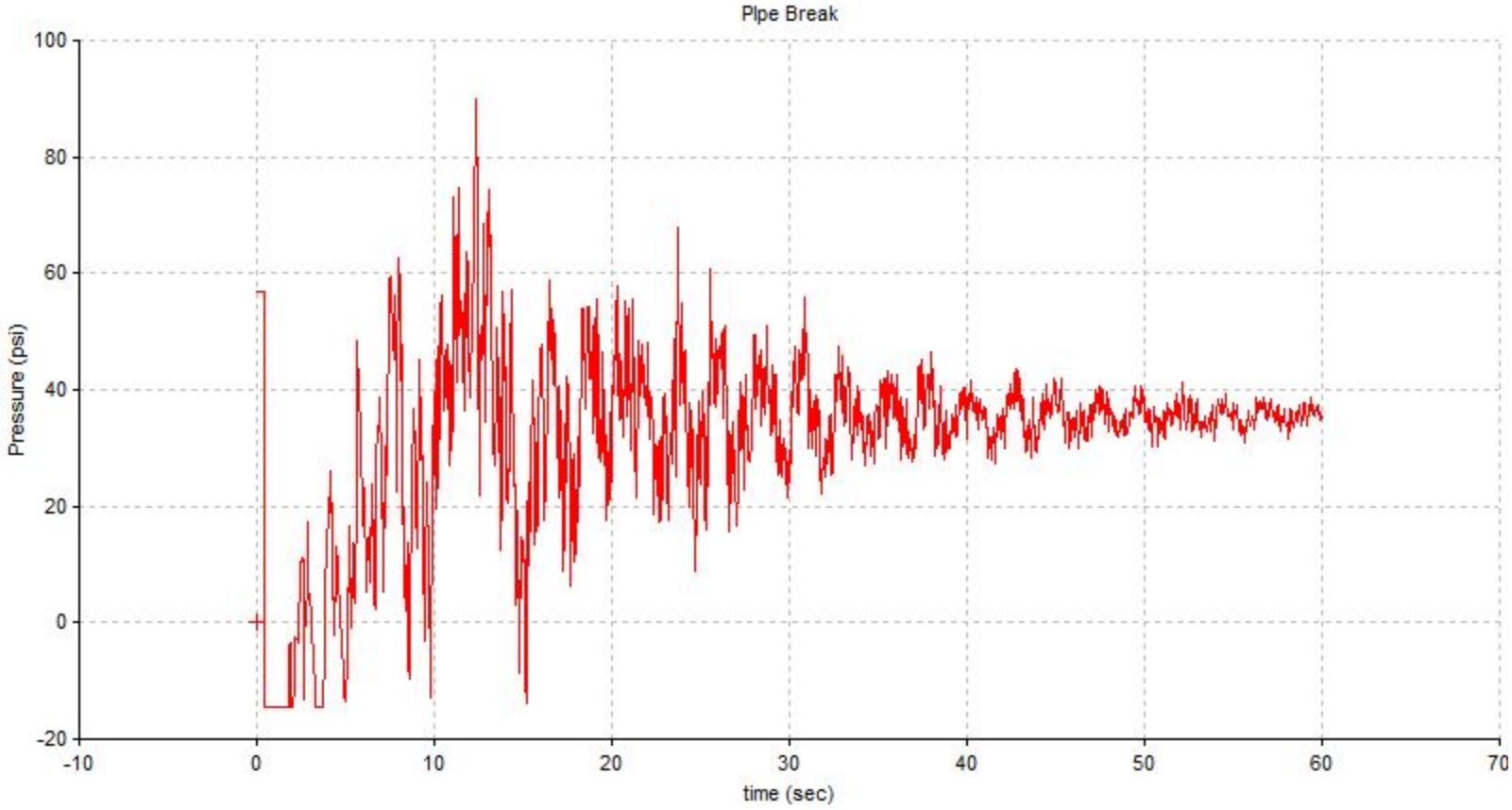



After effected pipes are highlighted, valves may be clicked to open or close. Pipe Break area will be extended or limited accordingly.

Once pipe break is set up, go to Analyze and select Surge analysis.




If you answer "OK", then Change Data for other transient events (e.g. pump shutdown, valve closure, etc) will be ignored and an analysis of the breaking of the selected pipe will take place.



## 1.5.15 Units Pipe2000 : KYPipe / Surge

Units KYPipe / Surge

| Flow | length | diameter | roughness (Darcy Weisbach) | kinematic viscosity (Darcy Weisbach) | demand | elevations, grades, tank level* | pressure | velocity | head loss | pump head | pump flow | pump power |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| cms | m | mm | mm | m*m/s | cms | m | kpa (gauge) | m/s | m | m | cms | kW |
| l/s | m | mm | mm | m*m/s | l/s | m | kpa (gauge) | m/s | m | m | l/s | kW |
| cfs | ft | in | mft | ft*ft/s | cfs | ft | psi (gauge) | ft/s | ft | ft | cfs | useful HP |
| mgd | ft | in | mft | ft*ft/s | mgd | ft | psi (gauge) | ft/s | ft | ft | mgd | useful HP |
| gpm | ft | in | mft | ft*ft/s | gpm | ft | psi (gauge) | ft/s | ft | ft | gpm | useful HP |

* Remember that elevations, grades, and tank levels are all measured from the same datum (elevation datum). If the elevation of the pipe connection to a reservoir (or tank) is 100 ft (m) and the reservoir (initial tank level) is 20 ft (m) above that then the grade for the reservoir (or initial tank level) is 120 ft (m) (NOT 20 ft (m)).

**Loss Element** - table of pressure drop (ft or m) versus flow in defined units.

**Sprinkler Constants** - (flow rate in gpm (l/s))/(sprinkler pressure drop in psi (kpa))^0.5.

Example: flow = 10 gpm, pressure drop = 4 psi

K = 10 gpm/(4psi)^0.5 = 5

- See the Sprinkler/Blowoff Constant tool under Tools in the Pipe2018 main menu

**Valve Resistance** - (head drop in ft (m))/(flow in cfs (cms))^2

Example: flow = 1000 gpm (2.228 cfs), pressure drop = 15 ft

R = 15ft/(2.228cfs)^2 = 3.022

- See the Resistance Calcluation tool, under Tools in the Pipe2018 main menu.

For Cv, flow coefficient, the units are number of gpm per 1 psi pressure drop.

**Regulator Setting** - PRV, PSV - same as pressure (psi or kpa)

FCV - flow in defined units

**Additional Units for Surge Devices:**

**Surge Protection Devices Inflow and Outflow Resistances** - (head drop in ft (m))/(flow in cfs (cms))^2

For all types of Surge Tanks, Pressure Relief Valves, Surge Anticipation Valve, Rupture Disk, see Resistance Tool - dropdown options.

**Surge Tank Diameter** - ft. or m

**Surge Tank Maximum Level (Spilling Surge Tank)** - ft. or m. above pipe center line (not above Elevation Datum)

**Air/Vacuum Valve Diameters** - inches or mm

**Air Vacuum Valve Initial Air Volume** - cubic feet or cubic meters

For more units information, see also 'Surge Elements and Data Requirements' in the Pipe2018 Help file.

## 1.5.16 Positive Displacement Pump in Surge

A positive displacement pump may be modeled in Surge through the use of the periodic input function. See Surge Elements and Data Requirements / Creating a Transient - Periodic Variations for examples using this feature.

**Positive Displacement (reciprocating) Pump Example**

Figure 1 shows a Pipe2018 model to simulate the action of a Positive Displacement pump. This pump provides a periodically varying flow into the piping system. The elevation and initial pressure are shown at each node and the length and initial flowrate for each pipe. There is an initial demand of 50 gpm at the node noted. This demand is assumed to be pressure dependent and will increase and decrease with pressure changes.

All pipes have a diameter of 8 inches, a Hazen Williams roughness of 100 and a wave speed of 3600 ft/s as shown.

To simulate the action of the positive displacement pump a periodic input is simulated for the inflow at the pump node. For this example it is assumed that the pump provides a periodic inflow varying from 0 to 400 gpm at a rate of 1 cycle/second. Figure 2 shows the Pipe2018 input (Other Data/Periodic Input) to set this condition. A surge analysis is required to do carry out this simulation and a time of 10 seconds is chosen. Note the simulation must be sufficiently long to reach a repeatable solution. This is determined by inspecting the results.

Figures 3 and 4 show the pressure and flow variations at the pump. Note that a repeatable periodic response was attained after 3-4 cycles.

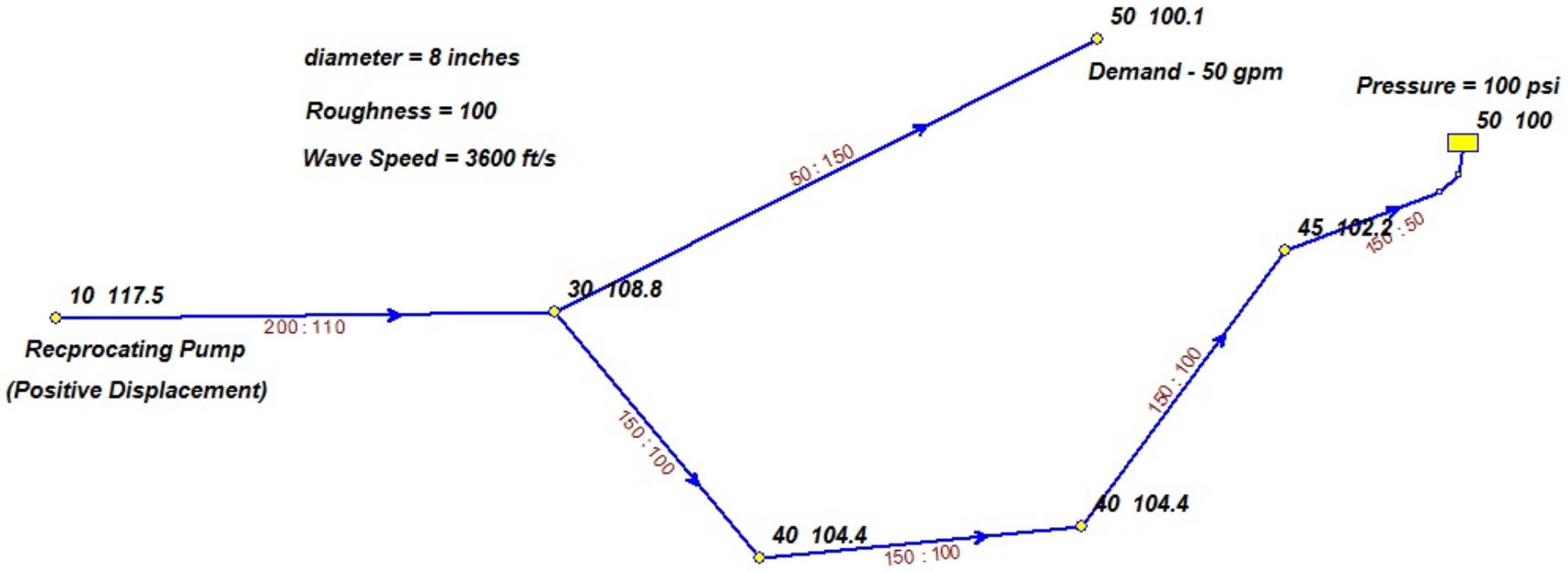



**Figure 1 Pipe2018 Model**

Meters | Library Elements | Periodic Input | Active Valves

Type: 0=Node 1=Connecting Rerservoir

| NODE | TYPE | PERIOD | PHASE SHIFT | AMPLITUDE | AVERAGE |
|---|---|---|---|---|---|
| J-4 | 0 | 1.0 | 0 | 200 | -200 |

**Figure 2 Setting up a Positive Displacement Pump**

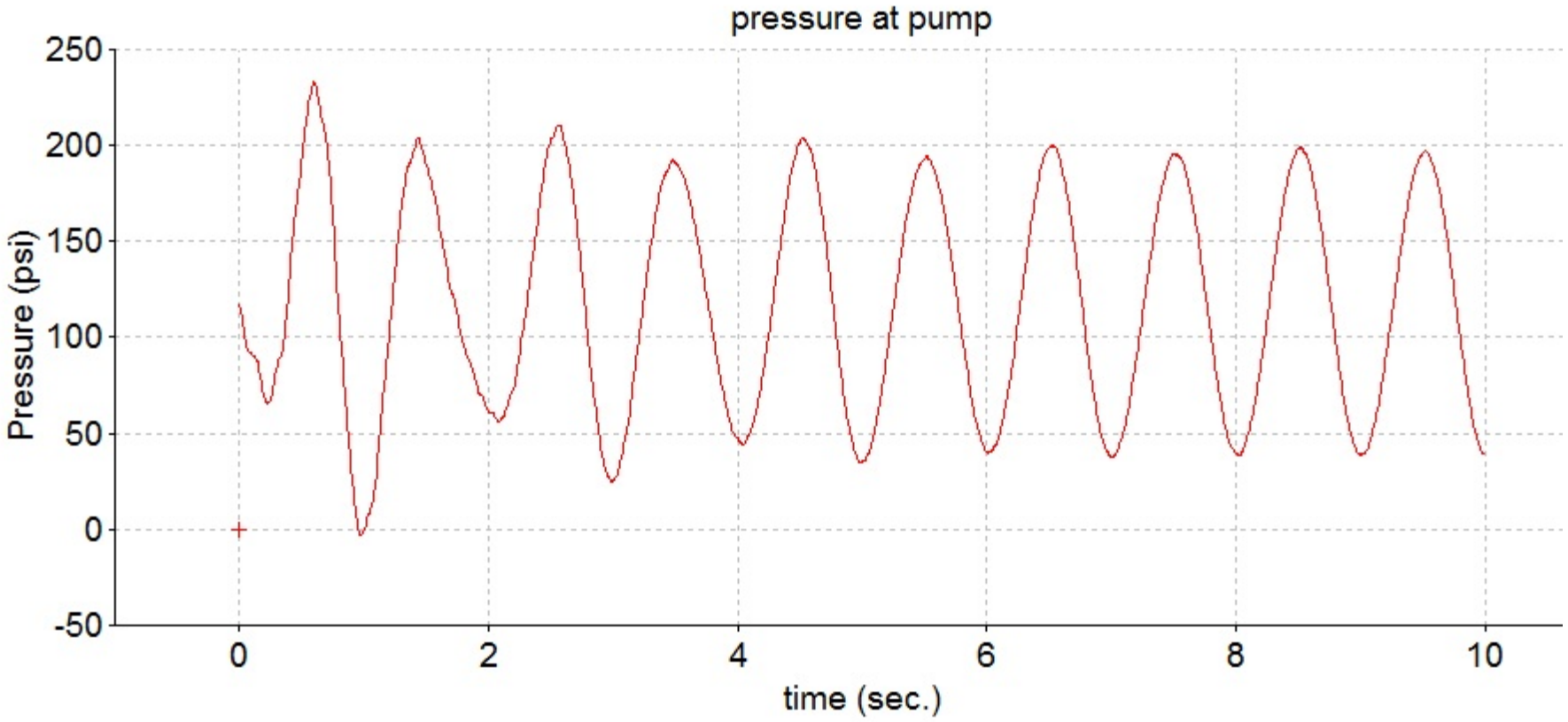



**Figure 3 Pressure Variation at Pump**

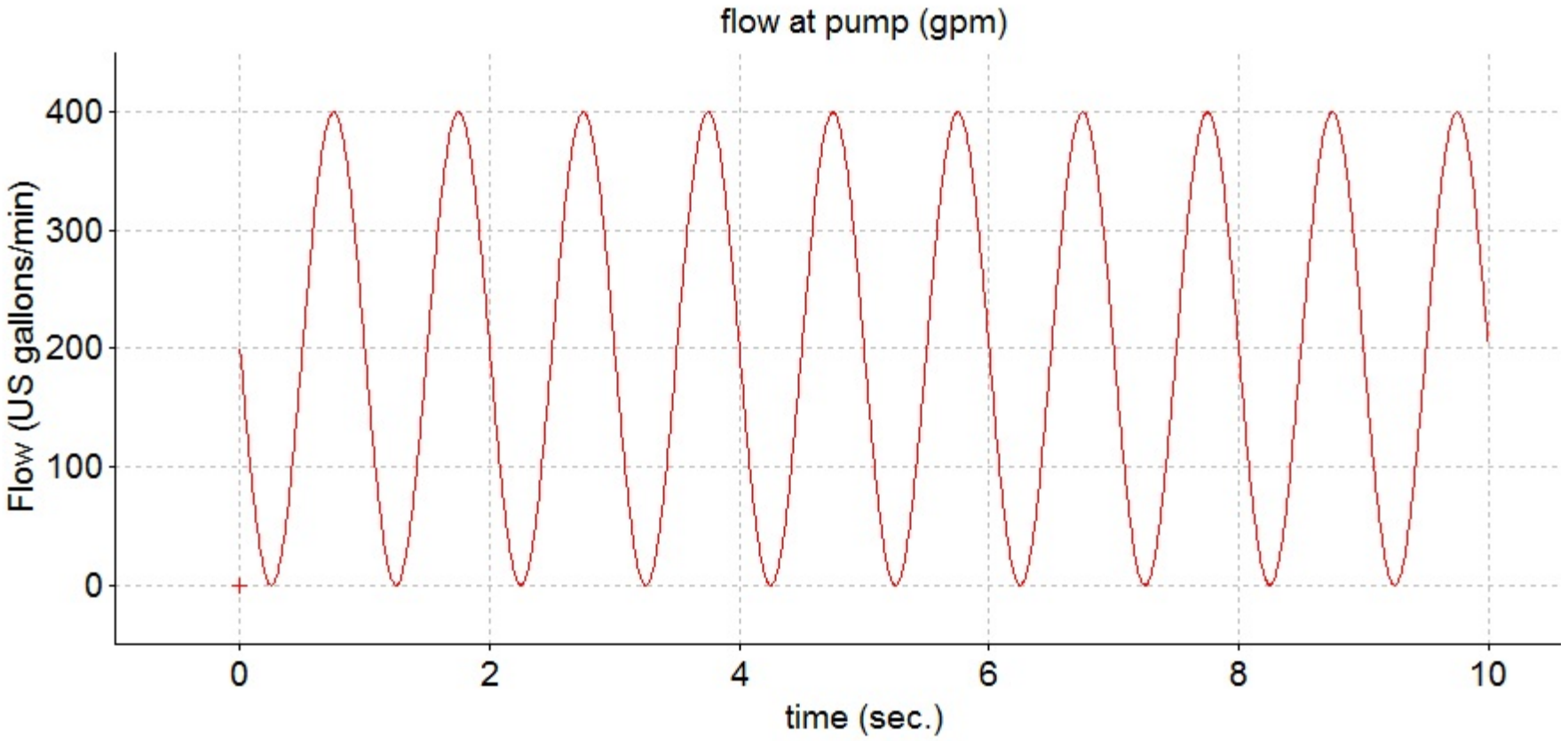



**Figure 4 Flow Variation at Pump**

## 1.5.17 Stabilization Settings for Surge

**These options are found in the System Data for Surge models.**

**Stabilization Settings**

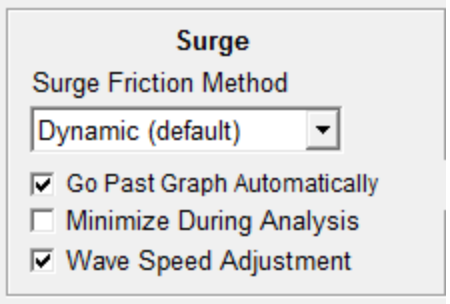
Transient flow models are unable to realistically account for some effects which reduce cavitation effects resulting in pressure spiking and unstable responses. Surge has several inputs which affect the formation and collapse of cavities and these inputs can be used to provide insight into situations resulting in severe pressure spiking.

**Time Steps for Cavity Collapse**. This feature was added to account for phenomena such as air release when vapor cavities occur. Air will cushion the cavity collapse. As the cavity collapses and the water columns rejoin a momentum calculation provides the pressure spike required to rejoin the water columns. The computational time increment is used as the default so that the deceleration of the water columns and resulting pressure spike occurs over this very short time increment resulting in greater pressure spikes. We know that this is the worst case and even a small amount of air in the cavity can result in the collapse occurring over several time increments (still very fast but can result in considerably smaller pressure spikes).

The bottom line is we know the default of one time increment is the very worst case and a small cushioning due to air can result in significantly lower pressure spikes. The problem is we really don't have the ability to actually compute this and are limited to being able to show the effects of slowing down the collapse. We really can't make a recommendation on a number to use although we are confident that there will normally be some cushioning and we want to provide our users with the capability to gain some insight into this effect.

**CV Settings for Inertial Effects**: This stabilization settings was primarily meant to address the numerical instability issues rather than addressing the inertia related issues of check valves. By numerical instabilities we mean that the flow through the pipeline is close to zero but not exactly zero and flowrate rapidly oscillates between a small positive and negative value (but practically close to zero). The CV (in the model) also oscillates (open & shut) accordingly causing CV chattering and pressure spikes. Designating the CV to continue to move in the same open/close mode when it's open area is within the user specified value (10% open for example) will provide a delay in the CV response. This will alleviate the CV chattering to some degree. Again, we really can't make a recommendation on a number to use although we are confident that there will normally be some CV response delay and we want to provide our users with the capability to gain some insight into this effect.

## 1.5.18 Surge Preferences




**Surge Friction Method** -

Steady State (Old Method) - The Static Friction Method uses a constant resistance based on the initial flow and pressure drop using DH = R*Q*Q where DH is the Head drop, Q is the flow rate and R is the Resistance. Both head loss equations expressed like this have a resistance that varies with flow. For dynamic friction a new resistance is calculated for each time increment.

For example consider Q = 200 gpm (.446 cfs), D = 4 inches, Length = 200 feet and HWC (roughness) = 120. For these conditions the head loss = 2.36 feet. Therefore the resistance = 2.36/(.446)^2 = 11.89.

If this flow changes to 400 gpm (.891 cfs), the loss predicted using the constant resistance = 11.89 (.891)^2 = 9.44 feet. However, using the HW equation the loss is 8.53 feet.

Dynamic (default) - Friction Method results in a better final steady state calculation for Surge analyses. If we use dynamic friction we would get a better result because the resistance would be adjusted to account for the change in flow. Dynamic Friction is the default setting. The reason why the Static Friction Method is still available is to accommodate older models with the ability to duplicate the original results obtained with the Static Friction Method.

Dynamic (Interpolate f in a transition zone) - Applies to cases where the Reynolda number is between 2000 (laminar) and 4000 (turbulent). Calculations for f are made using both laminar and turbulent formulas and the average value is used. This is because both formulas may or may not apply since either condition can exist. Suppose the R is 2750, then two friction factors are calculated: f1 = R/64 and f2 based on Jain equation. Then f is calculated using f = {f1 * (R-2000)/2000 } + {f2 * (4000-R)/2000)}.

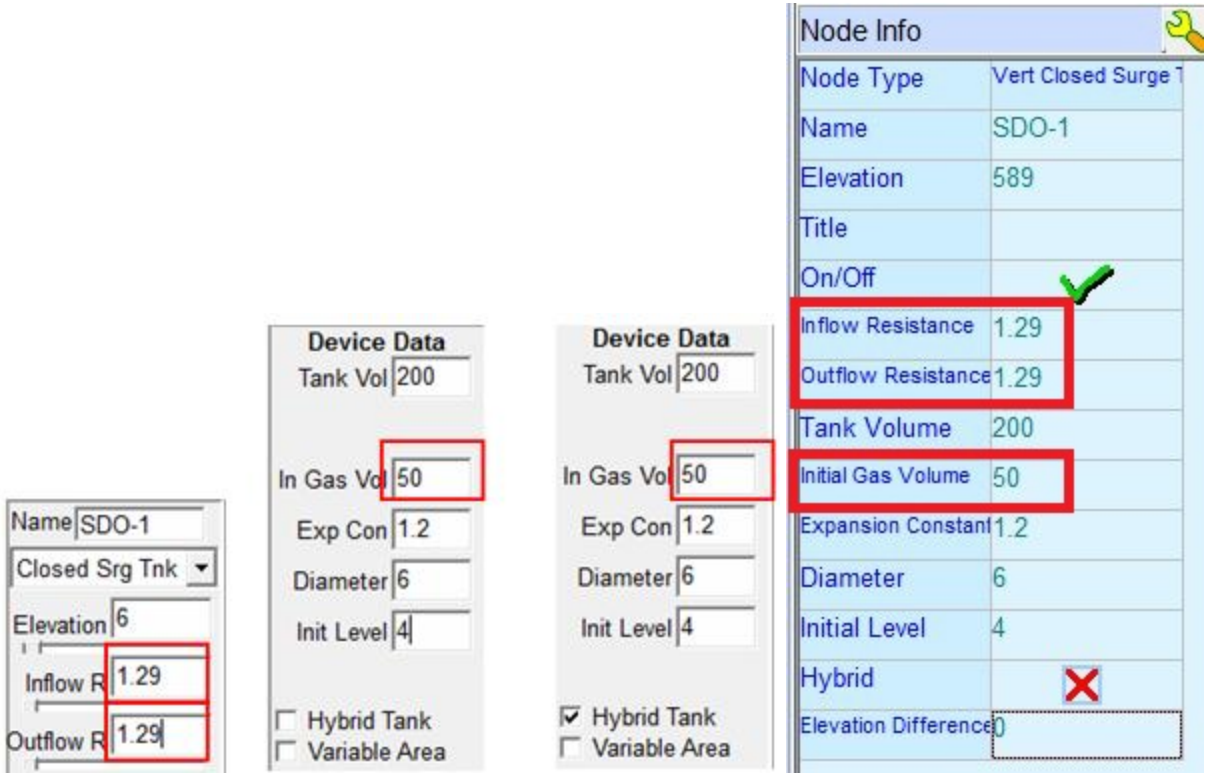
**Go Past Graph Automatically / Minimize During Analysis** - affects the dynamic screen plot that appears while a Surge Analysis is running.

**Wave Speed Adjustment** - The wave speed is only adjusted for short pipes (less than the length accuracy) where both the length is increased and the wave speed decreased by the same percentage. For example if the length accuracy is 40 feet and there is a 10 foot pipe - without wave speed adjustment the length would be adjusted to 40 feet with the same wave speed (say 4000 f/s). If the wave speed adjustment is used the length would be increased by 50% (to 20 feet) and the wave speed decreased by 50% (to 2000 f/s) so the wave travel time would be one time increment.

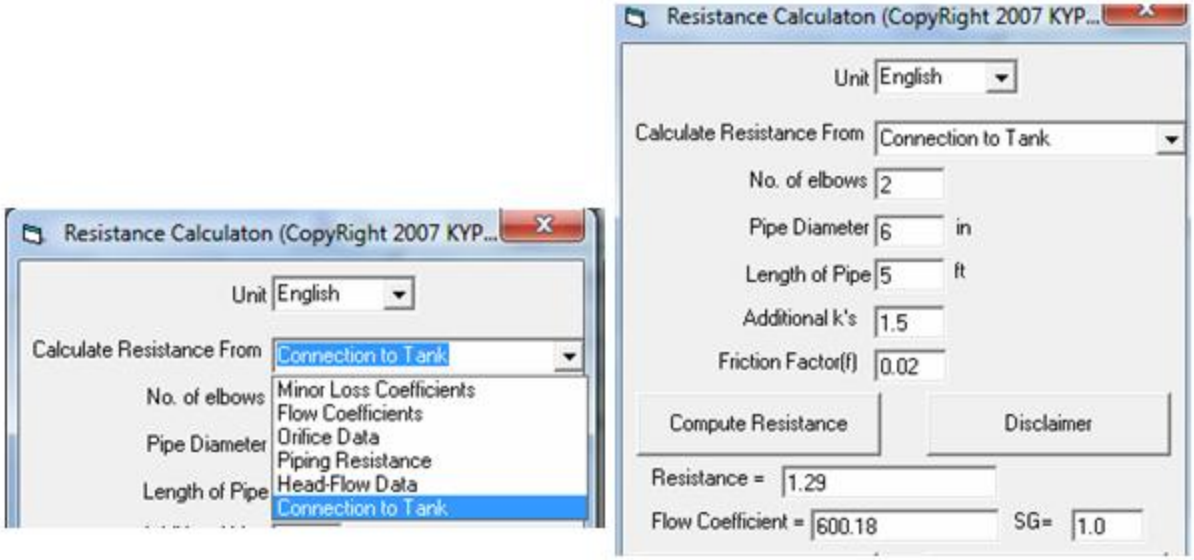
## 1.5.19 Surge Tank Data

### Surge Tank Data

Surge Tank data requirements are discussed in detail in this section. The Surge Data Boxes are shown below.




For the great majority of applications the data highlighted is the important data The Inflow and Outflow resistances are based on the size of the pipe connecting the surge tank to the pipeline and the fittings included in the connection. A Surge Tool (Resistance Calculations) shown below calculates the resistance value to enter.




The Initial Volume of gas along with the Inflow & Outflow Resistances pretty much determine how the surge tank will perform and adjusting the Initial Volume until the specified pressure high and low limits are attained is normally how the surge tank is sized. The surge tank should be sized 20% larger than the maximum gas volume for the smallest Initial Gas Volume which meets the objectives.

As noted for the great majority of surge tank simulations all that is required for the Diameter and Initial Level is a reasonable estimate. Section 1 provides additional details.
The Initial Gas Volume (**compressed volume under the initial pressure**) is the most important input data for accurate surge tank modeling. For a compressor surge tank this is a known value (the compressor is set to maintain this value). However, for precharged surge tanks (Bladder Tanks and Hybrid Tanks) this value must be computed and Surge2010 has a tool to make this computation (Sections 3 & 4). Finally for Hybrid Surge Tanks which operate under very low heads and Open Surge Tanks the diameter, initial level and tank shape can significantly affect the analysis and Surge provides Tools to accurately compute these values (Section 2)
**1) Reasonable Estimates for Diameter and Initial Level:** Modeling surge tanks requires input data on the geometric characteristics of the tank (equivalent diameter) and the initial conditions (water height ) for the surge tank. However, for most applications all that is required is a reasonable estimate for this input data. This is because the changes in water level in the surge tank usually will have a negligible effect on the results. To illustrate this an analysis of a pump trip was carried out using a 1000 gallon (133.5 ft^3) surge tank. It is assumed that this is a 4 foot diameter tank approximately 12 feet long. It initially has 225 gallons (30 ft^3) of compressed air. The pipeline is a 7000 foot long, 12 inch pipeline carrying 2000 gpm.

The analysis was carried out assuming two situations:
1) A vertical 4 foot diameter tank with 6 feet of water under the initial line pressure
2) A horizontal surge tank modeled as a 6 foot equivalent diameter vertical tank with 2 feet of water under the initial line pressure.

Figure 1 compares results (line pressure at surge tank (upper) and gas volume in surge tank (lower)) for the two variations in surge tank geometries for two

different initial line pressures. Note: all plots show **two** sets of results which are so similar that the differences can not be distinguished. There results clearly illustrate that reasonable estimates for the diameter and initial level are all that will be needed for the great majority of surge tank modeling.

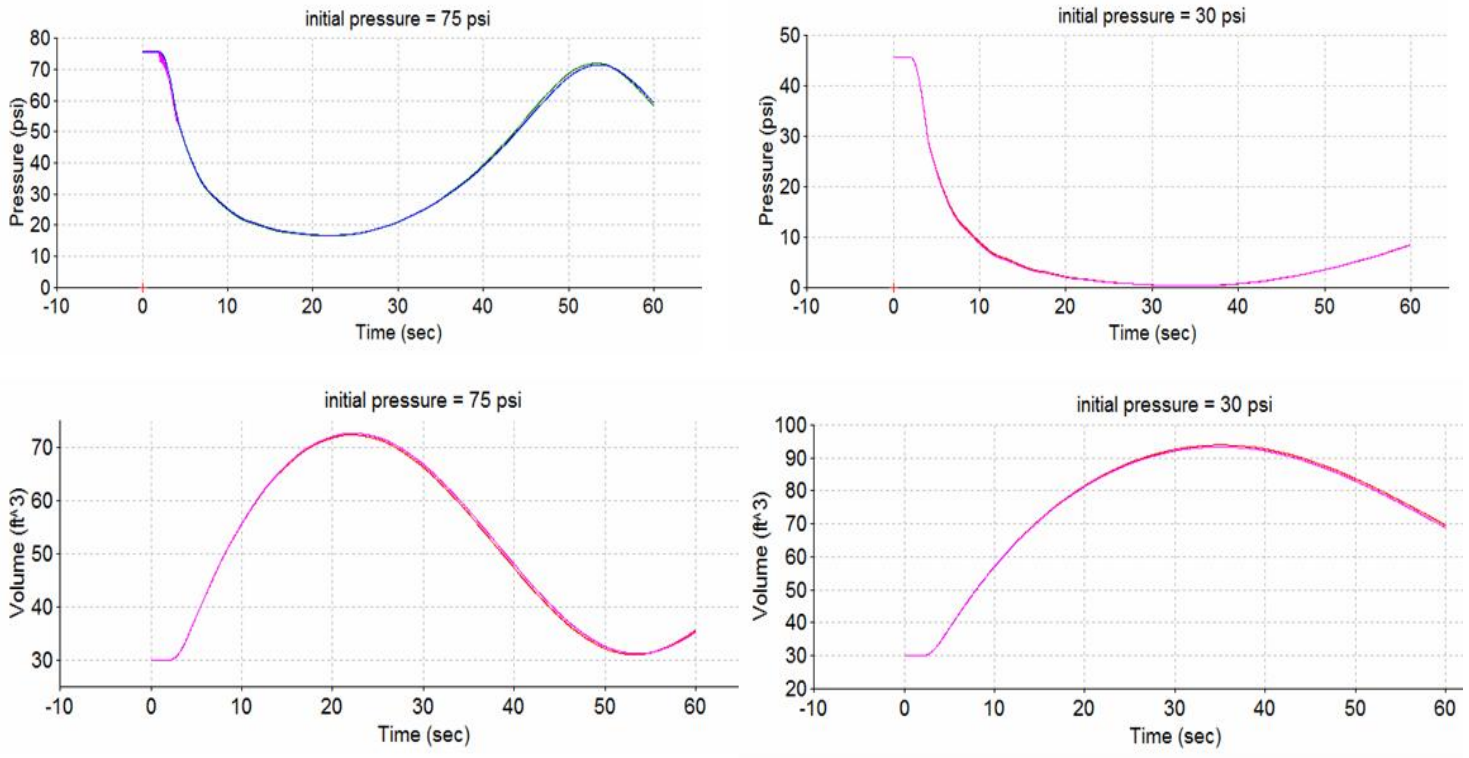


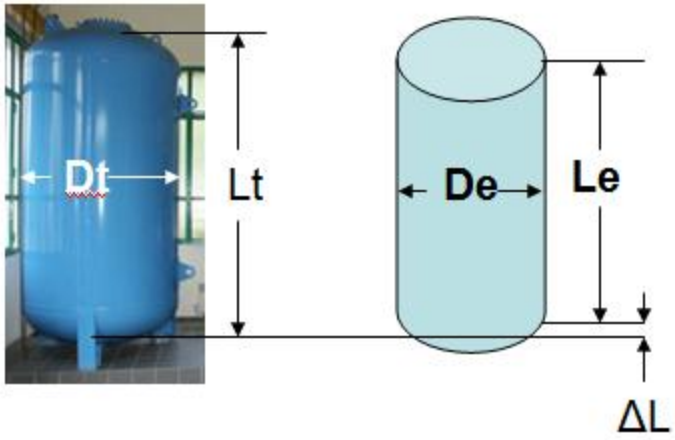

**Figure 1 Results comparing different surge tank geometries.**

For cases where the surge tank is initially operating under low pressure (say < 20) psi) and is expected reach pressures around 0 psi the change in water level can significantly affect the results. This is especially true for hybrid tanks which admit air when the water level drops to a switching level where the pressure < 0 and perform as an open surge tank until the water level goes above the switching level and the surge tank operates again as a closed surge tank..

**2) Geometric Characteristics:** The vast majority of surge tanks are cylindrical vessels with rounded ends aligned either vertically or horizontally such as those shown below.

For virtually all transient analysis these tanks can be modeled as a vertical cylindrical tank and with negligible loss of accuracy. For a vertical tank as shown below the equivalent diameter, De, and the volume, Vt, of the modeled tank is assumed to be the same as the actual tank while the length, Le, of the equivalent tank is computed as :

De = Dt
Le = Length of tank = Vt/ΠD**T**^2/4



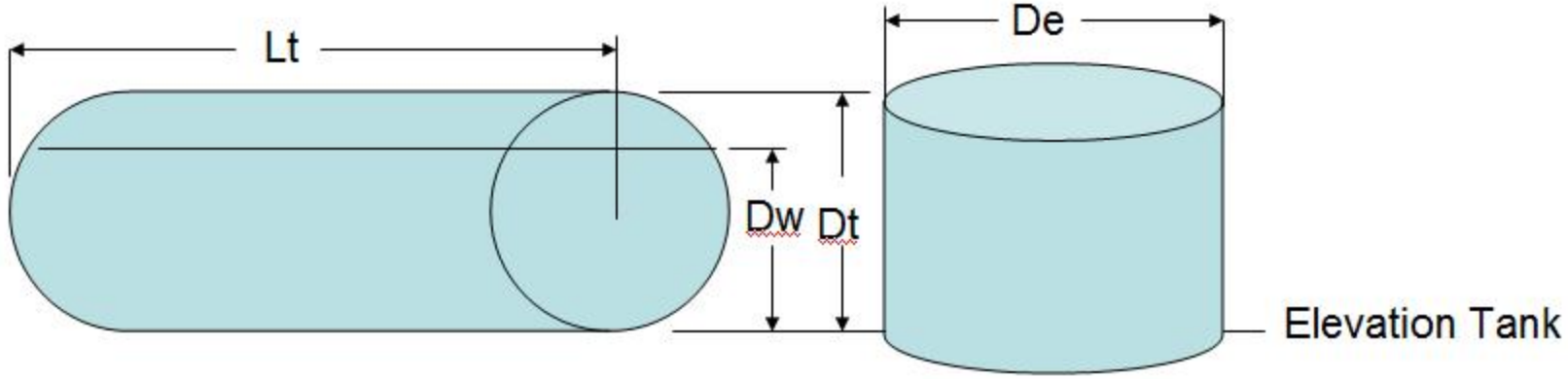

**Example Calculation:**
**Inputs:**

De = Dt = 4 ft.
Vt = 1000 gallons (133.5 ft^3)
Lt = 12 ft.

**Calculations:**

Le = 133.5/(Π*4*4/4) = 10.62 ft
ΔL = (L – Lt)/2 = 0.64 ft.
Note the elevation of the bottom of the equivalent tank will be raised by a small amount (ΔL). However, for all except very low head surge tanks this refinement can be neglected.

The calculations for determining the equivalent diameter for a horizontal tank to be modeled as a constant diameter vertical tank is shown below. For this calculation it is assumed that the volume, Vt, and the vertical dimension of the tank will remain the same which is the tank diameter ( Dt).




Dt = diameter tank
Vt = Volume Tank
Lt = Length of tank = Vt/ΠDt^2/4
De = Equivalent Diameter = (4*Vt/ΠDt)^.5
ΔE = 0

**Example Calculation:**

**Inputs:**

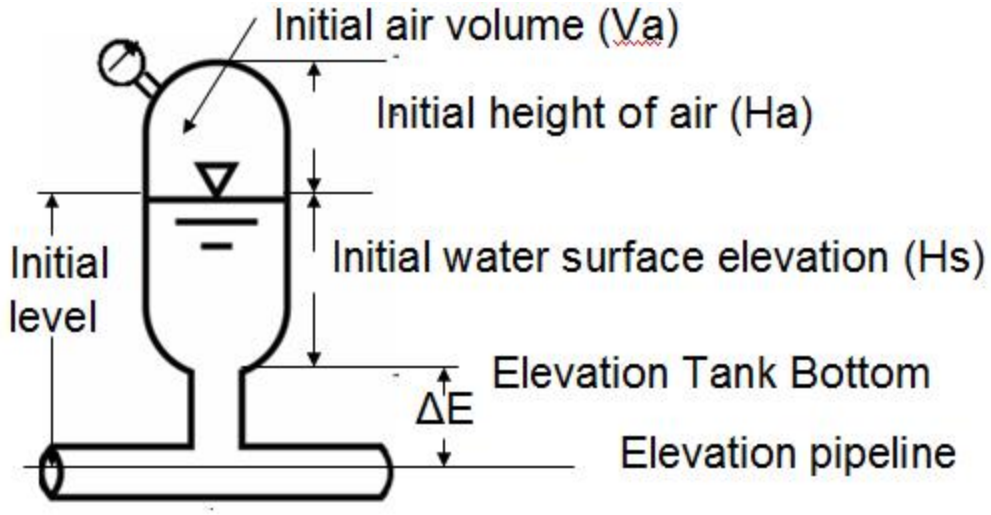
Dt = 4 ft.
Vt = 1000 gallons (133.5 ft^3)

**Calculations:**

Lt = 133.5/(Π*4*4/4) = 10.62 ft
Le = Dt = 4 ft.
De = (4*133.5/(Π*4))^5 = 6.52 ft

**3) Initial Calculations for Compressor Tank**

A compressor tank is designed to maintain a specified initial volume of air under the initial conditions so this important parameter does not require a calculation. The initial level of the liquid surface (Hw) is a simple calculation as shown below. Again except for very low pressure operation and hybrid tanks a reasonable estimate of the value of Hw will provide very accurate results.

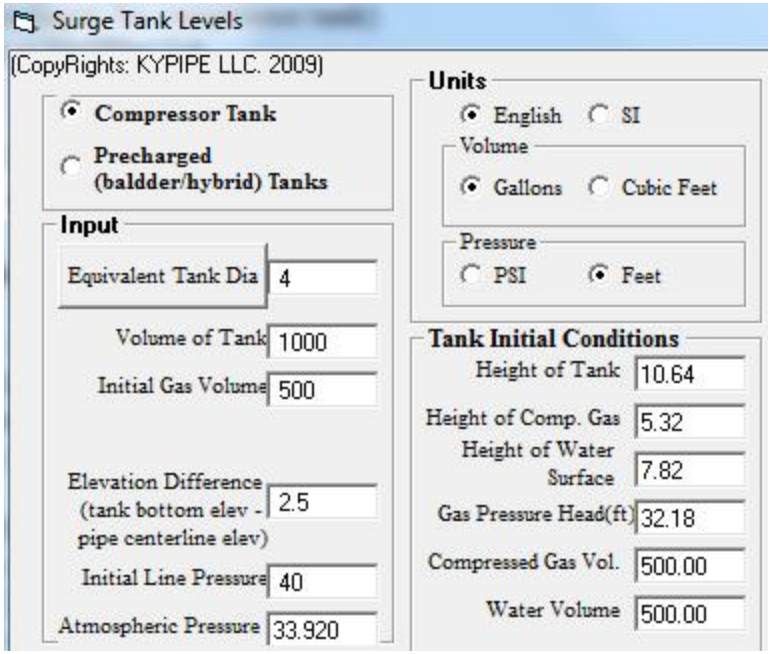



De = equivalent diameter of tank (input)
Vt = Volume Tank (input)

Va = Volume of Air (input for compressor tank)
Ht = Height of tank = Vt/ΠDe^2/4
Ha= (Va/Vt)Ht
Hs = Ht – Ha
ΔE = elevation diff. (bottom to pipeline) – (input)
Hw = initial liquid level (above pipeline centerline) = Hs + ΔE

**Example Calculation:**
**Inputs:**
De = 4 feet (input)
Vt = 1000 gallons (input)
Va = 500 gallons (input)
ΔE = 2.5 ft. (input)
**calculations:**
Ht = (1000/7.49)/(4*4* Π/4) = 10.64 ft.
Ha= (Va/Vt) Ht = 5.32 ft.
Hs = Ht-Ha = 5.32 ft.
Hw = Hs + ΔE = 7.82 ft.

There is a Tool provided to do this calculation (**Surge Tank Level**). The screen for this tool is shown below using the data above and showing the computations and identical results. Note that the selection for Compressor Tank is checked.



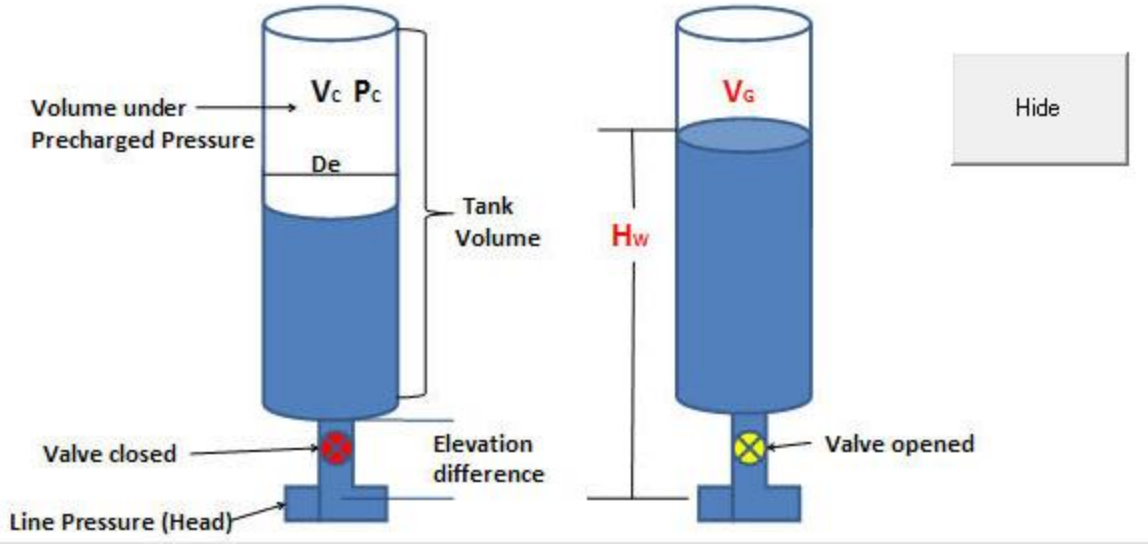

**4) Initial Calculations for Precharged Tank (Bladder and Hybrid)**

The calculations for a precharged tank are more involved because the initial volume of air is known for a precharged condition but not for the initial condition (initial head in the pipeline) which is required input data. Therefore, this important input data must be calculated.

The help screen for the tool provided for this calculation (Surge Tank Level) shows the situation for calculating the initial air volume and liquid level. We know the volume of air and the precharge pressure prior to pressurizing the surge tank. Assuming an isothermal compression of the gas we can determine the compressed volume of gas. The computation is complicated by the fact that we know the pressure in the pipeline but the pressure in the gas is reduced due to the column of liquid between the pipeline and gas. The calculation may be done by assuming the pressure in the gas is equal to that in the pipeline and computing the height of the liquid column and then repeating the calculation accounting for the decrease in pressure due to the liquid column.

Two example calculations are presented. The first is for a bladder tank where the precharged pressure is normally the entire tank volume. The second example is for a hybrid where a known volume of air is initially at atmospheric pressure prior to opening the valve and compressing the air.

1) Run Steady State to Calculate Initial Line Head
2) Input Precharge Volume (Vc), Precharge Head (pressure ) (Pc), Tank Volume, Initial Line Head (pressure), Equivalent Diameter (De) and Elevation Difference into Tool
3) Calculate and Input Initial Gas Volume ($V_G$) and Height of Water Surface (above pipe CL) (HW) into Surge2010



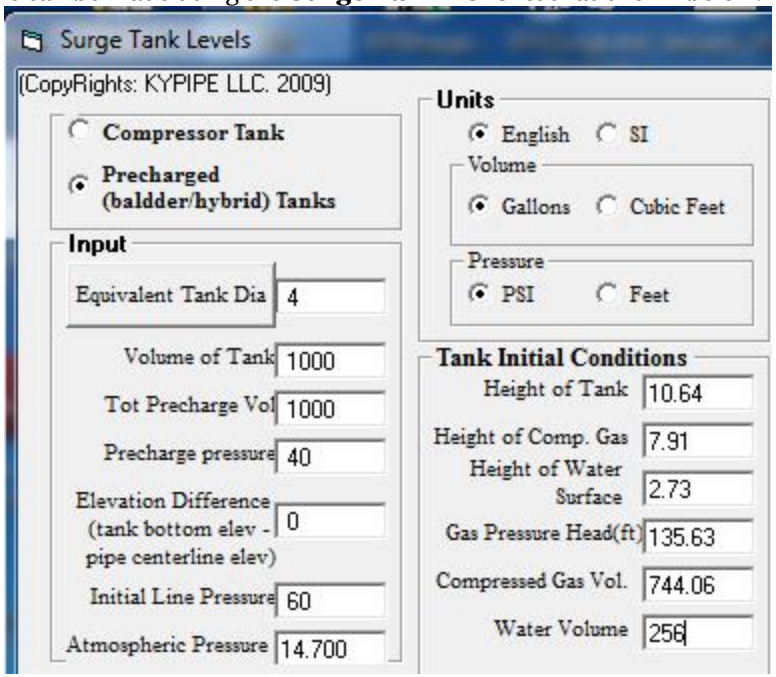

**Example 1 - Precharged Bladder Tank Calculations** :

De = 4 feet (input)
Vt = 1000 gallons (input)
Va = 1000 gallons (input)
Pc = 40 psi, Pi = 60 psi (input)

Va = 1000(40+14.7)/(60+14.7) = 732 gal.
Ha = (732/1000)10.62 = 7.78 ft.
Hs = 10.62 – 7.78 = 2.84 ft.
***Adjust air pressure for water height :***
Pi = 60 – 2.84/2.31 = 58.77psi = 135.7 ft.

Va = 1000(54.7)/(73.47) = 744 gallons
Ha = (744/1000)10.62 = 7.9 feet
Hs = 10.62 – 7.9 = 2.72 feet

These calculations can be made using the **Surge Tank Level** tool as shown below:

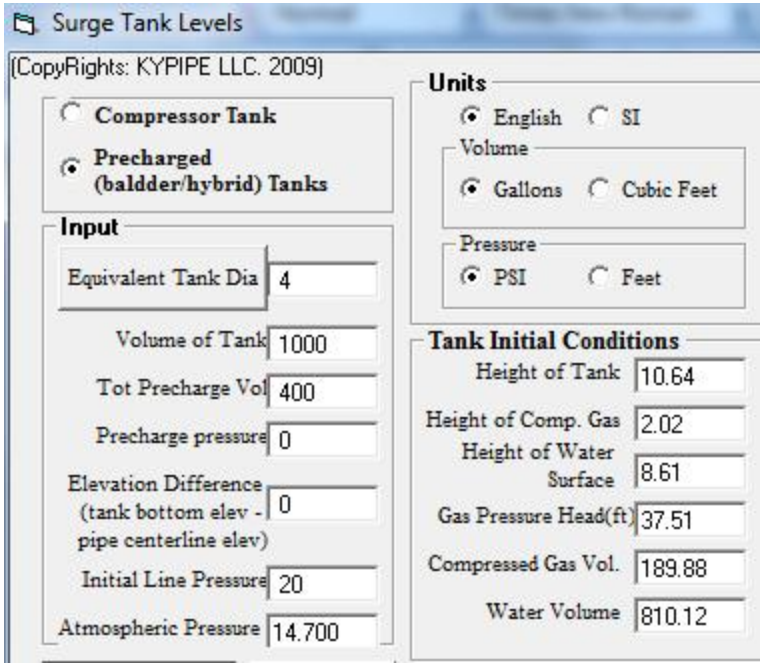



**Example 2 – Hybrid Tank Calculations:**
De = 4 feet (input)
Vt = 1000 gallons (input)
Va = 400 gallons (input)
Pc = 0 psi, Pi = 20 psi (input)

Va = 400(0+14.7)/(20+14.7) = 169.5 gal.
Ha = (169/1000)10.62 = 1.69 ft.
Hs = 10.62 – 1.69 = 8.93 ft.
***Adjust air pressure for water height :***
Pi = 20 –8.93/2.31 = 16.13 psi = 37.2 ft.
Va = 400(14.7)/(30.83) = 190 gallons
Ha = (190/1000)10.62 = 2.02 feet

Hs = 10.62 – 2.02 = 8.62 feet

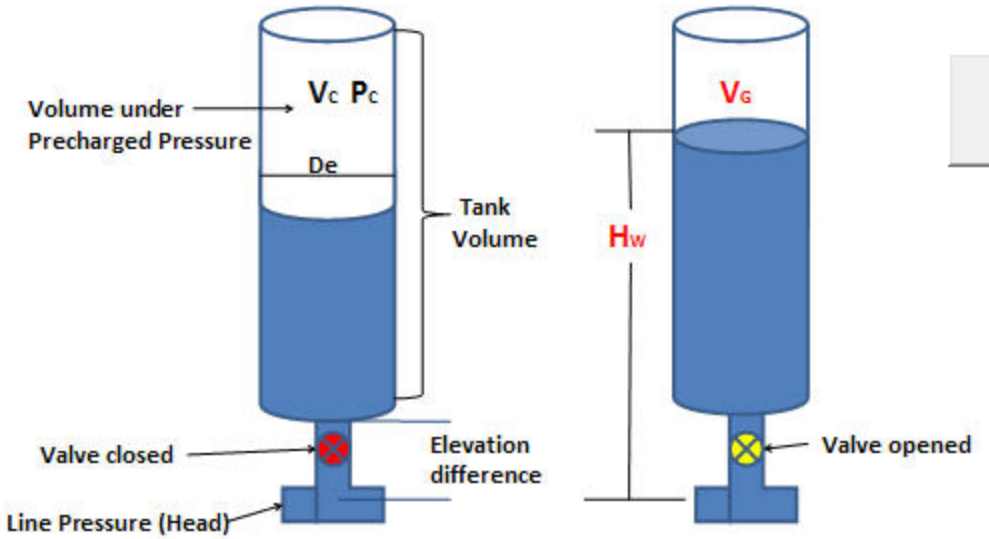
These calculations can be made using the **Surge Tank Level** tool as shown below:

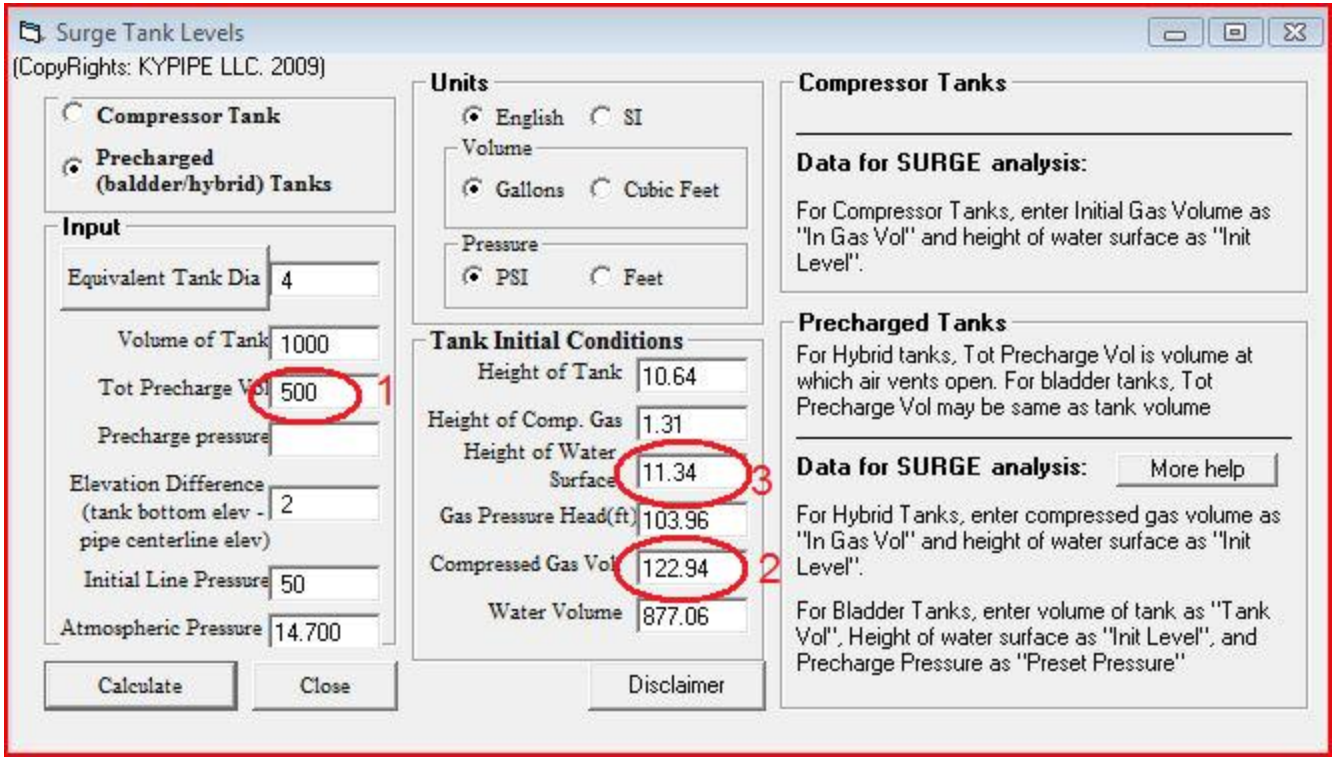



The In(itial) Gas Vol(ume) for Surge Tanks refers to the initial volume of air in the tank. To determine what this is for a hybrid tank with a dipping tube, use the tool called Surge Tank Levels under Tools in the main menu as shown below.

## Tool for Precharged Surge Tank (Bladder & Hybrid Tanks)

1) Run Steady State to Calculate Initial Line Head
2) Input Precharge Volume (Vc), Precharge Head (pressure ) (Pc), Tank Volume, Initial Line Head (pressure), Equivalent Diameter (De) and Elevation Difference into Tool
3) Calculate and Input Initial Gas Volume ($V_G$) and Height of Water Surface (above pipe CL) (HW) into Surge2010






Select Compressor or Precharged

## 1.5.20 Modeling Air in Pipeline for Surge

Modeling air in pipelines is a tricky problem. All surge analysis software programs in the world assume that the pipeline is always filled with water. This is true even when air is allowed into the pipeline, all the air is assumed to be right at the air valve and NOT in the pipeline.

Air valves keep track of how much air comes into the pipeline, and whether there will be air slam problems. We can use this feature to model empty columns as well.

This approach is sometimes used to model filling of long pipelines. Estimate the volume of air in pipeline (empty column volume from water level to upper check valve), put that as initial air volume at the air valve located at top of pipeline before check valve. The size of air valve should be modeled correctly as well. If there is a large diameter air valve before the check valve then water column velocity will be very high and there can be sudden deceleration of water column when all air is out of pipeline. Therefore, it is recommended to have a smaller air valve before the check valve, which will minimize the water column velocity.

With this approach the influence of sudden velocity reduction is demonstrated along with the surge effects, but it is **only an approximate method.** All surge

programs assume that pipelines are filled with water right from time 0.

### 1.5.21 Time Step in Surge

Time Increment used for the Surge simulation may be viewed in the Report in the Data Summary as shown:

```
              #*****    SURGE PROGRAM - VERSION 9.0     *****#
              **               Copyrighted by                **
              **                Dr. Don J. Wood              **
              **             Dr. Srini Lingireddy            **
              **              Lexington, KY, USA             **
              **                  March 2012                 **
              #*********************************************#

 INPUT DATA FILE NAME  = C:\Users\Public\DOCUME~1\KYPIPE~1\Surge\Bladder_.D
 OUTPUT DATA FILE NAME = C:\Users\Public\DOCUME~1\KYPIPE~1\Surge\Bladder_.O

 RUN DATE =     03/19/12
 RUN TIME =     09:24:49

 ------------------------------------------------
 # of Increments for Cavity collapse:  1
 CV Setting for Inertial effects:      0.00000000E+00
 ------------------------------------------------
*************************************************************************
                              SYSTEM DATA
*************************************************************************

 THE FOLLOWING DEFAULT OVERRIDES HAVE BEEN DEFINED:

     Liquid Specific Gravity =  1.00000000
      Time Increment Factor  =  1
     Flow Conversion Factor  =  448.859985
     Head Conversion Factor  =  1.00000000

 --------------------------------
 Fixed Demands at Junction Nodes.
 --------------------------------

 Total Simulation Time =   120.0 sec
 Time Increment        = 0.01230 sec
```

Time Increment (Dt) is calculated as Dt = La/WSmax where La is the length accuracy and WSmax is the maximum wave speed. A process is used that may come

up with a larger value which assures that all pipes have a model length within the Length Accuracy.

## 1.5.22 Surge Report Definitions

**Time Step for Cavity Collapse**- Time Step for Cavity Collapse is an integer representing the number of time increments for complete collapse of a vapor cavity. The default is 1, implying that the cavity collapses in one time increment, however small the time increment is. The purpose of this setting is to "soften" the effects of cavity collapse. Since air is normally released when cavitation occurs a softening effect is expected. Without allowing for this, excessive pressure spiking may occur. The factor is applied to the simulation time step. Therefore if your simulation time step is 1/100 second (this value is shown in the Report), then a value of 10 here would have the cavity collapse in 1/10 second.

**CV Setting for Inertial Effects**- With check valves that are opening and closing repeatedly, this allows the user to incorporate inertal effects into the modeling of the check valve. CV Setting for Inertial Effects is a number between 0 and 1. The default is 0 and will result in no inertial effects. If the CV is in the closing mode (reverse flow through the element) and the CV open ratio is greater than CV Setting, then Pipe2018 continues to close the CV until it is completely closed EVEN if the flow changes sign (flow in the forward direction). In other words, it is assumed that the CV continues to close once it reaches certain closing position due to the inertial effects of the disc or flap. The use of this setting will reduce pressure spiking due to check valve action. The default of zero should be applied for CVs that are initially closed.

In general no inertia effect (Zero) results in a "rattling" of the flap. The intended effect of the CV Setting for Inertial Effects is to slow down the rate (frequency) of the "rattling" flap by requiring the CV action (opening or closing) to be completed before swing the opposite direction. Due to inertia the swing will complete after reaching a certain % of the opening or closing before the flap can go in the opposite direction. This tends to reduce spiking somewhat because the fluctuation rate is slowed.

**Time Increment**(Dt) is calculated as Dt = La/WSmax where La is the length accuracy and WSmax is the maximum wave speed. A process is used that may come up with a larger value which assures that all pipes have a model length within the Length Accuracy.

**Component -** means devices such as pumps, tanks and surge protection devices. In this case, it refers to the pump

**Travel Increments -** the number of time increments required for the wave to go from one end of the pipeline to the other end. If the length of pipeline is 4000m and wave speed for that pipe is 4000m/s and computational time interval (same as Time Increment) is 0.01s then the number of time increments for that pipeline would be 100.

**Total Number of Time Increments -** is the sum total of time increments of all pipelines in the model multiplied by 2 (wave to travel down and return).

C/GA - C is the celerity or wave speed and G is gravitational acceleration, and A is area. C/GA times the change in flowrate gives potential change in head, Joukowsky's equation.

**Segment Resistance** - The resistance applied to each segment that the pipe is divided into. This is the resistance used for the Wave Method "friction orifice" calculation.

**Why Different Wave Speed?** Wave Speed Adjustment is used. This is default setting under System Data | Preferences. Here is definition:

**Wave Speed Adjustment** - The wave speed is only adjusted for short pipes (less than the length accuracy) where both the length is increased and the wave speed decreased by the same percentage. For example if the length accuracy is 40 feet and there is a 10 foot pipe - without wave speed adjustment the length would be adjusted to 40 feet with the same wave speed (say 4000 f/s). If the wave speed adjustment is used the length would be increased by 50% (to 20 feet) and the wave speed decreased by 50% (to 2000 f/s) so the wave travel time would be one time increment.

**For Components:**

**Initial Flow** - is flow through pump. Head #1 and Head #2 are head for inlet and outlet side of pump.

**Pos #1 and Pos #2 -** are position 1 and 2. This means it is the location of the component, which is the pump.

**Characteristics A, B, C -** are internal values used to plot the curve for pump operation.

**Connecting Positions**- In surge analysis, each pipe is modeled with its own unique node numbers. For example, if there are two pipes are connected at a junction node 5, node 5 will be common to both pipes for steady state analysis and as seen on the GUI. End nodes for the first pipe might be 4 and 5 and for the second pipe 5 and 6 (with 5 as common node). This is NOT acceptable for surge analysis as both pipes need to have unique end positions. We will rename the starting position of second pipe as say 10 instead of 5 and now the second pipe will have unique end positions 10 and 5. Now we need to tell the Surge program what this new number is and connecting positions under junction data represent those new position numbers. For example, if there are two node numbers listed under Connecting Positions column (eg. 55 and 56), these numbers represent the new starting node numbers for the two pipes connected at the node

**Check Valve Resistance**- is head drop (m) over the flow squared (cms). R = H/Q^2

**Time Reverse Gradient** - the total time that the pressure is < 0 (sub atmospheric) at that node.

Special symbols:

**zz~~temp** represents a temporary node created internally to complete calculations for an SDO (surge protection device such as Air Valve or Surge Tank) located at the end of a pipe, the SDO having only one pipe connection instead of two. In earlier versions of the program, locating an SDO at the end of a pipe with only one pipe connection was not allowed. SDOs with only one connection are handled in later versions by internally adding a small temporary pipe and node so the two required connections are made

## 1.5.23 Role of Pump Inertia Values in Transient Analysis of Pipe Line System

**Role of Pump Inertia Values in Transient Analysis of Pipe Line System**

Sathishkumar[1], Sarankumar[2] , Samir Raval[3] , Srini Lingireddy[4]

**Abstract**

The combined inertia of pumps and motors, including shafts and couplings, is required for the analysis of transient flow situations associated with sudden stopping of pumps. Accurate value of pump inertia information is frequently not available at the design time and needs to be determined by empirical means in order to simulate pump trip events. Underestimated values of pump and motor inertia can result in over sizing of surge protection system while overestimated

values may result in under sizing of surge protection system and consequent failure of pipeline system.

Empirical equations were generated by different groups of researcher based on pump characteristic data collected several years ago. The variation in predicated and actual values for certain pumps appears to be quite significant (±200%). Though the regression methodology employed appears to be one of the reasons for such variation, the manufacturing practices in different parts of the world could be another reason for such variation. For example, the combined pump inertia values for large pumps manufactured in India appear to be several times higher than the values estimated from empirical equations. This study was aimed at collecting more recent data from different parts of the world and employ better curve fitting techniques (such as neural networks) to arrive at more accurate empirical equations for computing pump motor inertia values.

**Introduction**

Interest in large water transmission main projects has increased many folds in recent years especially in emerging markets such as India and China. Transient analysis for arriving at appropriate surge protection system is an essential component in the design process of large pumping mains. Pump trip event resulting from power failure is generally the most severe transient condition for most transmission mains. It is a common practice to use the non-dimensional four quadrant pump characteristics in the form of Suter curves for modeling pump trip events. Moment of inertia of the moving parts (impeller, entrained liquid, shaft, couplings, prime mover etc) plays a major role in determining the severity of transient resulting pump trip events. Moment of inertia of the pumping machine may be defined as (Wood et. al 2005):

$$I = \sum_{i=1}^{S} W_i R_i^2 \quad (1)$$

where I is moment of inertia in $Nm^2$, $W_i$ is weight in Newtons and $R_i$ is radius of gyration in meters of $i^{th}$ moving part, and S is the total number of moving parts of the pumping machine. Though most manufacturers provide this information upon request, it may be essential to have a rough estimate for the moment of inertia at the preliminary design stage to estimate the potential cost of surge protection system at the tendering stage.

**Empirical Equations**

Donsky (1961) has proposed the following empirical equation for estimating the moment of inertia:

$$I = 3550\left[\frac{HP}{N}\right]^{1.435}$$

(2)

where I is moment of inertia in lb-ft$^2$, HP is the motor horse power, and N is rated pump speed in RPM. Thorley (1991) has proposed similar empirical equations to account for the light weight design of the pumps manufactured after 1960s:

$$I_p = 0.33423 \times \left[\frac{P_{kW}}{(N/1000)^3}\right]^{0.844}$$

$$I_M = 0.0422 \times \left[\frac{P_{kW}}{(N/1000)}\right]^{1.48}$$

(3) and (4)

where $I_P$ and $I_M$ are moments of inertia of pump and motor respectively, $P_{kW}$ is motor power in kilo watts, N is rated speed in RPM. These empirical equations were developed based on more than 500 data points gathered from a diverse group of pump and motor manufacturers. The shape of the equations was arrived at by comparing the laws of geometric similarities of pumping machinery with homologous laws of pumps. The coefficients of equations 3 and 4 were determined by

regression analysis relating the power coefficient $\frac{P_{kW}}{(N/1000)^3}$ with manufacturer provided inertia values for pumps and $\frac{P_{kW}}{(N/1000)}$ with manufacturer provided inertia values for motors.

Figure 1 shows a sample plot of the pump inertia data points extracted from Thorley (1991) plotted on logarithmic scale. Though the data appear to correlate very well on the log scale, a closer look at some of the data points indicates that the deviation between actual (manufacturer suggested) and estimated (Thorley's equation) values is quite large. Figure 2 shows the same plot of same data points on a normal scale to illustrate the deviations in actual and estimated values.

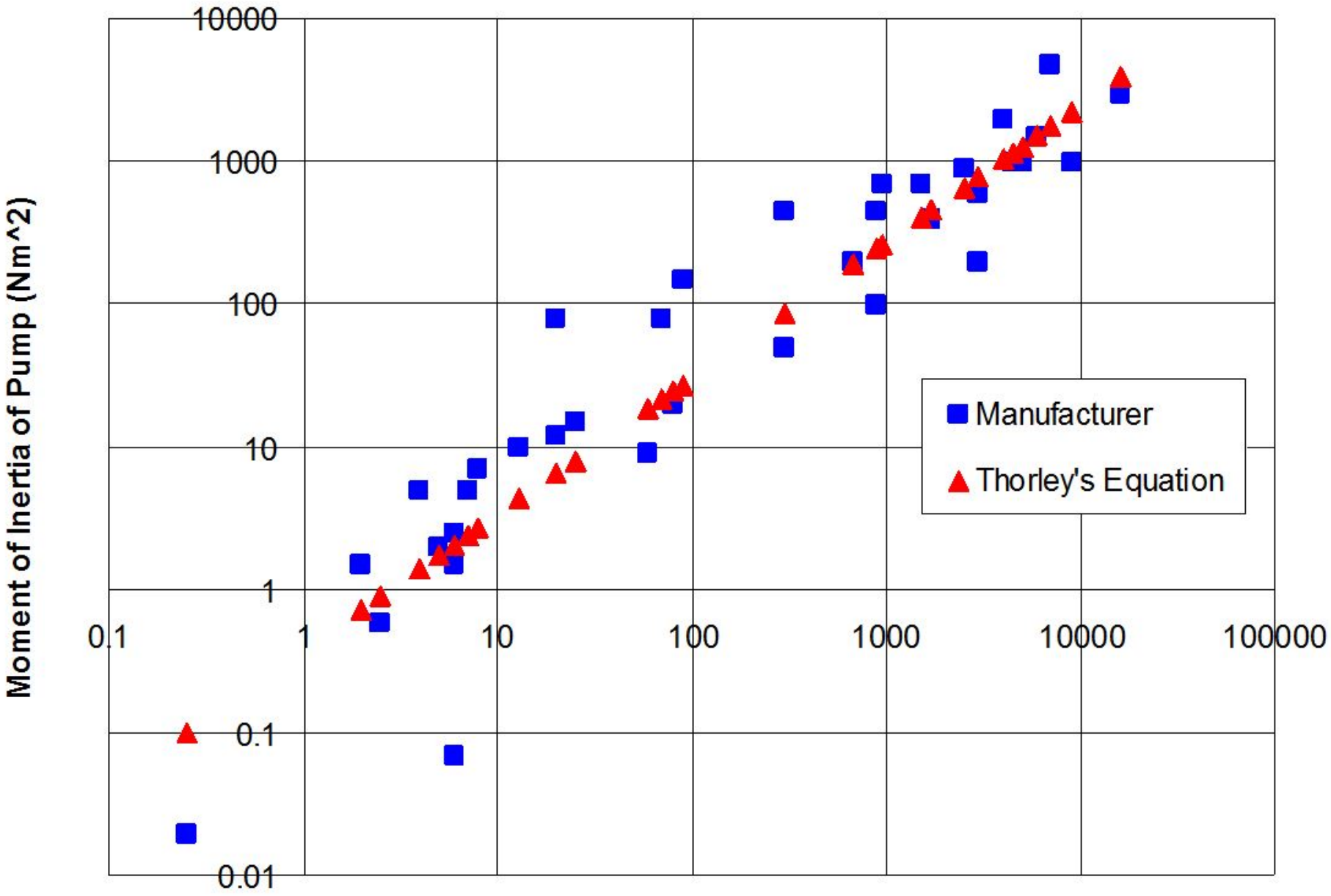

Moment of Inertia of Pump (Nm^2)
10000
1000
100
10
1
0.1
0.01
0.1
1
10
100
1000
10000
100000
Manufacturer
Thorley's Equation

Figure 1. Sample data of pump inertia values plotted on logarithmic scale (Thorley 1991)

The effect of inertia on transient pressures generated by a pump trip is demonstrated on an example large scale water transmission system.

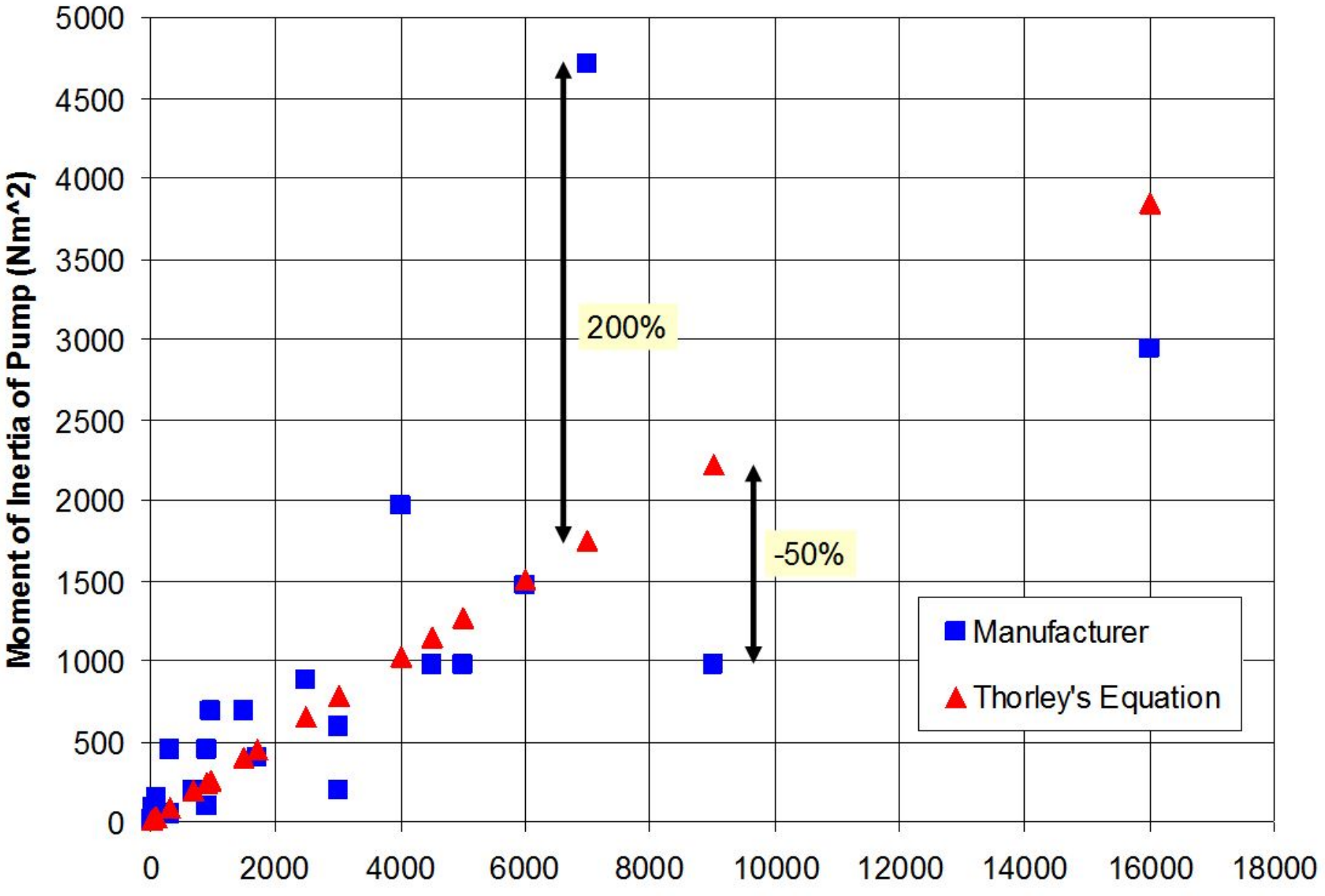

Moment of Inertia of Pump (Nm^2)
5000
4500
4000
3500
3000
2500
2000
1500
1000
500
0
0
2000
4000
6000
8000
10000
12000
14000
16000
18000
200%
-50%
Manufacturer
Thorley's Equation

Figure 2. Sample data of pump inertia values plotted on normal scale to illustrate the scale of deviation between actual (manufacturer suggested) and estimated values (Thorley's equation)

**Effect of Inertia on Transient Pressures**

The authors have been associated with several large scale water transmission main design projects in the recent years and have realized the huge deviations in manufacturer suggested inertia values compared to the values computed by using Thorley's equations. The following case study shows the implications of gross under/over estimation of combined inertia (pump and motor) values on large scale pipeline systems.

The pipeline system was designed to transfer 14$m^3$/s of water through a 3m diameter of steel pipe over a distance of 40km lifting water by nearly 150m. The transient modeling of this pipeline system was carried out using the Surge2008 program of KYPIPE LLC, USA (www.kypipe.com).

Figure 3 shows transient pressure envelope over the pipeline system during a 300 second simulation using the pump and motor inertia values (4113252 $Nm^2$) supplied by the manufacturers. The simulation considered two number of 70$m^3$ bladder vessels as primary surge protection devices and several other secondary surge protection devices. Figure 4 shows the transient pressure envelope for the same system (with the same protection) but using the combined pump and motor inertia values (161828 $Nm^2$) computed using Thorley's empirical equations. As evident from Figures 3 and 4, there is nearly 10bar difference in maximum pressures between the two cases and would require nearly double the volume of bladder vessels to bring down the pressures to acceptable limits. Though the error is more on the conservative side, the initial estimates for surge protection system with gross underestimates for moment of inertia values could discourage some of the contractors from the bidding process for EPC (Engineering, Procurement and Construction) or BOOT (Build, Own, Operate and Transfer) projects.

Figure 3 Pressure head envelope with manufacturer provided combined inertia

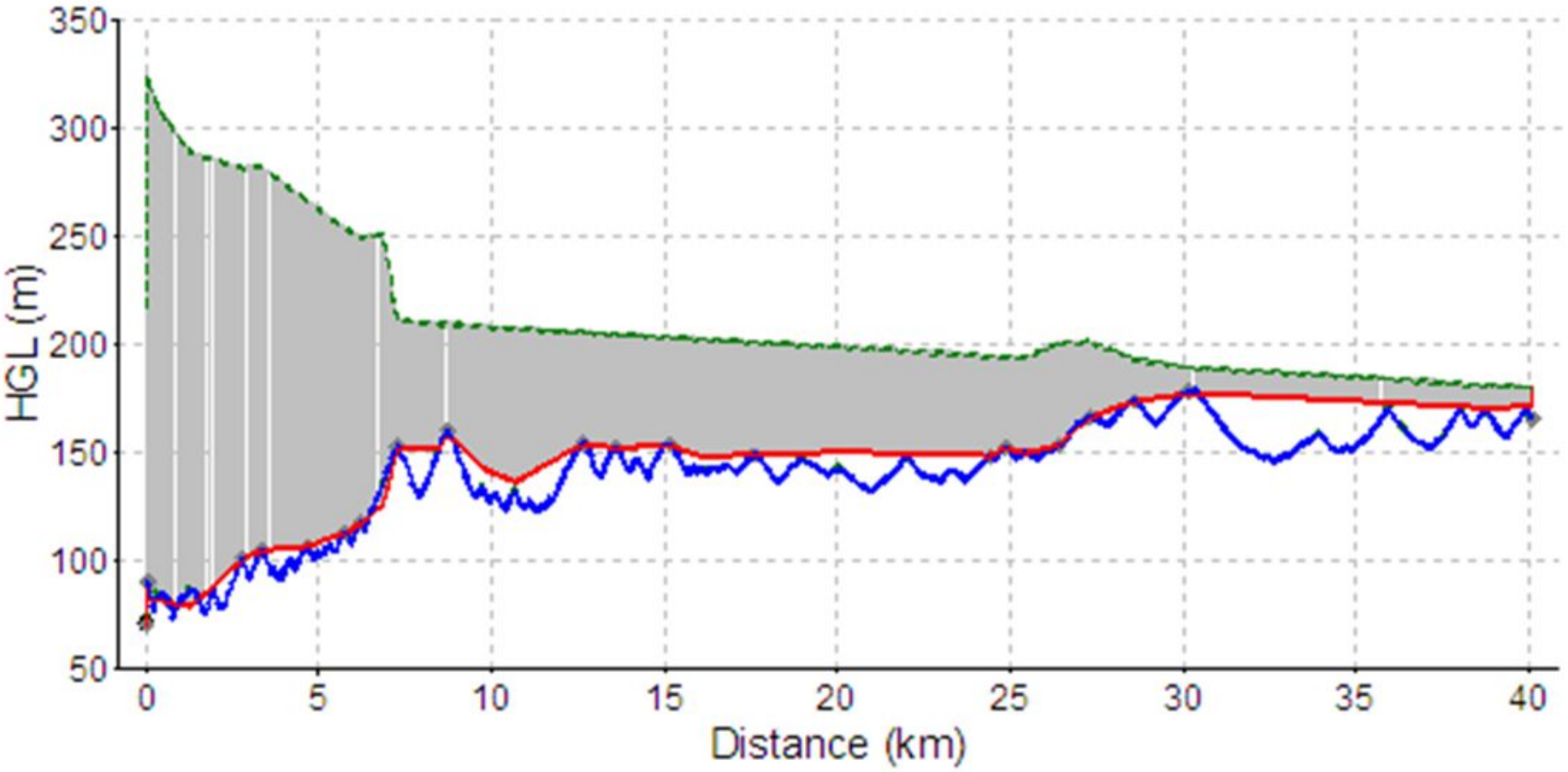
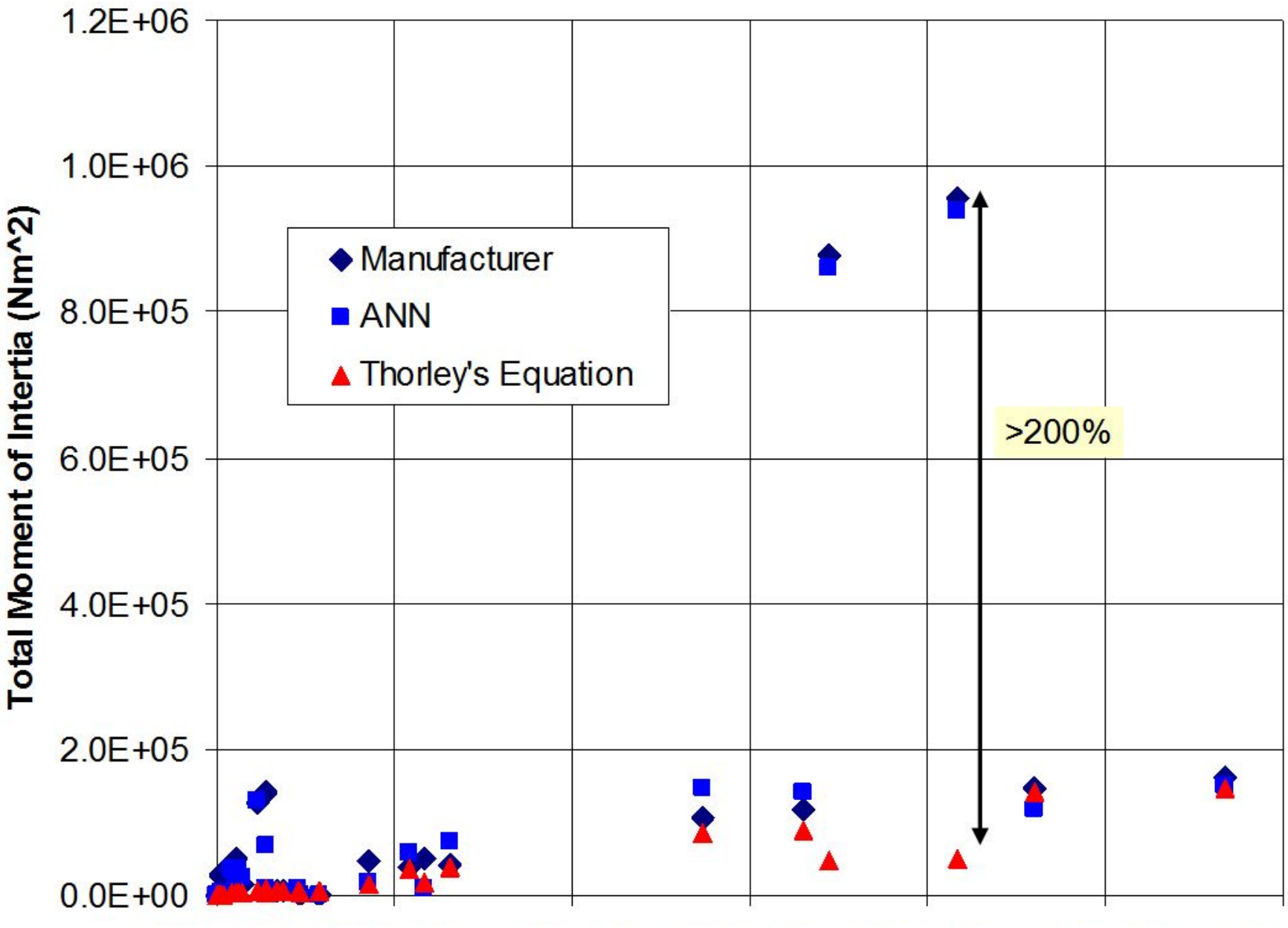



Figure 4. Pressure head envelope with empirical combined inertia (Eqs. 3 and 4)

**Advanced Prediction Tools**

Attempts were made to improve the accuracy of estimated combined moment of inertia values using advanced regression tools based on artificial neural networks (ANN). Data from several different manufacturers was compiled and trained using a ANN tools to obtain a better correlation between input values (rated flow, rated head, rated speed, best efficiency) with the manufacturer provided combined pump and motor inertia values. The database comprised about 110 data points and the data was divided in the ratio of 3:1::Training:Testing. The latest version of Neurosort (Neelakantan et. al,., 2007) was used for this exercise. Figure 5 shows the ANN predicted values compared to manufacturer values as well as values predicted using Thorley's equation (Equations 3 and 4) plotted as functions of power coefficient $\frac{P_{kW}}{(N/1000)^3}$. As evident from this Figure, the ANN predictions are much closer to the manufacturer provided values while the deviation between manufacturer values and values predicted using Thorley's equations off by more than 200% for some data points. Figure 6 shows the same data plotted for the lower ranges of power coefficient. The ANN predictions are far superior compared to Thorley's equations even in the lower ranges of moment of inertia values demonstrating the superiority of the proposed approach.


1.2E+06
1.0E+06
8.0E+05
6.0E+05
4.0E+05
2.0E+05
0.0E+00
Total Moment of Intertia (Nm^2)
Manufacturer
ANN
Thorley's Equation
>200%

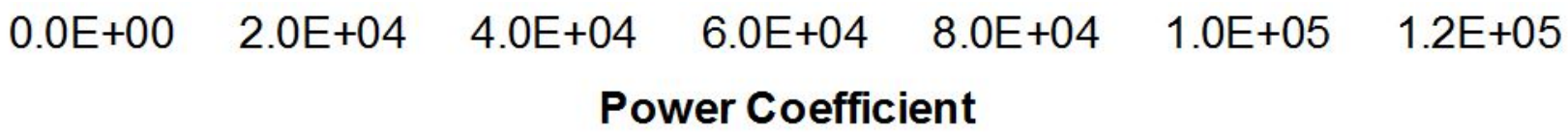
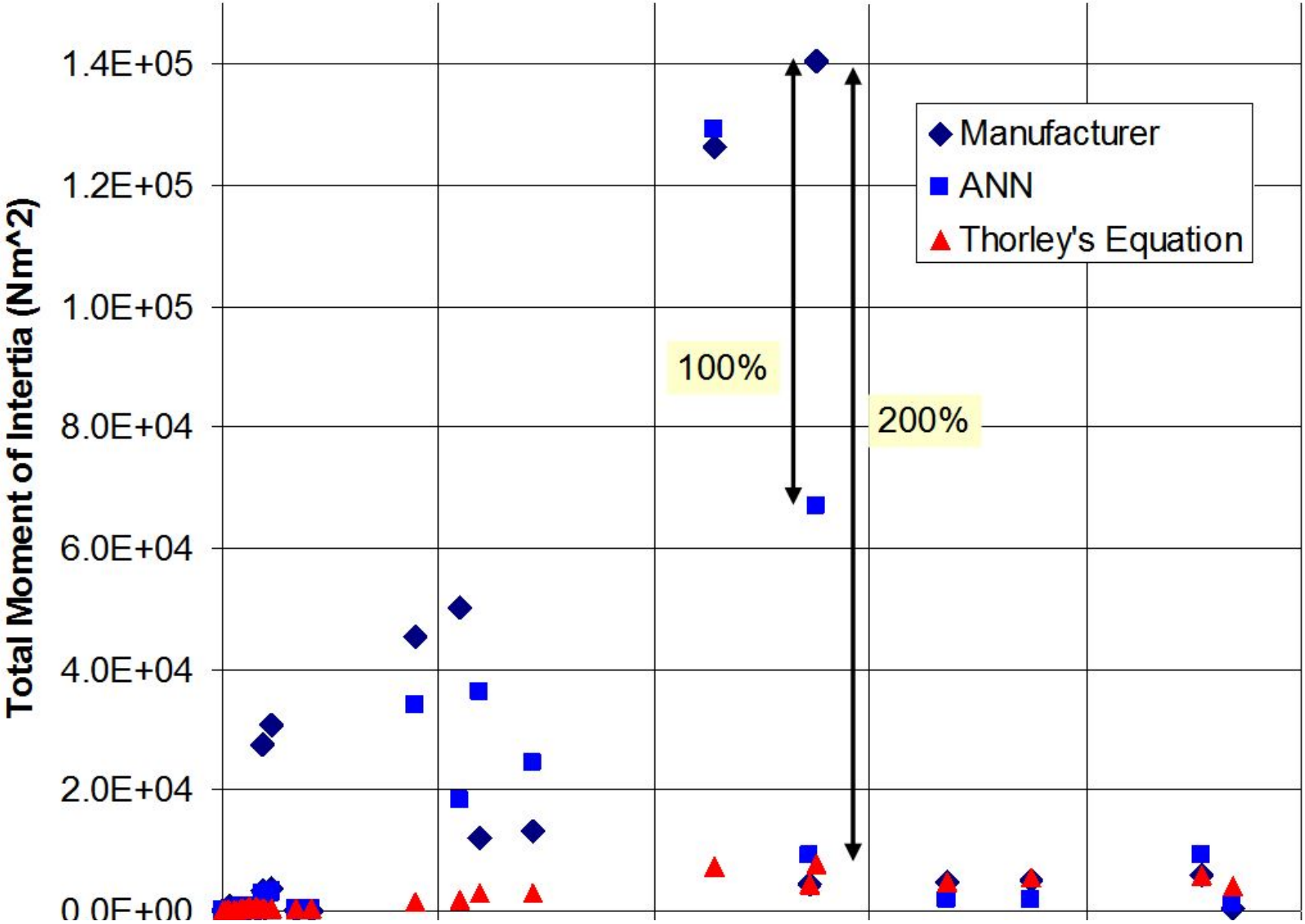



Figure 5. Actual (manufacturer) inertia values compared to ANN and Thorley Equations predictions.


Total Moment of Intertia (Nm^2)
1.4E+05
1.2E+05
1.0E+05
8.0E+04
6.0E+04
4.0E+04
2.0E+04
0.0E+00
Manufacturer
ANN
Thorley's Equation
100%
200%

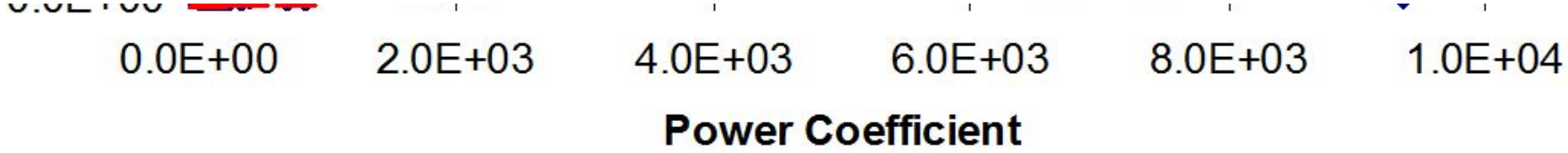



Figure 6. Actual (manufacturer) inertia values compared to ANN and Thorley Equations predictions – lower ranges of inertia values.

**Summary and Conclusions**

The need for precise estimation of combined pump and motor inertia was demonstrated on an example large scale water transmission pipeline project transferring raw water of 14$m^3$/s over a distance of 40km through a 3m diameter steel pipeline lifting water by nearly 50m. Moment of inertia predictions by the existing regression tools could be off by as much as 200% and might lead to gross under/over estimation of the ultimate surge protection requirement. The proposed tool based on artificial neural network appears to provide more accurate estimate for the combined pump and motor inertia values both in upper (>10000 $Nm^2$) and lower ranges of inertia values.

**References**

Donsky, B. (1961). "Complete pump characteristics and the effects of specific speed in hydraulic transients." *Journal of Basic Eng*., ASME, Vol. 12, pp. 685-691.

Neelakantan, T., Chandramouli, V., Lingireddy, S., Brion, G., (2007) "NeuroSort - Artificial Neural Network Tool for water systems", University of Kentucky, Lexington, KY, USA.

Thorley, A.R.D. (1991). *Fluid Transients in Pipeline Systems*. D. & L. George Ltd Publ., Herts, UK.

Wood, D.J., Lingireddy, S., and Boulos, P.F. (2005) *Pressure Wave Analysis of Transient Flow in Pipe Networks,* MWH Press, USA

## 1.5.24 EPA Surge

Inlcuded in Pipe2012 and later versions.

To download a full manual, go to www.epasurge.com

EPA-Surge is a powerful transient analysis program included with the Surge model. This new feature has been designed specifically for EPANET users so that transient analysis can be performed on EPANET models easily. This program will utilize a working EPANET file to define a piping system and the initial conditions for initiating a transient analysis. The transient analysis can be initiated by valve action or pump action or both. A wide variety of surge protection devices can be incorporated into the analysis.

EPA-Surge empowers the EPANET user by allowing them to model and analyze transients in an EPANET model without re-entering data. The simple wizard will transfer EPANET data into a surge model with only four quick steps. Surge control devices and events can be specified and results can be displayed in tables, graphs, contours and animations. Download the free demo and sample this new feature .

## Four Simple Steps to Analyze Transients in an EPANET Model.

### Step One: Prepare Baseline EPANET File

The EPANET file should define a single initial condition for transient analysis. This involves setting the following parameters:

- Reservoir & Tank Levels
- Demands
- Pump status (on/off) & Speed Ratio
- Valve status (on/off) & Setting
- Regulator Settings

### Step Two: Load EPANET (.inp) File & Start Wizard

Generate an .inp from EPANET's file menu (File | Export | Network)

**Step Three: Complete Four Surge Wizard Tasks**

- Set Wave Speed
- Set Demands & Demand Options
- Insert Surge Protection Devices
- Modify Baseline Data with Operational Control Screen (*OCS)
  *OCS displays and provides the capability to modify the following settings: 1) Tank and Reservoir Levels and ON/OFF switch 2) Pump ON/OFF status, initial speed ratio and number operating for multiple pumps 3) Valve ON/OFF status, type and initial setting 4) Regulator settings

**Step Four: Run Analysis**

Final Results

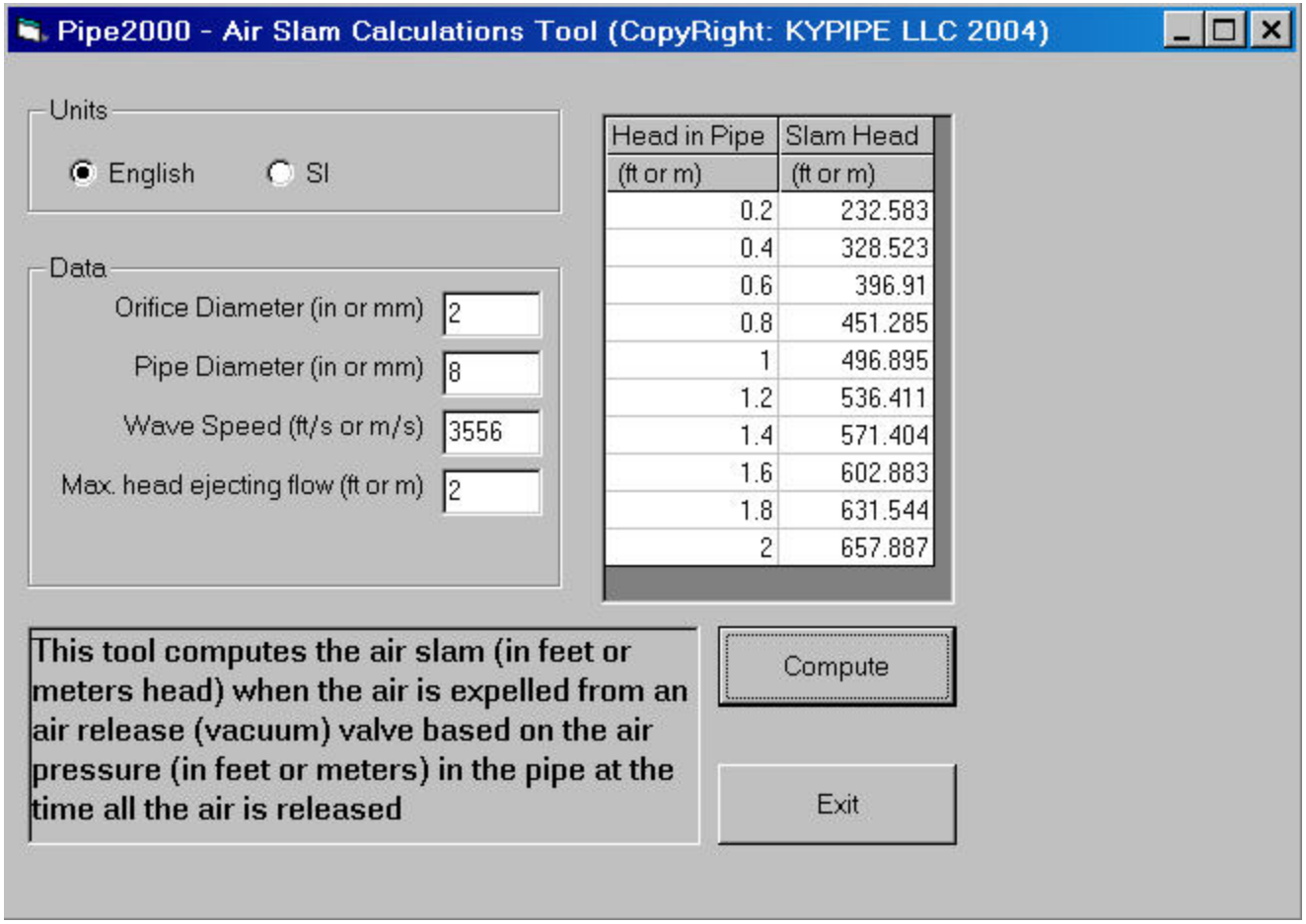
- Steady State
- Surge

## 1.5.25 Surge Tools

## 1.5.25.1 Air Slam Pressure Surge Tool

### Air Slam Pressure Surge

This tool is used to estimate the surge pressure potential due to air expulsion from an air release/vacuum valve. It calculates the pressure surge generated by water column impact following the expulsion of air from an air release/vacuum valve. Provide the required data displayed on the screen and an upper limit for the air pressure just prior to the slam and the tool computes surge pressures for a range of air pressures up to the maximum specified.

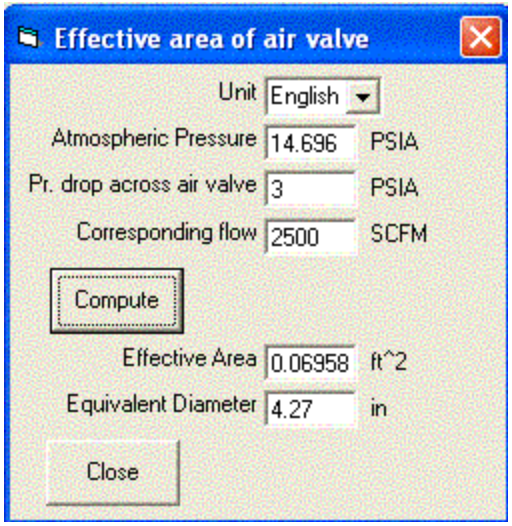



## 1.5.25.2 Air Valve Orifice Size Tool

**Air Valve Orifice Size**

This tool will calculate the orifice size required for an air valve to flow a specified volumetric flowrate at a specified pressure drop. This will aid in air valve selection.

See also Air Vacuum Valves and Three Stage Valves



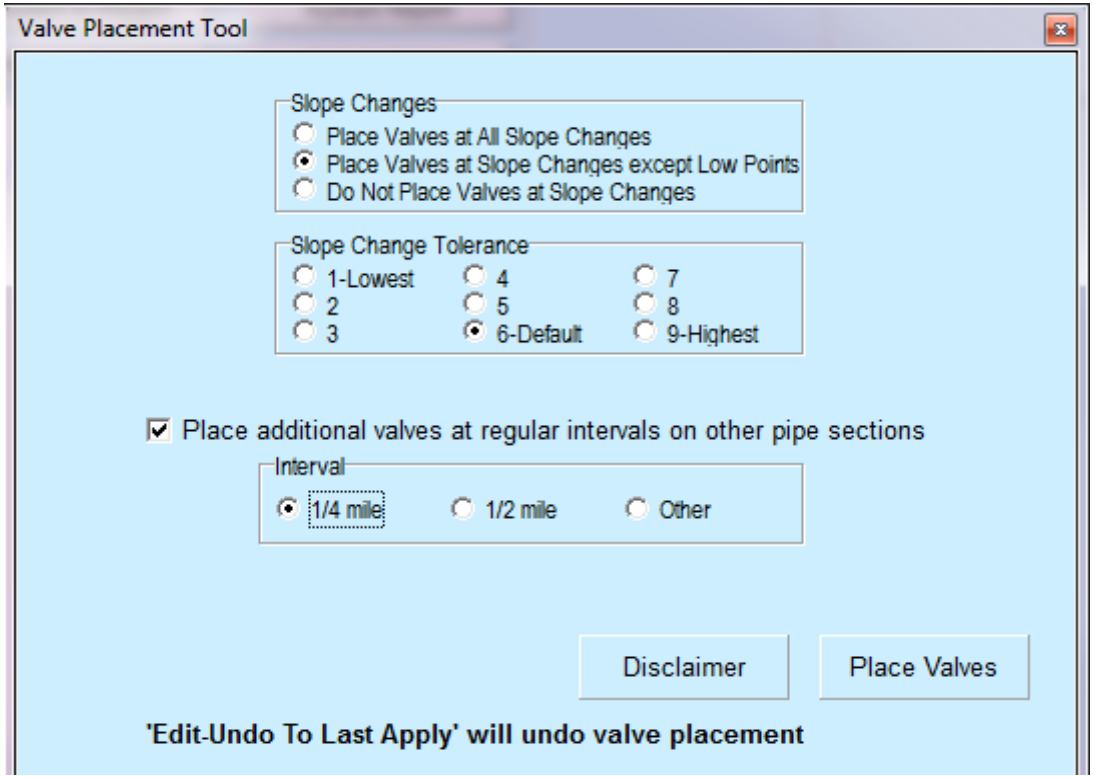

## 1.5.25.3 Air Valve Placement Tool

### Air Valve Placement Tool

For Surge only, available in both KYnetic and Classic.

In pipelines air valves are often placed at high points, where large slope changes occur, or sometimes at regular intervals. The Air Valve Placement Tool will automatically insert air valves based on these criteria, which may be adjusted by the user.

The tool is located under Facilities Management (“Other” section in KYnetic)




**Place Valves at All Slope Changes**. Slope Change Tolerance settings are important. Lower settings will place valves at less severe slope changes. Higher settings will place fewer air valves.

**Place Valves at Slope Changes Except Low Points**. Same as above but low points are not included.

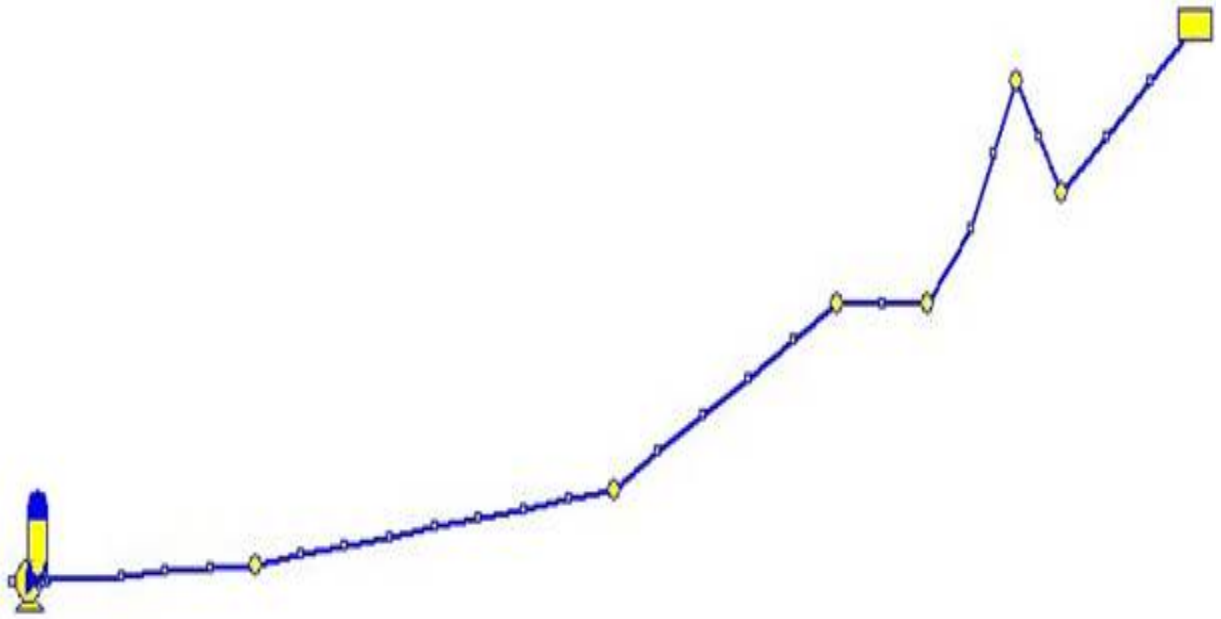
**Do Not Place Valves at Slope Changes.** Slope changes are not considered in valve placement, only regular intervals as defined in the bottom half of the menu.

**Place additional valves at regular intervals on other pipe sections**. Specify the distance between air valves. The units will change (mile and ft, or m) according to the units used in the model.

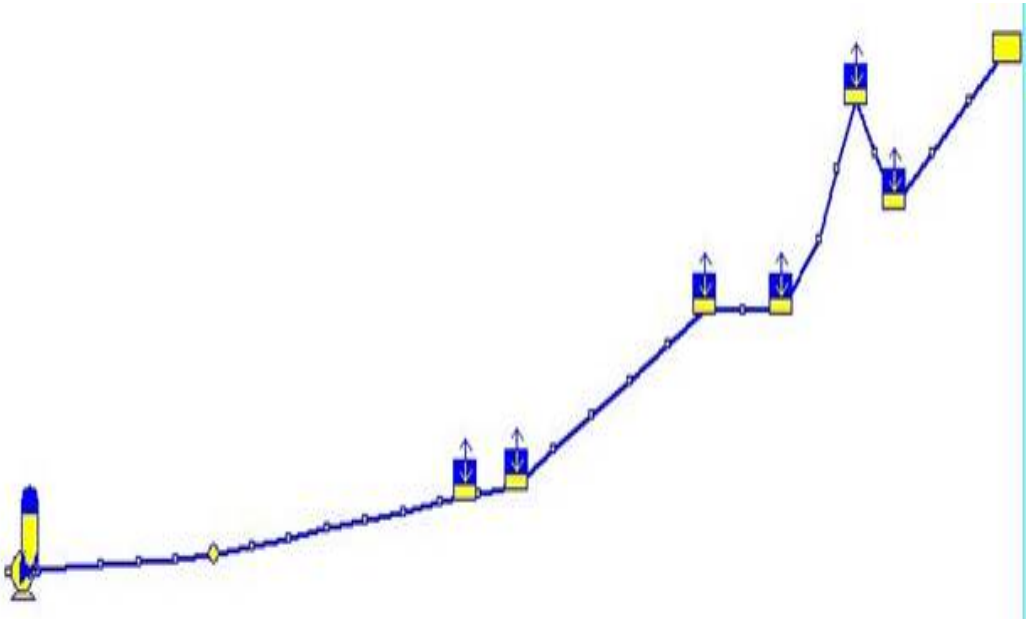
**To undo valve placement.** If the initial valve placement is not satisfactory, use Edit – Undo to last apply (in the main menu of the Pipe program) to start over.

Example:

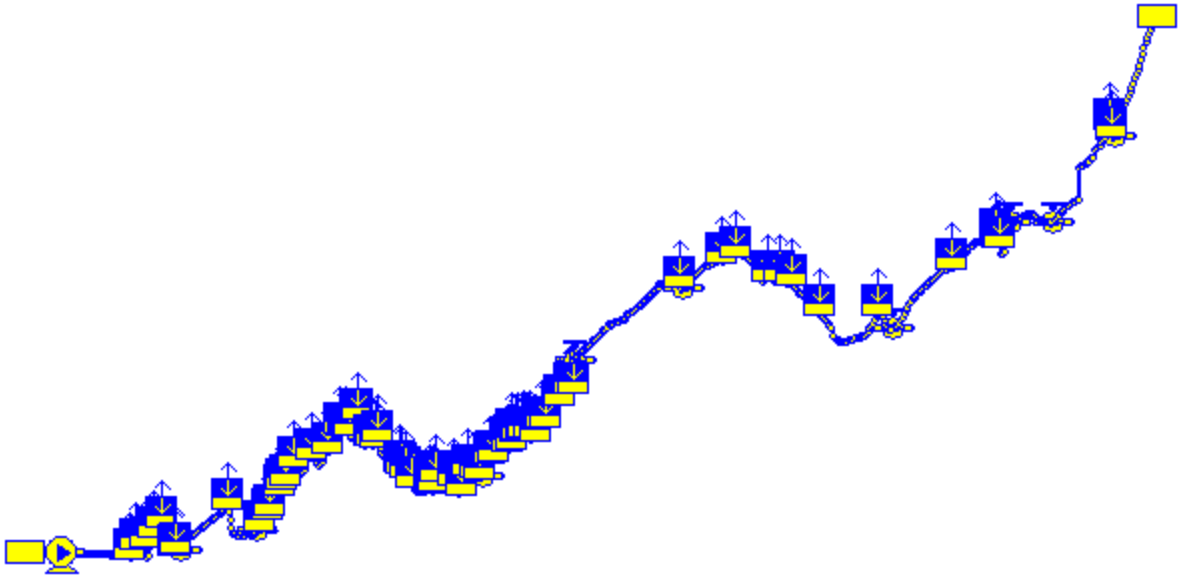
Initial system before air valve placement:

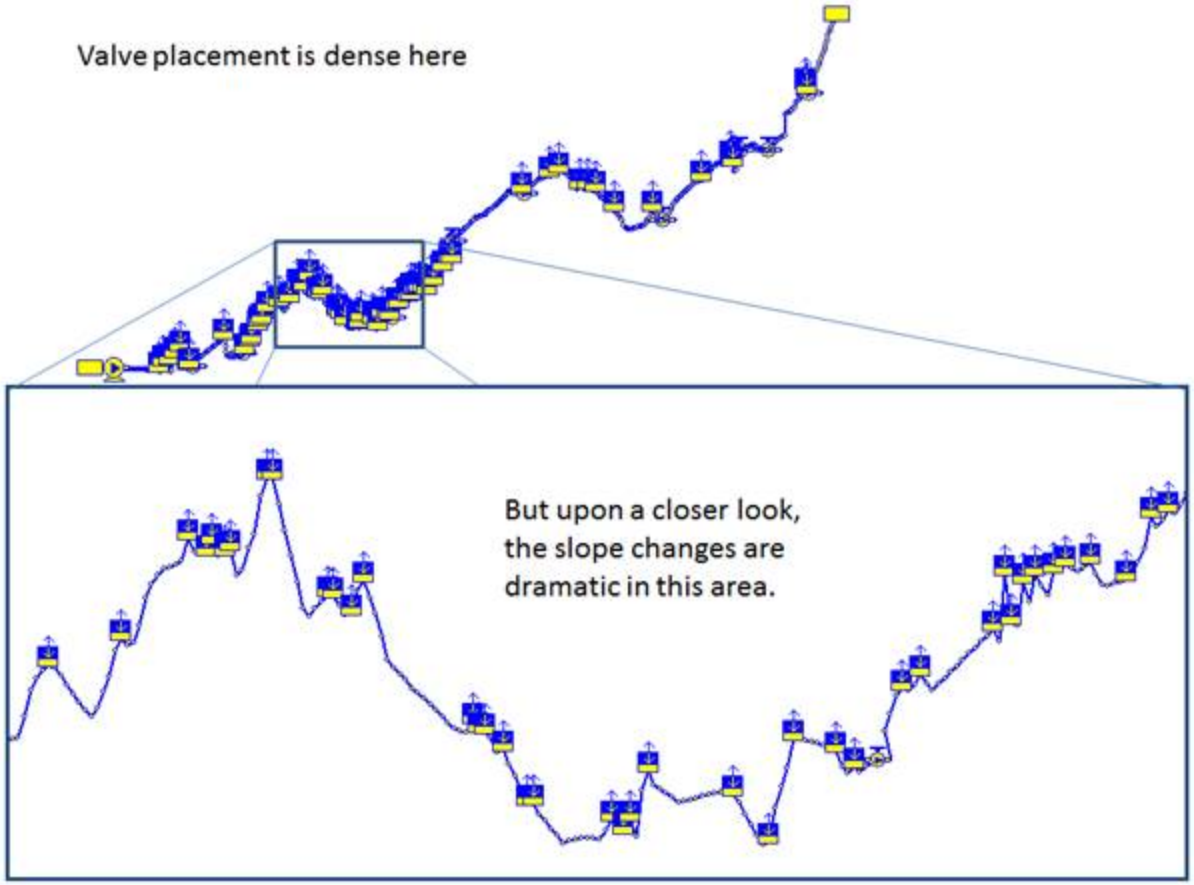


Settings for the tool:



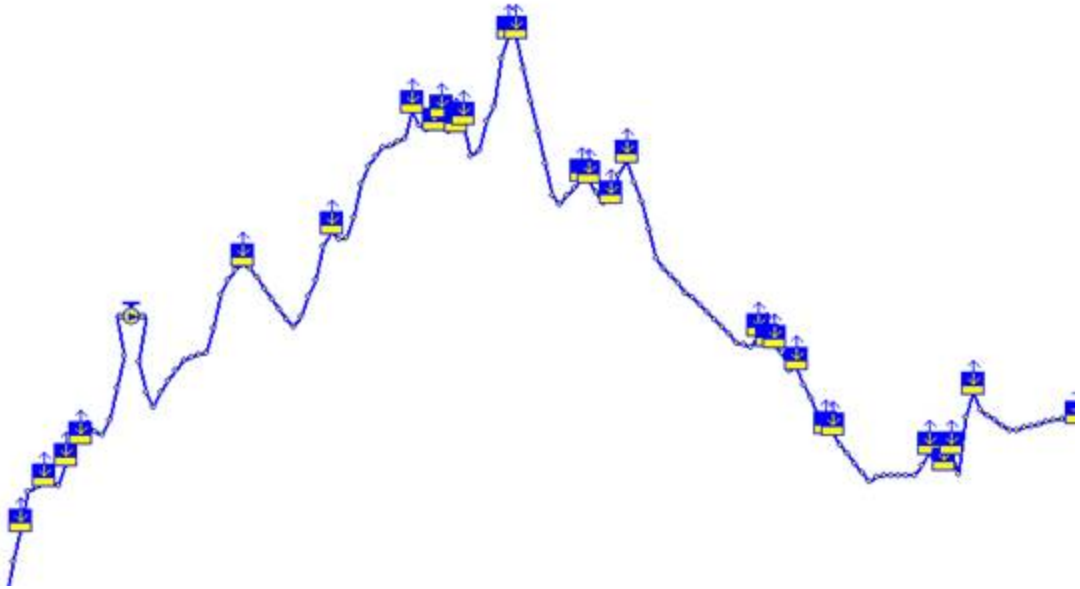
After air valve placement:





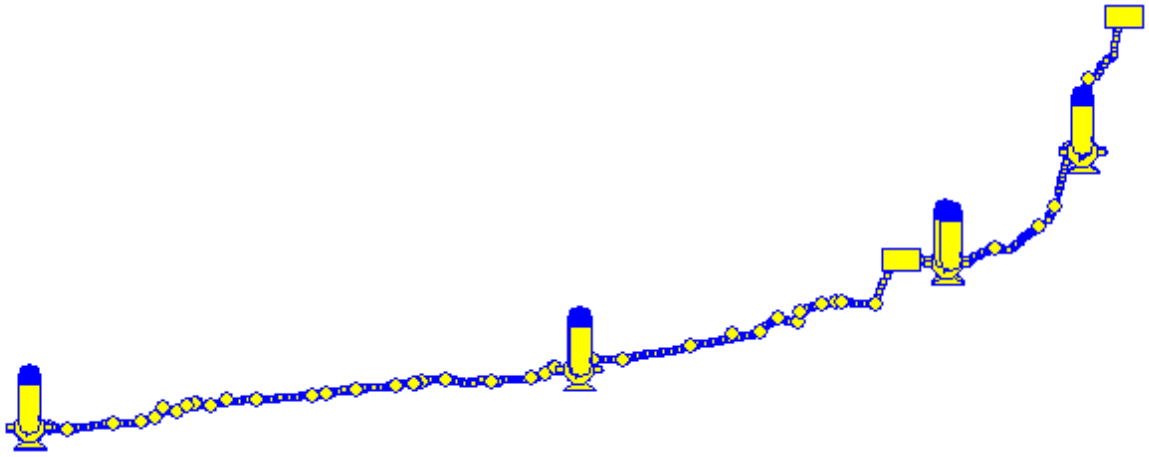

Notice that nodes other than junctions and intermediate nodes will not be replaced. See Active Valve at a high point below:

After Air Valve Placement, Active Valve is still present:

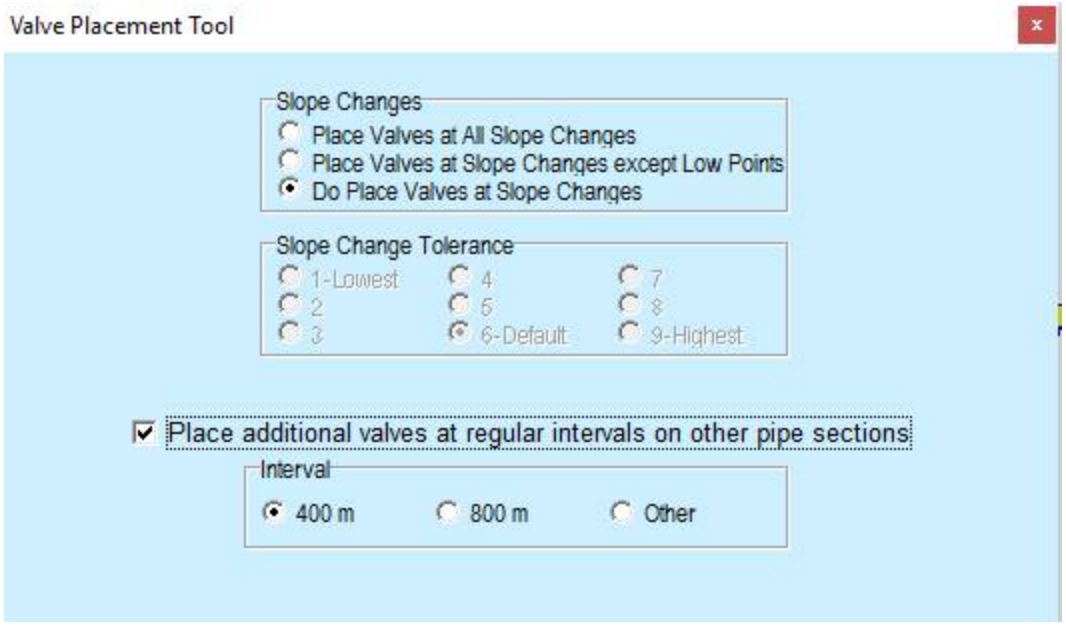


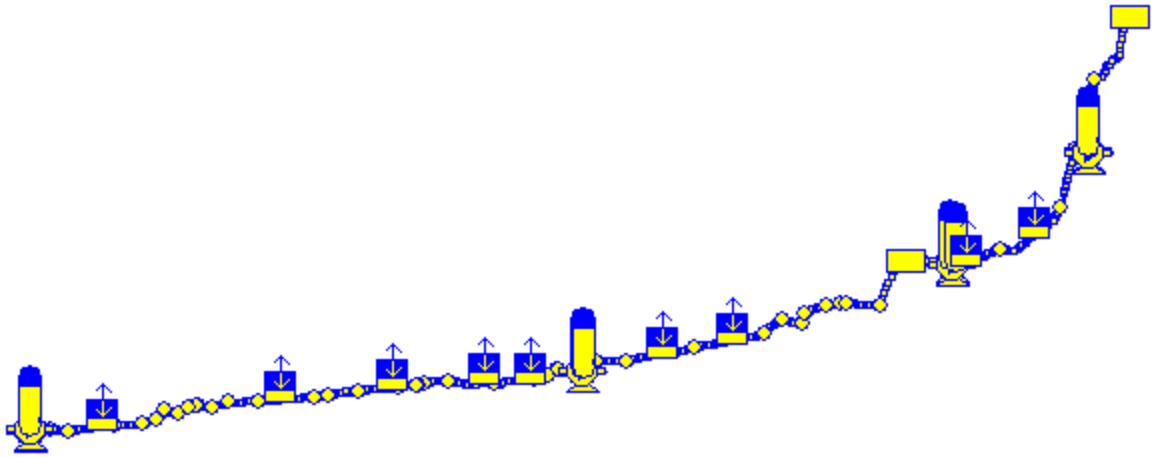
---

Another example:



Using the interval setting at 400 m:





**Notes about slope change calculation.** AWWA M51 only puts forth very general guidelines for placement of air valves. No specific formula for determining slope is defined. In developing the Air Valve Placement tool, a formula was created to determine slope change and allow for a range of tolerances.

Slope-difference = 0.00125 * 2^(Tolerance-1)

For any two segments the Slope-difference is the absolute value of the difference in the slopes.

A Tolerance of 1 gives a Slope-difference of 0.000125 and 9 makes the Slope-difference 0.32.

Each node or intermediate node that connects two pipes or segments is considered. If the slope change is less than the maximum slope difference, the node is removed from consideration (keeping the pipes). After the node is removed from consideration, all nearby nodes are reevaluated. In this way a series of very small gradual changes can't slip through. Nodes where three pipes come together are not considered.

## 1.5.25.4 Bladder Precharge Tool

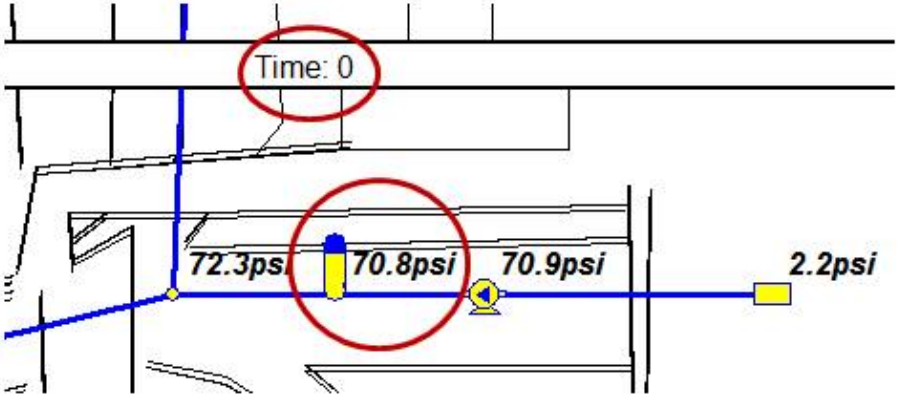
### Bladder Precharge

This tool will allow the user to use the results obtained for a Closed Surge Tank to size a Bladder Tank and determine the required precharge pressure. This Bladder Tank will provide the same results as the Closed Surge Tank. The Initial Volume is input for the closed surge tank and the Maximum Volume is determined by the Volume plot for the surge tank (Make sure the initial maximum volume has been reached). The initial air pressure is computed by the initial steady state analysis. See also Bladder Tanks.

# Using Closed Surge Tank Results to Size a Bladder Tank

1) Run a Surge analysis of a model with an appropriately-sized Closed Surge Tank

2) View the initial (steady state) Pressure results for the Closed Surge Tank. Note the External pressure value in the Results Table. This is the Initial Air Pressure which will be input into the Bladder Precharge Tool.

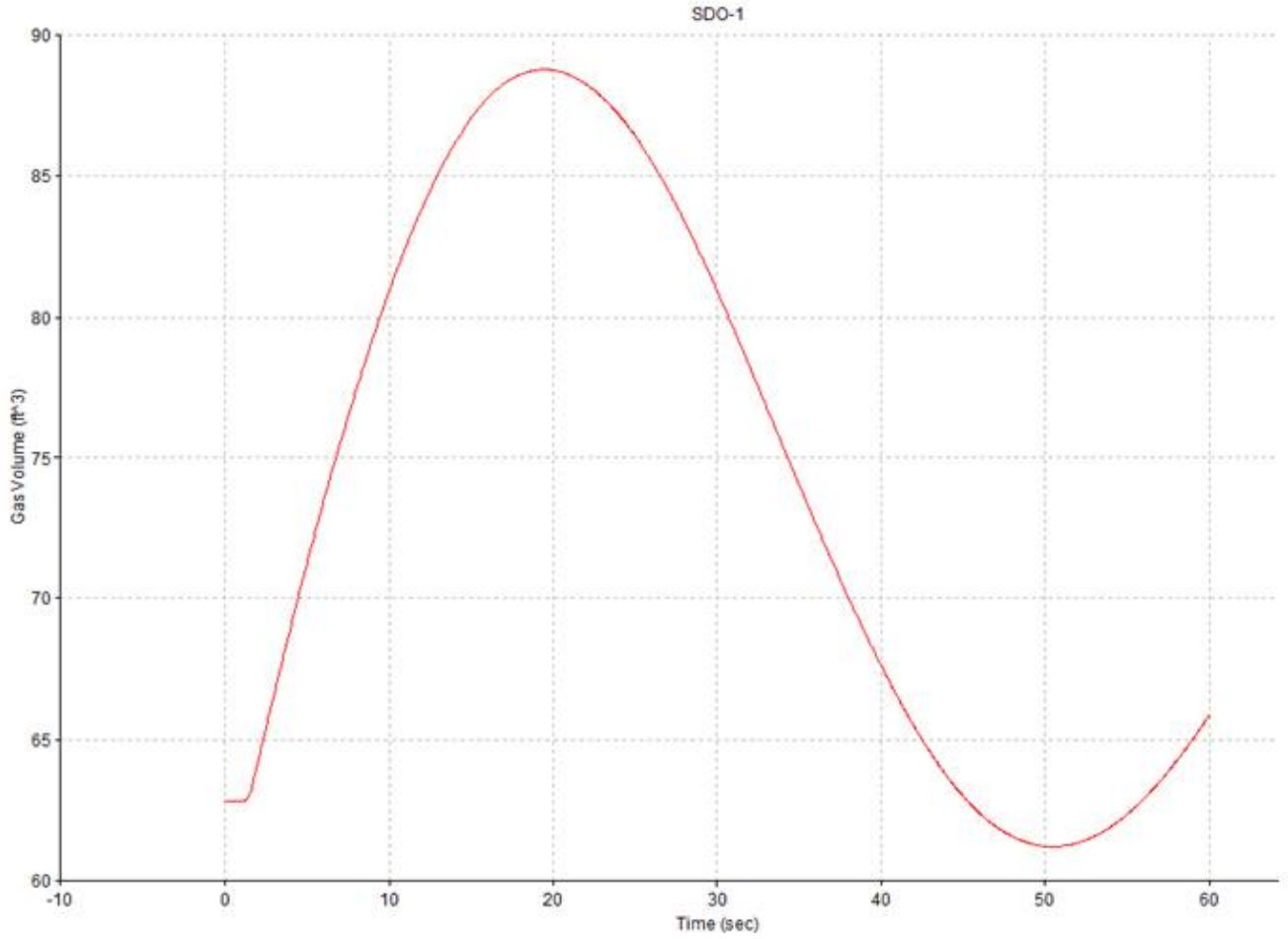



**Pressure**

| Time | Internal | External |
|---|---|---|
| 0 | 70.8 | 69.0 |
| 0.0494 | 70.8 | 69.0 |
| 0.0988 | 70.8 | 69.0 |
| 0.1482 | 70.8 | 69.0 |

Initial Air Pressure = 69.0 psi

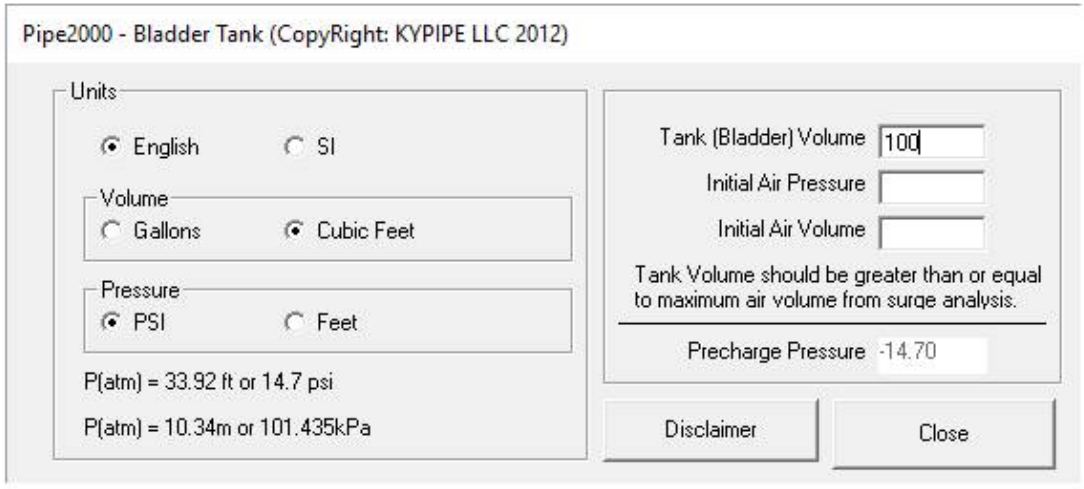
3) Look at the gas volume results for the Closed Surge Tank. Note the maximum volume value.




Maximum Gas Volume = 89 ft^3

# Estimated Bladder Tank Volume

The Maximum Gas Volume is a good estimate for minimum size of the Bladder Tank. A value equal to or larger than Maximum Gas Volume is input into the Bladder Precharge Tool as shown. Instead of 89 ft^3, we will use 100 ft^3 to be conservative.



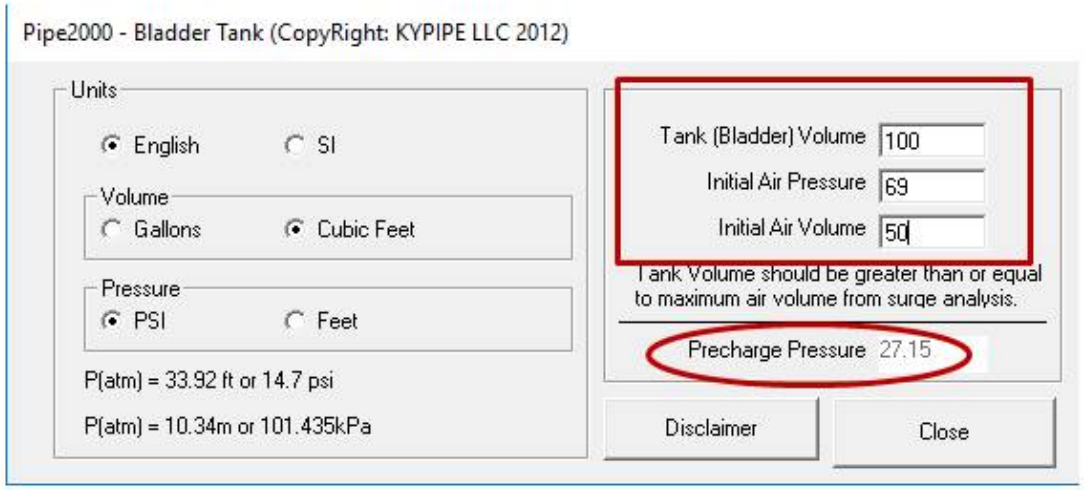
# Estimated Initial Air Volume

This value is estimated by the modeler. Usually it is desired for the bladder under steady state conditions to fill a certain percent volume of the bladder tank. Let's assume we want the bladder to be 50% of the tank volume at steady state.

Estimated Tank Volume = 100 ft^3

Initial Air Volume = 0.5 (100 ft^3) = 50 ft^3

Input in the estimated Initial Air Volume and the estimated Bladder Tank Volume.



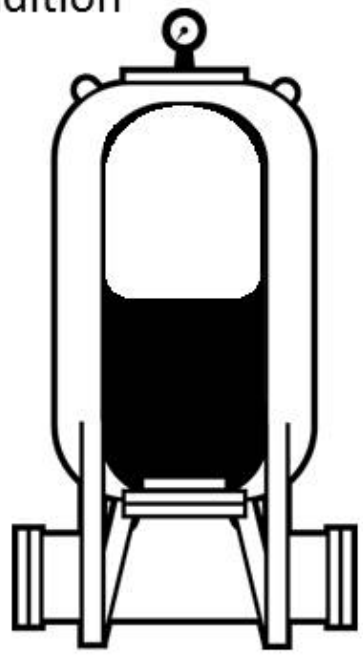

Precharge Pressure = 27.15 psi

**How is this value calculated?**

# Using Boyle's Law, $P_1V_1 = P_2V_2$

$V_1$ = volume of bladder at initial condition
$P_1$ = ABSOLUTE pressure at initial condition

At initial (steady state) conditions, we have stated that want the air volume to be 50% (for this example) of the total volume of the tank. This volume should occur at the Initial Line Pressure. Therefore.....



Initial Steady State

**$V_1$ = 50 ft^2**

***$P_1$ = 69 psig + 14.7 psi = 83.7 psia**

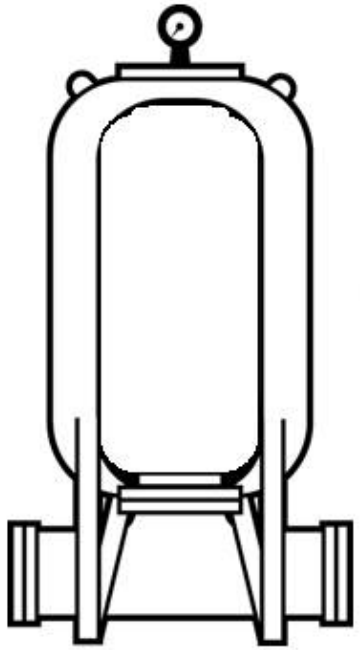
*note the Bladder Precharge Tool makes this conversion

note the Bladder Precharge Tool makes this conversion internally. Input the **GAUGE** pressure value into the tool.

# Using Boyle's Law, $P_1V_1 = P_2V_2$

$V_2$ = volume of bladder at precharge condition
$P_2$ = ABSOLUTE precharge presssure

It is assumed that the bladder is fully expanded in the tank when it is precharged by the manufacturer. Therefore the Volume is the volume of the tank and the Precharge Pressure is what we are calculating.....



Precharge at atmospheric pressure

**$V_2$ = 100 ft^2**

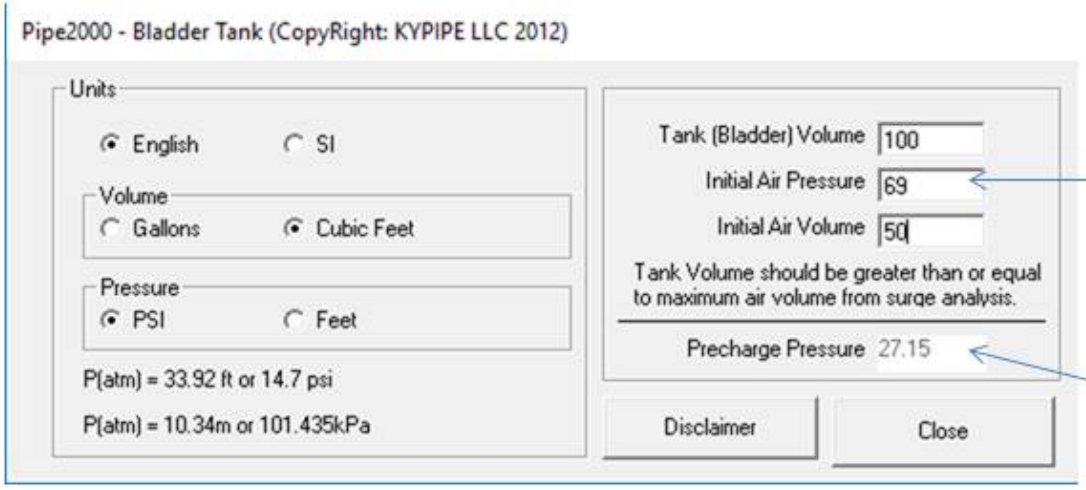
**$P_2$ = Abs. Precharge Pressure**

**$V_1$ = 50 ft^2**
**$P_1$ = 83.7 psi**

**$V_2$ = 100 ft^2**
**$P_2$ = Abs. Precharge**

Absolute Precharge Pressure
50*83.7/100 =41.85 psia

Gauge Precharge Pressure
41.85 psi – 14.7 psi = 27.15 psig




Note, Initial Pressure is input as Gauge Pressure

And Precharge is given as Gauge Pressure.

Low pressure example. SI units.

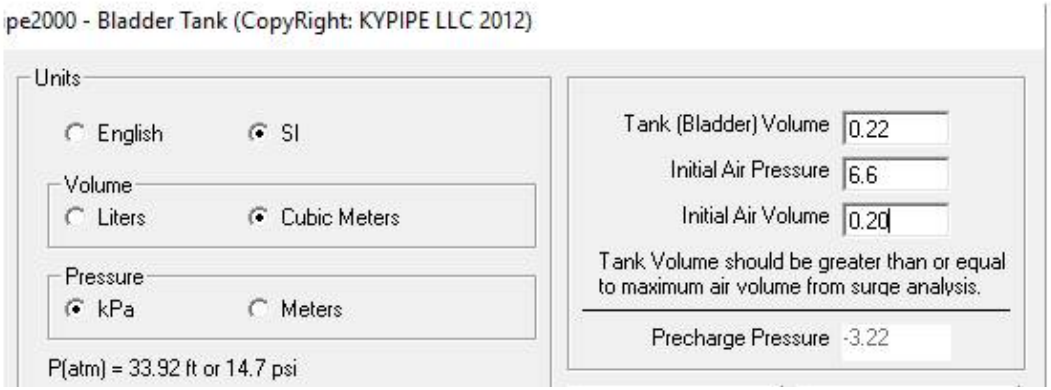
V1 = 0.2
P1(abs) = 6.6 + 100 = 106.6

V2 = 0.22

P2(abs) = 0.2 (106.6)/0.22 = 96.9

P2(g) = 96.9 – 100 = -3.1

Rounding errors might make units slightly off, but the problem with low pressures is demonstrated.



## 1.5.25.5 EPS2Surge

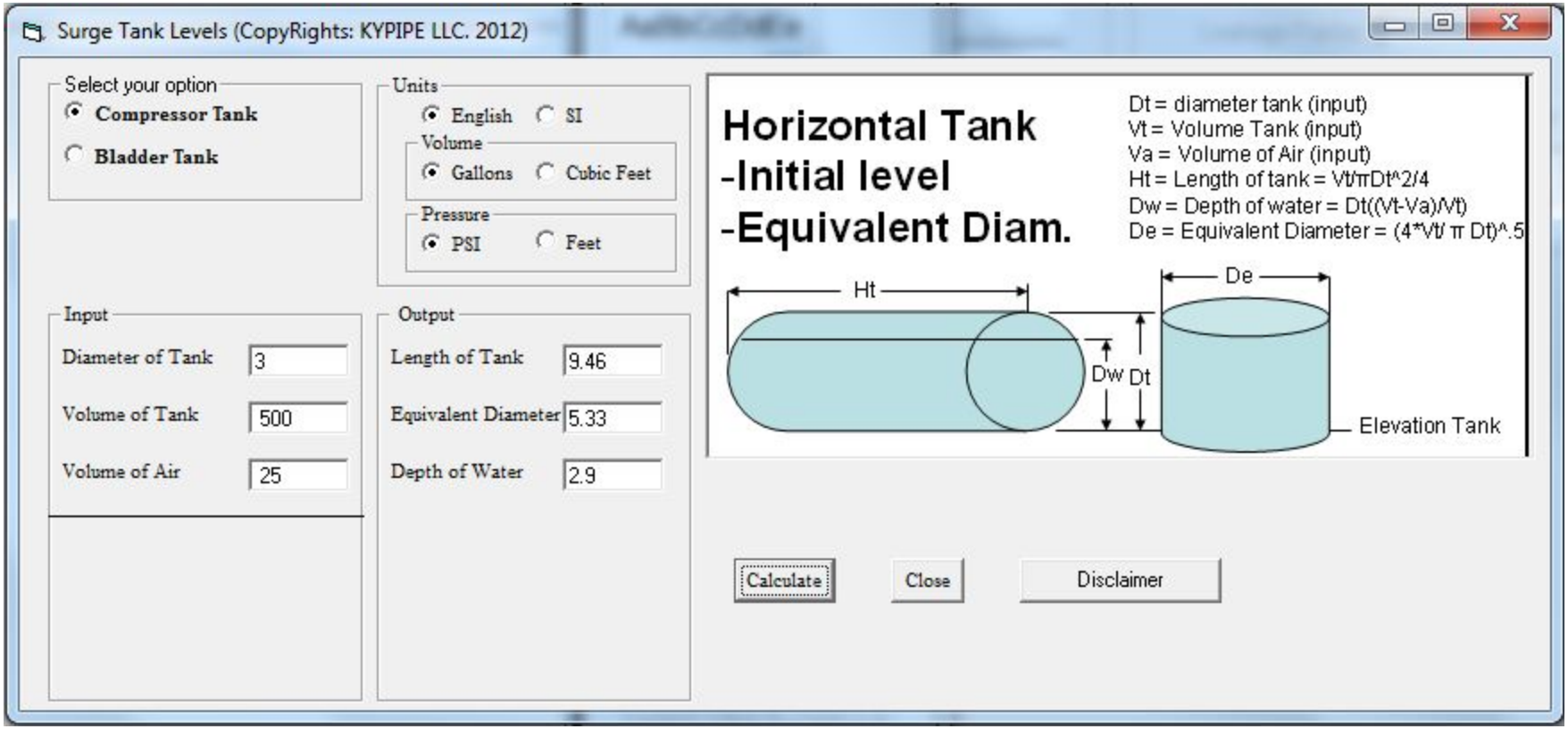
EPS to Surge is a Pipe2014 feature.

If you check the **EPS2Surge** box and then run an eps analysis and then click the **EPS 2 Surge** button it will modify the existing system in this way:

- Remove all change patterns, demand patterns, and global demand factor.
- Make the system not EPS.
- Make all nodes single demand type.
- Remove all control switches, skeletonization, and constraints.
- Turn all closed elements to open.
- Remove all residential meters.
- Set new demands based on the last time simulation of the last EPS analysis.
- Set new tanks and reservoirs levels based on the last EPS.
- Set pump speed ratios and on/off status based on last EPS.

It is not really a Surge file, just a steady state case.

## 1.5.25.6 Equivalent Surge Tank Converter Tool

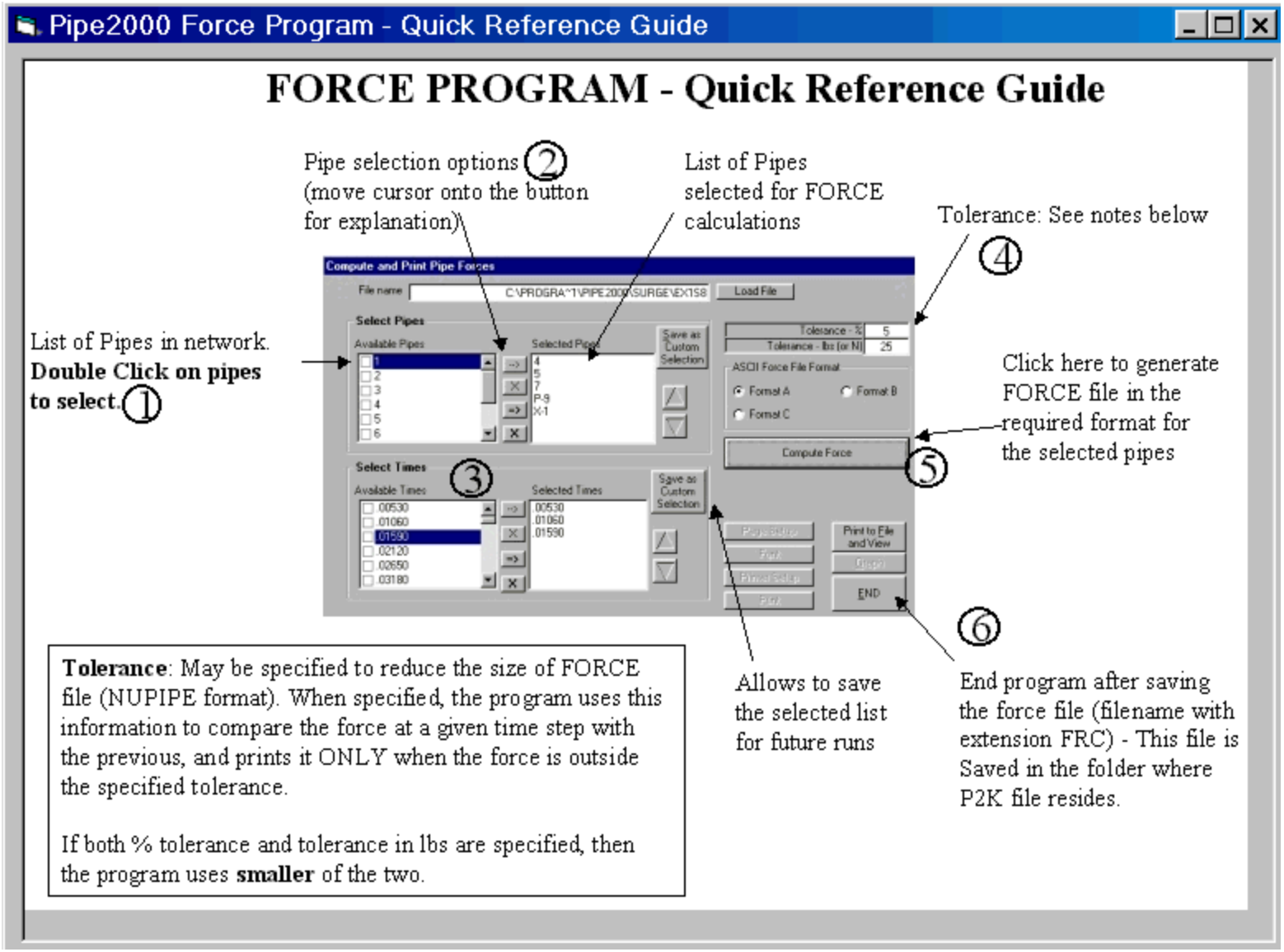



Used to find an equivalent diameter for when a surge tank is horizontal. In Pipe2018, Horiztontal Surge tank designation makes this equivalent calculation unnecessary.

## 1.5.25.7 Force Calculations Tool

**Force Calculations**

This tool calculates unbalanced forces for straight pipe segments. A force file is generated (print to file and view button) which then can be used with third party software to compute pipe stresses and forces at joints. This allows the user to create a file of dynamic forces which can be used as input into various pipe stress programs. The Quick Reference Guide button provides additional details as shown.




There are four Force File Formats available:

Format A - compatible with versions of NuPipe

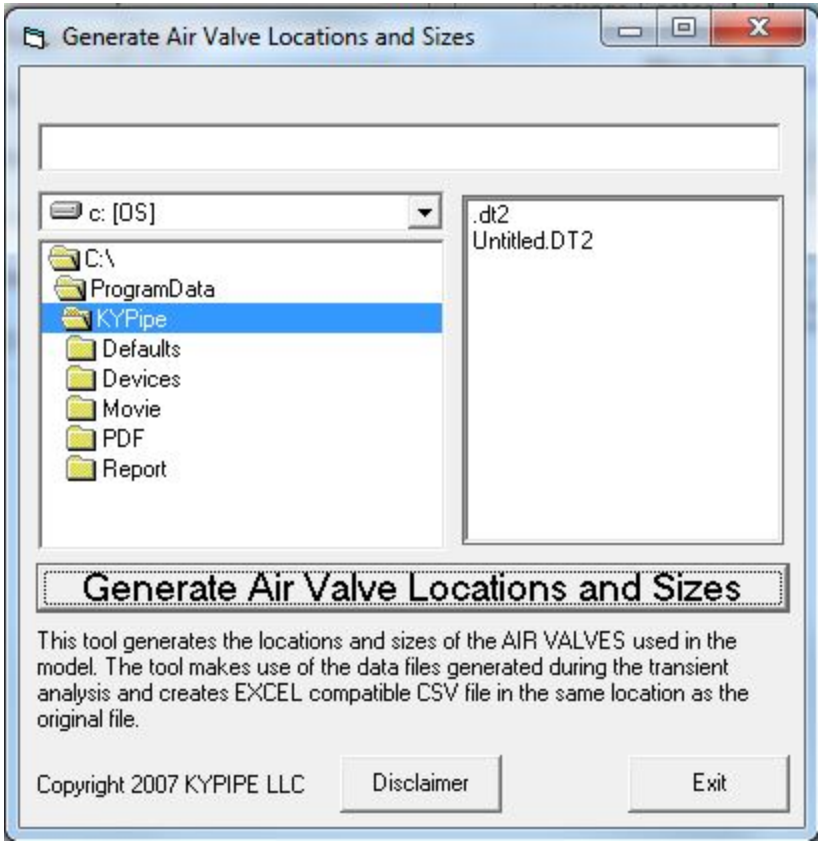
Format B and D - compatible with Caesar or Caesar II

Format C - a generic format

The force equation used in Surge uses the following definitions for specific weight and mass density.

Specific weight (Greek letter gamma) is weight per unit volume and therefore employs N/m^3 as the units. Mass density (Greek letter rho) as the mass per unit volume and hence uses kg/m^3 as the units. Since the force term resulting from pressure is pressure * area (p * A) or head*sp. weight*area (H * gamma * A) or Head* mass density * acce. gravity * Area (H * rho * g * A). Since the g term in the equation converts the mass units into weight units, we should use 1030kg/m^3 (instead of 10101N/m^3).

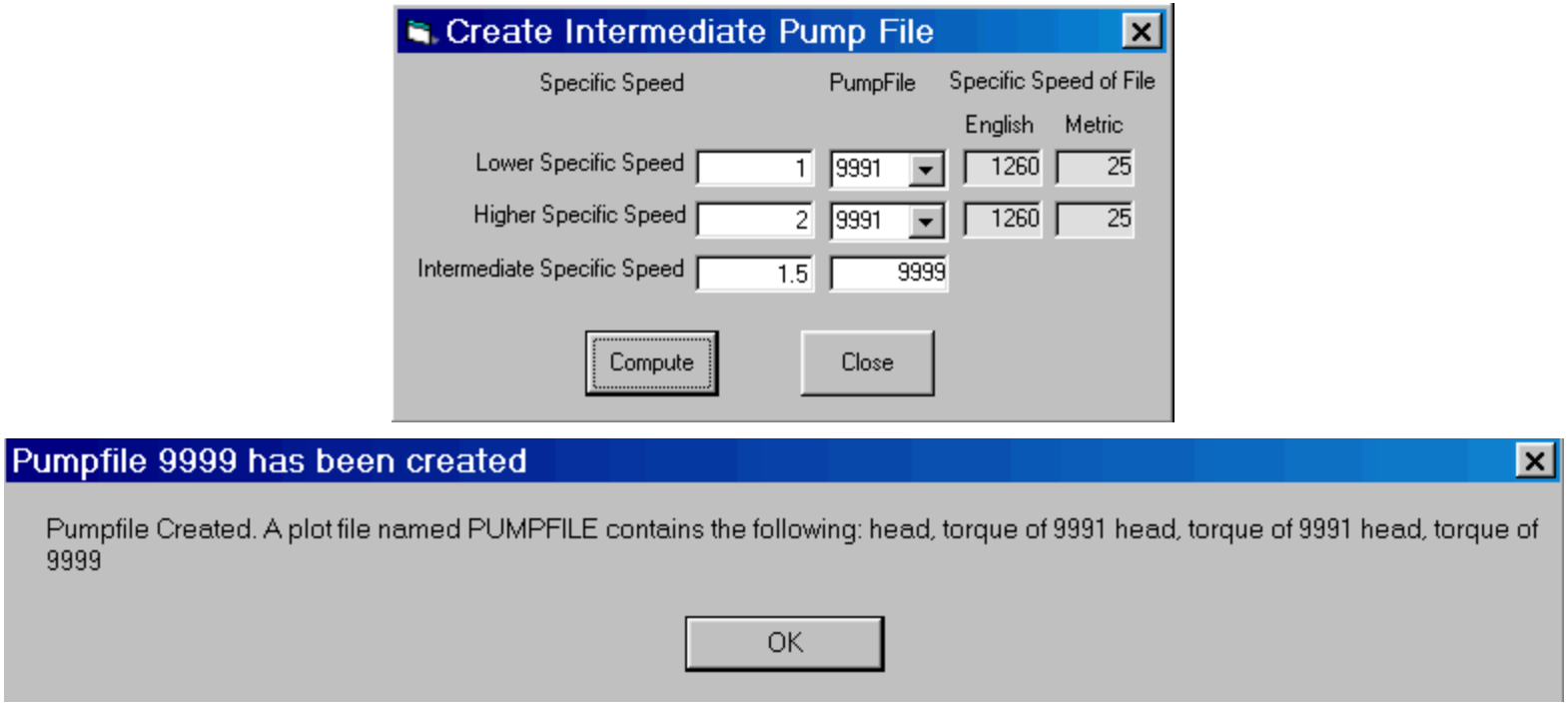
## 1.5.25.8 Generate Air Valvel Locations Tool




## 1.5.25.9 Generate Intermediate Pump File Tool

**Generate Intermediate Pump File**

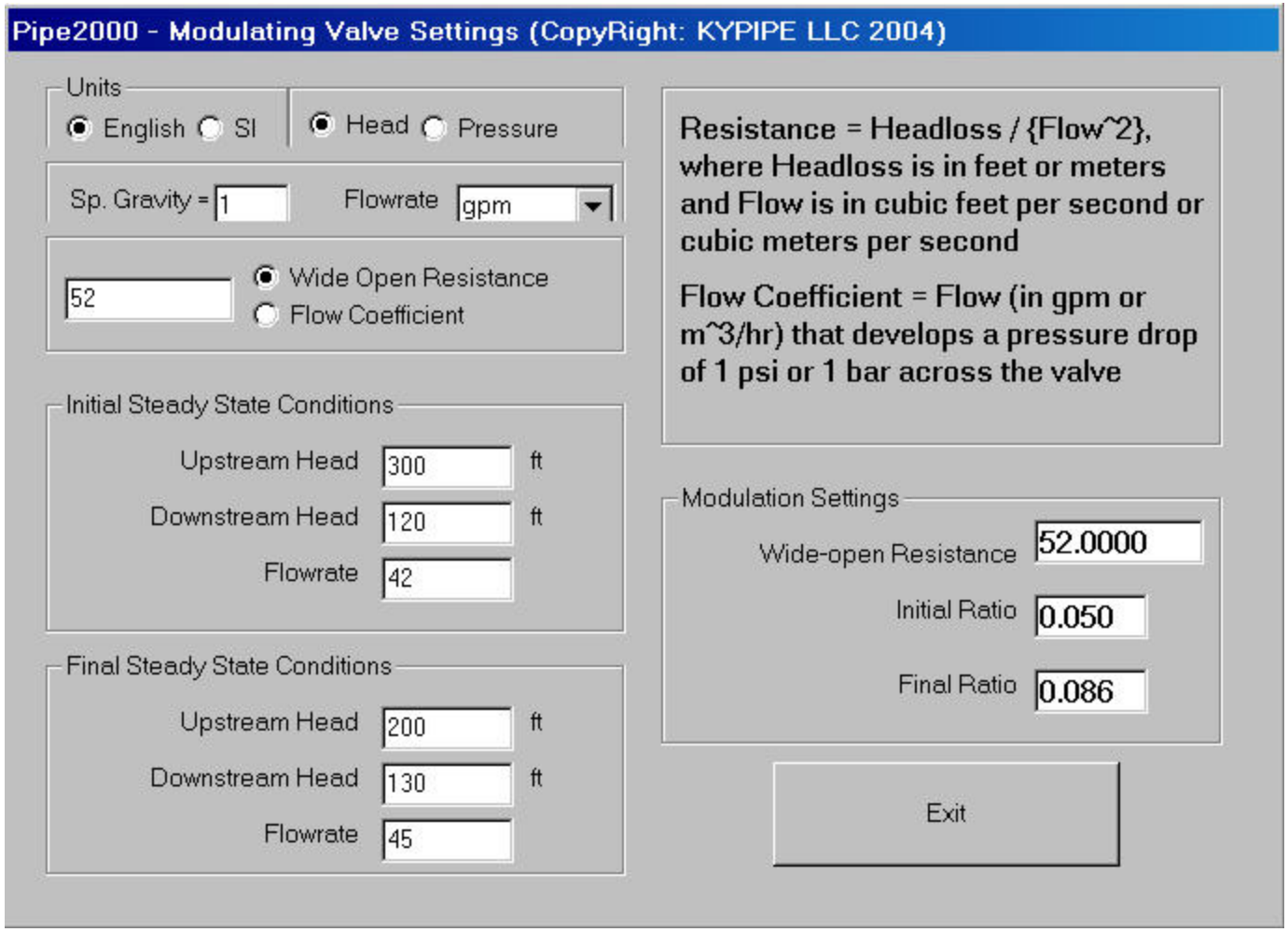
This tool will create a pump file at an intermediate specific speed by interpolating the data from two standard pump files. See also pump file.




## 1.5.25.10 Modulating (Regulating) Valve Tool

### Modulating (Regulating) Valve

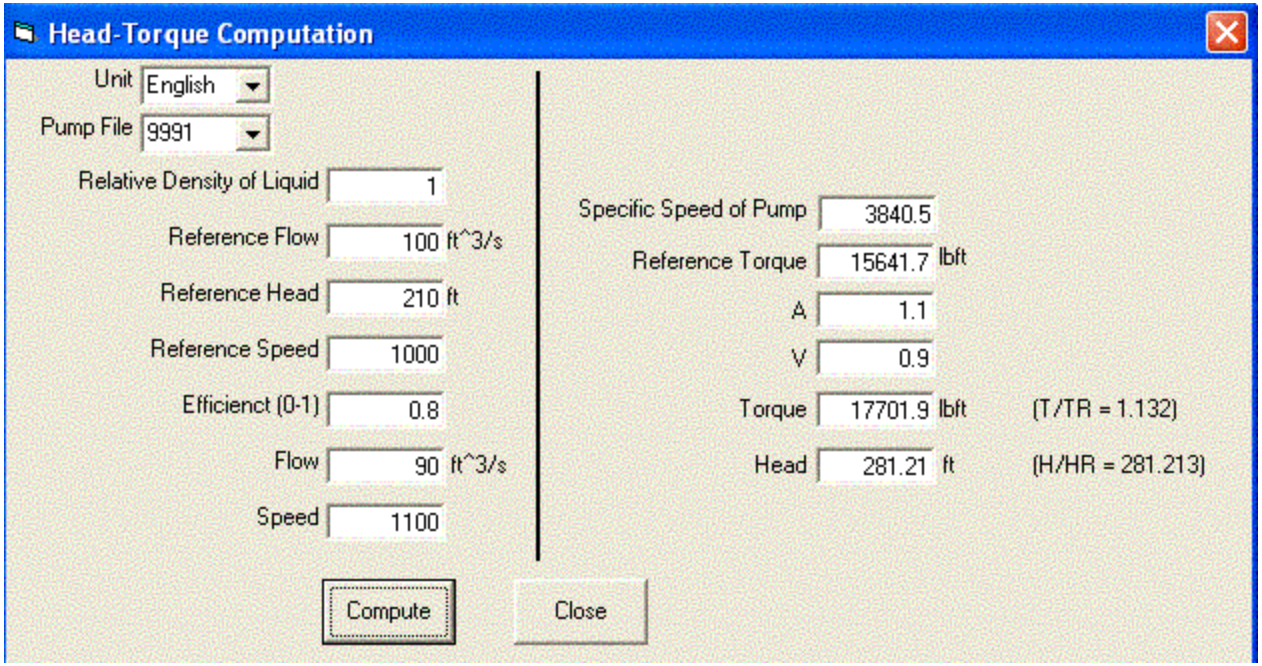
For use with Pipe2000 : Surge Version 2 or later versions to model modulating (regulating) valves. This tool allows the user to calculate the data needed to use an Active Valve as a Modulating Regulating Valve. For normal applications regulating valves are assumed to maintain their initial setting during the transient analysis. Using an Active Valve and the data provided by this tool the valve will modulate from an initial setting to a final setting over a specified time period. To use this tool the user should run the steady state for both the initial and final conditions and input valve operating conditions data as shown on the input screen.




## 1.5.25.11 Pump File Characteristics Tool

**Pump File Characteristics**

Based on specified pump file data, this tool will calculate the torque and head at a specified flow and speed. This tool may be used to generate data points for a Head/Flow curve dictated by the pump file. See also Pump Files for Surge

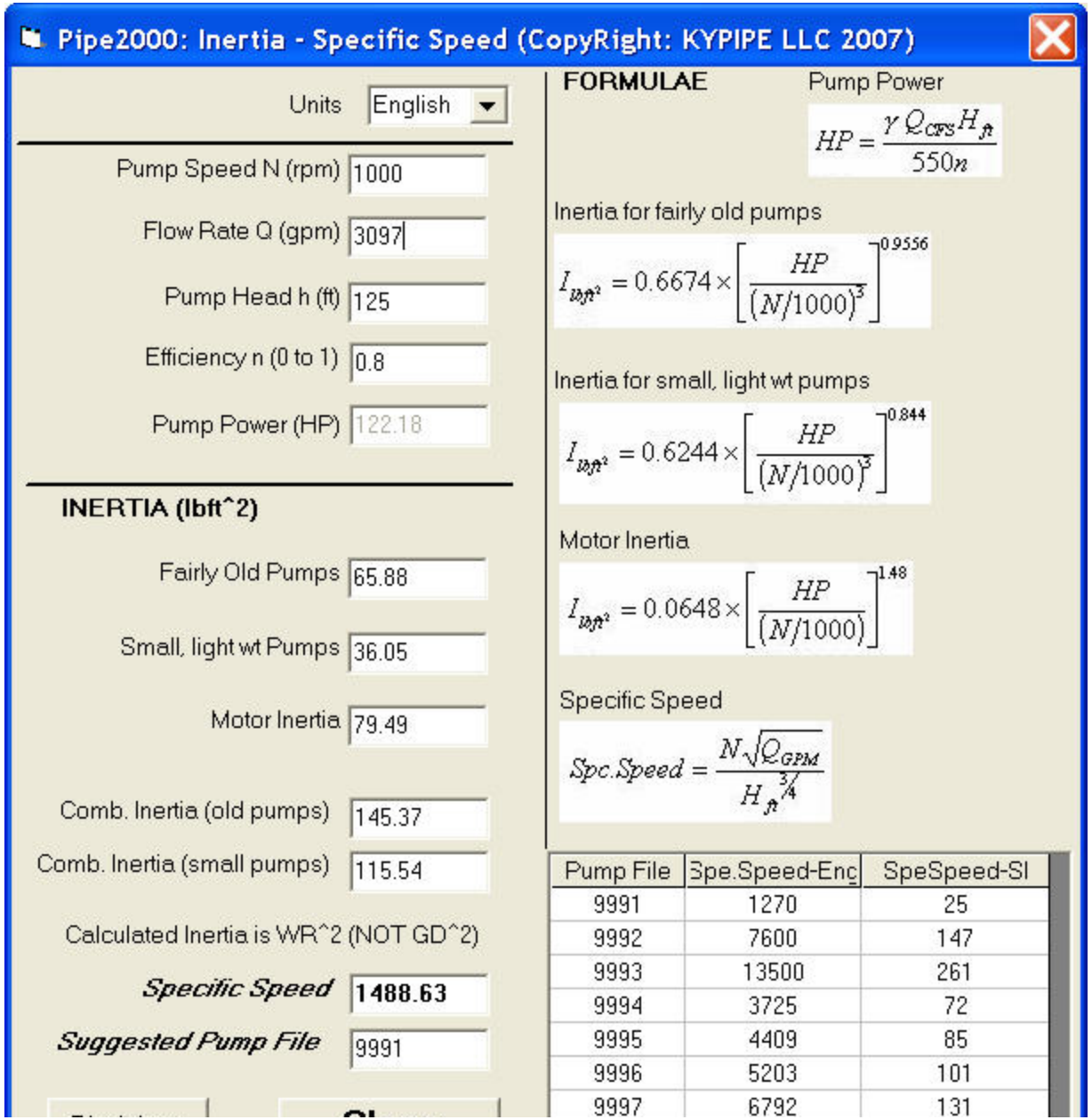



## 1.5.25.12 Select Pump File Calculate Inertia Tool

**Select Pump File/Calculate Inertia**

See also article Role of Pump Inertia Values in Transient Analysis of Pipe Line Systems. This paper summarizes the problems associated with the empirical data and equation that use with the Select Pump File Calculate Inertia tool.

Based on rated pump data this tool selects the appropriate pump file to use for a pump trip analysis. This tool will also calculate the motor and pump inertia using the properties of the pump and curve fits of available data. See also pump file.


Pipe2000: Inertia - Specific Speed (CopyRight: KYPIPE LLC 2007)
Units English
Pump Speed N (rpm) 1000
Flow Rate Q (gpm) 3097
Pump Head h (ft) 125
Efficiency n (0 to 1) 0.8
Pump Power (HP) 122.18
INERTIA (lbft^2)
Fairly Old Pumps 65.88
Small, light wt Pumps 36.05
Motor Inertia 79.49
Comb. Inertia (old pumps) 145.37
Comb. Inertia (small pumps) 115.54
Calculated Inertia is WR^2 (NOT GD^2)
Specific Speed 1488.63
Suggested Pump File 9991
FORMULAE
Pump Power
$HP = \frac{\gamma Q_{CFS} H_{ft}}{550\eta}$
Inertia for fairly old pumps
$I_{lbft^2} = 0.6674 \times \left[\frac{HP}{(N/1000)^3}\right]^{0.9556}$
Inertia for small, light wt pumps
$I_{lbft^2} = 0.6244 \times \left[\frac{HP}{(N/1000)^3}\right]^{0.844}$
Motor Inertia
$I_{lbft^2} = 0.0648 \times \left[\frac{HP}{(N/1000)}\right]^{1.48}$
Specific Speed
$Spc.Speed = \frac{N\sqrt{Q_{GPM}}}{H_{ft}^{3/4}}$
Pump File | Spe.Speed-Enc | SpeSpeed-SI
9991 1270 25
9992 7600 147
9993 13500 261
9994 3725 72
9995 4409 85
9996 5203 101
9997 6792 131

The motor and pump inertia is normally difficult to obtain from the manufacturer and a reasonable estimate can be calculated using **Inertia/Specific Speed** tool.

Inertia selection: The Combined (motor + pump) is often a good estimate but it is very important for the modeler to use judgement and knowledge of existing pump characteristics. Since pumps have gotten lighter, newer pumps have a lower inertia so the user should select older or newer based on the age of the pump. Inertia selection is approximate, just a best guess. Inertia data from the manufacturer is always best, but is often not available.

Old pumps generally refer to those manufactured in th e1960's or earlier. Use of high speed computers starting in mid 60s changed the manufacturers designed pumps. Use of finite element techniques and other fluid dynamics methods helped reduce the size of pumps for delivering the same rated head, flow, and efficiency compared to pumps designed earlier. For the same capacity, pumps have become lighter and saved a lot of manufacturing cost. So pumps manufactured prior to 60s are generally bulky and those manufactured after 60s are small and light weight.

The definition of Inertia is WR^2 where W is the weight (N) of moving part of pumping machine (impeller, shaft, motor etc), R is the radius of gyration of the moving part (m). Therefore the units for WR^2 would be Nm^2. However, some manufacturers provide the WR^2 in terms of kg-m^2 instead of Nm^2 in that case the user should convert the inertia in Kg-m^2 to Nm^2 by multiplying inertia in kg-m^2 with g (9.81m/s^2). Also, some manufacturers provide the inertia in terms of what they call GD^2 instead of WR^2 where GD^2 = 4 * WR^2 (they use diameter instead of radius in estimating the inertia). Our tool uses empirical equations that compute inertia as WR^2 instead of GD^2. Our Surge program requires the inertia as WR^2 and NOT as GD^2.

Inertia in SI units. Manufacturer supplied values are often in GD^2 units . For example, if the manufacturer provided GD^2 value is 4.193 kg-m^2, then the equivalent WR^2 value for the combined pump and motor inertia would be 4.193/4 = 1.048 kg-m^2 or 10.28N-m^2.

## 1.5.25.13 Spike Track Tool

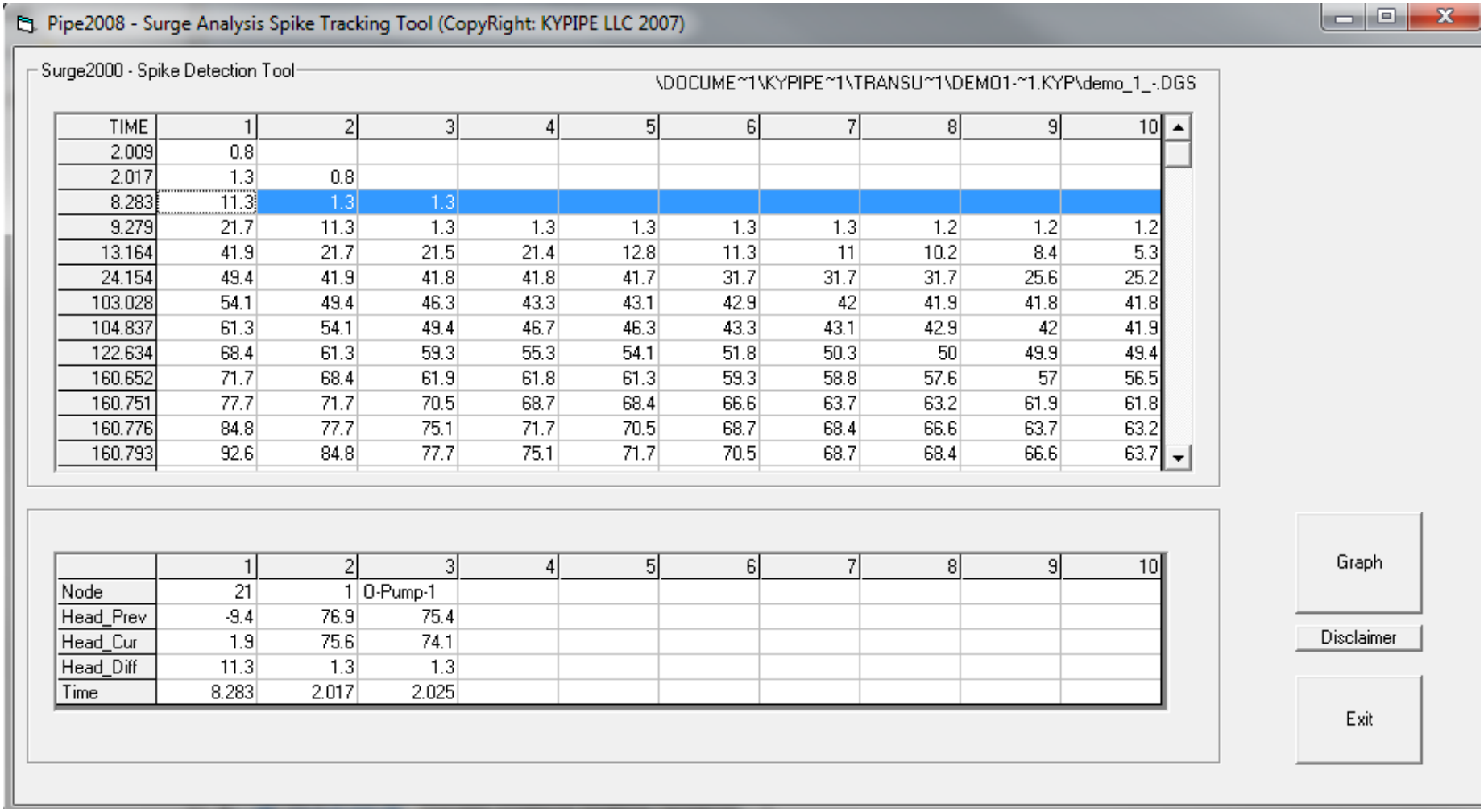
### Spike Track

In Analysis Setup menu, run analysis with "Use Former Analysis Method" option checked.

This tool tracks the origin of pressure spikes following the Surge analysis. This helps identify the events and elements which produce the extreme pressures. The tool is very useful for tracing where the largest changes occur at a glance.

The main objective is to determine where the maximum change in pressure occurs at any given point in time. For example, the first row in the image below shows 2.009s under time column and 0.8 under column 1. That is the maximum difference in pressure of 0.8 occurred at time 2.009 (from the beginning of simulation - no change in data for first two seconds in this model). The maximum change in pressure at time 2.017 (row 2) was 1.3.

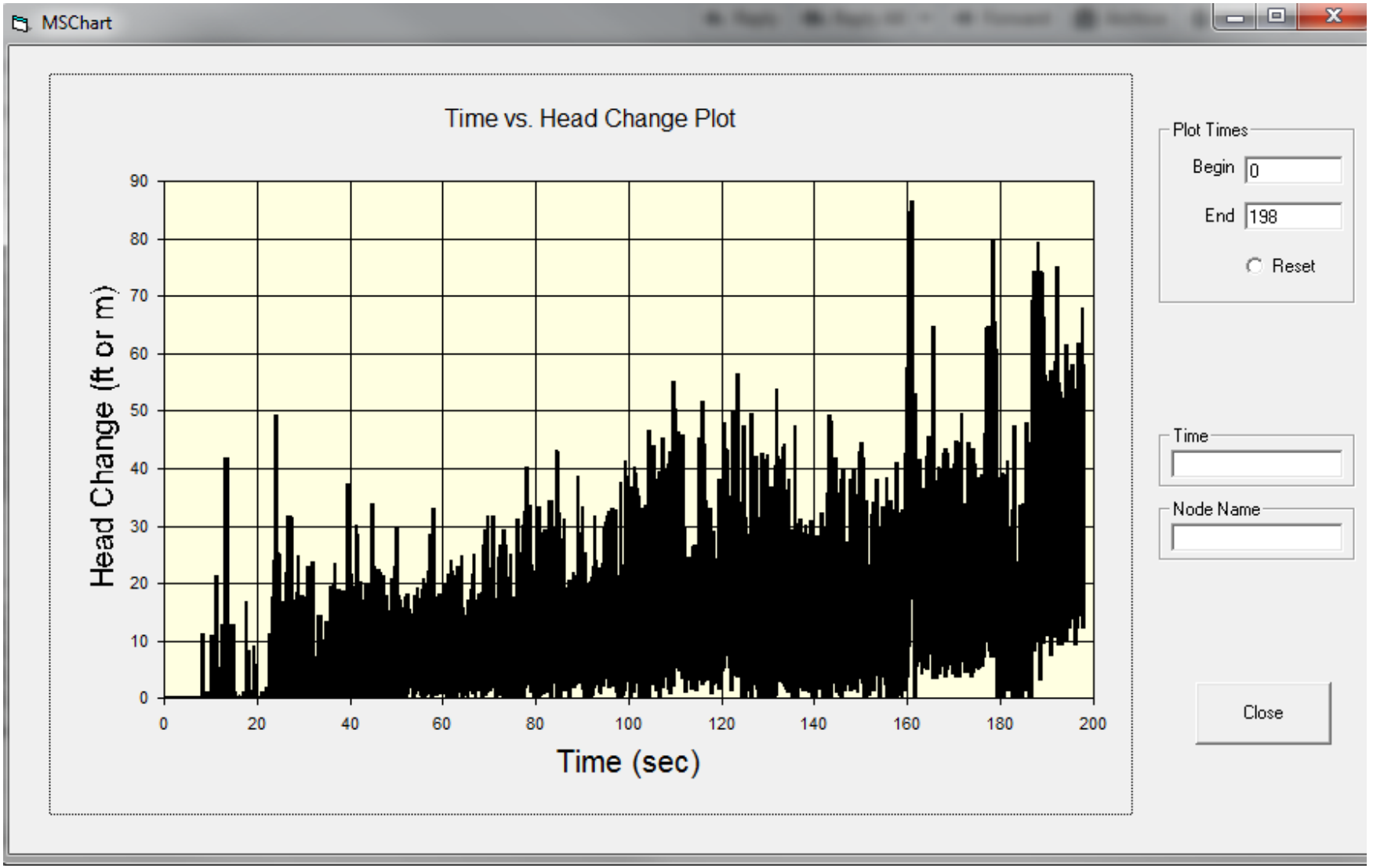
Change in pressure larger than 1.3 occurred at 8.283s and it was 11.3 (value under column 1). All other pressure changes throughout the network were less than 1.3 till 8.282s. When the cursor is moved to any particular row (such as 8.283s row in the image), the corresponding locations where these pressure changes occurred will be listed in the bottom window. Largest change in pressure of 11.3 was at node 21 where the head changed from -9.4 to 1.9. The previous higher change value was at node 1 where the pressure dropped from 76.9 to 75.6 which had occurred at time 2.017. Scrolling down, the 10 largest pressure change values along with their locations are seen.



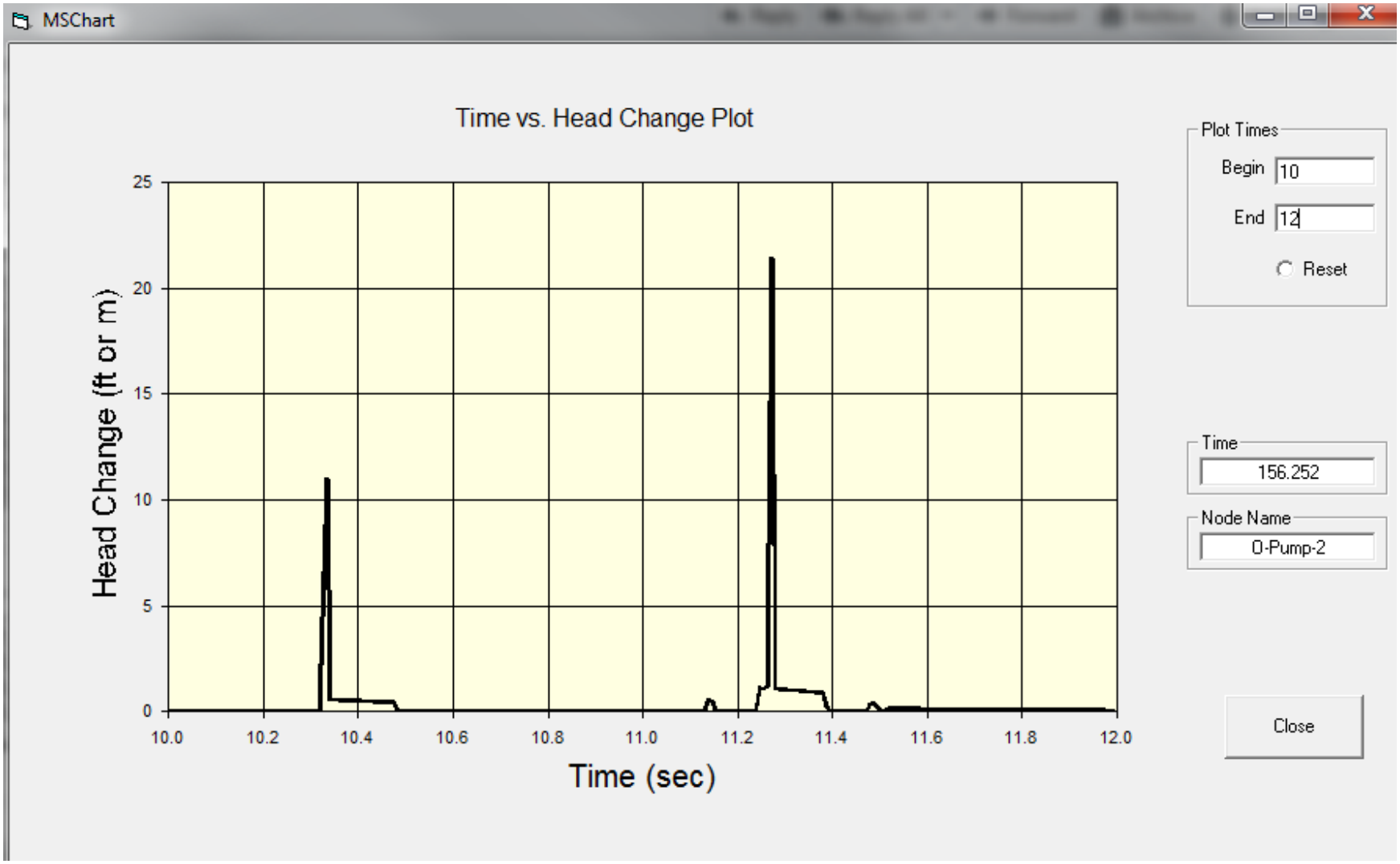

| TIME | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|---|---|---|---|---|---|---|---|---|---|---|
| 2.009 | 0.8 | | | | | | | | | |
| 2.017 | 1.3 | 0.8 | | | | | | | | |
| 8.283 | 11.3 | 1.3 | 1.3 | | | | | | | |
| 9.279 | 21.7 | 11.3 | 1.3 | 1.3 | 1.3 | 1.3 | 1.3 | 1.2 | 1.2 | 1.2 |
| 13.164 | 41.9 | 21.7 | 21.5 | 21.4 | 12.8 | 11.3 | 11 | 10.2 | 8.4 | 5.3 |
| 24.154 | 49.4 | 41.9 | 41.8 | 41.8 | 41.7 | 31.7 | 31.7 | 31.7 | 25.6 | 25.2 |
| 103.028 | 54.1 | 49.4 | 46.3 | 43.3 | 43.1 | 42.9 | 42 | 41.9 | 41.8 | 41.8 |
| 104.837 | 61.3 | 54.1 | 49.4 | 46.7 | 46.3 | 43.3 | 43.1 | 42.9 | 42 | 41.9 |
| 122.634 | 68.4 | 61.3 | 59.3 | 55.3 | 54.1 | 51.8 | 50.3 | 50 | 49.9 | 49.4 |
| 160.652 | 71.7 | 68.4 | 61.9 | 61.8 | 61.3 | 59.3 | 58.8 | 57.6 | 57 | 56.5 |
| 160.751 | 77.7 | 71.7 | 70.5 | 68.7 | 68.4 | 66.6 | 63.7 | 63.2 | 61.9 | 61.8 |
| 160.776 | 84.8 | 77.7 | 75.1 | 71.7 | 70.5 | 68.7 | 68.4 | 66.6 | 63.7 | 63.2 |
| 160.793 | 92.6 | 84.8 | 77.7 | 75.1 | 71.7 | 70.5 | 68.7 | 68.4 | 66.6 | 63.7 |

| | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|---|---|---|---|---|---|---|---|---|---|---|
| Node | 21 | 1 | 0-Pump-1 | | | | | | | |
| Head_Prev | -9.4 | 76.9 | 75.4 | | | | | | | |
| Head_Cur | 1.9 | 75.6 | 74.1 | | | | | | | |
| Head_Diff | 11.3 | 1.3 | 1.3 | | | | | | | |
| Time | 8.283 | 2.017 | 2.025 | | | | | | | |

Click on the graph button and the following screen will appear. The maximum change in pressure at every time step is plotted. Click on any of the lines and the time and node name is displayed.



The begin and end times may be set for closer observation.



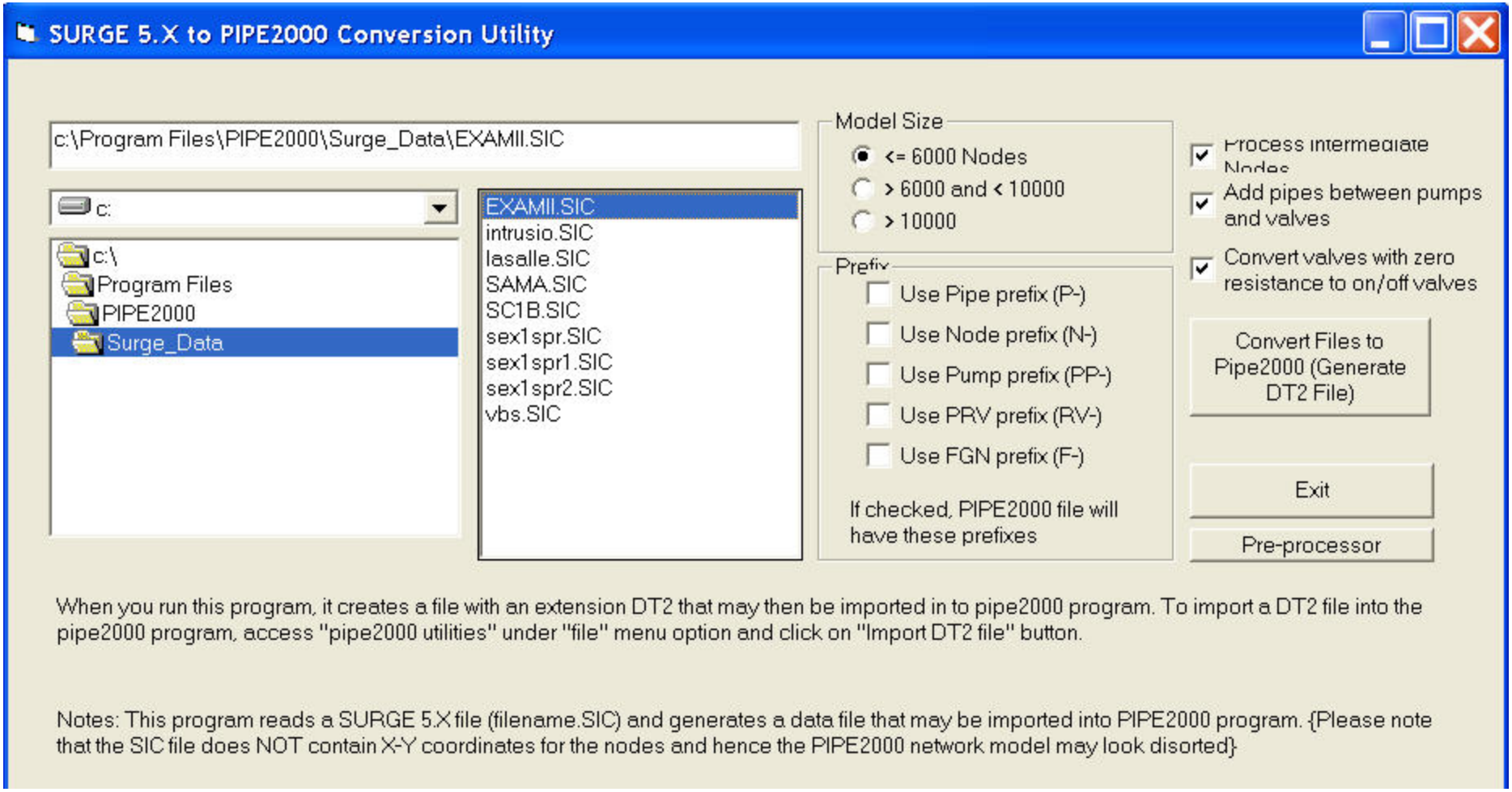
## 1.5.25.14 Surge5 Conversion

Surge5 Conversion

Pipe2000 Utilities / Data Exchange

---

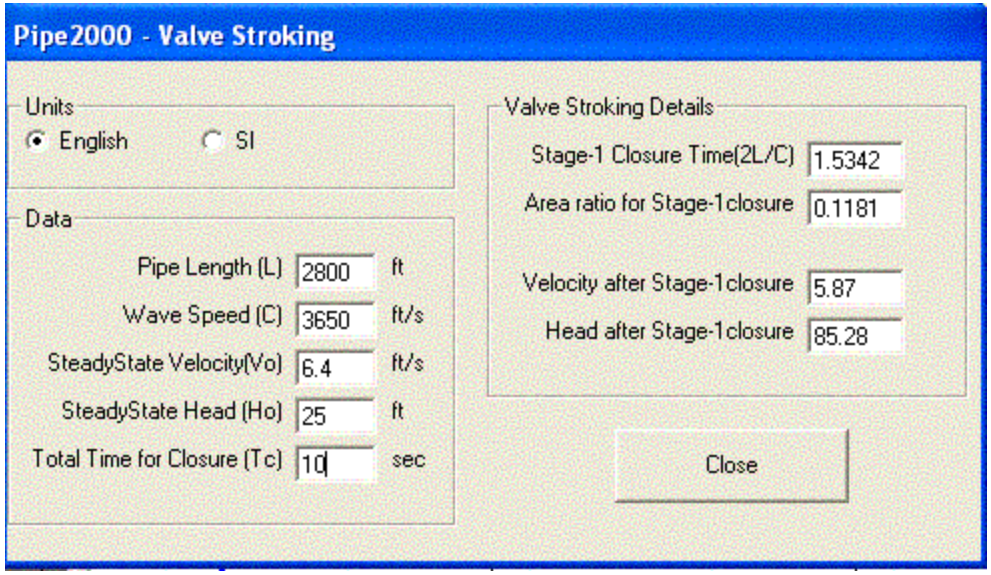
Click File | Pipe2000 Utilities or Import Export (KYnetic) to conduct a Surge 5 conversion. Follow the instructions in the window.




## 1.5.25.15 Valve Stroking Tool

**Valve Stroking**

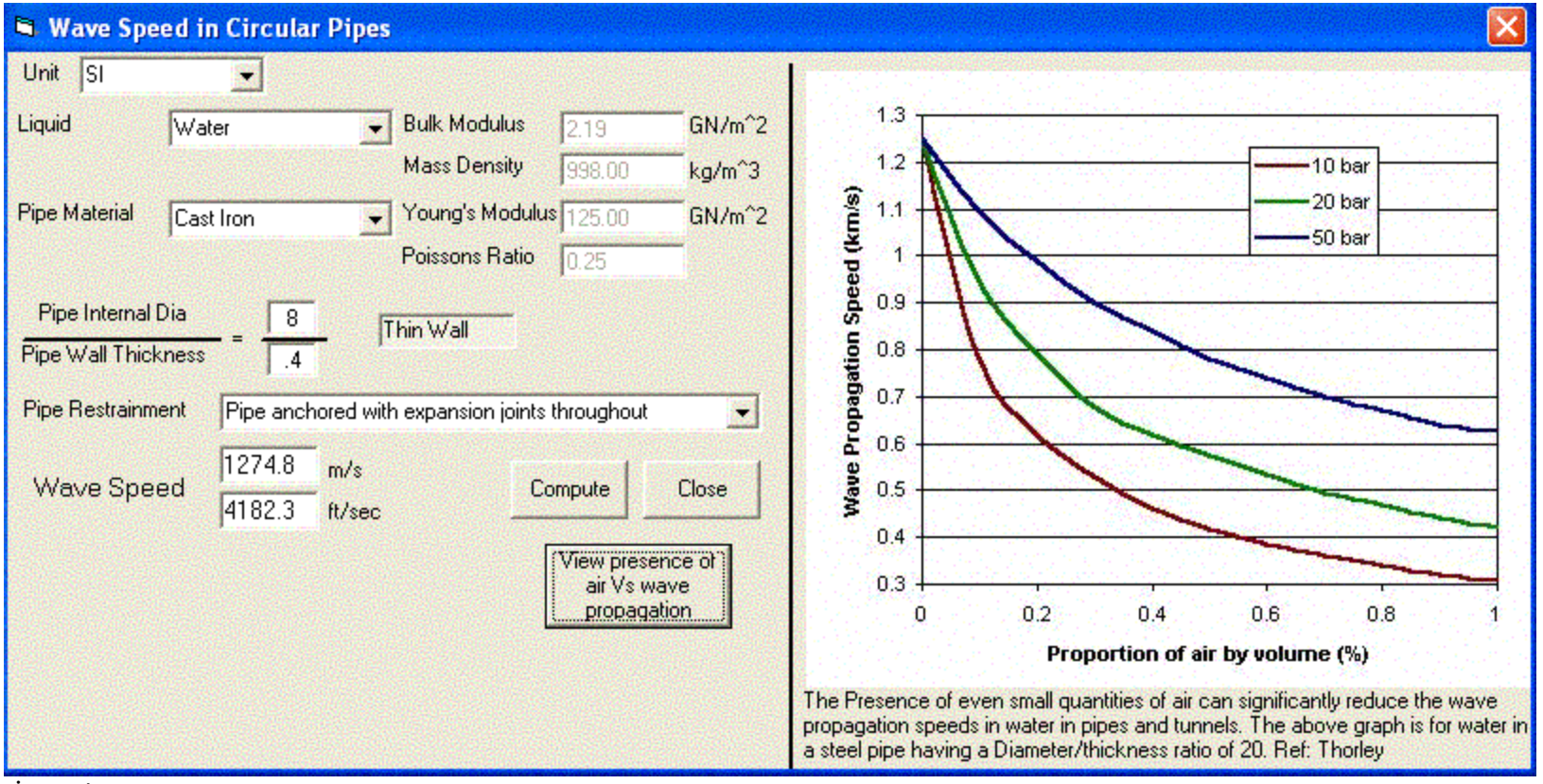
This tool calculates the optimum 2-stage valve closure based on the pipeline characteristics and the valve closure time. See also Active Valves for Surge analysis, how to create a transient.




## 1.5.25.16 Wave Speed Tool

### Wave Speed

This tool calculates the speed of the pressure wave in a pipe based on the characteristics of the pipe, the liquid, and the restraint applied to the pipeline. See also wave speed.




Pipe Restrainment

Pipe is anchored against longitudinal movement - this is used for restrained and buried pipe.

Pipe is anchored at the upstream end only - this is for pipe not buried, restrained only intermittantly

Pipeline has expansion joints throughout - pipeline with expansion joints regardless of restrainment

For composite pipes, Composite YoungsMod = (YoungMod1 * Thickness1 + YoungMod2 * Thickness2) / Total Thickness

## 1.7 GoFlow/KYPipe Fire Sprinkler Analysis

### 1.6.1 NFPA Standards in KYPipe and GoFlow

# NFPA Requirements

Using KYPipe, GoFlow, and Surge

# Hydraulic Calculations

KYPipe, GoFlow, and Surge meet the formula requirements for calculations as described in

NFPA 13 - 22.4.2
NFPA 15 - 8.5
NFPA 16 - 7.4

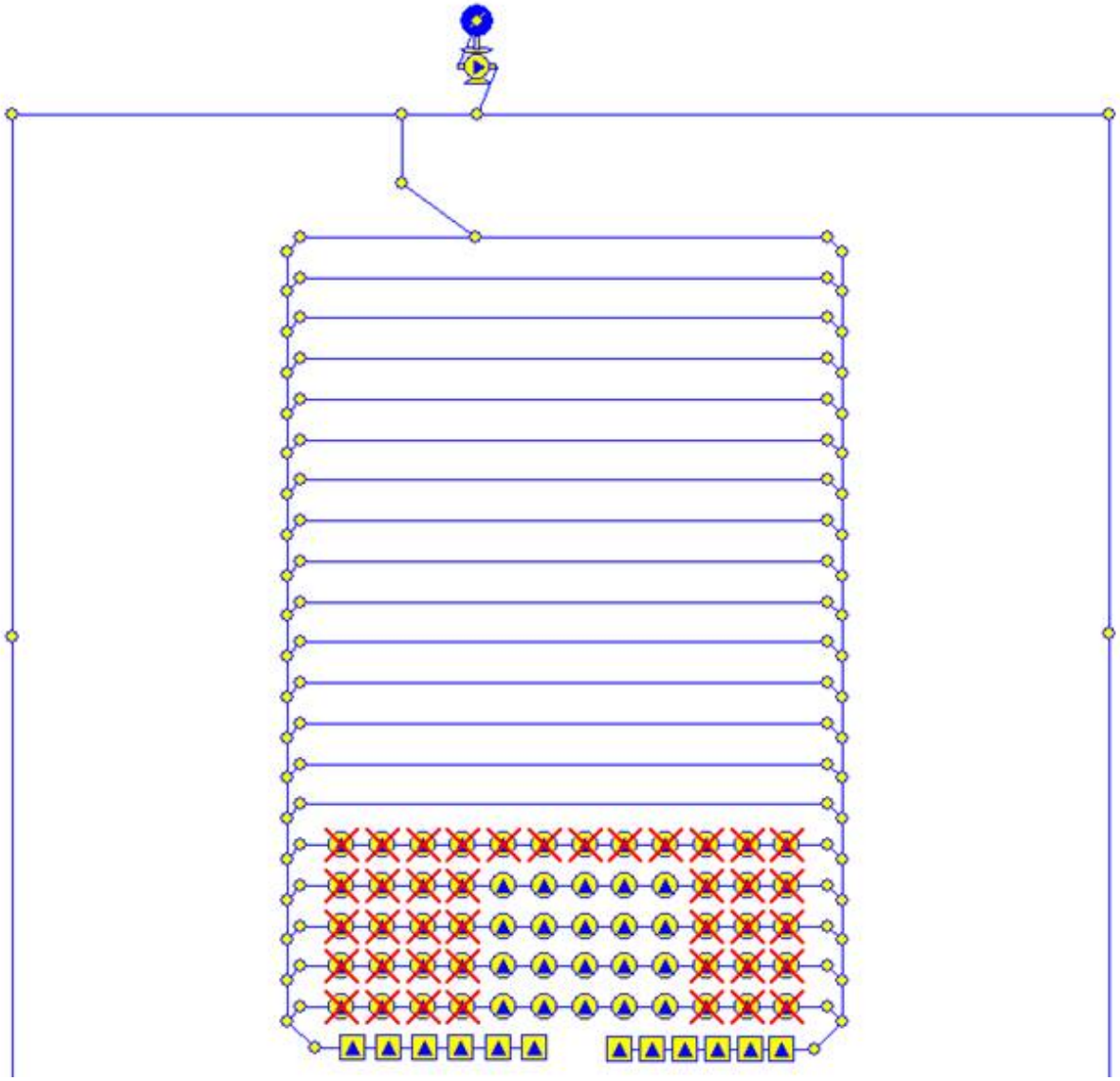
Analysis scenarios and conditions outlined in these standards may be created within KYPipe and GoFlow.

Surge is limited to basic steady state analysis. Surge specializes in unsteady flow analysis. Therefore KYPipe or GoFlow is required to carry out some specific steady analysis scenarios such as Required Capacity, Remote Location, and ESFR.

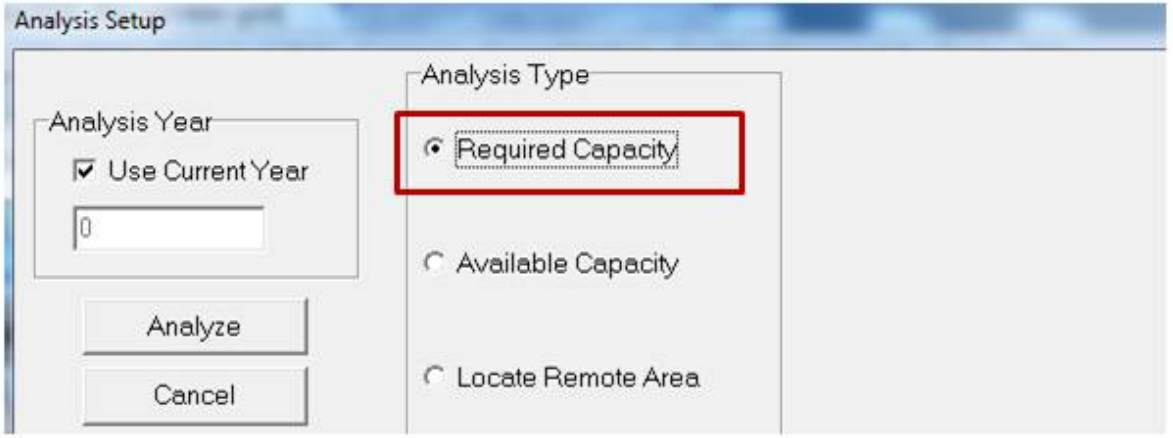
# NPFA 13 and 15 Reporting

with KYPipe and GoFlow

A sprinkler system is developed within KYPipe or GoFlow.



# Run an Analysis



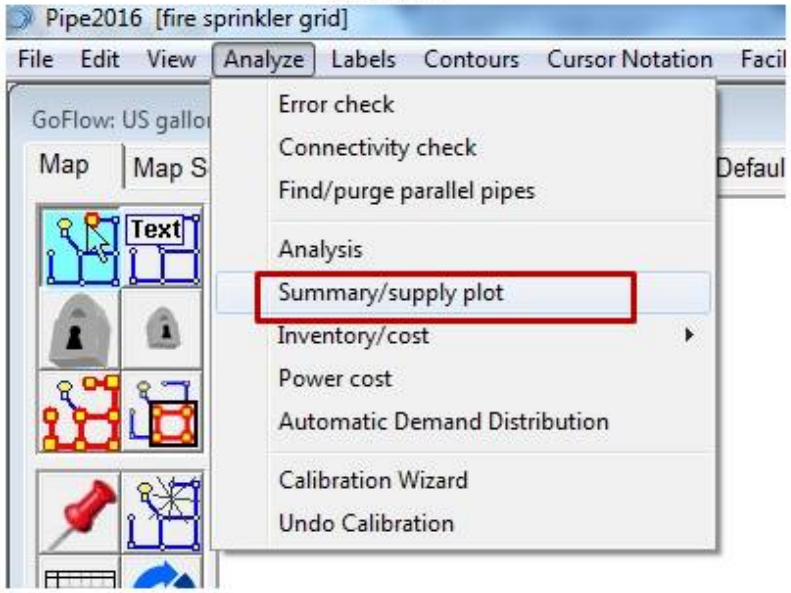

Required Capacity analysis produces information required for NFPA reporting.

# NFPA 13 and 15 Reporting Features

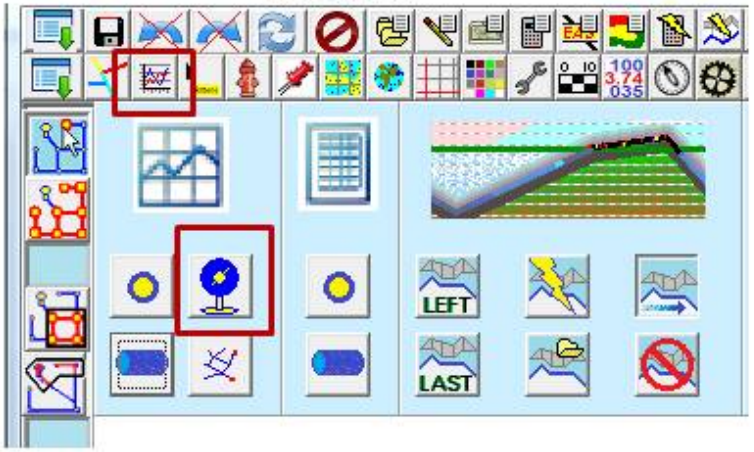
After analysis has been completed, open Summary/Supply Plot feature.

Both program interfaces are shown.

Classic




KYnetic

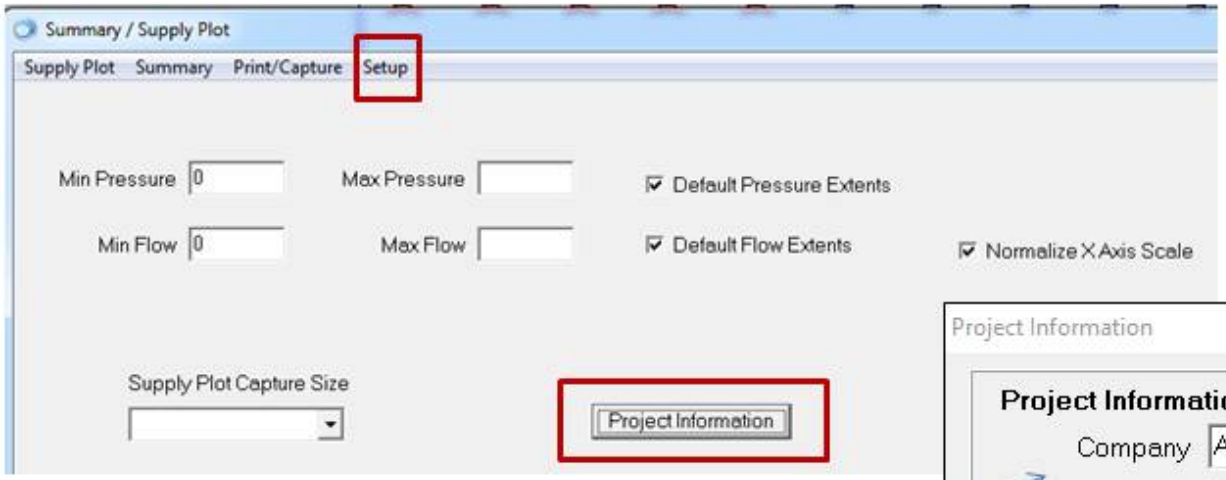


# Summary Sheet Requirements

**NPFA 13 22.3.2 Summary Sheet.** The summary sheet shall contain the following information, where applicable:

(1) Date
(2) Location
(3) Name of owner and occupant
(4) Building number or other identification
(5) Description of hazard (for storage applications, the commodity classification, storage height, and rack configuration shall be included)
(6) Name and address of contractor or designer
(7) Name of approving agency
(8) System design requirements, as follows:
    (a) Design area of water application, ft 2 (m 2 )
    (b) Minimum rate of water application (density), gpm/ft 2 (mm/min). Where sprinklers are listed with minimum water application in gpm (L/min) or pressure in psi (bar), the minimum rate of water application shall be indicated in gpm (L/min) or pressure, psi (bar).
    (c) Area per sprinkler, ft 2 (m 2 )
(9) Total water requirements as calculated, including allowance for inside hose, outside
hydrants, and water curtain and exposure sprinklers
(10) Allowance for in rack sprinklers, gpm (L/min)
(11) Limitations (dimension, flow, and pressure) on extended coverage or other listed
special sprinklers

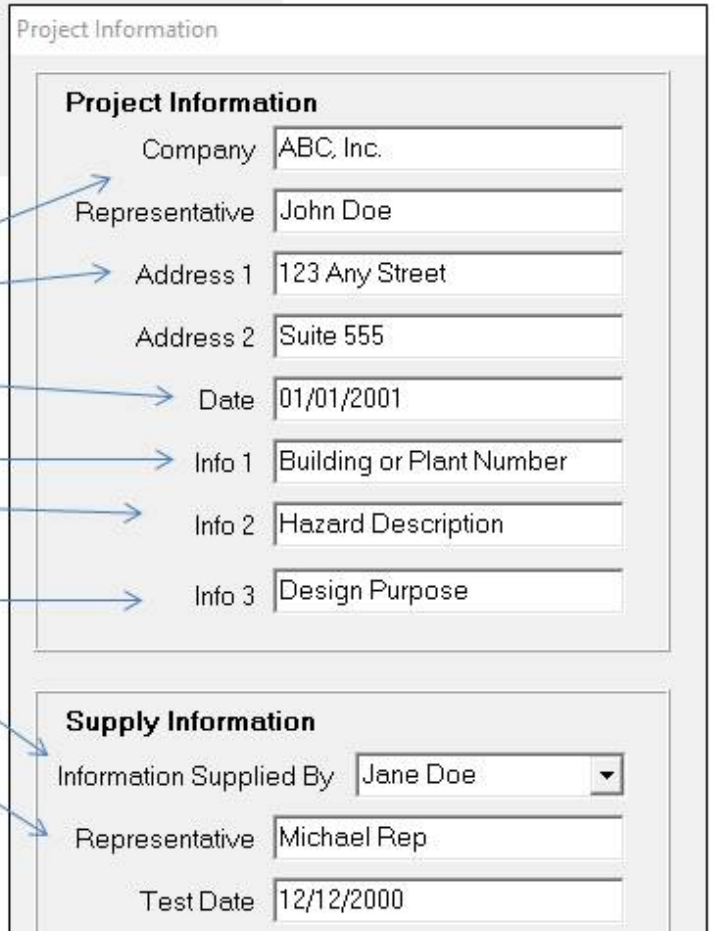
**NPFA 15 8.3.2 Summary Sheet.** The summary sheet *[for sample summary sheet, see Figure B.1(a)]* shall contain all of the following information where applicable:
(1) The date
(2) The location
(3) The name of the owner and occupant
(4) The building or plant number
(5) The description of the hazard
(6) The name and address of the contractor and calculator
(7) The name of the authority having jurisdiction
(8) The design purpose
(9) The rates of the water application (density) and applied areas in gpm/ft2 [L/min)/m2]
(10) The total system water requirements as calculated, including allowance for hose streams
(11) The total designed water demand with number of systems designed to operate simultaneously at a reference
point, preferably the source of supply, including hose streams and other fire protection equipment
(12) Water supply information

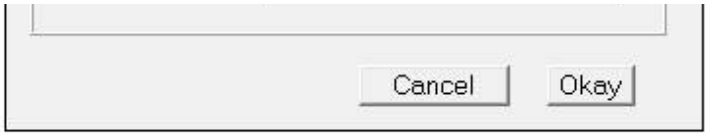



Upon opening Summary/Supply Plot, click Setup.

**NPFA 13 and 15 Summary Sheet.** The summary sheet *[for sample summary sheet, see Figure B.1(a)]* shall contain all of the following information where applicable:

(1) The date
(2) The location
(3) The name of the owner and occupant
(4) The building or plant number
(5) The description of the hazard
(6) The name and address of the contractor and calculator
(7) The name of the authority having jurisdiction
(8) The design purpose




Cancel
Okay

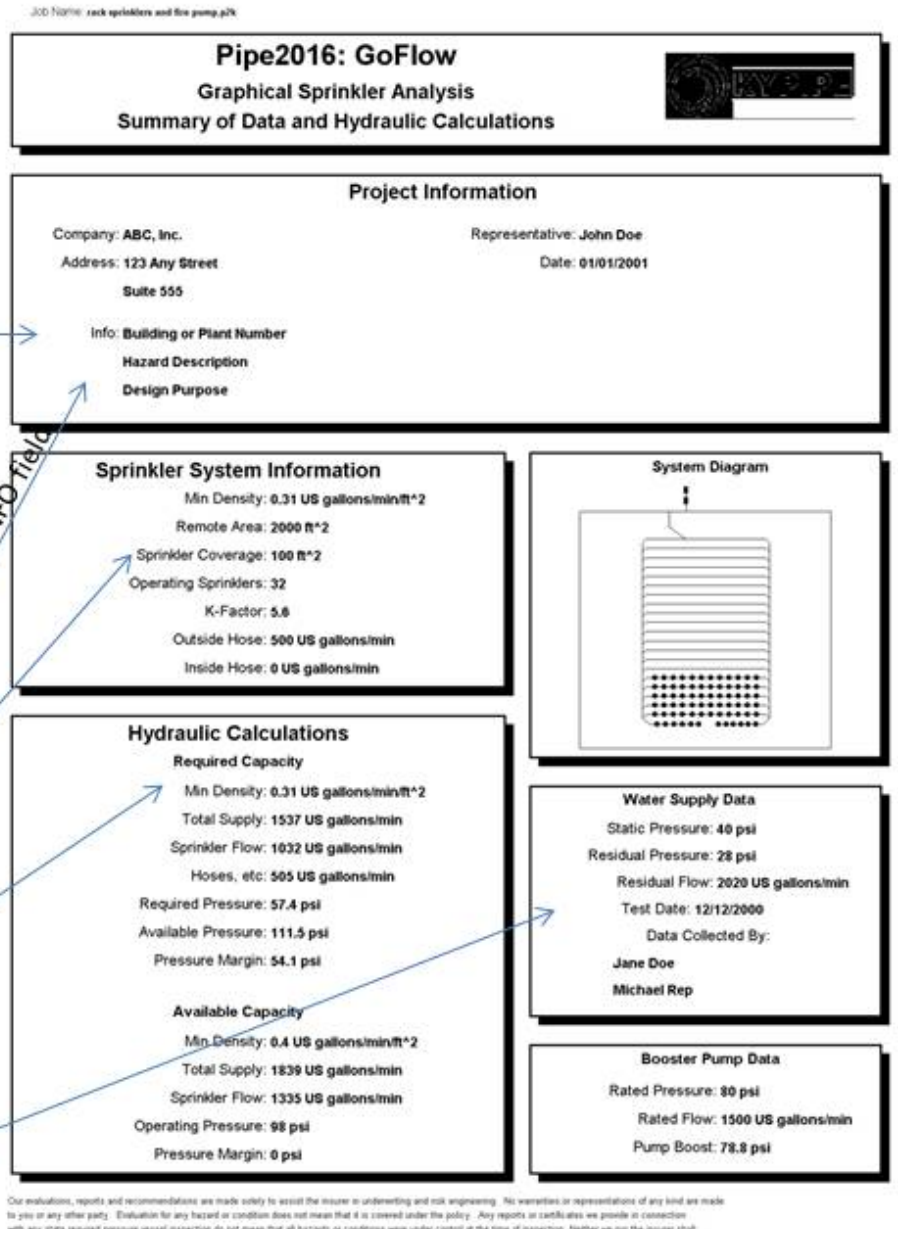
## Summary Sheet - NFPA 13

Some items are automatically placed in the Summary.

**NPFA 13 22.3.2 Summary Sheet.** The summary sheet shall contain the following information, where applicable:

(1) Date
(2) Location
(3) Name of owner and occupant
(4) Building number or other identification
(5) Description of hazard (for storage applications, the commodity classification, storage height, and rack configuration shall be included)
(6) Name and address of contractor or designer
(7) Name of approving agency
(8) System design requirements, as follows:
  (a) Design area of water application, ft 2 (m 2 )
  (b) Minimum rate of water application (density), gpm/ft 2 (mm/min). Where sprinklers are listed with minimum water application in gpm (L/min) or pressure in psi (bar), the minimum rate of water application shall be indicated in gpm (L/min) or pressure, psi (bar).
  (c) Area per sprinkler, ft 2 (m 2 )
(9) Total water requirements as calculated, including allowance for inside hose, outside hydrants, and water curtain and exposure sprinklers

Add to INFO field



(10) Allowance for in rack sprinklers, gpm (L/min)
(11) Limitations (dimension, flow, and pressure) on extended coverage or other listed special sprinklers

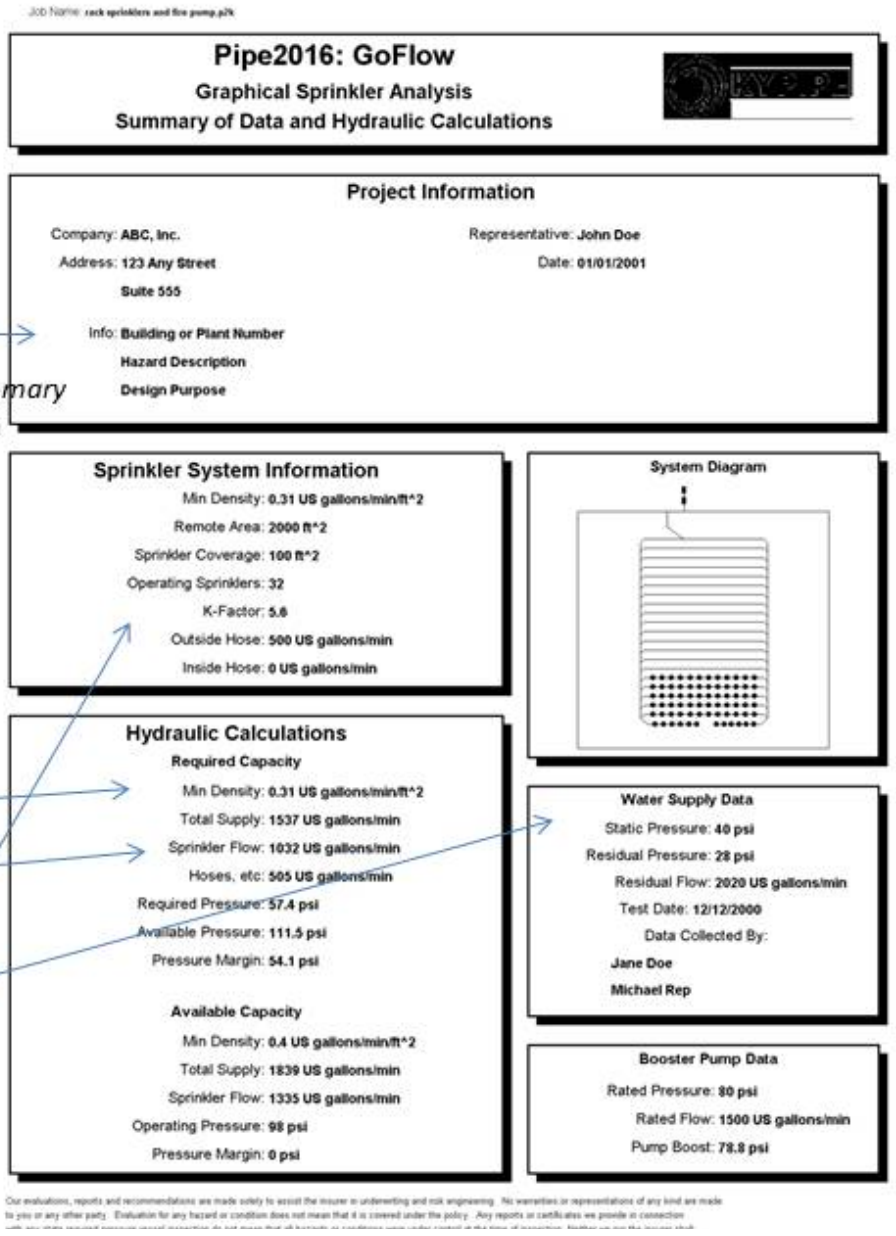
Include in INFO

## Summary Sheet - NPFA 15

Some items are automatically placed in the Summary.

**NPFA 15 8.3.2 Summary Sheet.** The summary sheet *[for sample summary sheet, see Figure B.1(a)]* shall contain all of the following information where applicable:

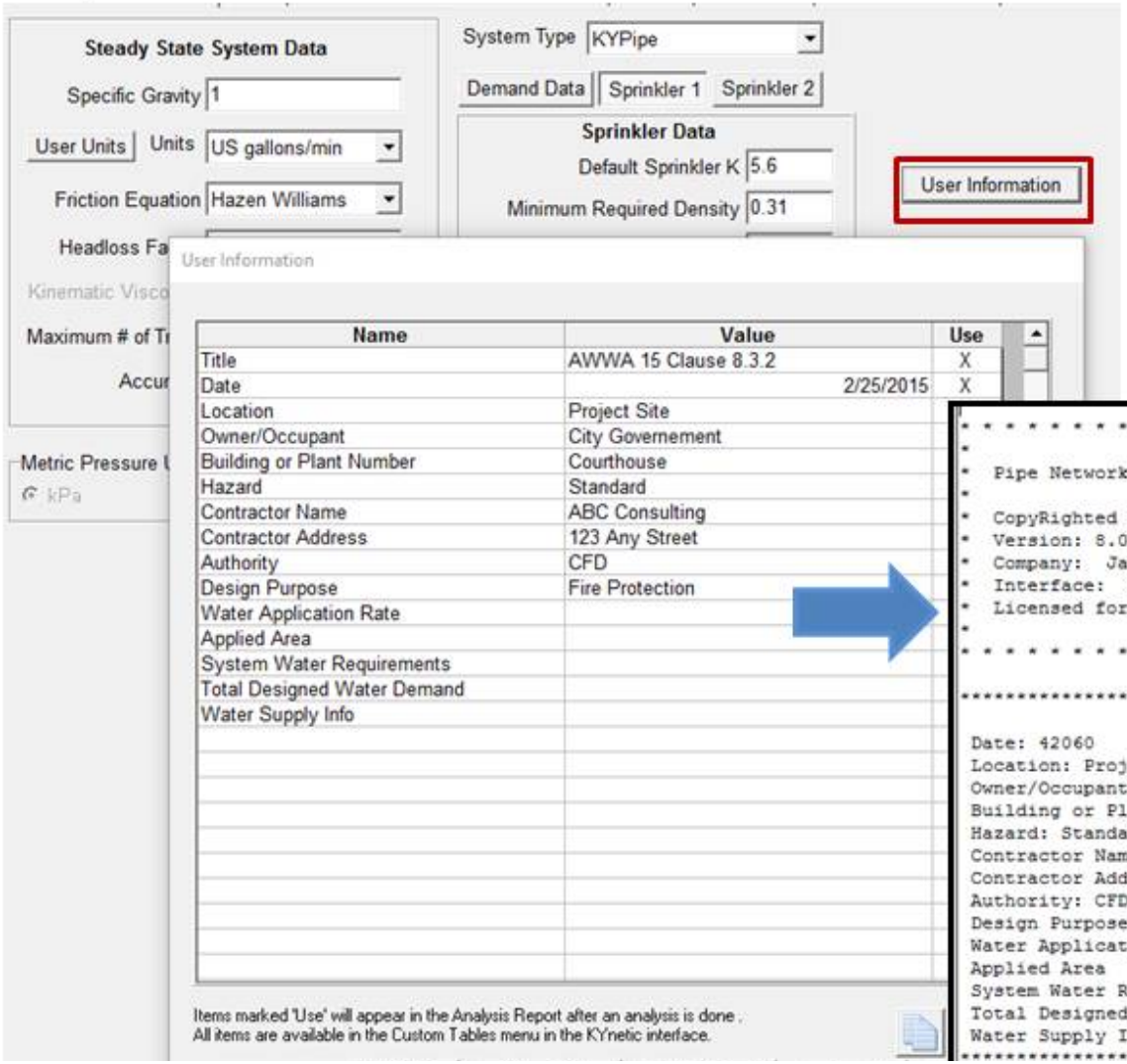
(1) The date
(2) The location
(3) The name of the owner and occupant
(4) The building or plant number
(5) The description of the hazard
(6) The name and address of the contractor and calculator
(7) The name of the authority having jurisdiction
(8) The design purpose
(9) The rates of the water application (density) and applied areas in gpm/ft2 [L/min)/m2]
(10) The total system water requirements as calculated, including allowance for hose streams
(11) The total designed water demand with number of systems designed to operate simultaneously at a reference point, preferably the source of supply, including hose streams and other fire protection equipment
(12) Water supply information

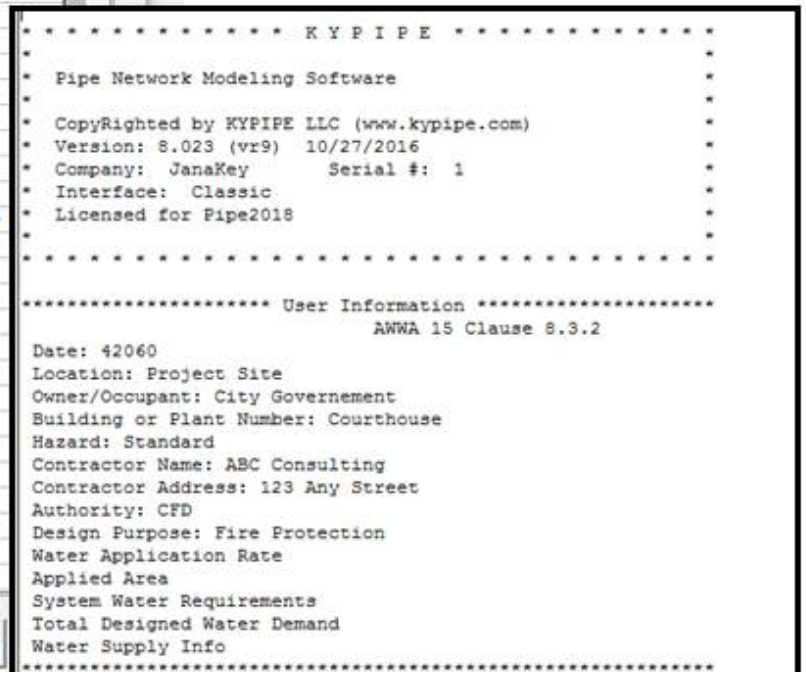


with any state required pressure vessel inspection do not mean that all hazards or conditions were under control at the time of inspection. Neither we nor the insurer shall be liable to you or any other person for the use of any information provided or statements made by any of our employees or agents during the performance of an evaluation.

# NFPA Summary - Additional Information - also Available for **Surge**

## System Data menu

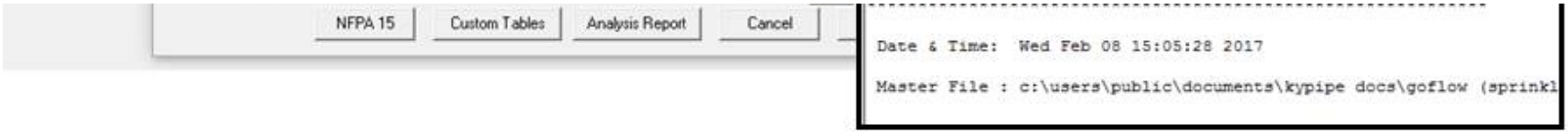


```
* * * * * * * * * * * * K Y P I P E * * * * * * * * * * * *
*                                                          *
*  Pipe Network Modeling Software                          *
*                                                          *
*  CopyRighted by KYPIPE LLC (www.kypipe.com)              *
*  Version: 8.023 (vr9)  10/27/2016                        *
*  Company:  JanaKey       Serial #:  1                    *
*  Interface:  Classic                                     *
*  Licensed for Pipe2018                                   *
*                                                          *
* * * * * * * * * * * * * * * * * * * * * * * * * * * * * *

********** User Information **********
                    AWWA 15 Clause 8.3.2
Date: 42060
Location: Project Site
Owner/Occupant: City Governement
Building or Plant Number: Courthouse
Hazard: Standard
Contractor Name: ABC Consulting
Contractor Address: 123 Any Street
Authority: CFD
Design Purpose: Fire Protection
Water Application Rate
Applied Area
System Water Requirements
Total Designed Water Demand
Water Supply Info
************************************************************
```

Any data may be added at the beginning of the Report.

Blank rows may be used or an NPFA 15 format form is available.


NFPA 15
Custom Tables
Analysis Report
Cancel
Date & Time: Wed Feb 08 15:05:28 2017
Master File : c:\users\public\documents\kypipe docs\goflow (sprinkl

# Detailed Worksheets

**NPFA 13 22.3.3 Detailed Worksheets.** Detailed worksheets or computer printout sheets shall
contain the following information:
(1) Sheet number
(2) Sprinkler description and discharge constant *(K)*
(3) Hydraulic reference points
(4) Flow in gpm (L/min)
(5) Pipe size
(6) Pipe lengths, center to center of fittings
(7) Equivalent pipe lengths for fittings and devices
(8) Friction loss in psi/ft (bar/m) of pipe
(9) Total friction loss between reference points
(10) Inrack sprinkler demand balanced to ceiling demand
(11) Elevation head in psi (bar) between reference points
(12) Required pressure in psi (bar) at each reference point
(13) Velocity pressure and normal pressure if included in calculations
(14) Notes to indicate starting points or reference to other sheets or to clarify data shown
(15)* Diagram to accompany gridded system calculations to indicate flow quantities and directions for lines with sprinklers operating in the remote area
(16) Combined Kfactor calculations for sprinklers on drops, armovers, or sprigs where calculations do not begin at the sprinkler

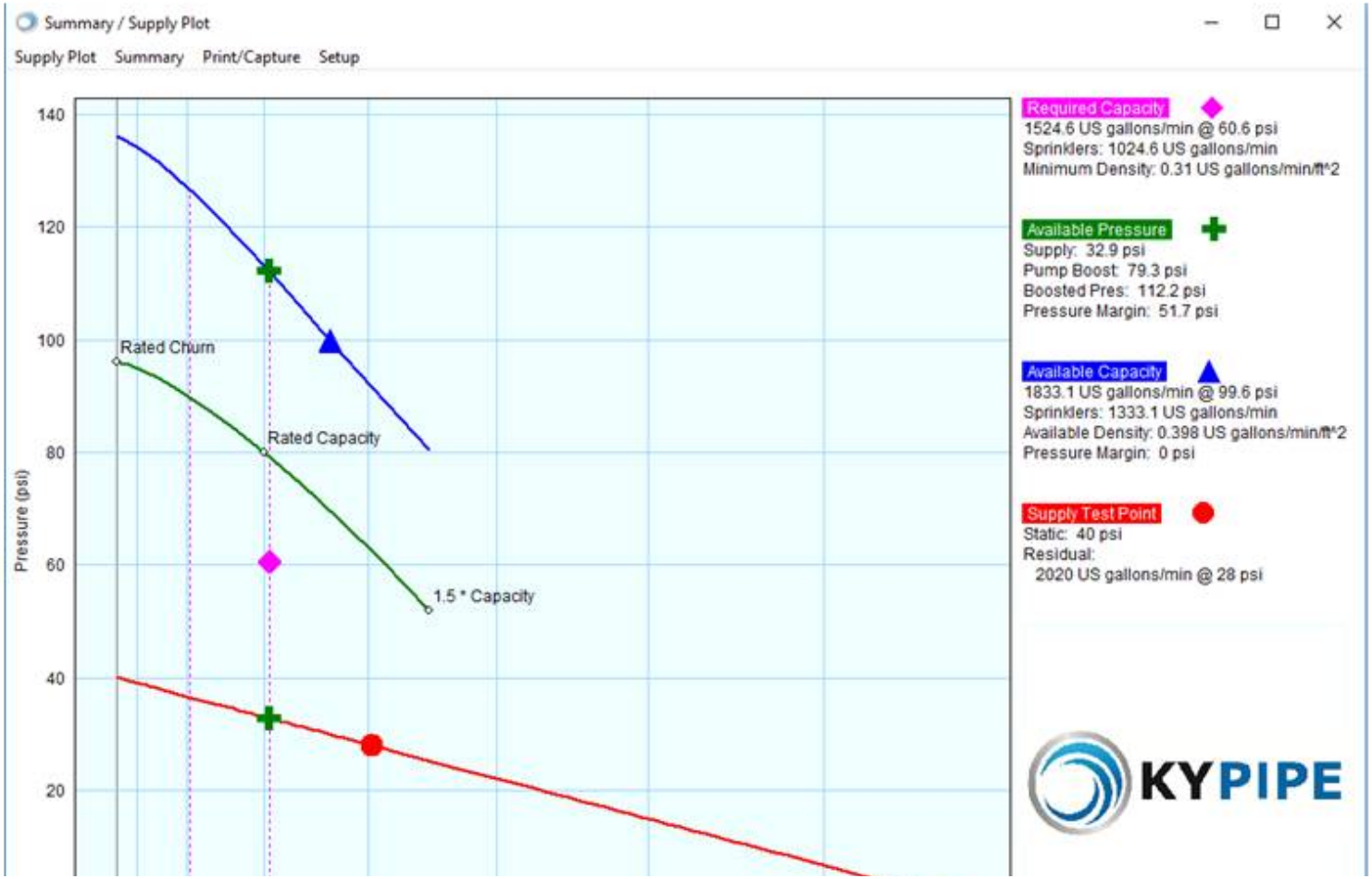
**NPFA 15 8.3.3 Detailed Worksheets.** Detailed worksheets or computer printout sheets *[for sample worksheet, see Figure B.1(b)]* shall contain all of the following information:
(1) Sheet number, date, job number, and identification of calculations covered
(2) Description of discharge constant (K) (or provide the discharge curve or tabulation) for each nozzle type
(3) Hydraulic reference points
(4) Flow in gpm (L/min)
(5) Pipe size in in. (mm)
(6) Pipe lengths, center to center of fittings (or cut lengths) in ft (m)
(7) Equivalent pipe lengths for fittings and devices in ft (m)
(8) Friction loss in psi (bar) between reference points
(9) Total friction loss in psi (bar) between reference points
(10) Elevation head in psi (bar) between reference points
(11) Required pressure in psi (bar) at each reference point
(12) Velocity pressure and normal pressure if included in calculations
(13) Notes to indicate starting points, reference to other sheets, or to clarify data shown
(14) Combined *K*-factor calculations for nozzles on drops, armovers, or sprigs where calculations do not begin at a nozzle
(15) Where extending existing equipment, hydraulic calculations indicating the previous design, volume, and pressure at points of connection, and adequate additional calculations to indicate effect on existing systems

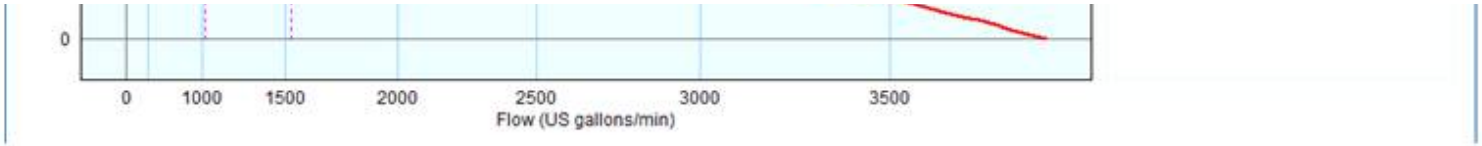
While some of the information will be provided by the consultant,

many of these worksheets can be easily produced within the standard presentation capabilities of KYPipe and GoFlow.

## Graph Sheet -> Supply Plot

**NPFA 13 22.3.4 Graph Sheet.** A graphic representation of the complete hydraulic calculation shall be plotted on semi-exponential graph paper (*Q* 1.85 ) and shall include the following:
(1) Water supply curve
(2) Sprinkler system demand
(3) Hose allowance (where applicable)
(4) In-rack sprinkler demand (where applicable)




0
0
1000
1500
2000
2500
3000
3500
Flow (US gallons/min)

**NFPFA 13 22.3.5 Computer Generated Hydraulic Reports.**

**22.3.5.1* General.** Hydraulic calculations shall be prepared on form sheets that include a summary sheet, a graph sheet, a water supply analysis, a node analysis and detailed worksheets. The data shall be presented in the order shown in Figure 22.3.5.1(a), Figure 22.3.5.1(b), Figure 22.3.5.1(c), and Figure 22.3.5.1(d). The requirement of 22.3.5 shall be effective starting January 1, 2008.

**HYDRAULIC CALCULATIONS**
**for**

Project name: _____
Location: _____
Drawing no.: _____ Date: _____

Design
- Remote area number: _____
- Remote area location: _____
- Occupancy classification: _____
- Density: _____ gpm/ft²
- Area of application: _____ ft²
- Coverage per sprinkler: _____ ft²
- Type of sprinklers calculated: _____
- No. of sprinklers calculated: _____
- In-rack demand: _____
- Hose streams: _____
- Total water required (including hose streams): _____ gpm @ _____
- Type of system: _____
- Volume of dry or preaction system: _____ gal

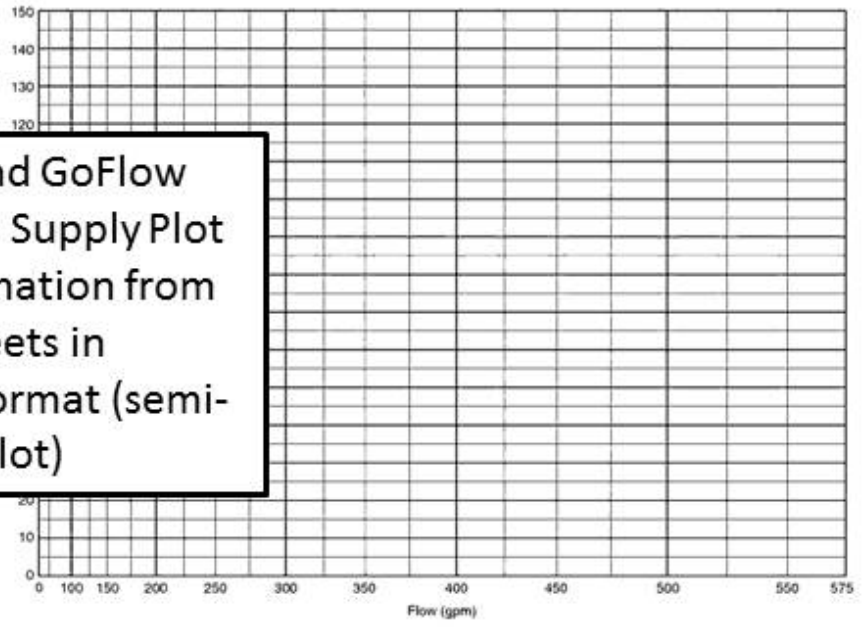
Water supply information
- Date: _____
- Location: _____
- Source: _____

Name of contractor: _____
Address: _____
Phone number: _____
Name of designer: _____
Authority having jurisdiction: _____

Notes: (Include peaking information or gridded systems here.) _____

**FLOW TEST SUMMARY SHEET**
$N^{1.85}$

Contract name and number: _____




Notes: _____

The KYPipe and GoFlow Summary and Supply Plot contain information from these two sheets in appropriate format (semi-exponential plot)

FIGURE 22.3.5.1(a) Summary Sheet.

© 2006 National Fire Protection Association NFPA 13 (p. 1 of 1)

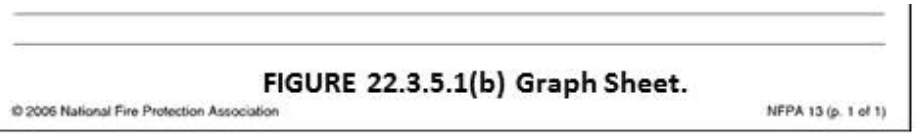


FIGURE 22.3.5.1(b) Graph Sheet.

© 2006 National Fire Protection Association NFPA 13 (p. 1 of 1)

**SUPPLY ANALYSIS**

| Node at Source | Static Pressure | Residual Pressure | Flow | Available Pressure | Total Demand | Required Pressure |
|---|---|---|---|---|---|---|
| Data | Data | Data | Data | Data | Data | Data |

**NODE ANALYSIS**

| Node Tag | Elevation | Node Type | Pressure at Node | Discharge at Node | Notes |
|---|---|---|---|---|---|
| Data | Data | Data | Data | Data | Data |
| Data | Data | Data | Data | Data | Data |
| Data | Data | Data | Data | Data | Data |
| Data | Data | Data | Data | Data | Data |
| Data | Data | Data | Data | Data | Data |
| Data | Data | Data | Data | Data | Data |
| Data | Data | Data | Data | Data | Data |
| Data | Data | Data | Data | Data | Data |
| Data | Data | Data | Data | Data | Data |
| Data | Data | Data | Data | Data | Data |
| Data | Data | Data | Data | Data | Data |
| Data | Data | Data | Data | Data | Data |
| Data | Data | Data | Data | Data | Data |
| Data | Data | Data | Data | Data | Data |

© 2006 National Fire Protection Association NFPA 13 (p. 1 of 1)

**FIGURE 22.3.5.1(c) Supply and Node Analyst Sheet.**

## GoFlow report table

The Following Ceiling Sprinklers are Operating:

| Name | Sprink K | Elevation (ft) | Flow (gpm ) | Pressure (psi) |
|---|---|---|---|---|
| 8 | ******** | 19.50 | 31.00 | 30.64 |
| 9 | 5.60 | 19.50 | 30.90 | 30.41 |
| 10 | 5.60 | 19.50 | 30.89 | 30.38 |
| 11 | 5.60 | 19.50 | 30.91 | 30.43 |
| 12 | 5.60 | 19.50 | 31.06 | 30.73 |
| 26 | 5.60 | 19.50 | 33.00 | 34.68 |
| 27 | 5.60 | 19.50 | 33.00 | 34.67 |
| 28 | 5.60 | 19.50 | 33.00 | 34.67 |
| 29 | 5.60 | 19.50 | 33.00 | 34.67 |
| 30 | 5.60 | 19.50 | 33.00 | 34.68 |
| 42 | 5.60 | 19.50 | 33.18 | 35.06 |
| 43 | 5.60 | 19.50 | 33.18 | 35.05 |
| 44 | 5.60 | 19.50 | 33.17 | 35.05 |
| 45 | 5.60 | 19.50 | 33.18 | 35.05 |
| 46 | 5.60 | 19.50 | 33.18 | 35.06 |
| 58 | 5.60 | 19.50 | 33.44 | 35.61 |
| 59 | 5.60 | 19.50 | 33.44 | 35.60 |
| 60 | 5.60 | 19.50 | 33.44 | 35.60 |
| 61 | 5.60 | 19.50 | 33.44 | 35.60 |
| 62 | 5.60 | 19.50 | 33.44 | 35.61 |

Total Ceiling Sprinkler Flow Delivered = 652.84(gpm )

Job name: _____ Sheet number: _____

**PIPE INFORMATION**

| Node 1 | Elev 1 (ft) | K factor | Flow added —this step (q) | Nominal ID | Fittings—quantity and length | L ft | C Factor | total (Pt) | NOTES |
|---|---|---|---|---|---|---|---|---|---|
| Node 2 | Elev 2 (ft) | | Total flow (Q) | Actual ID | | F ft<br>T ft | Pf per foot (psi) | elev (Pe)<br>frict (Pf) | |
| data 1 | data 1 | data 1 | data 1 | data | data | data | data | data 1 | |
| data 2 | data 2 | | data | data | data | data | data | data | data |
| | | | | | data | data | | data | |
| data 1 | data 1 | data 1 | data 1 | data | data | data | data | data 1 | |
| data 2 | data 2 | | data | data | data | data | data | data | data |
| | | | | | data | data | | data | |
| data 1 | data 1 | data 1 | data 1 | data | data | data | data | data 1 | |
| data 2 | data 2 | | data | data | data | data | data | data | data |
| | | | | | data | data | | data | |
| data 1 | data 1 | data 1 | data 1 | data | data | data | data | data 1 | |
| data 2 | data 2 | | data | data | data | data | data | data | data |
| | | | | | data | data | | data | |
| data 1 | data 1 | data 1 | data 1 | data | data | data | data | data 1 | |
| data 2 | data 2 | | data | data | data | data | data | data | data |
| | | | | | data | data | | data | |
| data 1 | data 1 | data 1 | data 1 | data | data | data | data | data 1 | |
| data 2 | data 2 | | data | data | data | data | data | data | data |
| | | | | | data | data | | data | |
| data 1 | data 1 | data 1 | data 1 | data | data | data | data | data 1 | |
| data 2 | data 2 | | data | data | data | data | data | data | data |
| | | | | | data | data | | data | |

## KYPipe and GoFlow
## Pipe Data and Pipe Results

P I P E L I N E  D A T A

STATUS CODE: XX -CLOSED PIPE CV -CHECK VALVE

| PIPE NAME | NODE NAMES #1 | #2 | LENGTH (ft) | DIAMETER (in) | ROUGHNESS COEFF. | MINOR LOSS COEFF. |
|---|---|---|---|---|---|---|
| 1 | 2 | 3 | 1.00 | 2.00 | 120.0000 | 1.00 |
| 2 | 3 | 4 | 10.00 | 2.00 | 120.0000 | 1.00 |
| 3 | 4 | 5 | 10.00 | 2.00 | 120.0000 | 0.00 |
| 4 | 5 | 6 | 10.00 | 2.00 | 120.0000 | 0.00 |
| 5 | 6 | 7 | 10.00 | 2.00 | 120.0000 | 0.00 |
| 6 | 7 | 8 | 10.00 | 2.00 | 120.0000 | 0.00 |
| 7 | 8 | 9 | 10.00 | 2.00 | 120.0000 | 0.00 |
| 8 | 9 | 10 | 10.00 | 2.00 | 120.0000 | 0.00 |
| 9 | 10 | 11 | 10.00 | 2.00 | 120.0000 | 0.00 |
| 10 | 11 | 12 | 10.00 | 2.00 | 120.0000 | 0.00 |

P I P E L I N E  R E S U L T S

STATUS CODE: XX -CLOSED PIPE CV -CHECK VALVE

| PIPE NAME | NODE NUMBERS #1 | #2 | FLOWRATE gpm | HEAD LOSS ft | MINOR LOSS ft | LINE VELO. ft/s | ML+ML/ 1000 ft/f | HL/ 1000 ft/f |
|---|---|---|---|---|---|---|---|---|
| 1 | 2 | 3 | 74.50 | 0.15 | 0.90 | 7.61 | 1046.45 | 147.70 |
| 2 | 3 | 4 | 74.50 | 1.48 | 0.90 | 7.61 | 237.58 | 147.70 |
| 3 | 4 | 5 | 74.50 | 1.48 | 0.00 | 7.61 | 147.70 | 147.70 |
| 4 | 5 | 6 | 74.50 | 1.48 | 0.00 | 7.61 | 147.70 | 147.70 |
| 5 | 6 | 7 | 74.50 | 1.48 | 0.00 | 7.61 | 147.70 | 147.70 |
| 6 | 7 | 8 | 74.50 | 1.48 | 0.00 | 7.61 | 147.70 | 147.70 |
| 7 | 8 | 9 | 43.50 | 0.55 | 0.00 | 4.44 | 54.54 | 54.54 |

**FIGURE 22.3.5.1(d) Detailed Work Sheet.**

## Example Sections from Full GoFlow Report

```
*********************************************
A N A L Y S I S   R E S U L T S   S U M M A R Y
*********************************************


   Available Pressure at Supply:    32.9
   Pump Boost:                      79.2
   Boosted Pressure:                112.1

   Required Pressure:               55.9

   Pressure Margin:                 56.1
```

```
Sprinkler System Data Summay:
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Characteristics of Main Supply:       Supply-A
  Static Pressure =   40.0(psi)
  Residual Pressure =   28.0(psi)
  Residual Flow =  2020.0(  gpm)
  Elevation of Pressure Gage =   0.0(ft)

Flow Requirements:
  Outside Hose Flow Requirement (at supply) = 500.0(  gpm)
  There is NO Inside Hose Requirement Specified
  Total Sprinkler System Flow Required (Base of Riser) =     0.0(  gpm)

Sprinkler Information:
  Maximum Area Per Sprinkler Head =   100.0(SqFt)
  Required Minimum Flow (per Sprinkler) =    31.0(  gpm)
  Required Minimum Flow Density =     0.3(gpm/SqFt)

Other Sprinkler System Data:
  Default Value for C-Factor =   120.0
  Number of Booster Pumps =   0
```

```
Summary of Sprinkler Operation:
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~
```

| Name | Elevation (ft) | Sprink K | Length (ft) | Diameter (in) | Elev_Diff (ft) | Elbows | Type |
|---|---|---|---|---|---|---|---|
| 8 | 19.50 | 5.60 | 0.00 | 0.00 | 0.00 | 0 | Ceil |
| 9 | 19.50 | 5.60 | 0.00 | 0.00 | 0.00 | 0 | Ceil |
| 10 | 19.50 | 5.60 | 0.00 | 0.00 | 0.00 | 0 | Ceil |
| 11 | 19.50 | 5.60 | 0.00 | 0.00 | 0.00 | 0 | Ceil |
| 12 | 19.50 | 5.60 | 0.00 | 0.00 | 0.00 | 0 | Ceil |
| 26 | 19.50 | 5.60 | 0.00 | 0.00 | 0.00 | 0 | Ceil |

| | | | | | | | |
|---|---|---|---|---|---|---|---|
| 27 | 19.50 | 5.60 | 0.00 | 0.00 | 0.00 | 0 | Ceil |
| 28 | 19.50 | 5.60 | 0.00 | 0.00 | 0.00 | 0 | Ceil |
| 29 | 19.50 | 5.60 | 0.00 | 0.00 | 0.00 | 0 | Ceil |

## Example Sections from Full Report, continued.

```
************************************************
****** Summary of Sprinkler System Operation ******
************************************************

The Following Ceiling Sprinklers are Operating:
```

| Name | Sprink K | Elevation (ft) | Flow (gpm ) | Pressure (psi) |
|---|---|---|---|---|
| 8 | 5.60 | 19.50 | 30.63 | 29.88 |
| 9 | 5.60 | 19.50 | 30.51 | 29.65 |
| 10 | 5.60 | 19.50 | 30.50 | 29.62 |
| 11 | 5.60 | 19.50 | 30.52 | 29.67 |
| 12 | 5.60 | 19.50 | 30.67 | 29.96 |
| 26 | 5.60 | 19.50 | 32.59 | 33.82 |
| 27 | 5.60 | 19.50 | 32.59 | 33.81 |
| 28 | 5.60 | 19.50 | 32.59 | 33.81 |
| 29 | 5.60 | 19.50 | 32.59 | 33.81 |
| 30 | 5.60 | 19.50 | 32.59 | 33.82 |
| 42 | 5.60 | 19.50 | 32.77 | 34.19 |
| 43 | 5.60 | 19.50 | 32.76 | 34.18 |
| 44 | 5.60 | 19.50 | 32.76 | 34.18 |
| 45 | 5.60 | 19.50 | 32.76 | 34.18 |
| 46 | 5.60 | 19.50 | 32.77 | 34.19 |
| 58 | 5.60 | 19.50 | 33.02 | 34.73 |
| 59 | 5.60 | 19.50 | 33.02 | 34.72 |
| 60 | 5.60 | 19.50 | 33.02 | 34.72 |
| 61 | 5.60 | 19.50 | 33.02 | 34.72 |
| 62 | 5.60 | 19.50 | 33.03 | 34.73 |

```
Total Ceiling Sprinkler Flow Delivered =  644.71(gpm  )

Min. Density Provided by Ceiling Sprinklers =    0.31(gpm/SqFt)
```

```
The Following Rack Sprinklers are Operating:
```

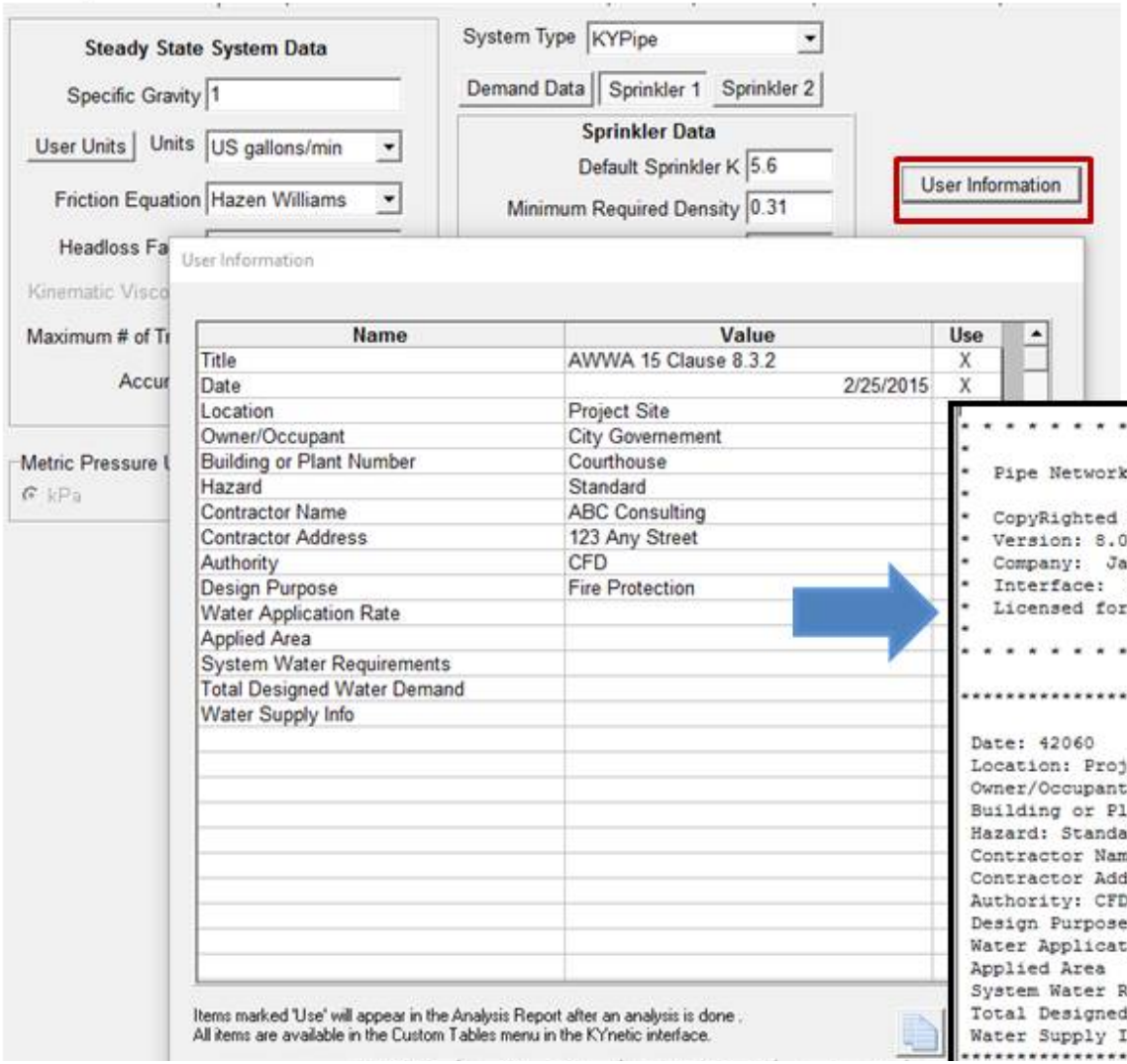
| Name | Sprink K | Elevation (ft) | Flow (gpm ) | Pressure (psi) |
|---|---|---|---|---|
| 102 | 5.60 | 19.50 | 32.97 | 34.61 |
| 103 | 5.60 | 19.50 | 32.13 | 32.87 |
| 104 | 5.60 | 19.50 | 31.57 | 31.74 |
| 105 | 5.60 | 19.50 | 31.24 | 31.07 |
| 106 | 5.60 | 19.50 | 31.08 | 30.76 |
| 107 | 5.60 | 19.50 | 31.04 | 30.68 |
| 109 | 5.60 | 19.50 | 32.95 | 34.57 |
| 110 | 5.60 | 19.50 | 32.11 | 32.84 |
| 111 | 5.60 | 19.50 | 31.55 | 31.70 |
| 112 | 5.60 | 19.50 | 31.22 | 31.04 |
| 113 | 5.60 | 19.50 | 31.06 | 30.73 |
| ******** | | 19.50 | 31.00 | 0.00 |

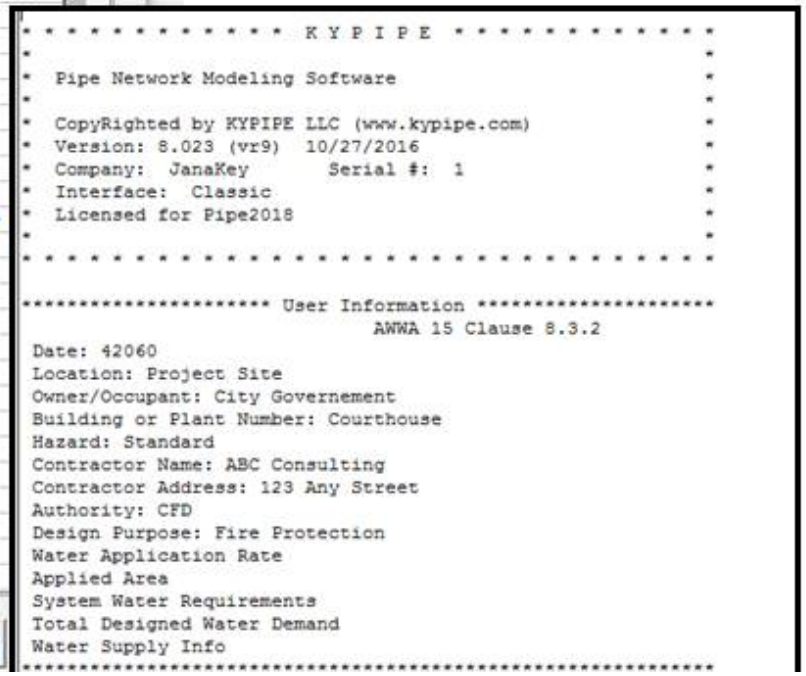
```
Total    Rack Sprinkler Flow Delivered =  379.92(gpm  )

Min. Density Provided by Rack Sprinklers =    0.31(gpm/SqFt)

Outside Hosestream Flow of   500.0(gpm  )
at the Main Supply Operates at -23.2(psi)

Results for Main Water Supply:      Supply-A :
  Static Pressure    =   40.0(psi)
  Residual Pressure  =   28.0(psi)
  Residual Flow      =  2020.0(gpm  )

  Operating Pressure=  -23.2(psi)
  Available Flow     =   500.0(gpm  )
```

### 1.6.2 User Information in Report

# NFPA Summary - Additional Information - also Available for **Surge**

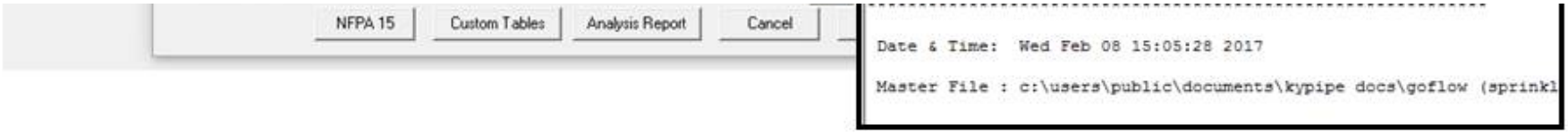
## System Data menu




Any data may be added at the beginning of the Report.

Blank rows may be used or an NPFA 15 format form is available.




## 1.6.3 GoFlow

GoFlow

---

GoFlow provides a simple, fast graphical environment for designing and carrying out the hydraulic analysis of fire sprinkler systems. The pipe system can be laid out very quickly using a mouse. A number of features have been incorporated to simplify and accelerate the task of developing pipe system layouts and corresponding data files. GoFlow provides comprehensive, effective results presentation which are in accordance with NFPA 13 requirements.

GoFlow operates in the Pipe2018 graphical environment. The procedures for system layout and data entry are covered in the many audio/video tutorials. Additional tutorials are provided for GoFlow and these include

- GoFlow1 - GoFlow elements
- GoFlow2 - Sprinkler system layouts - 10 operations
- QSP1 - QuickStart example - system layout
- QSP2 - QuickStart example - data entry
- QSP3 - QuickStart example - analysis and results

A QuickStart example steps you through the entire process of preparing a model and analyzing a sprinkler system. See GoFlow QuickStart.

See also GoFlow - System Data

- GoFlow - Nodes
- GoFlow - Generate System
- GoFlow - Auto Orthogonal

## 1.6.4 GoFlow - Quickstart Example

GoFlow - Quickstart Example

---

# GoFlow QuickStart Example

This will guide you through the complete layout development, data entry and hydraulic analysis of a simple sprinkler system. We suggest that you run GoFlow in as high a resolution as your monitor can display such that it can be comfortably read. We recommend the following monitor settings:

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

A simple example sprinkler system is shown in Figure 1. All the required data is displayed. This example will be used to guide you through the layout, data entry and hydraulic analysis. Note that several audio/video tutorials (QSP1, QSP2, and QSP3) are available also to guide you through this example.

## Step 1 - Initial Preparation

Initial steps include file selection, background preparation and system data input. Make sure that you are in Layout Mode and Auto-orthogonal is selected

**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 input** - The GoFlow System Data screen appears, KYnetic menu icon:

.

You need to enter or modify all pertinent data. The units displayed (English (g.p.m.)) are acceptable as are all the default values displayed. You need to enter the following data:

| | |
|---|---|
| Outside Hose Demand | 100 |
| Pipe Schedule (select) | Standard |
| Minimum Required Density | 0.1 |
| Minimum Area/Sprinkler | 400 |

**Click on *MAP* tab or *Return to Map* icon (KYnetic) to return to the GoFlow drawing region.**

**c. background preparation** - You can import a drawing, a 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 and let GoFlow calculate pipe lengths.

**Click on *Grid* icon (KYnetic) or *Map Settings/Grids*** - The grid settings of 50 (major) and 10 (minor) are good for our demonstration so we will select them from the dropdown list and use them.

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

## Step 2 - System Layout and Data Entry for Pipes and Nodes

The map area, which appears on the screen, will show a region approximately 1000 x 1000 feet with the 10-foot grid lines displayed. Use the ***Zoom Window*** button to draw a box covering 100x100'. 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.

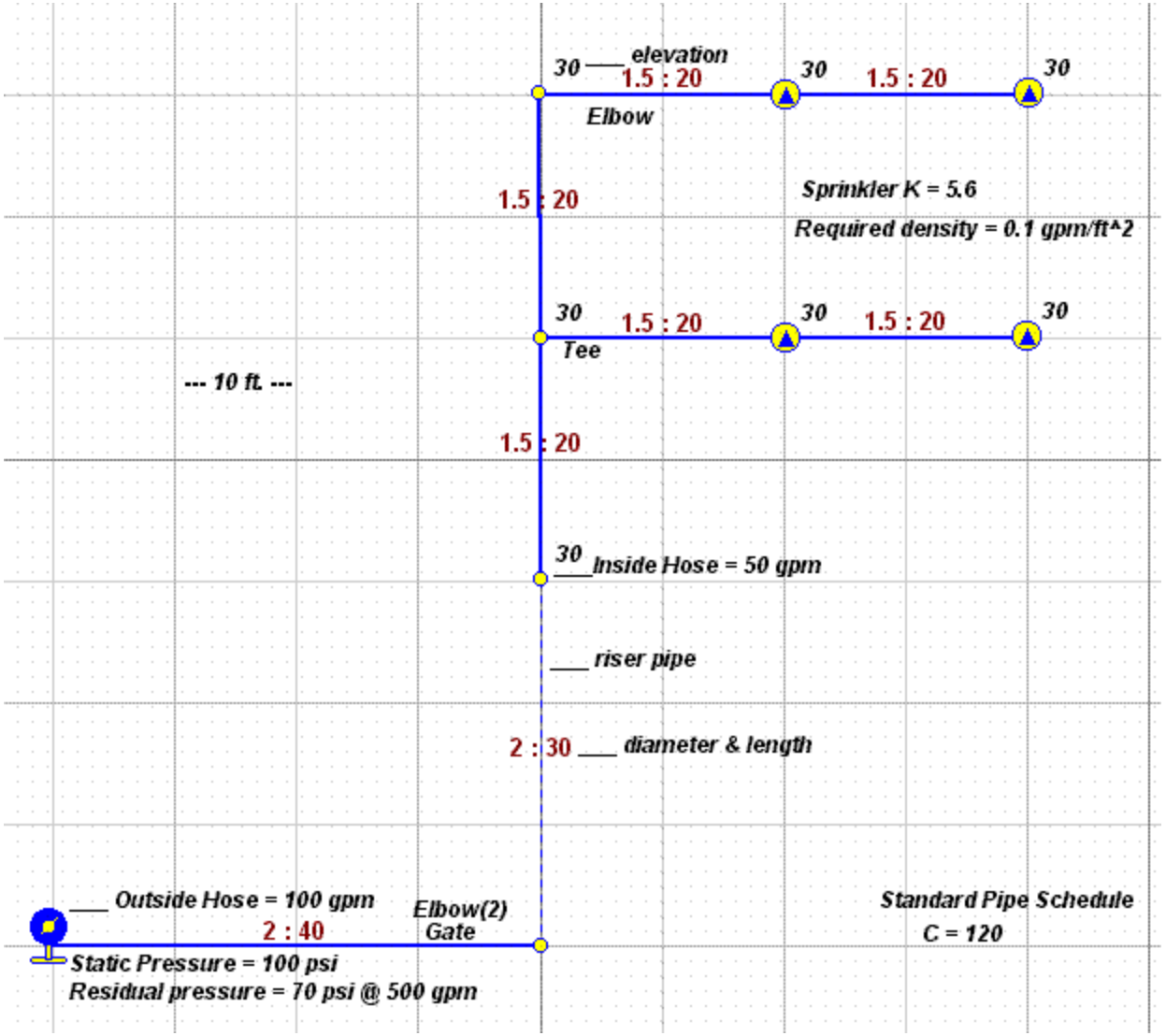



**Figure 1 Example pipe system**

**a.** **layout pipes** - 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 different node The following operations will produce the system**:**

1) **RC on a grid line intersection in the lower left corner to make first node**
2) **move mouse 40 feet (4 blocks) to right and RC**
3) **move mouse 30 feet up and RC**
4) **move mouse 20 feet up and RC**
5) **move mouse 20 feet up and RC**
6) **move mouse 20 feet to right and RC**
7) **move mouse 20 feet to right and RC**
8) **select node at (a) and move 20 feet to right and RC**
9) **move mouse 20 feet to left and RC**

Now all the pipes and nodes are laid out. Note all nodes are junction nodes and GoFlow has assigned pipe and node names.

**b.** **change node types** - Select any nodes which are different than junctions and change to the correct type. To do this select the junction node and select type from dropdown box under Node Information:

1) **Select nodes at Sprinkler locations (LC) and change node type to Sprinkler**

2) **Select node at Pressure Supply location and change node type to Pressure Supply**

The system should now look as shown below. **The AVI called QSP1 covers the steps described above.**

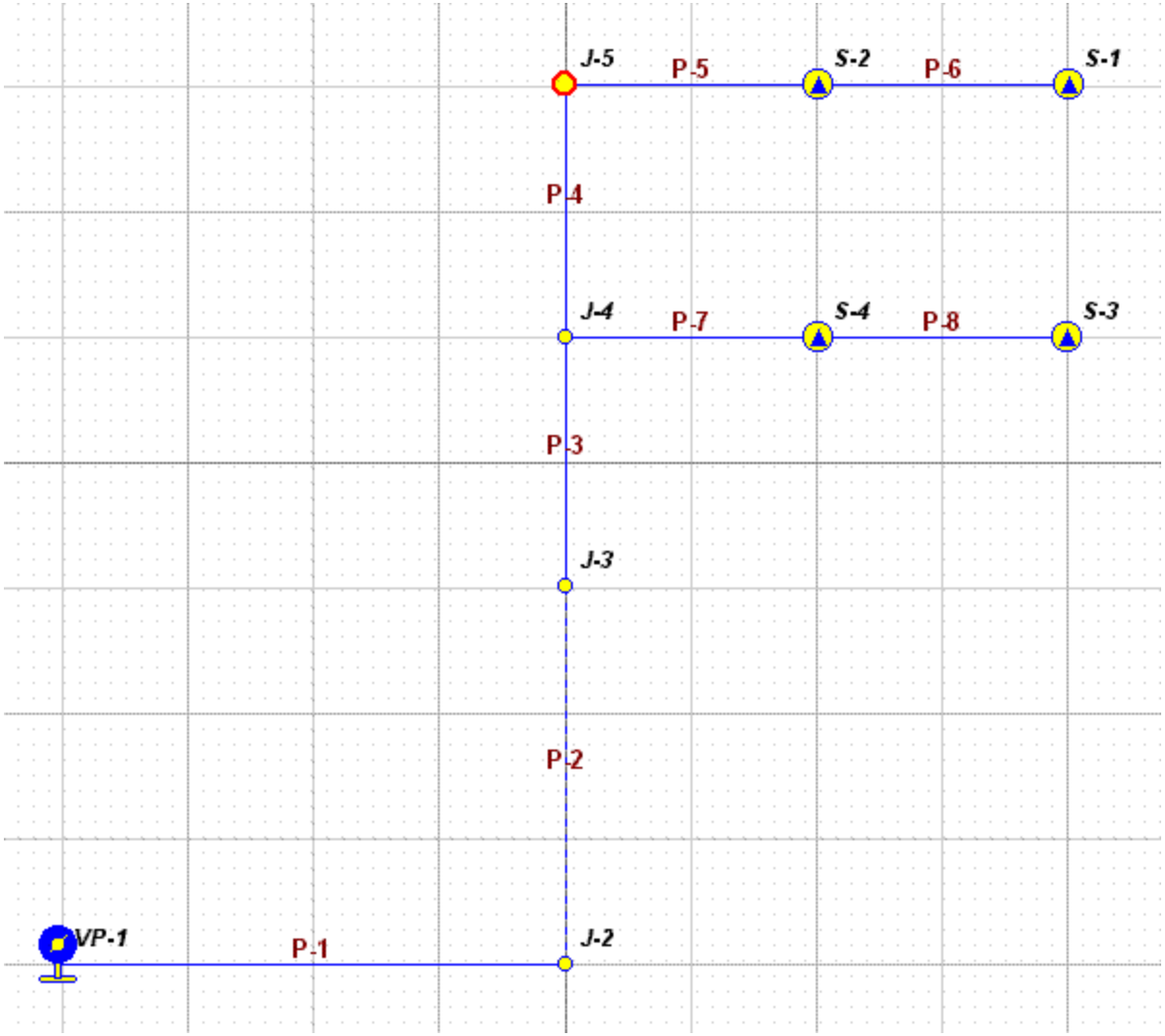



**Figure 2 Completed pipe system layout**

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 pull down list.** Note that default roughness values (120) are provided. Provide appropriate Fittings Data by checking the information in the Fittings Box (elbow for pipe with 90 degree bend, for example) The fittings are noted in Figure 1

2) **Select each Sprinkler and provide values shown for Elevation**

3) **Select each junction and provide Elevation. Provide a demand of 50 gpm at the node where the inside hose is shown. Enter this node name in the System Data to generate a report on the inside hose.**

4) **Select Pressure Supply and provide required data (static = 100 psi, residual = 70 psi at residual flow = 500 gpm)**

**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 Pop-up Menu (such as QS for QuickStart)**

## Step 3 - Analyze System and Review Results

**d. 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 to next step.

2 **Click *Analyze* (Main Menu) and select *Analyze System* and click *Analyze* on the Analysis box to accept the default** (Required Capacity). The Required Capacity is the hydraulic analysis performed with the required flow at the remote sprinkler head. This will calculate the pressure margin for your system. The Available Capacity is a hydraulic analysis using the capability of the pressure supply. For this analysis the flow at the remote sprinkler head will be above or below what is required.

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

**1** **Click on *Report*** 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. The tabulated output is shown in Figure 2. **Click on *Return to Map* icon or *Maps* tab**to go back to your system graphical display.

**2** **Click on *Labels* (Main menu) and select *Pipe Results* and *Node Results*** to show the pressures and flows. Figures 3 (Required Capacity) and 4 (Available Capacity) show a printout with the pressures and flows shown. You can click on the P selector at the bottom right to change the pipe results (to Velocity for example) and the N selector to change to the node results (to head, for example).

**3 Generate Water Supply Plot and Summary of Data and Hydraulic Calculations. Click on Analysis and select Supply Plot/Summary or use the KYnetic icons shown below.** This will generate the supply plot which is shown in Figure 5 and the Summary which is shown in Figure 6.

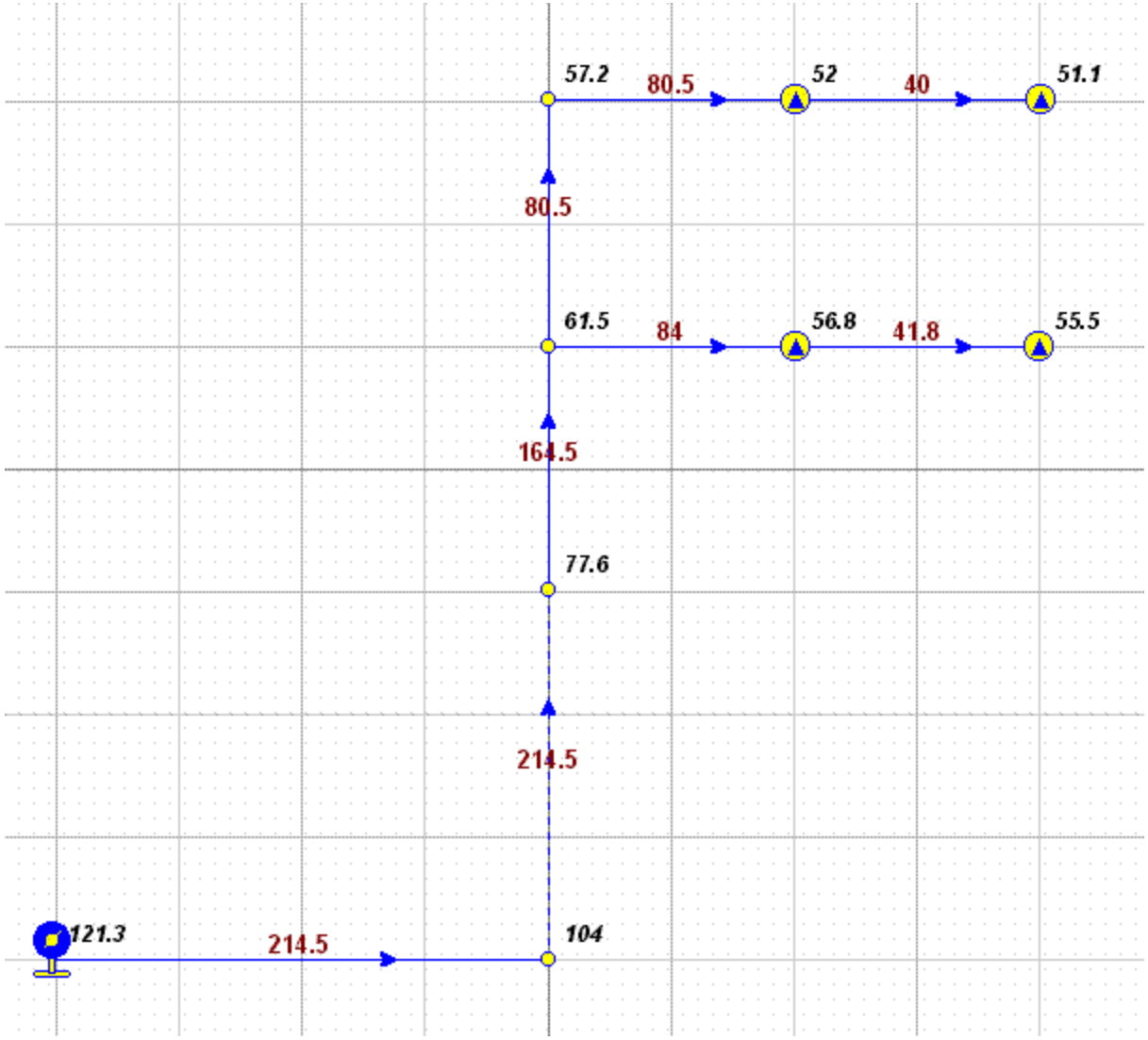
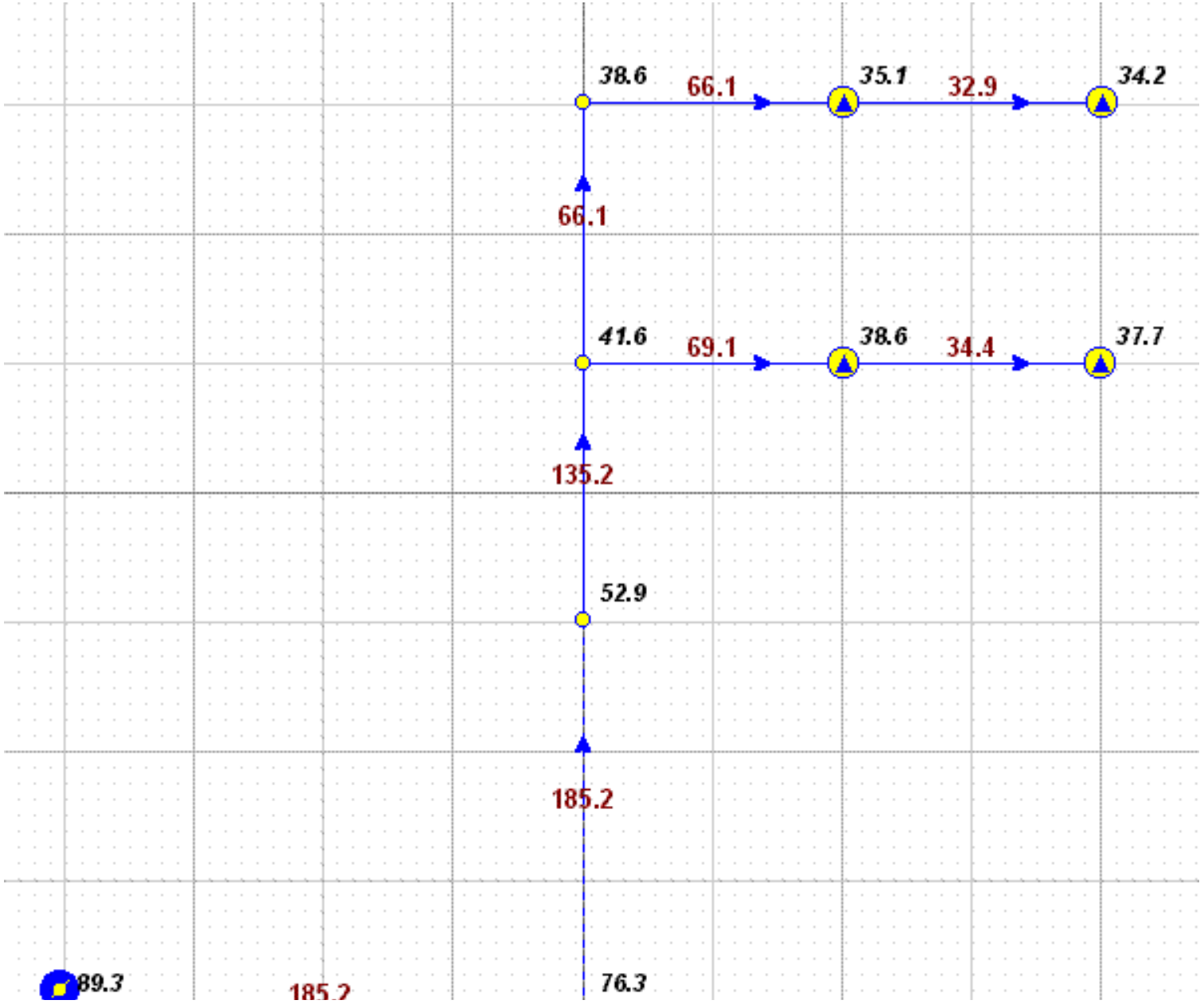



**Figure 3 Required Capacity - Pressure and Flow**


38.6
66.1
35.1
32.9
34.2
66.1
41.6
69.1
38.6
34.4
37.7
135.2
52.9
185.2
89.3
185.2
76.3

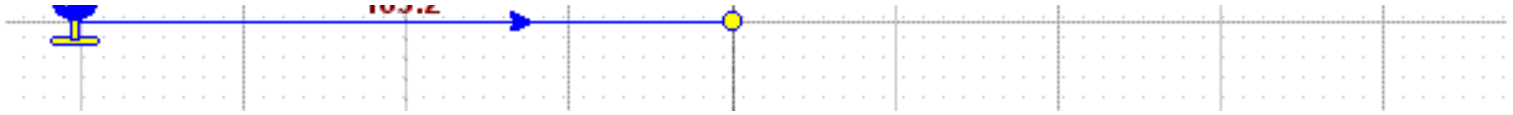


**Figure 4 Available Capacity - Pressures and Flow**

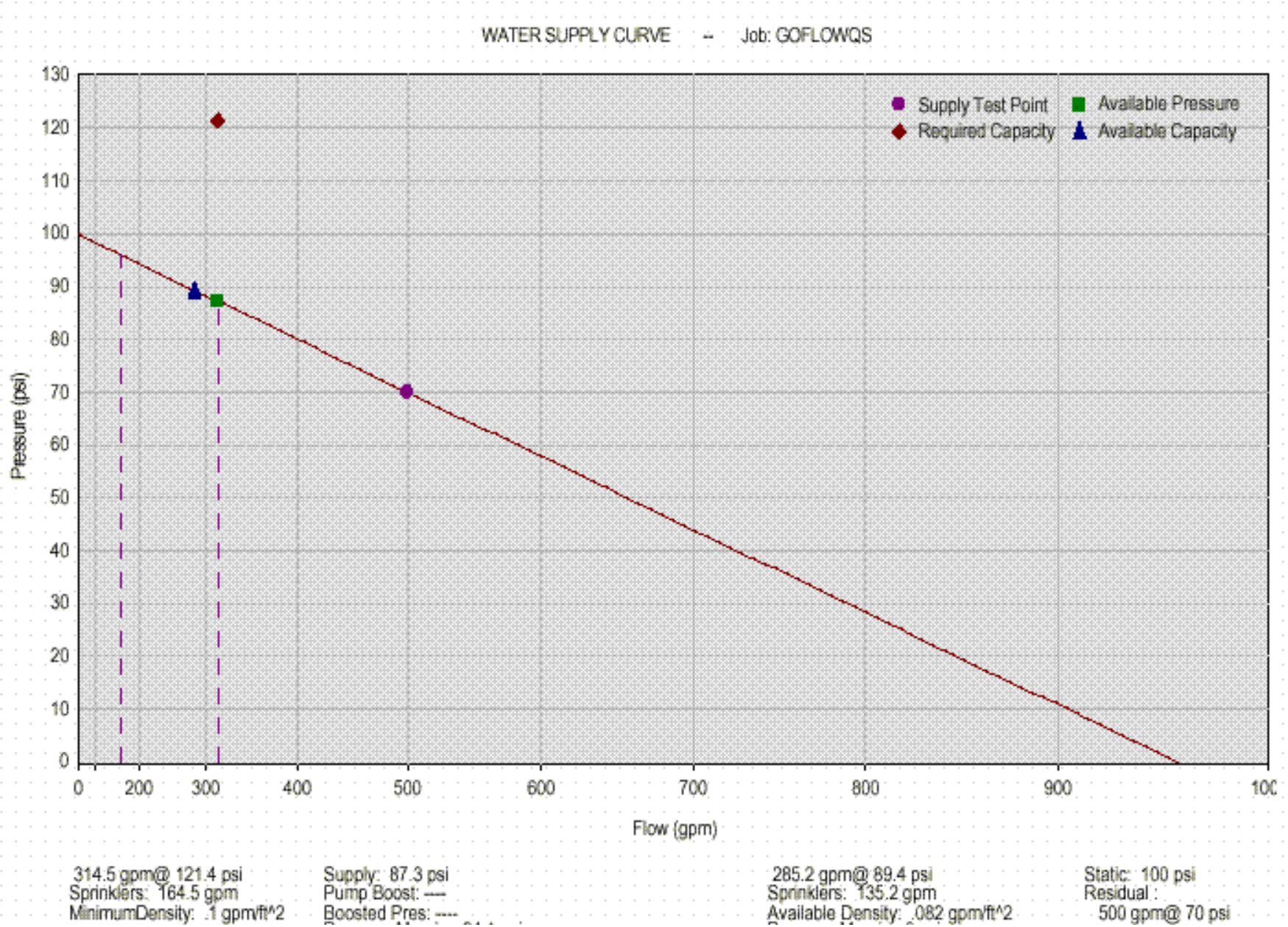

WATER SUPPLY CURVE -- Job: GOFLOWQS
Supply Test Point
Available Pressure
Required Capacity
Available Capacity
Pressure (psi)
130
120
110
100
90
80
70
60
50
40
30
20
10
0
0
200
300
400
500
600
700
800
900
Flow (gpm)
314.5 gpm@ 121.4 psi
Sprinklers: 164.5 gpm
MinimumDensity: .1 gpm/ft^2
Supply: 87.3 psi
Pump Boost: ----
Boosted Pres: ----
285.2 gpm@ 89.4 psi
Sprinklers: 135.2 gpm
Available Density: .082 gpm/ft^2
Static: 100 psi
Residual :
500 gpm@ 70 psi

**Figure 5 Supply Plot**

# GoFlow

Graphical Sprinkler Analysis Program
Summary of Data and Hydraulic Calculations

## Project Information

Company:
Address:

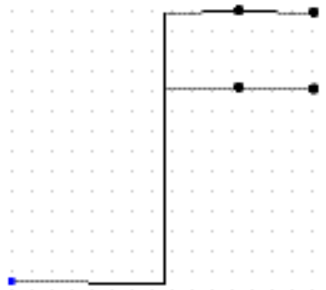
Engineer:
Date: 04-12-2001

Info: File Name - GOFLOWQS

## Sprinkler System Information

Min Density: .1 gpm/ft^2
Remote Area: 1500 ft^2
Sprinkler Coverage: 400 ft^2
Operating Sprinklers: 4
K-Factor: 5.6
Outside Hose: 100 gpm
Inside Hose: 0 gpm

## GOFLOWQS



## Hydraulic Calculations

**Required Capacity**
Minimum Density: .1 gpm/ft^2
Total Supply: 314.5 gpm
Sprinkler Flow: 164.5 gpm
Hoses, etc: 150 gpm
Required Pressure: 121.4 psi
Available Pressure: 87.3 psi
Pressure Margin: -34.1 psi

## Water Supply Data

Static Pres: 100 psi
Residual Pres: 70 psi
Residual Flow: 500 gpm

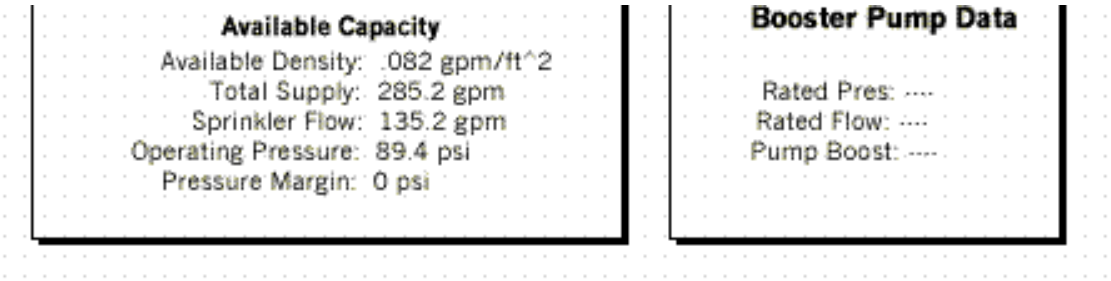



**Figure 6 Summary of Data and Hydraulic Calculations**

```
                    *** KYFire ***
HYDRAULIC ANALYSIS OF A FIRE PROTECTION SPRINKLER SYSTEM
              VERSION 1.2 - FEBRUARY 2001

DATE FOR THIS COMPUTER RUN = 04-03-2001 -  TIME = 17:43:13

DATA FILE NAME = C:\HELP_P~1\GOFLOW\QS



*****************************************************************
*************    SPRINKLER SYSTEM DATA SUMMARY    ***************
*****************************************************************



CHARACTERISTICS OF MAIN WATER SUPPLY A :
    STATIC PRESSURE =  100 PSI
    RESIDUAL PRESSURE =  70 PSI
    RESIDUAL FLOW =  500 GPM
    ELEVATION OF PRESSURE GAGE =  0 FT

FLOW REQUIREMENTS:
    OUTSIDE HOSE FLOW REQUIREMENT (AT SUPPLY) =  100 GPM
   THERE IS NO INSIDE HOSE REQUIREMENT SPECIFIED
    TOTAL SPRINKLER SYSTEM FLOW REQUIRED (BASE OF RISER) =  0 GPM
    ADDITIONAL FLOW REQUIREMENTS:
        FLOW REQUIREMENT =  50 GPM -  AT LOCATION (NODE) =  12

SPRINKLER INFORMATION:
    MAXIMUM AREA PER SPRINKLER HEAD =  400 SQ FT
    REQUIRED MINIMUM FLOW (PER SPRINKLER) =  40 GPM
    REQUIRED MINIMUM FLOW DENSITY =  0.1 GPM/SQ FT

OTHER SPRINKLER SYSTEM DATA:
    DEFAULT VALUE FOR C-FACTOR = 120
```

```
DEFAULT VALUE FOR C-FACTOR = 120
NUMBER OF BOOSTER PUMPS = 0
```

SUMMARY OF SPRINKLER OPERATION:

| LOCATION | ELEVATION | SPRINKLER K | PIPING TO SPRINKLER LENGTH | PIPING TO SPRINKLER DIAMETER | PIPING TO SPRINKLER ELEV. DIFF. | PIPING TO SPRINKLER TYPE |
|---|---|---|---|---|---|---|
| 17 | 30.0 | 5.6 | 0.000 | 0.000 | 0.000 | Ceil |
| 18 | 30.0 | 5.6 | 0.000 | 0.000 | 0.000 | Ceil |
| 19 | 30.0 | 5.6 | 0.000 | 0.000 | 0.000 | Ceil |
| 20 | 30.0 | 5.6 | 0.000 | 0.000 | 0.000 | Ceil |

**********
**********  PIPE DATA SUMMARY  **********
**********

NUMBER OF PIPES = 8
NUMBER OF JUNCTIONS = 9
NUMBER OF LOOPS = 0

| NUM. | FROM | TO | DIAM. (IN.) | HW-C | LENGTH (FT.) | EL. DIFF. (FT.) | ADDED LENGTH |
|---|---|---|---|---|---|---|---|
| 1 | A | 11 | 2.067 | 120 | 40.0 | 0.0 | 5.00 |
| 2 | 11 | 12 | 2.067 | 120 | 30.0 | 30.0 | 5.00 |
| 3 | 12 | 13 | 1.610 | 120 | 20.0 | 0.0 | 0.00 |
| 4 | 13 | 14 | 1.610 | 120 | 20.0 | 0.0 | 0.00 |
| 5 | 14 | 18 | 1.610 | 120 | 20.0 | 0.0 | 4.00 |
| 6 | 18 | 17 | 1.610 | 120 | 20.0 | 0.0 | 0.00 |

| | | | | | | | |
|---|---|---|---|---|---|---|---|
| 7 | 13 | 20 | 1.610 | 120 | 20.0 | 0.0 | 0.00 |
| 8 | 20 | 19 | 1.610 | 120 | 20.0 | 0.0 | 0.00 |

**Table 1 - GoFlow Data Summary**

```
*****************************************************************
****   HYDRAULIC ANALYSIS WITH REQUIRED FLOW AT REMOTE HEAD   ****
*****************************************************************



****SUMMARY OF SPRINKLER SYSTEM OPERATION****

 THE FOLLOWING CEILING SPRINKLERS ARE OPERATING

----------+-----------+---------+-------+--------
 LOCATION  SPRINKLER K ELEVATION   FLOW  PRESSURE
----------+-----------+---------+-------+--------
    17          5.60    30.00    40.00    51.02
    18          5.60    30.00    40.45    52.18
    19          5.60    30.00    41.78    55.66
    20          5.60    30.00    42.25    56.92
----------+-----------+---------+-------+--------

THE TOTAL CEILING SPRINKLER FLOW DELIVERED = 164.48 GPM

THE OUTSIDE  HOSESTREAM FLOW OF  100  GPM
AT THE MAIN SUPPLY OPERATES AT  87.28 PSI

THE MINIMUM DENSITY PROVIDED BY THE CEILING SPRINKLERS IS    0.1 GPM/SQ FT


 RESULTS FOR MAIN WATER SUPPLY A :
   STATIC PRESSURE =  100 PSI
   RESIDUAL PRESSURE =  70 PSI AT  500 GPM
   OPERATING PRESSURE = 121.38 PSI
   AVAILABLE PRESSURE =  87.28 PSI AT  314.48 GPM

********* PRESSURE MARGIN = -34.1 PSI *********
```

**********
********** SUMMARY OF HYDRAULIC CALCULATIONS **********
**********

| NUM. | NODES FROM-TO | FLOW (GPM) | PIPE (FT.) | EQV. FT | H-W -C- | DIA. IN. | PSI/ 100-FT | ELEV. PSI. | START PSI. | END PSI. | DIFF. PSI. |
|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | A 11 | 214.5 | 40.0 | 5.0 | 120 | 2.067 | 38.63 | 0.00 | 121.39 | 104.00 | 17.39 |
| 2 | 11 12 | 214.5 | 30.0 | 5.0 | 120 | 2.067 | 38.63 | 13.00 | 104.00 | 77.48 | 26.52 |
| 3 | 12 13 | 164.5 | 20.0 | 0.0 | 120 | 1.610 | 79.79 | 0.00 | 77.48 | 61.52 | 15.96 |
| 4 | 13 14 | 80.5 | 20.0 | 0.0 | 120 | 1.610 | 21.22 | 0.00 | 61.52 | 57.28 | 4.24 |
| 5 | 14 18 | 80.5 | 20.0 | 4.0 | 120 | 1.610 | 21.22 | 0.00 | 57.28 | 52.18 | 5.09 |
| 6 | 18 17 | 40.0 | 20.0 | 0.0 | 120 | 1.610 | 5.82 | 0.00 | 52.18 | 51.02 | 1.16 |
| 7 | 13 20 | 84.0 | 20.0 | 0.0 | 120 | 1.610 | 23.00 | 0.00 | 61.52 | 56.92 | 4.60 |
| 8 | 20 19 | 41.8 | 20.0 | 0.0 | 120 | 1.610 | 6.31 | 0.00 | 56.92 | 55.66 | 1.26 |

THE MAXIMUM VELOCITY OF 25.91 FT/S OCCURS BETWEEN NODES 12 AND 13

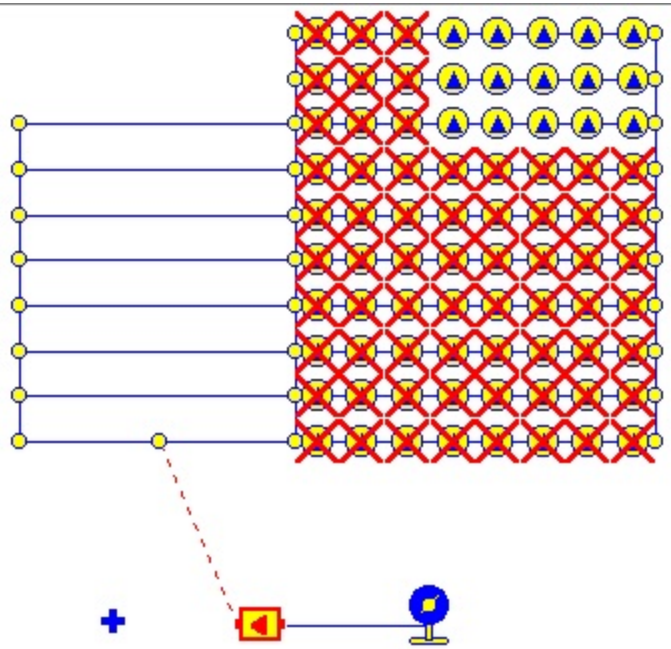
***** END OF SIMULATION *****

**Table 2 - GoFlow Tabulated Results**

## 1.6.5 GoFlow - Nodes

GoFlow - Nodes

There are a limited number of nodes available for GoFlow as shown below. Some additional description of the nodes and associated data inputs are described.



**Junction**

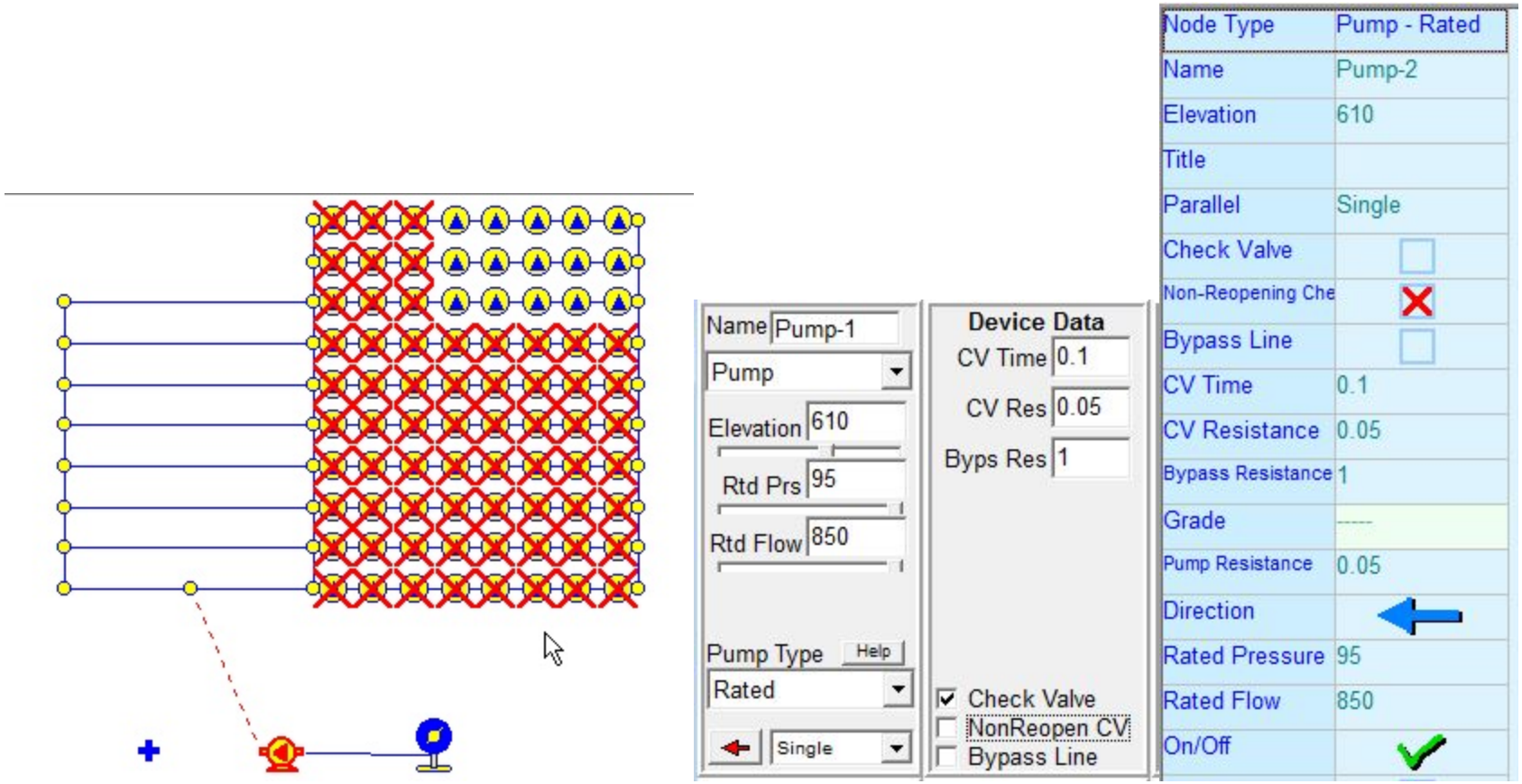
A demand may be specified at any junction. This may represent an inside hose or other requirement. For the inside hose the junction name should be indicated on the System Data | System Specifications screen so this will be referenced in the tabulated report.

**Reservoir**

See Pressure Supplies.

**Pumps**

Provide the Rated Pressure and Rated Flow for the fire pump as shown below




**Sprinkler (Rack Sprinkler)**

Sprinkler Constant - The sprinkler constant is defined as the flow divided by the square root of the pressure. Some typical values are presented (see Sprinkler Constant)

**Sprinkler Connections**

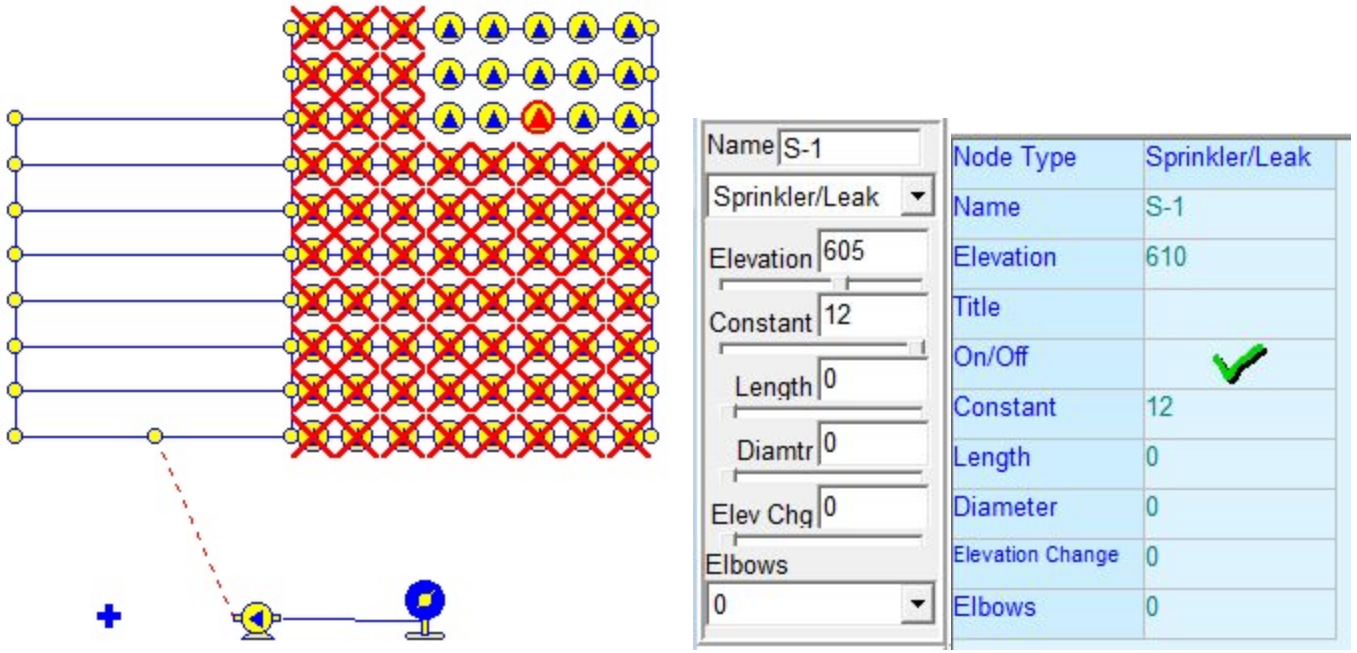
Leave the data fields blank if the sprinkler connects directly to a branch line. Data for sprinkler connections include the following

**Length** - the total length in feet (m) of the connecting pipe

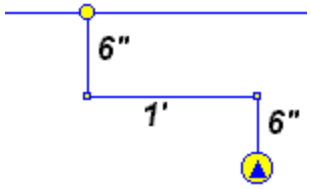
**Diameter** - the diameter in inches (mm) of the connecting pipe

**Elevation Change** - the change in elevation in feet (m) from the branch line to the sprinkler (use negative (-) sign if sprinkler is lower than branch line)

**Elbow** - click on number of elbows in connection




The data shown is for a connection such as the one shown below.

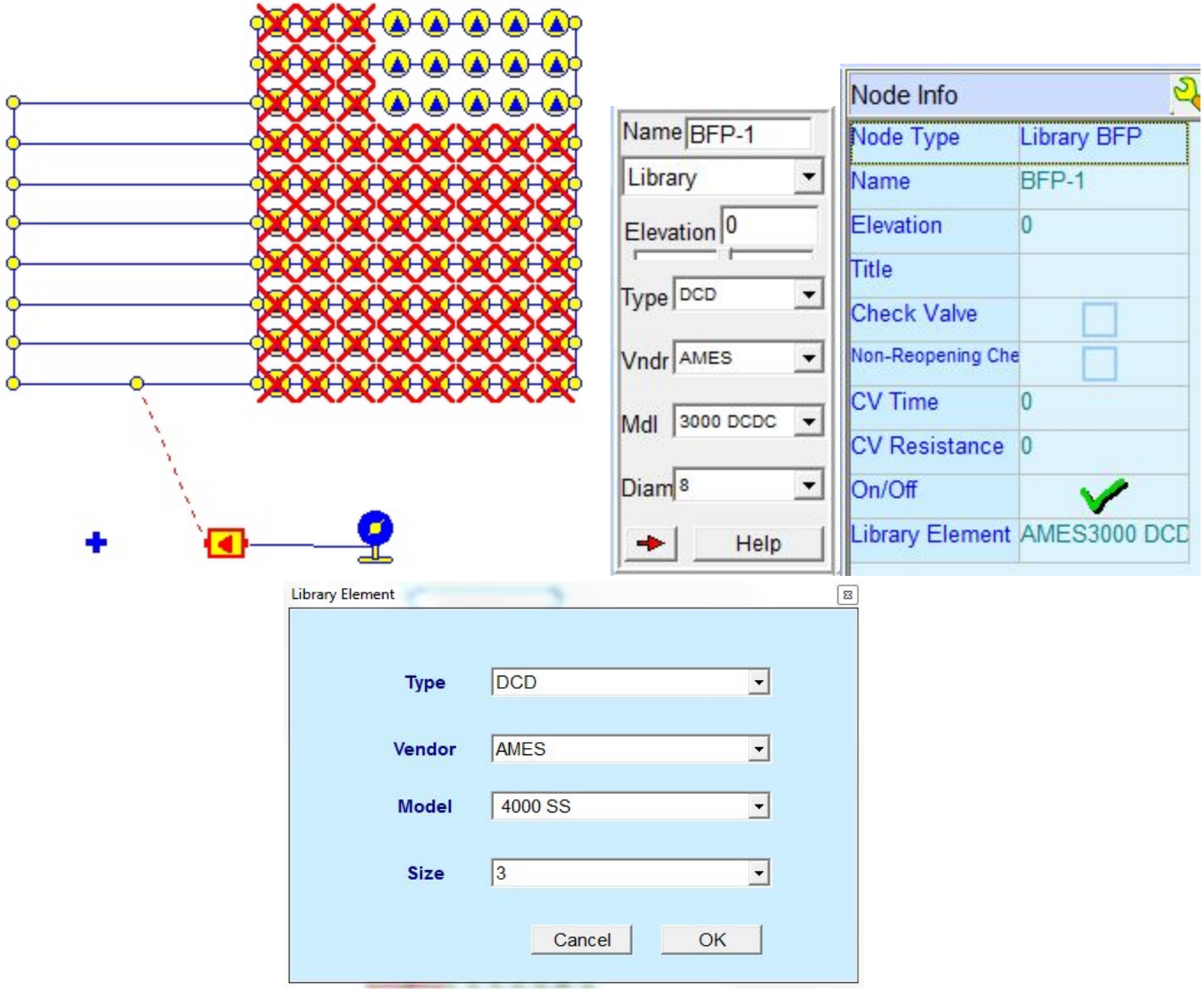



**Pressure Supplies**

All sprinkler systems should have one pressure supply which is designated as the main supply. Additional supplies can be modeled and will be designated as Not Main supply. If a supply is a constant level reservoir then a Reservoir node should be selected and the Grade (water level) input.

**BFP (Backflow Preventer)**

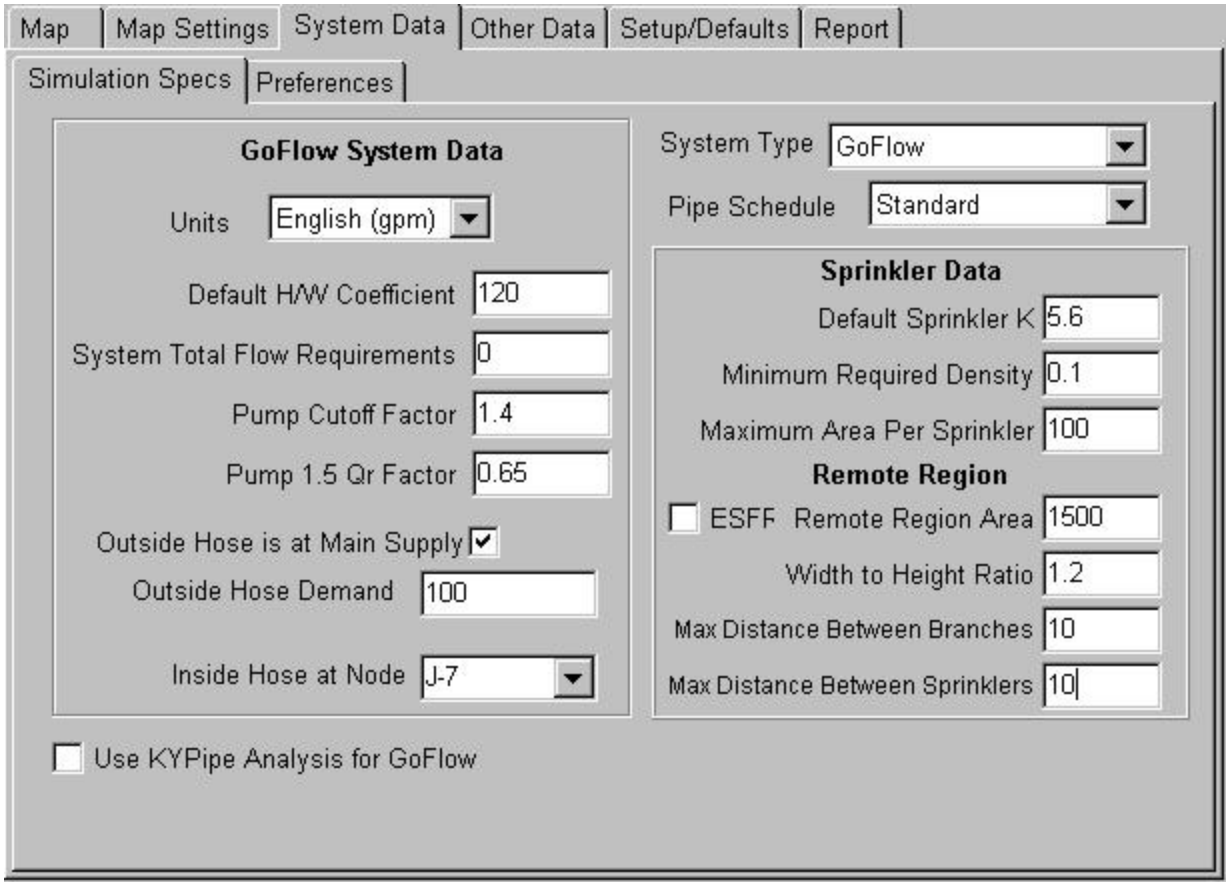
Backflow preventers prevent flow backflow into the supply line. Over two hundred BFPs are included in the library and you can select any of these by selecting the Type, Vendor, Model, and Diameter from the drop down lists provided as shown below. You can add new devices by accessing the Backflow Preventer Spreadsheet: Other Data | Loss Element (BFP).


Name BFP-1
Library
Elevation 0
Type DCD
Vndr AMES
Mdl 3000 DCDC
Diam 8
Help
Node Info
Node Type Library BFP
Name BFP-1
Elevation 0
Title
Check Valve
Non-Reopening Che
CV Time 0
CV Resistance 0
On/Off
Library Element AMES3000 DCD
Library Element
Type DCD
Vendor AMES
Model 4000 SS
Size 3
Cancel
OK

## 1.6.6 GoFlow - System Data

GoFlow - System Data

KYnetic System Data icon:



### GoFlow System Data

**Units**: Choose the units from the choices offered which include English (gpm) and three SI choices; liters/second, centimeters/hour, liters/minute. See GoFlow Units

**Default HW Coefficient**: This value will be used for any pipe entered without the Hazen Williams Coefficient defined.

**System Total Flow Requirements**: This is an optional input for the total flow delivered to the riser. It does not affect the hydraulic calculations.

**Pump Cutoff Factor**: The pump cutoff pressure (churn) is calculated as Rated Pressure x this factor (default value is 1.4)

**Pump 1.5 Qr Factor**: This defines the pressure at 1.5 x Rated Flow as the Rated Pressure x this factor.

**Outside Hose is at Main Supply**: Check this box of the Outside Hose is located at the main supply. If this is not checked then the next box will ask you to identify the node for the Outside Hose*.

**Outside Hose Demand**: The outside hose requirement in gpm if the location is the main supply.

**Inside Hose at Node**: The node location for the primary inside hose requirement*.

*Note: the node (junction) demands will define the magnitude of the hose requirements at these nodes. These requirements can be imposed at any junction in the sprinkler system.

**Use KYPipe Analysis for GoFlow** - This check box allows the user to use the KYPipe analysis engine to analyze the model. This enables such features as Changes and Demand Patterns, and added options for pumps and pressure supplies to be used within a fire sprinkler analysis.

**System Type** - For GoFlow users, this should be set to GoFlow (or GoFlow (KY) if Use KYPipe Analysis for GoFlow is selected). Other System Types are for use with other Pipe2018 models.

**Pipe Schedule**: Select the pipe schedule to be used for the Pipe Type from the drop down list. You can add schedules to this list. Note: it is important to select or provide the appropriate schedule prior to entering data. See Pipe Type.

### Sprinkler Data

**Default Sprinkler K**: the K factor for the principal sprinklers.

**Minimum Required Density**: the required density (in gpm/ft^2 or appropriate SI units) for the sprinklers.

**Maximum Area Per Sprinkler**: the maximum coverage area per sprinkler. The required flow for a sprinkler is the product of this and the previous entry.

### Remote Region

This data will be required only if the hydraulically remote area is to be determined.

**ESFR** - Stands for Early Suppression Fast Response. Refers to areas of lowest pressure. By checking this box, instead of defining the remote region are with an area and width to height ration, a configuration of 12 sprinklers in a 4 x 3 arrangement arrangement is used. The ESFR specification uses the design pressure specified and ignores the required density values above.

**Remote Region Area** - Rectangular area in ft ^ 2 or m ^2. The analysis will define the most remote area hydraulically (lowest flow).

**Width to Height Ratio**: this is used to size the width and height of the remote area. For the data shown, height x 1.2 width = 1500 or height = 35.4 and width =

42.4.

**Max Distance Between Branches** - in feet or meters

**Max Distance Between Sprinklers** - in feet or meters.

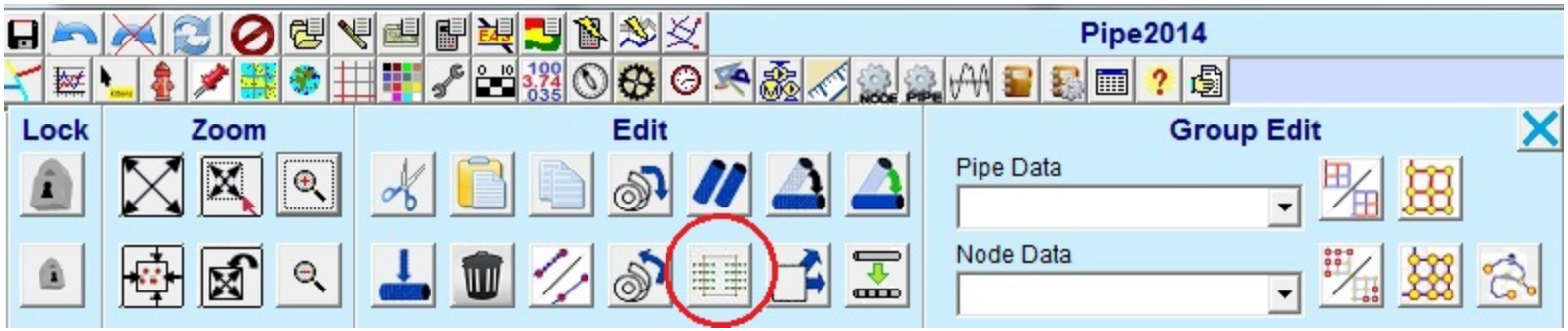
## 1.6.7 GoFlow - Generate System

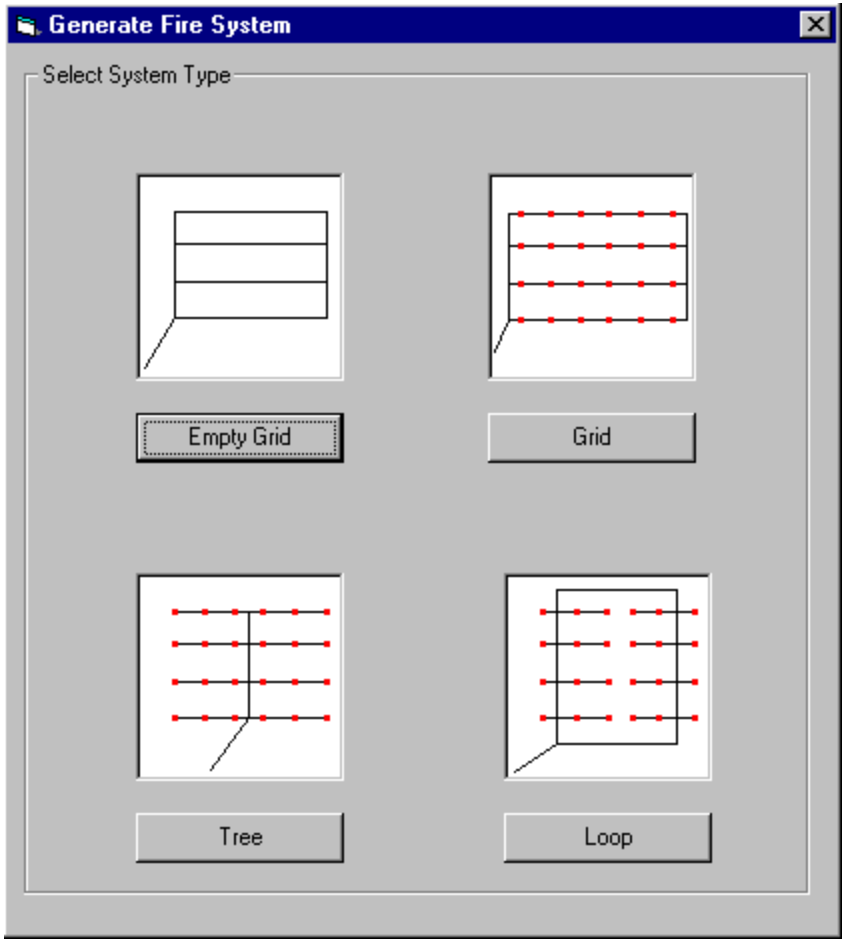
GoFlow - Generate System

---

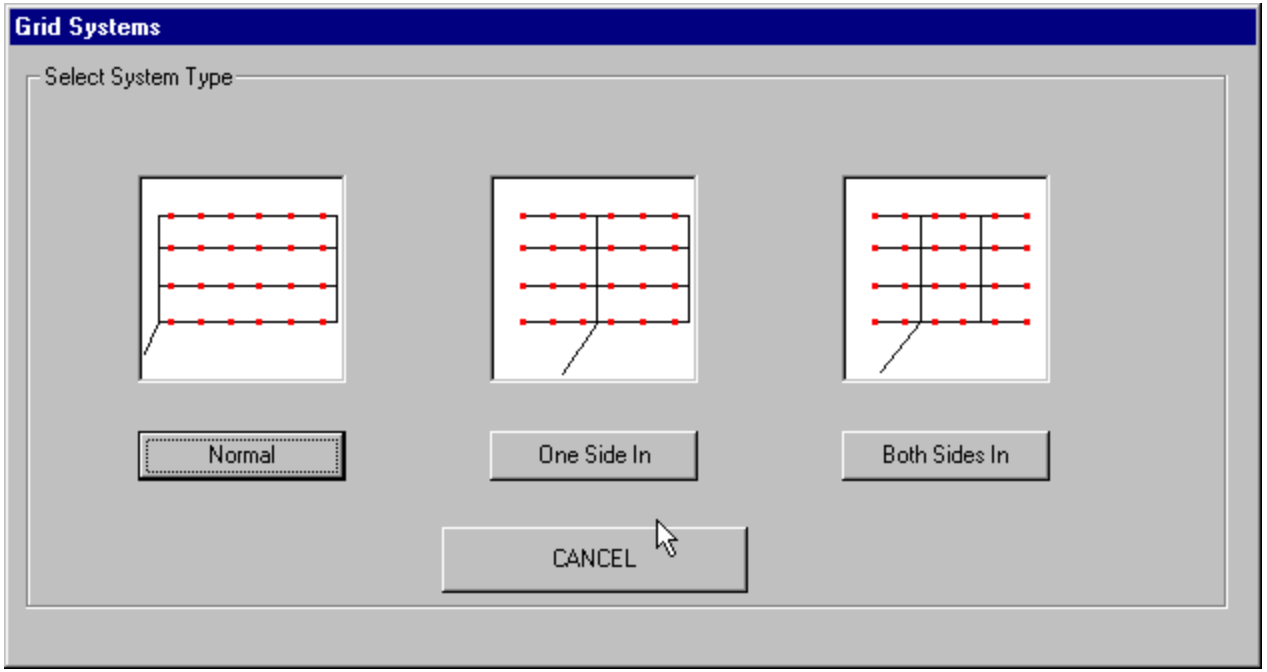
**Generating Systems**

Generating Systems is a very useful time savings feature. This is accessed as shown above by clicking on Edit and Generate System. Before accessing the feature, select the node (layout mode) where the generated system will be attached. The following selection appears.

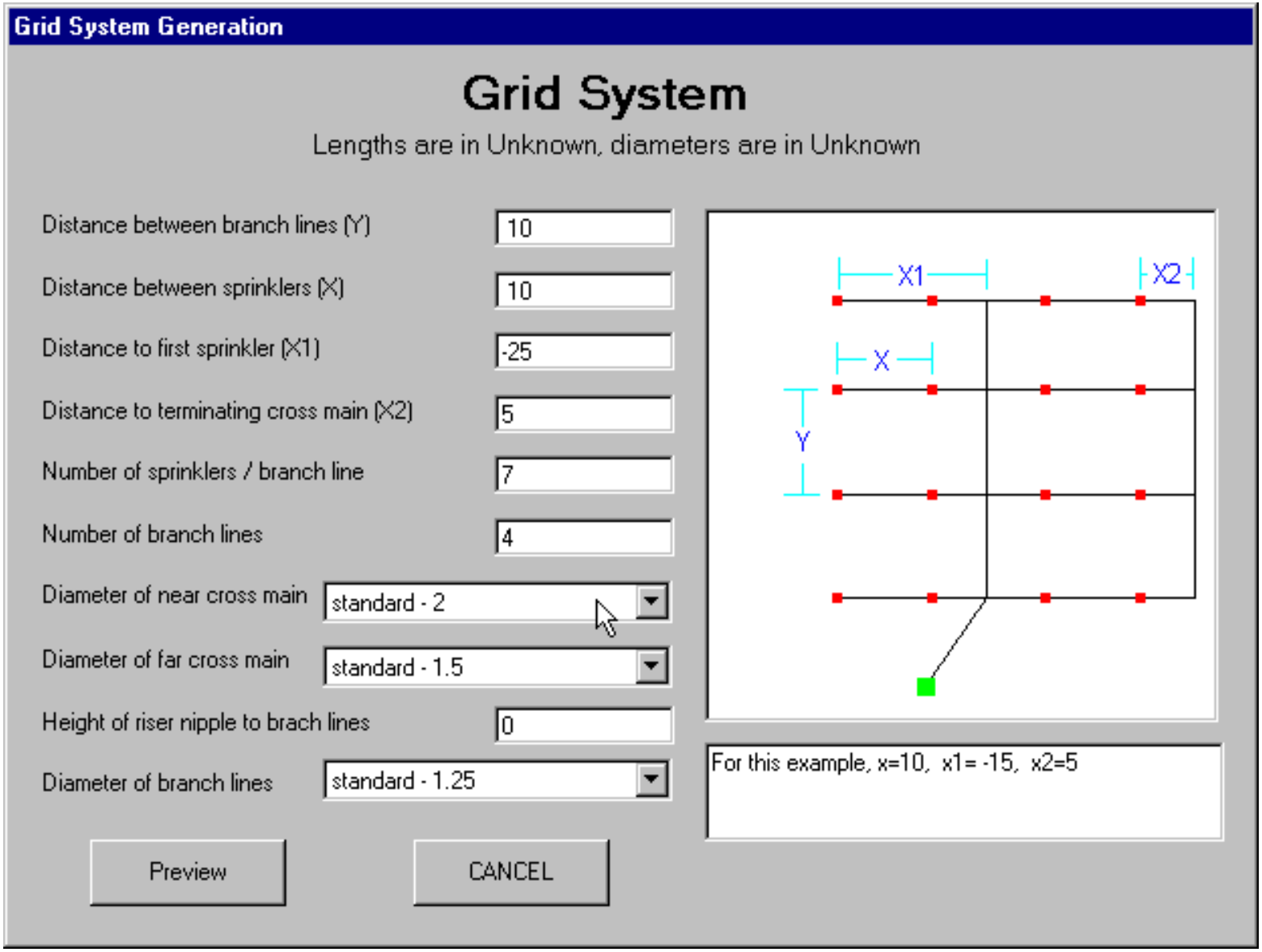




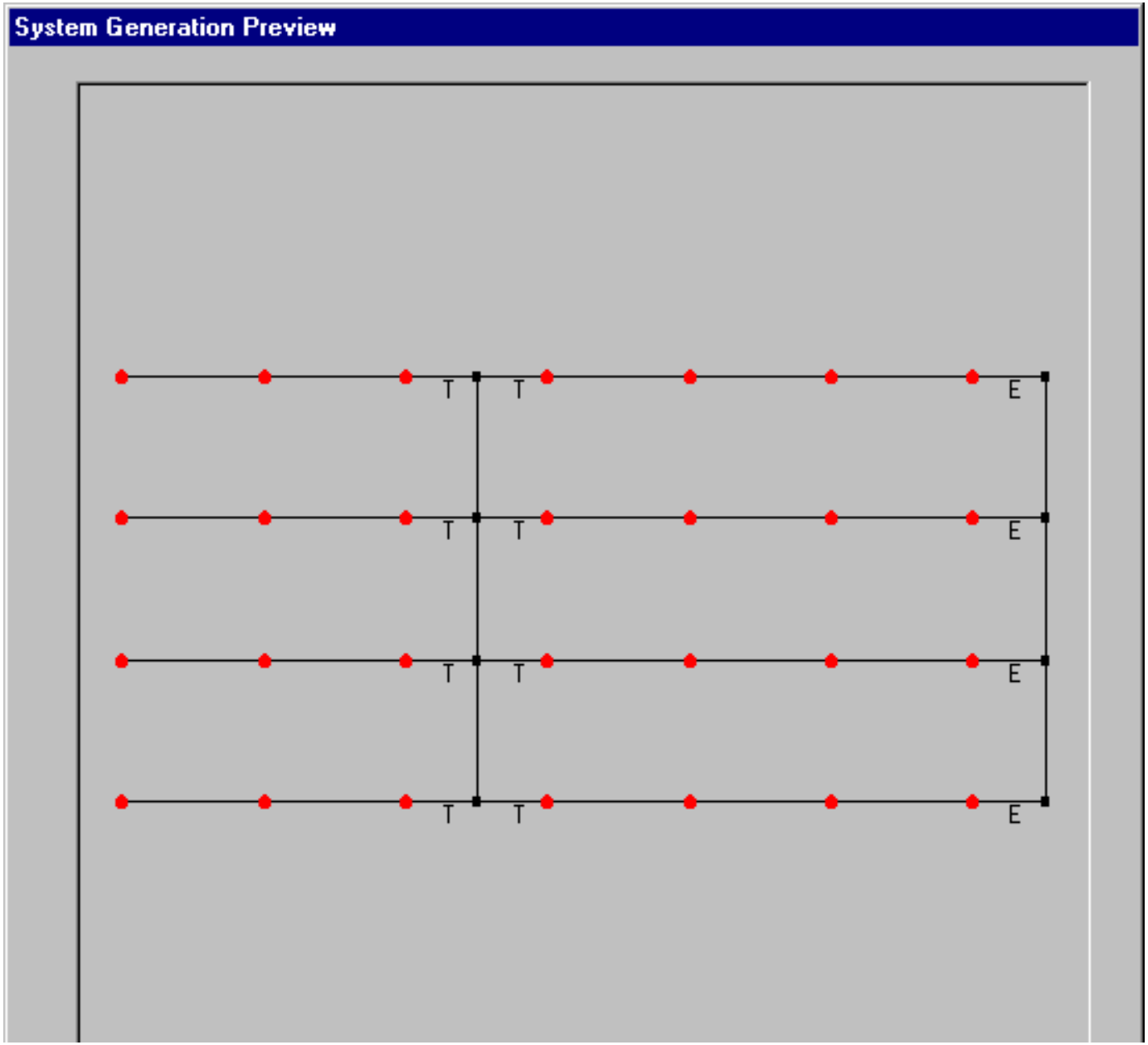

Click on the basic configuration and additional choices for that configuration will appear.

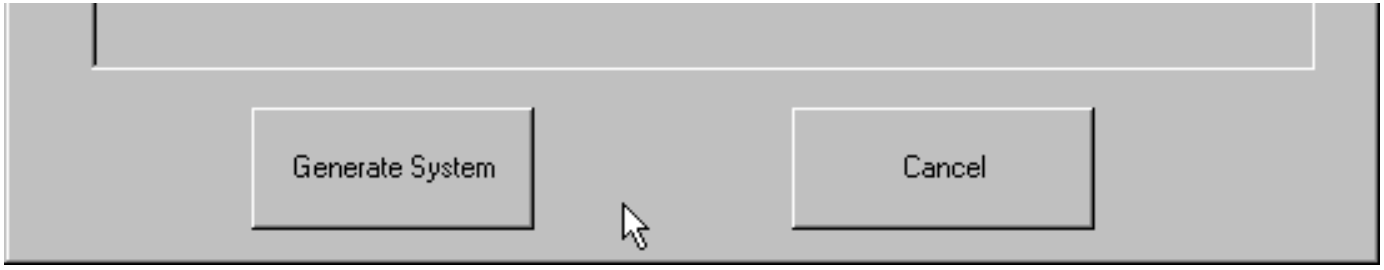



Select your configuration and the following menu will appear. You can enter the data specific to your configuration. Note the distance parameters are noted on the schematic. Distances must be entered as a negative if the orientation from the first to second location for that distance is to the left. For example shown below the distance from the cross main to first (leftmost) sprinkler is 15 feet to the left (-15) and from the last sprinkler to cross main is to the right (5). For the system being generated X1 will be -25.




Once the data is entered you can click preview to see the system before it is generated.


System Generation Preview
T
T
E
T
T
E
T
T
E
T
T
E



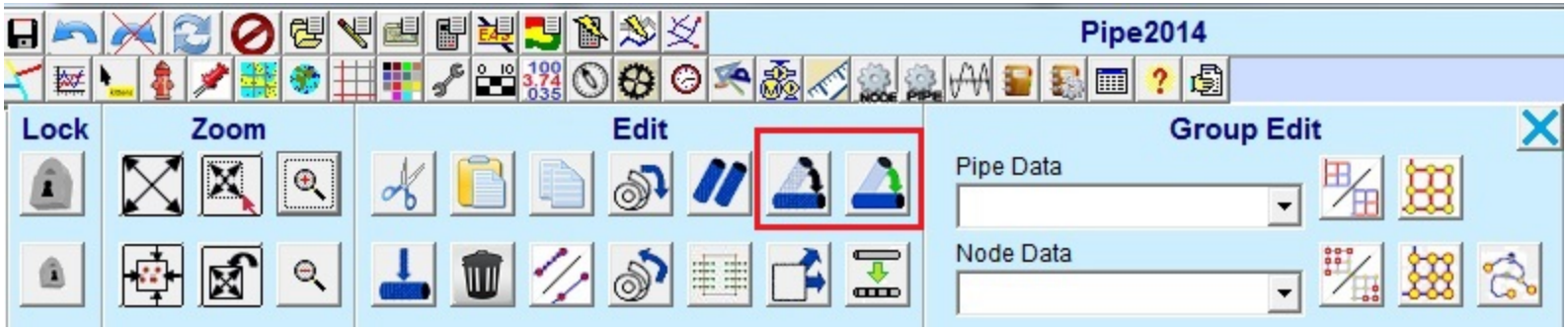

## 1.6.8 GoFlow - Auto Orthogonalize

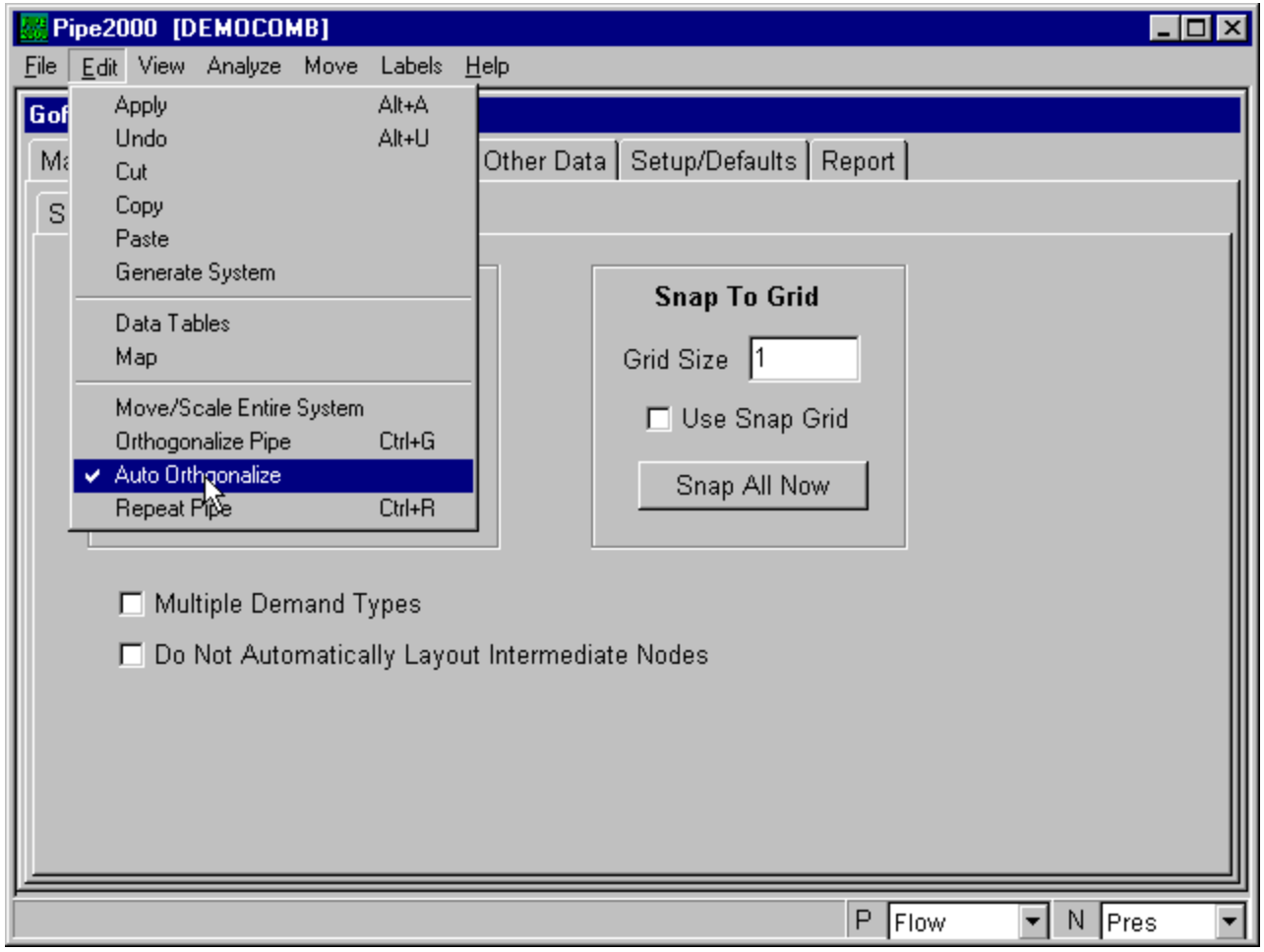
GoFlow - Auto Orthogonalize

---

**Auto Orthogonalize**

This feature should be turned on to automatically orthogonalize pipes. It is recommended that you normally operate with this switch on. Click on Edit and Auto Orthogonalize to turn on or off.


Pipe2014
Lock
Zoom
Edit
Group Edit
Pipe Data
Node Data




## 1.6.9 GoFlow - Booster or Fire Pump Water Supply

## Booster or Fire Pump Water Supply:

Enter fire pump test data into GoFlow using two flow/pressure points from the pump test curve as a pressure supply:

Enter the pump test results as a pressure supply as follows:

**Static Pr –** enter the discharge pressure at churn (no flow)

**Res Pr –** calculate the estimated flow of the sprinkler system using the flowing formula:

Flow = (1.1 x sprinkler density x remote area) + hose stream

Extrapolate the discharge pressure (not the net pressure) from the pump discharge curve corresponding to the above estimated flow. Enter this pressure as the **Res Pr** value in the node information window.

## ESFR Systems:

For analysis of ESFR systems, the minimum required density and remote area need to be handled differently than a conventional density/area design since ESFR system designs are based on 12 sprinklers operating at a specific pressure and the remote area is defined as 4 sprinklers on 3 branch lines.

On the System Data tab, enter minimum required density, remote region area, and width to height ratio data as follows:

**Minimum Required Density =** $K \times (P)^{½} / S$

where: K = sprinkler K-factor

P = ESFR design pressure (psi)
S = sprinkler spacing (ft²)

**Remote Area Region =** 12 x S

where: S = sprinkler spacing (ft²)

***Width to Height Ratio =** d x 4 / $(RA)^{½}$

where: d = sprinkler to sprinkler distance along branch line (ft.)
RA = remote area (ft²)

*Note: this number should be rounded down to the nearest tenth (i.e. 1.15 1.1)

## 1.6.10 GoFlow - Tree Generator Example

GoFlow - Tree Generator Example

---

### Tree Generator Example:

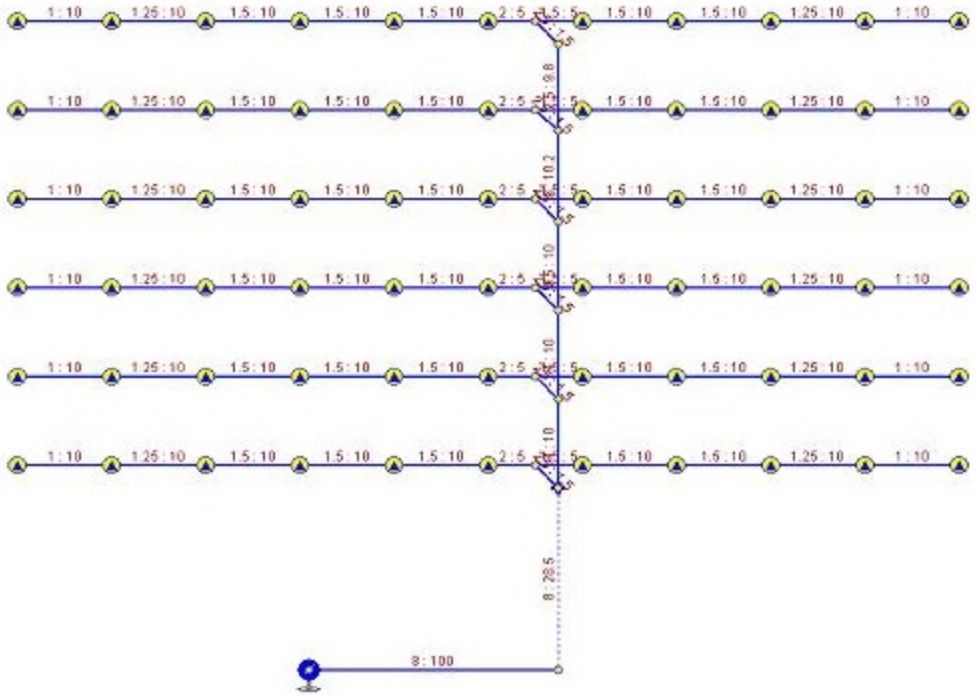
For this example, we will use the tree generator in GoFlow to analyze a wet pipe, center-fed, 1-2-5 extra hazard pipe schedule system. The system characteristics are as follows:

Sprinkler K-factor: 5.6
Sprinkler spacing: 10 ft.
Branch line spacing: 10 ft.
Pipe schedule: Schedule 40

Other information is as follows:

Required design: 0.40/2500 + 500 gpm outside hose
Public water supply: Static – 100 psi
Residual – 78 psi
Flow – 1250 gpm

**Sprinkler System Sketch:**



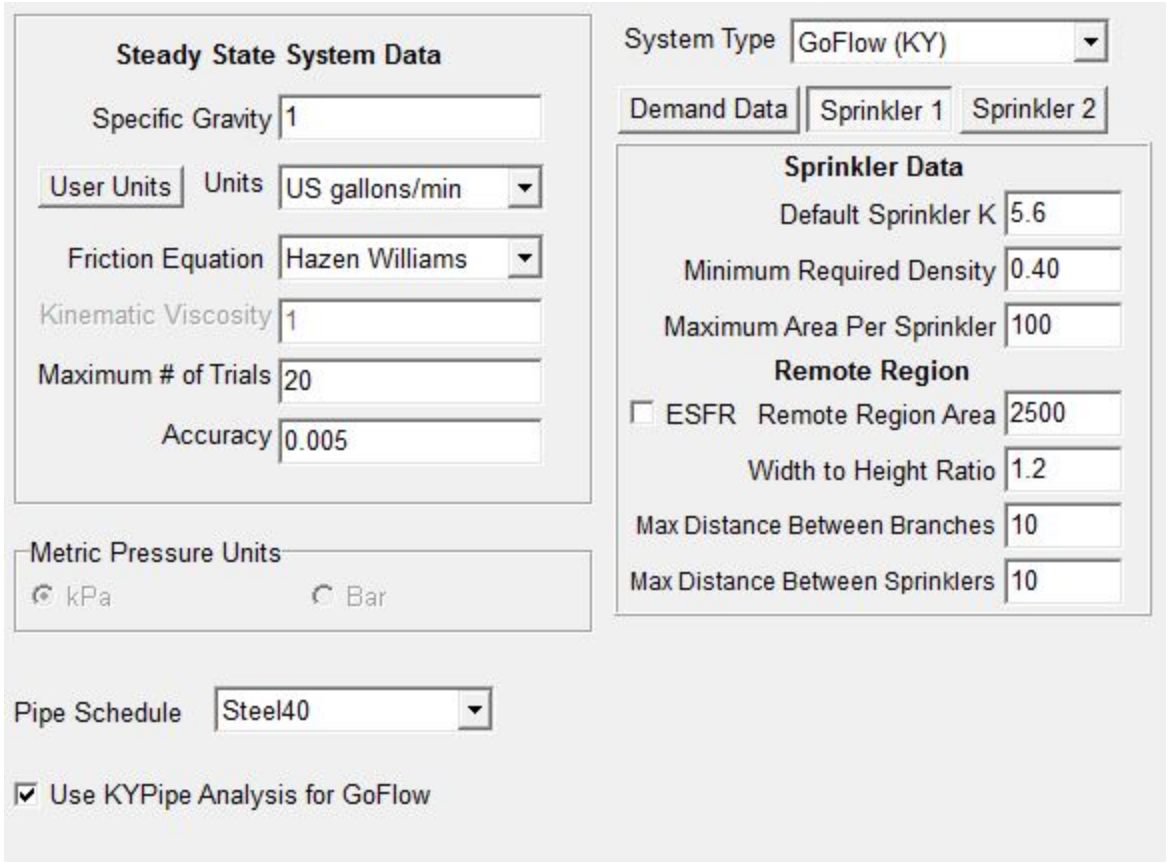

-

**Step-by-Step Instructions:**

1. Open Pipe2018 (make sure USB hardware key is installed); click OK

2. Go to System Data icon or System Data | Simulation Spec (Classic). Use SystemType drop down box, select “GoFlow (KY)”; click Okay

3. Maximize window and enter GoFlow System Data as shown below:



*Comments:*

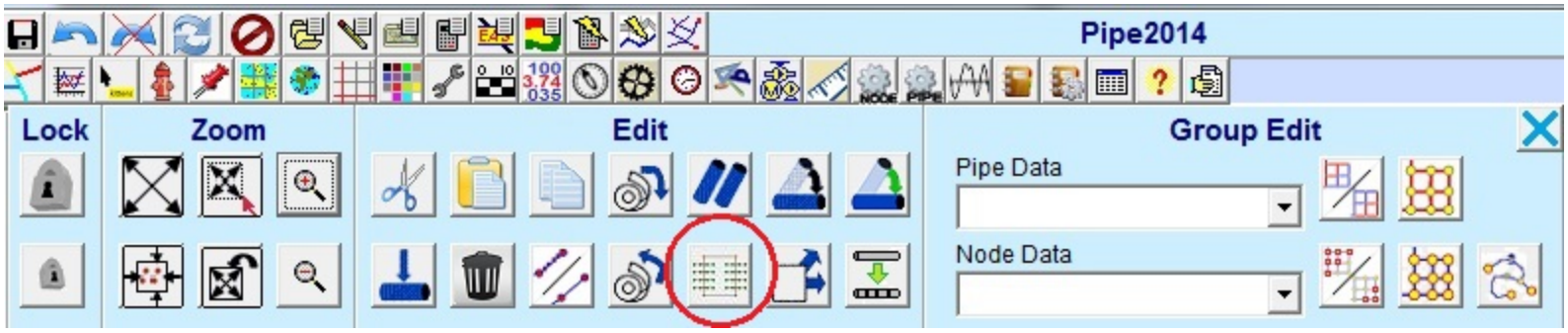
**System Total Flow Requirements** – This is where you can enter flow requirements at BOR; we will not typically use this field.

**Pump Cutoff Factor** – these fields are used only if there's a pump

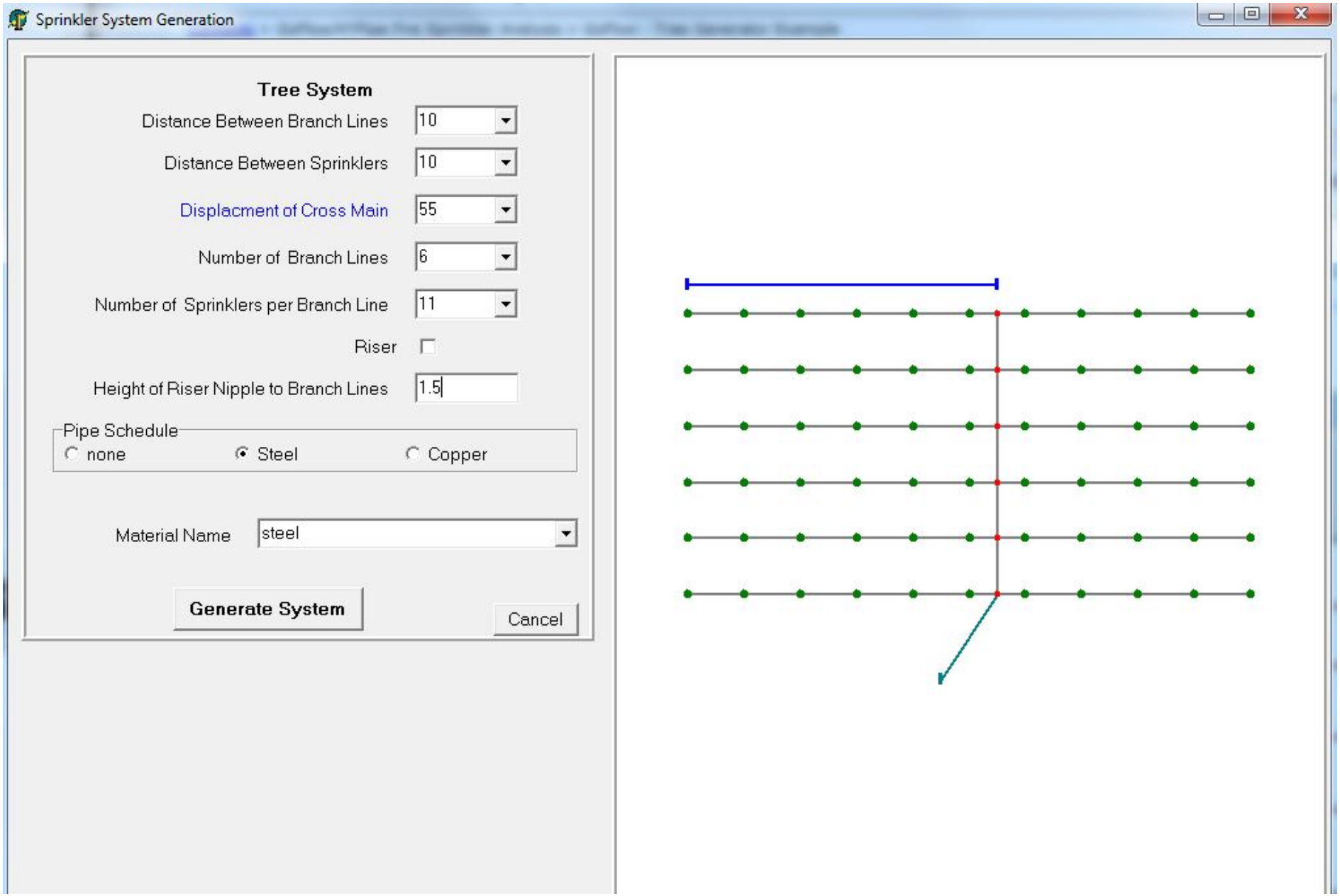
**Outside Hose is at Main Supply** – enter 500 for this example; the outside hose demand is applied at the water supply node (i.e. pump, pressure supply, tank, etc.)

**Pipe Schedule** – the default is Steel Schedule 40; the drop down box can be used to select a different pipe schedule (Steel 40 and Standard are the same pipe schedules).

4. Select "Edit", "Generate System"; select "Tree" for System Type, and "Normal".




Enter the following information based on the sketch:


Sprinkler System Generation
Tree System
Distance Between Branch Lines
10
Distance Between Sprinklers
10
Displacment of Cross Main
55
Number of Branch Lines
6
Number of Sprinklers per Branch Line
11
Riser
Height of Riser Nipple to Branch Lines
1.5
Pipe Schedule
none
Steel
Copper
Material Name
steel
Generate System
Cancel

Lengths are in [ft]. Diameters are in [in].

*Comments:*

**Displacement of cross main (X1)** – distance measured from cross main to last sprinkler on left side (55 ft. in this example)

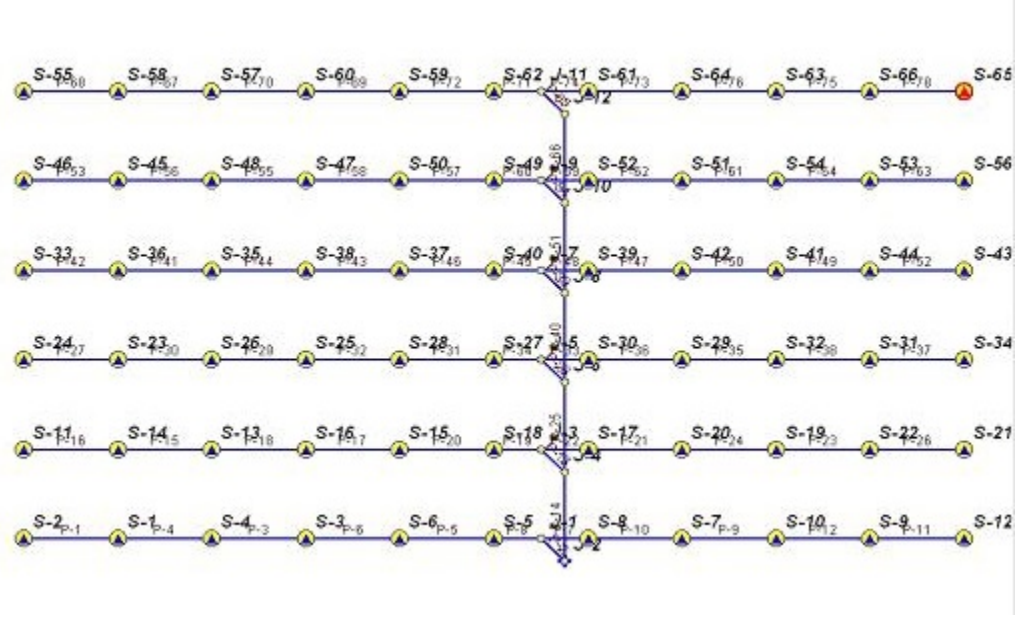
**Number of sprinklers / branch line** – total number of sprinklers on both sides of cross main (11 in this example)

**Number of branch lines -** 6 in this example (only need to include those in remote area)

**Height of riser nipple to branch lines** – 1.5 ft. in this example

5. Click “Preview”; this is where you can check to make sure the system information was entered correctly.

6. Click “Generate System”, then “Map” tab and you should see the following:



*Comments:* GoFlow assigns node values slightly different each time a system is generated and your nodes may not be labeled exactly as shown in the figure above. For example, node J-1 may be labeled node J-2, and vice-versa, or the sprinkler nodes may be slightly different. This will not affect the system's analysis.

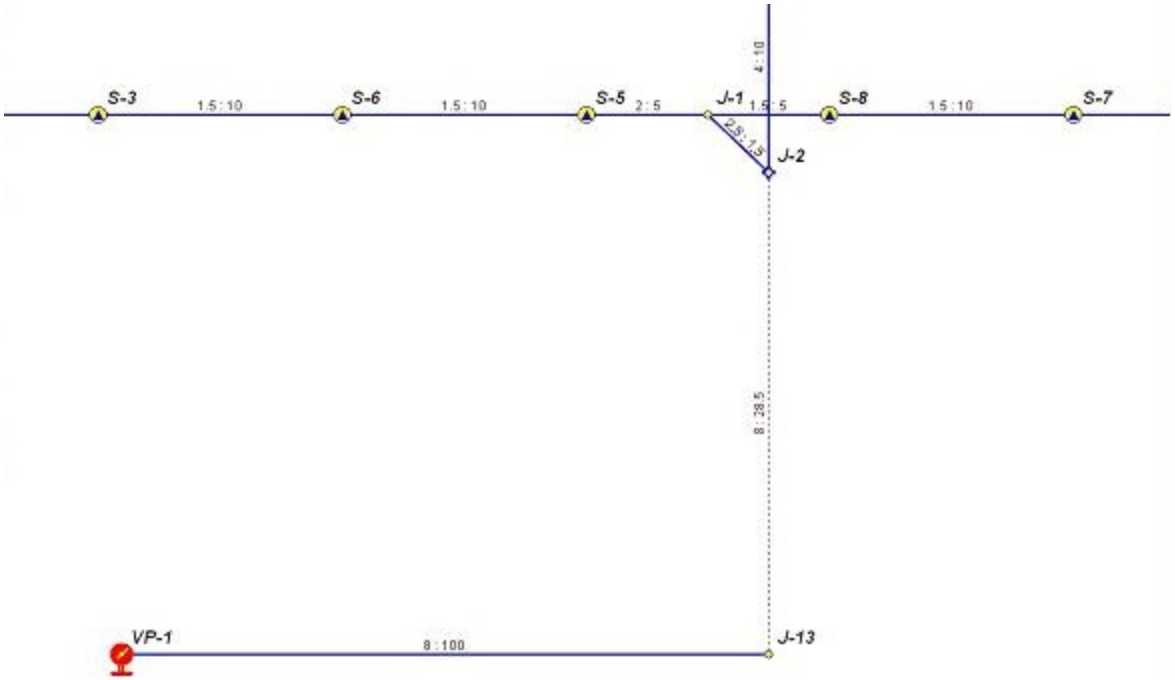
7. Enter piping & nodes from sprinkler system to water supply:

1. Left click (LC) on node J-2 (this will be the top of riser (TOR) node)
2. Move mouse directly below node J-2 and right click (RC) to enter node J-13 and pipe connecting node J-2 and J-13.
3. Move mouse to left of node J-13 and RC to enter node J-14 and connecting pipe.
4. Change node J-14 from a junction node to a pressure supply (i.e. public water supply) by LC on the drop down box in the Node Information window on right side of screen, and select "Pressure Supply".
5. Enter pressure supply information – Static Pr = 100; Res Pr = 78; Res flow = 1250 gpm (leave Elevation and Gauge Dif fields as 0).

8. Revise diameter & length of pipes from Pressure Supply (node VP-1) to TOR (node J-2):

1. Click on "Labels", "Pipe Diameter & Length" to display the current pipe diameters & lengths.
2. LC on pipe between nodes VP-1 and J-13.
3. In Pipe Information window, enter "8" for diameter, "100" for length, and check one box to the right of Gate valv in the Fittings window.
4. LC on pipe between nodes J-13 and J-2.
5. In Pipe Information window, enter "8" for diameter, "28.5" for length, and check Riser box. Check one box to the right of Alarm C\, one box for 90 elbow, and one box for Gate valv in the Fittings window.
6. Click on "Edit" and "Apply" to save all edits completed up this point.

The water supply piping should look as follows:

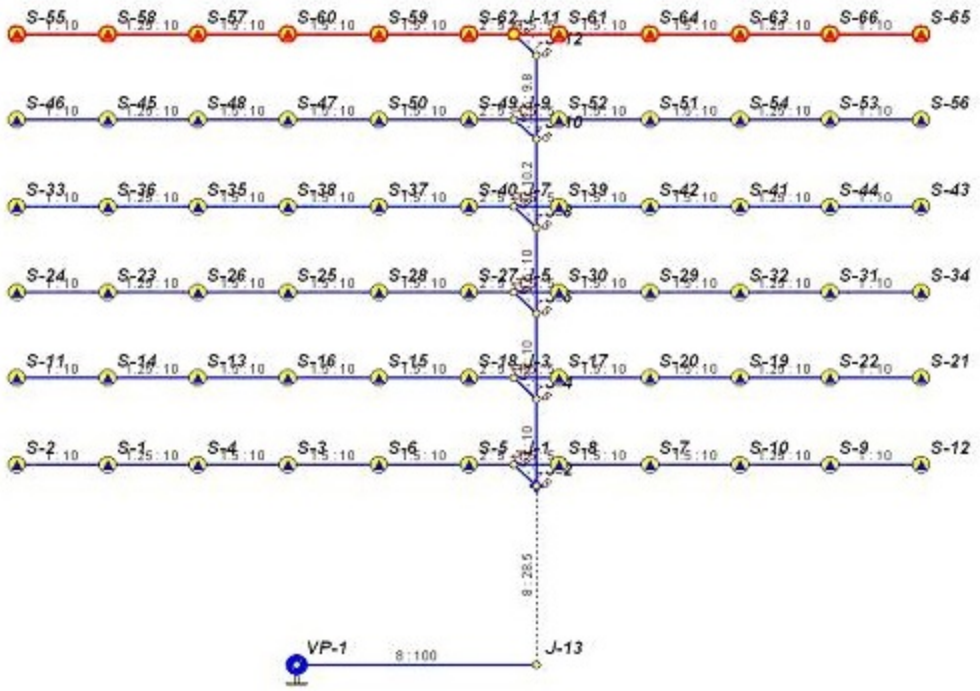



9. Revise the elevations of the cross main and branch line nodes:

1. LC on node J-2; enter 28.5 in Elevation on Node Information window.
2. LC on each cross main node (J-4, J-6, J-8, etc.) to revise the elevation to 28.5 as described above.

To change the branch line elevations, we will use the “G Box” selection feature:

c. LC on G Box on left side of screen. Draw a box around all of the sprinklers on the upper most branch line by holding down LC, and dragging yellow box around sprinklers. Release LC and the selected nodes and pipes will be highlighted in red as shown below (make sure only the nodes you want selected are highlighted):

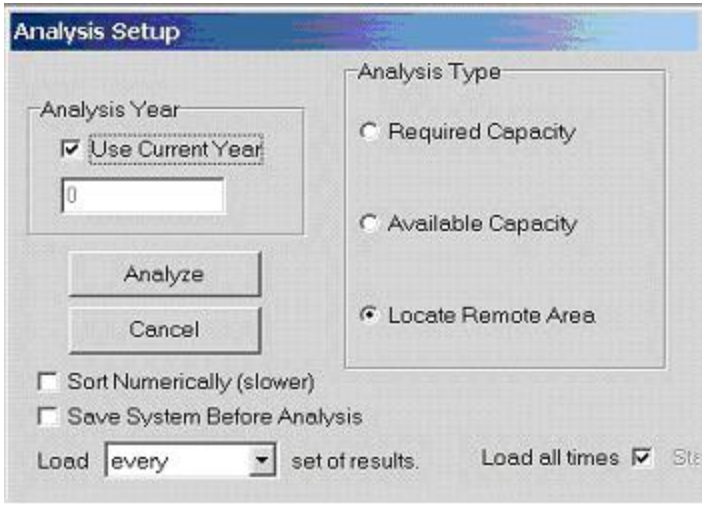



d. In the Edit Node Set window, select “Elevation” from the drop down box, enter “30” in the Value box, and LC “Proceed”. This will change the elevation of the entire branch line to 30 ft. Click “Layout” on left side of screen to exit the G Box mode.
e. Revise the elevations of the remaining 5 branch lines to 30 ft. as described above.
f. You can quickly verify all elevations are correct by LC on “Labels” and “Node Elevation”.
g. Click on “File”, Save As...” to name and save file in desired folder.

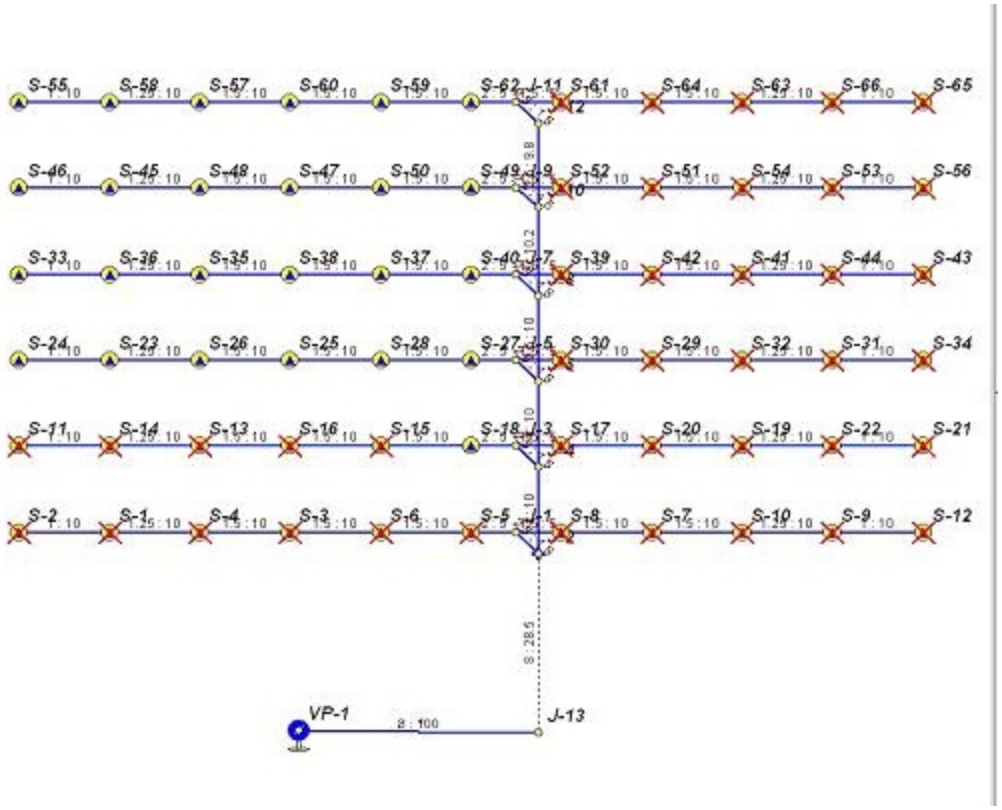
10. Conduct error checks and analysis:

1. Click on “Analyze” and “Error Check” (a window should pop up stating “No errors”).
2. Click on “Analyze” and “Connectivity Check”; click OK and LC on any pipe in the system (a window should pop up stating “System is all connected”); click OK.

3. Click on “Analyze” and “Analyze”; the following window should pop up:




d. Select “Locate Remote Area”; click “Analyze”. This will determine the remote area of the system as follow:



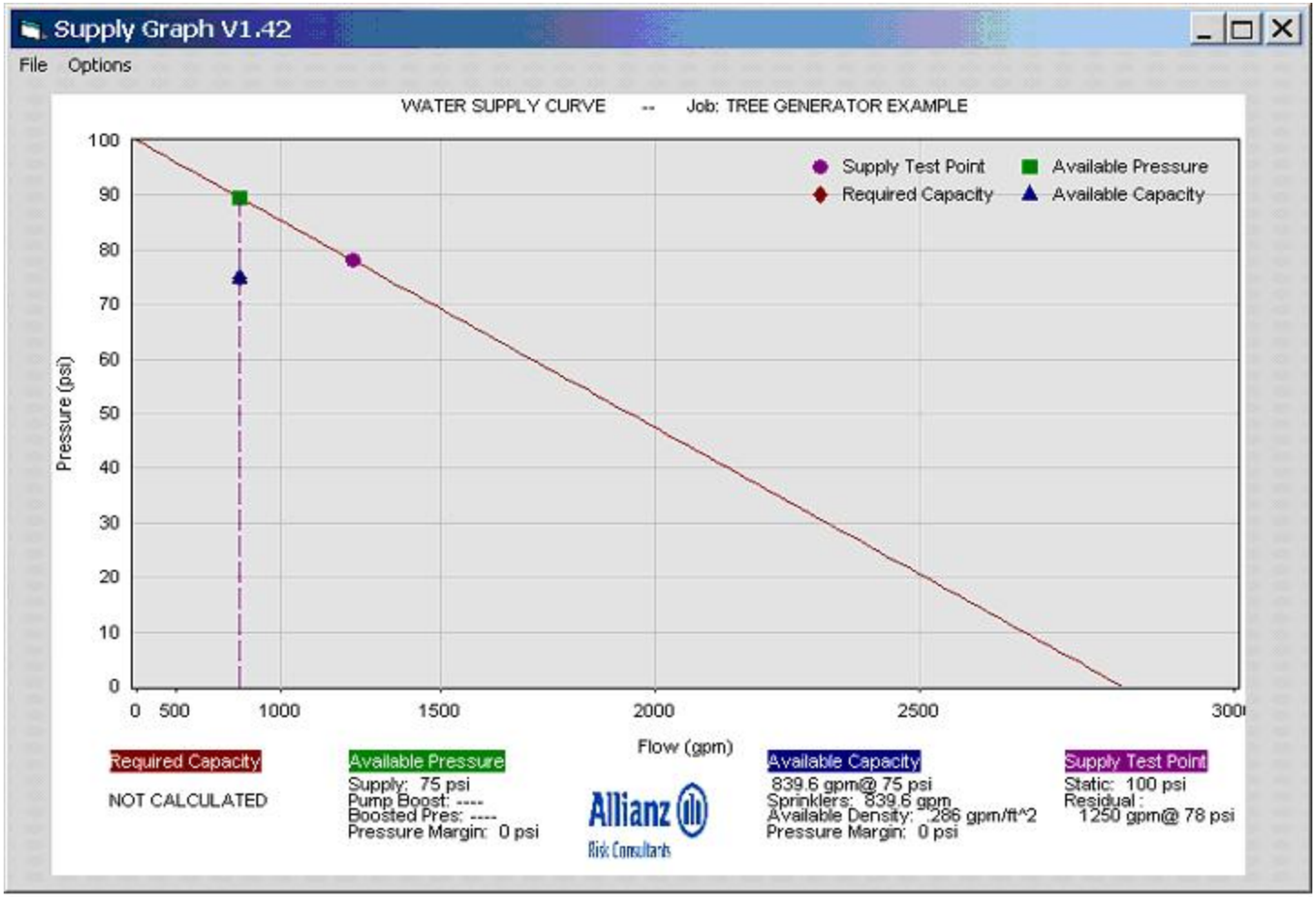

All of the sprinklers outside the remote area will be turned off to allow analysis of only the remote area sprinklers.

e. Click on “Analyze” and “Analyze”. A window will pop up; click on “Available Capacity” and “Analyze” to determine the maximum achievable density based on the water supply data entered. If you wanted to determine the water demand required to deliver the 0.40/2500, you would click on “Required Capacity” and “Analyze”.

11. Obtain results:

1. Click on “Report” tab to view calculations.
2. Click on Graphs, Tables and Profiles icon (KYnetic) or Analyze | Summary/Supply Plot (Classic) to view results and water supply curve as shown below:

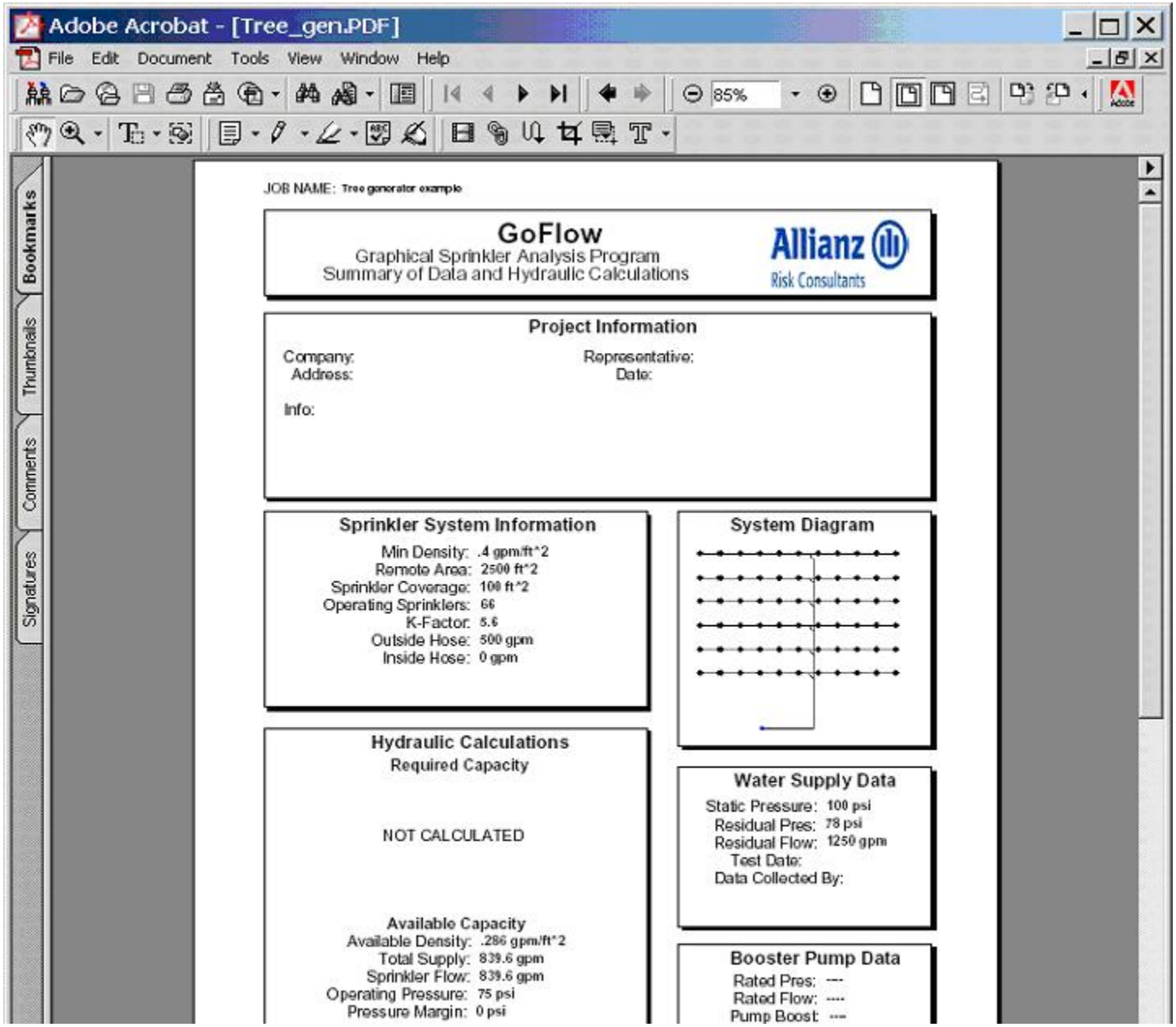
LEFT
LAST

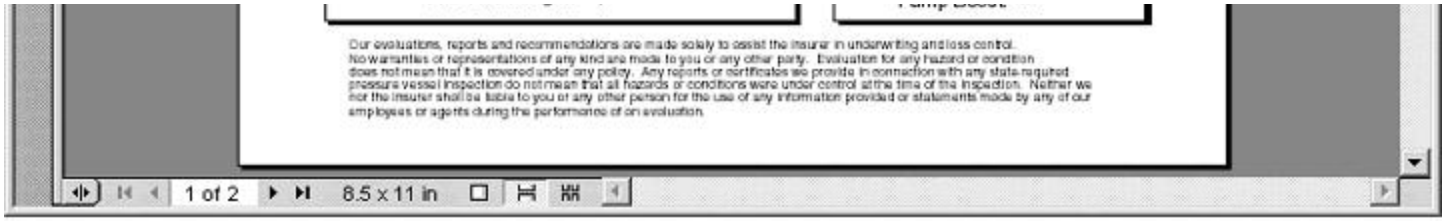



1.

c. Click on “Options”, “Supply Information” to enter water supply information and “Options”, “Project Information” to enter project information.

d. Results can be printed from the supply graph screen by clicking “File”, “Preview Report in PDF” as shown below:


Adobe Acrobat - [Tree_gen.PDF]
File Edit Document Tools View Window Help
85%
Bookmarks
Thumbnails
Comments
Signatures
JOB NAME: Tree generator example
GoFlow
Graphical Sprinkler Analysis Program
Summary of Data and Hydraulic Calculations
Allianz
Risk Consultants
Project Information
Company:
Address:
Representative:
Date:
Info:
Sprinkler System Information
Min Density: .4 gpm/ft^2
Remote Area: 2500 ft^2
Sprinkler Coverage: 100 ft^2
Operating Sprinklers: 66
K-Factor: 5.6
Outside Hose: 500 gpm
Inside Hose: 0 gpm
System Diagram
Hydraulic Calculations
Required Capacity
NOT CALCULATED
Available Capacity
Available Density: .286 gpm/ft^2
Total Supply: 839.6 gpm
Sprinkler Flow: 839.6 gpm
Operating Pressure: 75 psi
Pressure Margin: 0 psi
Water Supply Data
Static Pressure: 100 psi
Residual Pres: 78 psi
Residual Flow: 1250 gpm
Test Date:
Data Collected By:
Booster Pump Data
Rated Pres: ---
Rated Flow: ----
Pump Boost: ---




*Comments:*

Based on the results in the report (calculations or summary/supply plot), the system is only capable of delivering **0.286/2500** based on the current water supplies.

## 1.6.11 GoFlow - Grid Generator Example

GoFlow - Grid Generator Example

---

### **Grid Generator Example:**

For this example, we will use the grid generator in GoFlow to analyze a wet pipe grid. The system characteristics are as follows:

Sprinkler K-factor: 8.0
Sprinkler spacing: 10 ft.
Branch line spacing: 12 ft.
Pipe schedule: Schedule 10
Near main diameter: 6 in.
Far main diameter: 4 in.
Branch line diameter: 2 in.

Other information is as follows:

Required design: 0.30/2000 + 250 gpm outside hose
Public water supply: Static – 75 psi

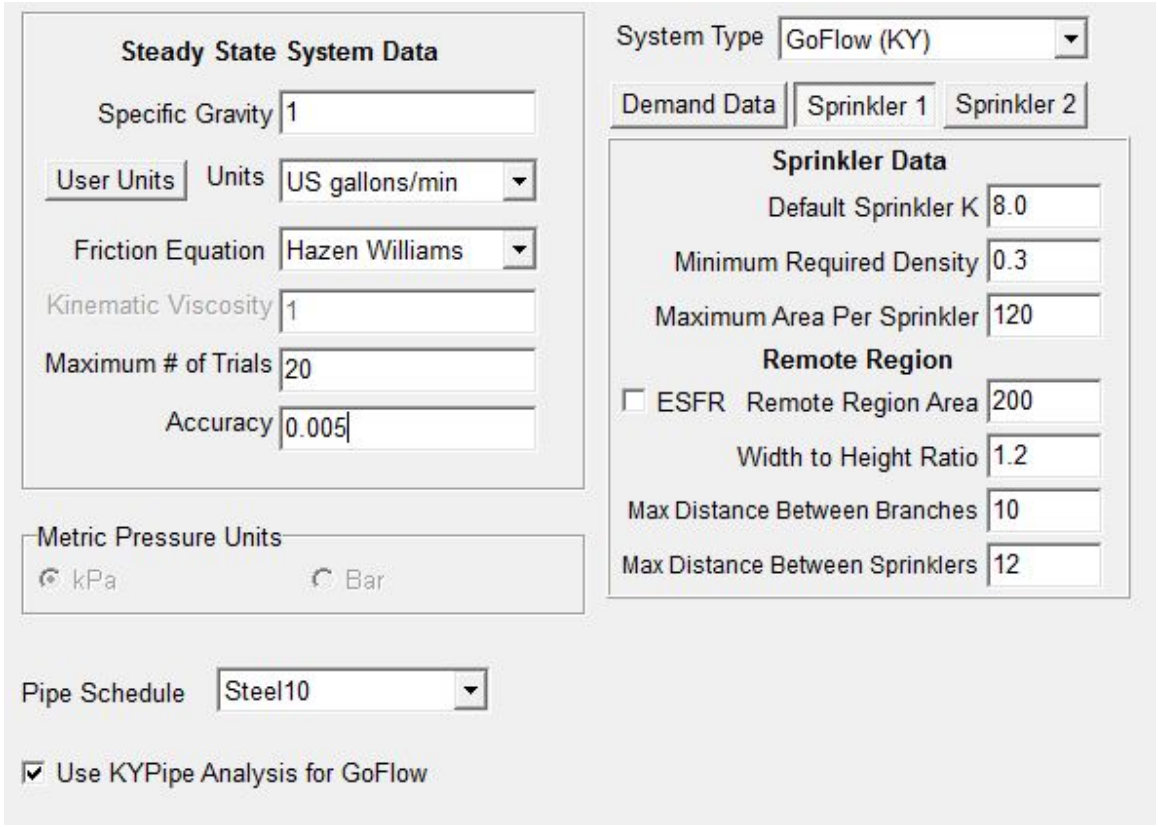
Residual – 62 psi
Flow – 1050 gpm

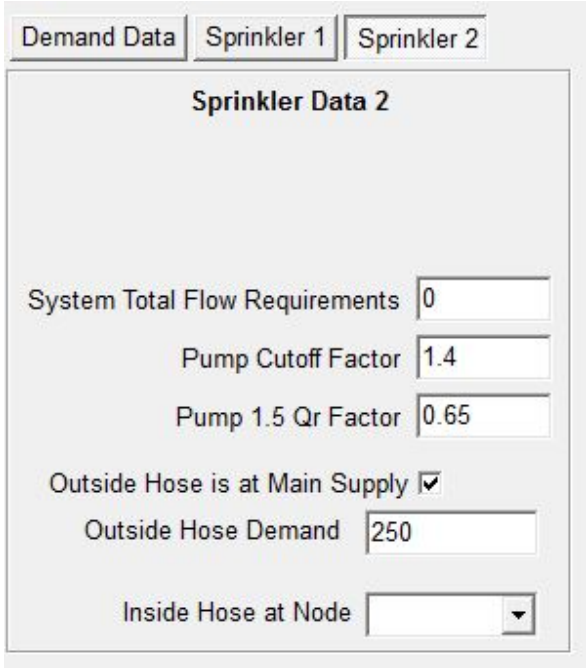
**Sprinkler System Sketch:**

**Step-by-Step Instructions:**

1. Open Pipe2018 (make sure USB hardware key is installed); click OK
2. 2. Go to System Data icon or System Data | Simulation Spec (Classic). Use SystemType drop down box, select “GoFlow (KY)”; click Okay
3. Maximize window and enter GoFlow System Data as shown below:

4.


Steady State System Data
Specific Gravity 1
User Units
Units US gallons/min
Friction Equation Hazen Williams
Kinematic Viscosity 1
Maximum # of Trials 20
Accuracy 0.005
Metric Pressure Units
kPa
Bar
Pipe Schedule Steel10
Use KYPipe Analysis for GoFlow
System Type GoFlow (KY)
Demand Data
Sprinkler 1
Sprinkler 2
Sprinkler Data
Default Sprinkler K 8.0
Minimum Required Density 0.3
Maximum Area Per Sprinkler 120
Remote Region
ESFR
Remote Region Area 200
Width to Height Ratio 1.2
Max Distance Between Branches 10
Max Distance Between Sprinklers 12




*Comments:*

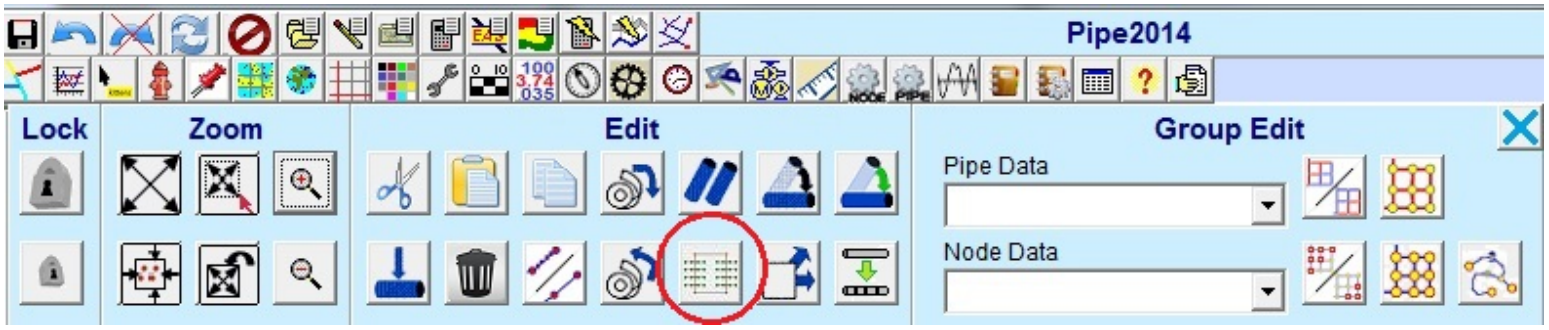
**System Total Flow Requirements** – This is where you can enter flow requirements at BOR; we will not typically use this field.

**Pump Cutoff Factor** – these fields are used only if there's a pump

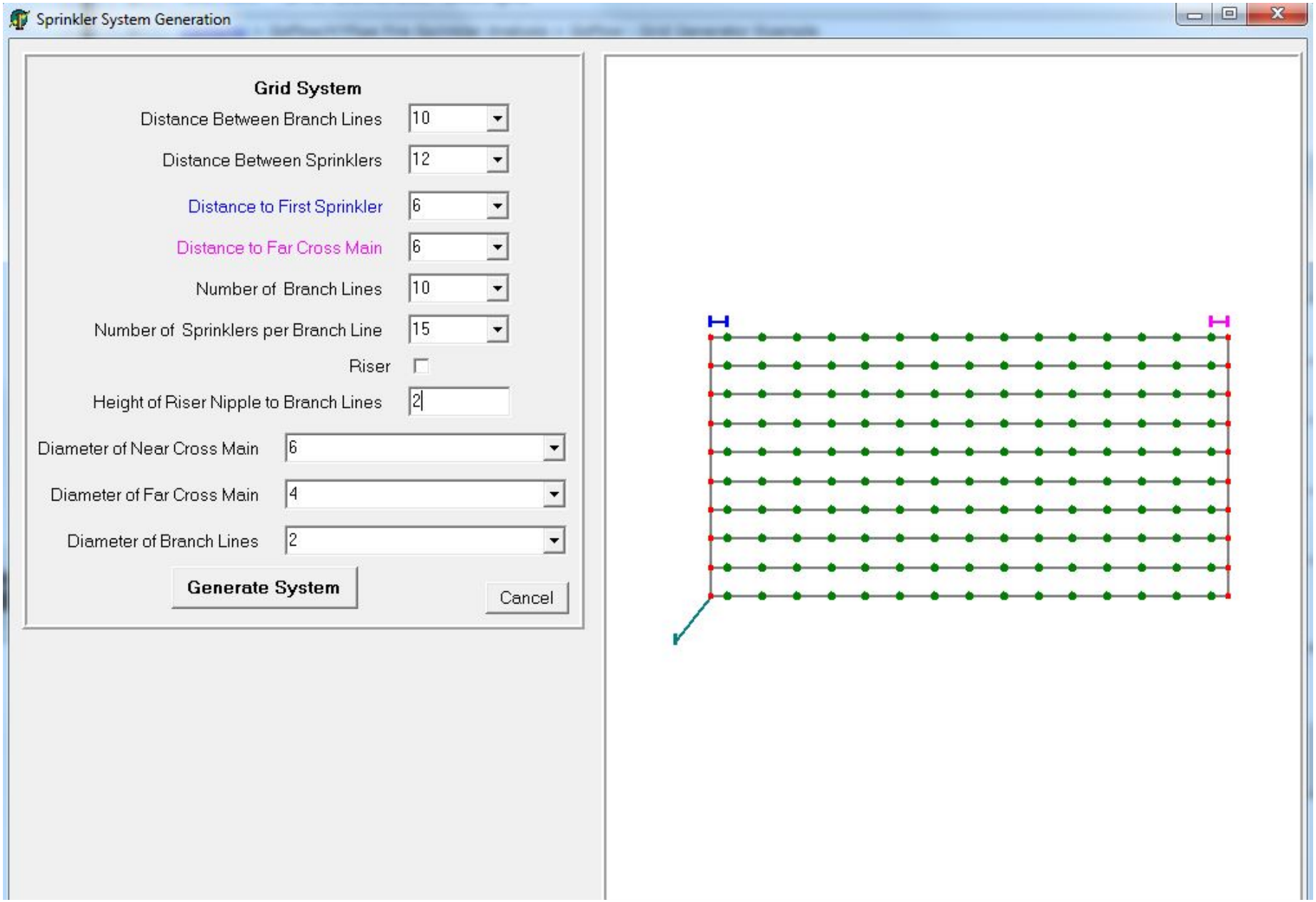
**Outside Hose is at Main Supply** – enter 250 for this example; the outside hose demand is applied at the water supply node (i.e. pump, pressure supply, tank, etc.)

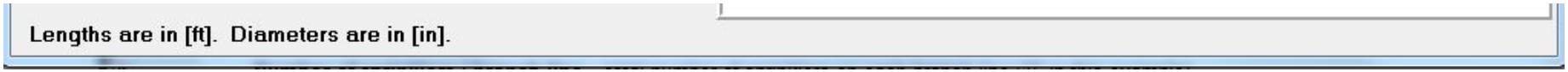
**Pipe Schedule** – the default is Steel Schedule 40; use the drop down box to select Steel Schedule 10 for this example.

d. Select "Edit", "Generate System"; select "Grid" for System Type, and "Normal".




Enter the following information based on the sketch:


Sprinkler System Generation
Grid System
Distance Between Branch Lines
10
Distance Between Sprinklers
12
Distance to First Sprinkler
6
Distance to Far Cross Main
6
Number of Branch Lines
10
Number of Sprinklers per Branch Line
15
Riser
Height of Riser Nipple to Branch Lines
2
Diameter of Near Cross Main
6
Diameter of Far Cross Main
4
Diameter of Branch Lines
2
Generate System
Cancel




*Comments:*

**Distance to first sprinkler (X1)** – distance measured from near cross main to first sprinkler (6 ft. in this example)

**Distance to terminating cross main (X2)** – distance measured from far cross main to first sprinkler (6 ft. in this example)

**Number of sprinklers / branch line** – total number of sprinklers on each branch line (15 in this example)

**Number of branch lines –** 10 in this example

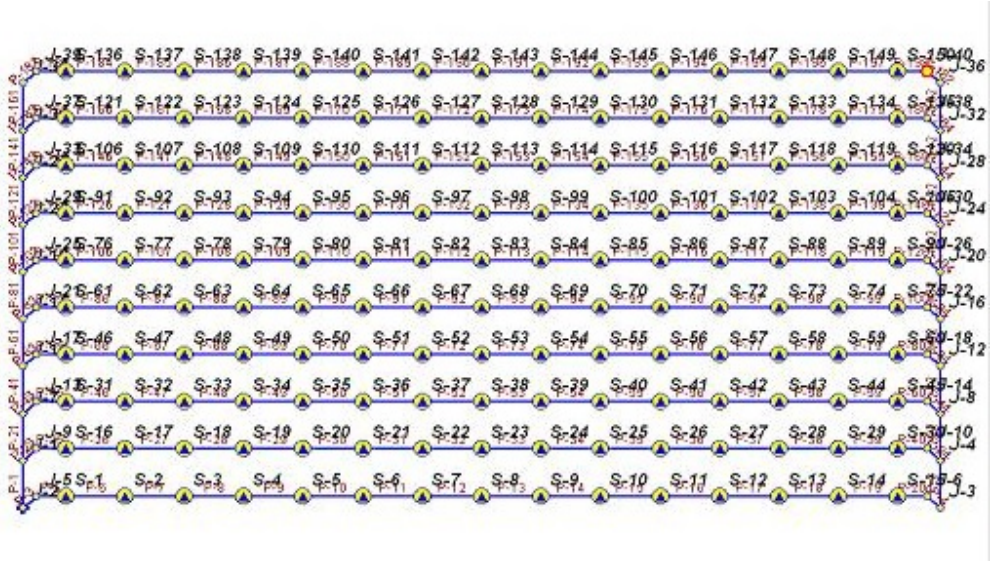
**Diameter of near cross main –** 6 in. in this example

**Diameter of far cross main –** 4 in. in this example

**Height of riser nipple to branch lines** – 2 ft. in this example

**Diameter of branch lines –** 2 in. in this example

5. Click “Preview”; this is where you can check to make sure the system information was entered correctly.
6. Click “Generate System”, then “Map” tab and you should see the following:



*Comments:* GoFlow assigns node values slightly different each time a system is generated and your nodes may not be labeled exactly as shown in the figure above. This will not affect the system's analysis.

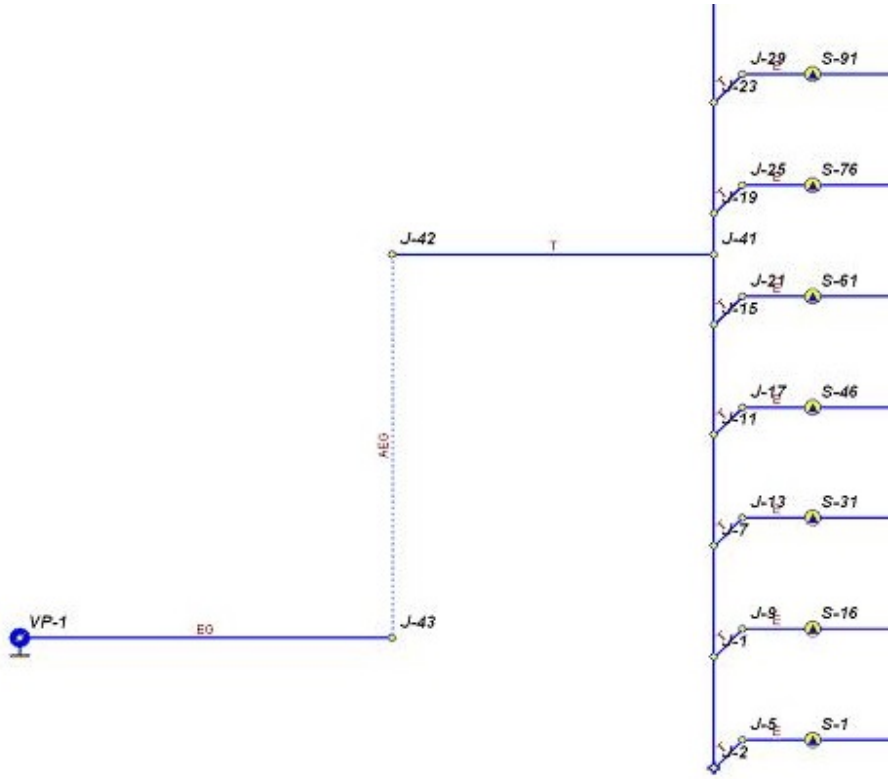
7. Enter piping & nodes from sprinkler system to water supply:

1. Left click (LC) on pipe P-81 located on left cross main (this will highlight the pipe in red).
2. In Pipe Information window on the right side of the screen, click on "Insrt"; this will insert an intermediate node in pipe P-81 (this will be our point of connection at the near cross main).
3. Use the horizontal scroll bar at the bottom of the screen to move the near main to the center of the screen (this will allow you to more easily view the piping that will be entered to the water supply).
4. LC on new intermediate node.
5. Move mouse directly left of the intermediate node and right click (RC) to enter node J-42 and pipe connecting node J-42 and intermediate node (J-41).
6. Move mouse directly down from node J-42 and RC to enter node J-43 and connecting pipe (this pipe will be the riser).
7. Move mouse directly left of node J-43 and RC to enter node J-44 and connecting pipe.
8. Change node J-44 from a junction node to a pressure supply (i.e. public water supply) by LC on the drop down box in the Node Information window on right side of screen, and select "Pressure Supply".
9. Enter pressure supply information – Static Pr = 75; Res Pr = 62; Res flow = 1050 gpm (Elevation and Gauge Dif fields should be 0).

8. Revise diameter & length of pipes from Pressure Supply (node VP-1) to point of connection to near main (node J-41):

1. Click on “Labels”, “Pipe Diameter & Length” to display the current pipe diameters & lengths.
2. LC on pipe between nodes VP-1 and J-43.
3. In Pipe Information window, enter “8” for diameter, “200” for length, check one box to the right of Gate valv, and one box to the right of 90 elbow in the Fittings window.
4. LC on pipe between nodes J-43 and J-42.
5. In Pipe Information window located in upper right corner of screen, enter “8” for diameter, “33” for length, and check Riser box. Check one box to the right of Alarm C\, one box for 90 elbow, and one box for Gate valv in the Fittings window.
6. LC on pipe between nodes J-42 and J-41.
7. In Pipe Information window located in upper right corner of screen, enter “6” for diameter and “15” for length. Check one box to the right of Tee in the Fittings window.
8. Click on “Labels”, “All Labels Off”, and “Pipe Fittings” to display the current pipe fittings.
9. Click on “Labels”, “Node Name” to also display the nodes.
10. Click on “Edit” and “Apply” to save all edits completed up this point.

The water supply piping should look as follows:



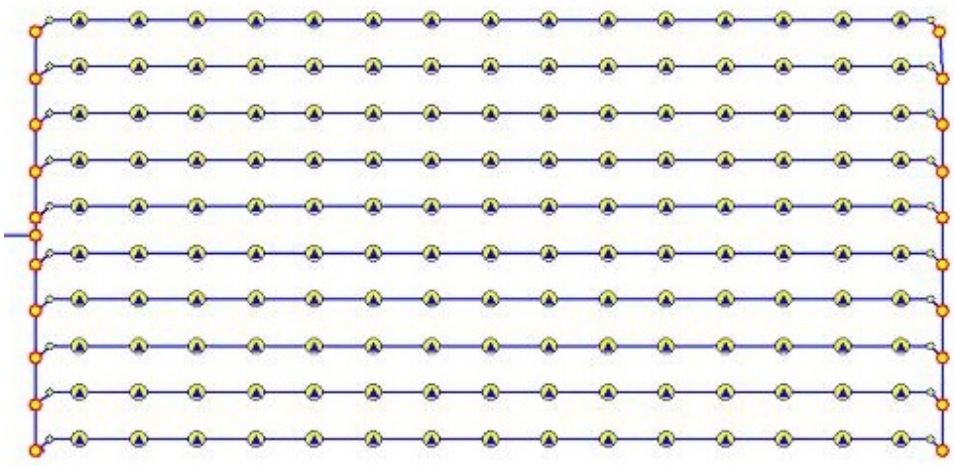

9. Revise the lengths of the near main pipes connected to the intermediate node:

1. LC on pipe between nodes J-41 and J-19.
2. In Pipe Information window, enter “6” for length.
3. LC on pipe between nodes J-41 and J-15.
4. In Pipe Information window, enter “6” for length

10. Revise the elevations of the cross main and branch line nodes:

1. Click on “Labels”, “All Labels Off” to clean up the screen.

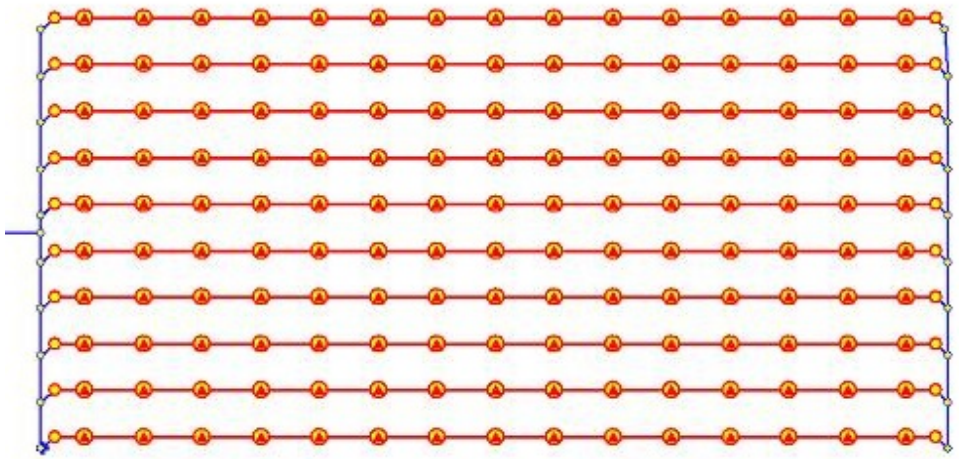
2. LC on Group on left side of screen
3. LC on all nodes located at bottom of riser nipples on near and far mains; each node selected will be highlighted in red as shown below (make sure only the nodes you want selected are highlighted):



d. In the Edit Node Set window, select “Elevation” from the Item to Edit drop down box, enter “33” in the Value box, LC “Proceed”, and click OK. The selected nodes will be highlighted in red as shown below (make sure only the nodes you want selected are highlighted):This will change the elevation of the highlighted nodes to 30 ft. Click “Layout” on left side of screen to exit the Group mode.

To change the branch line elevations, we will use the “G Box” selection feature:

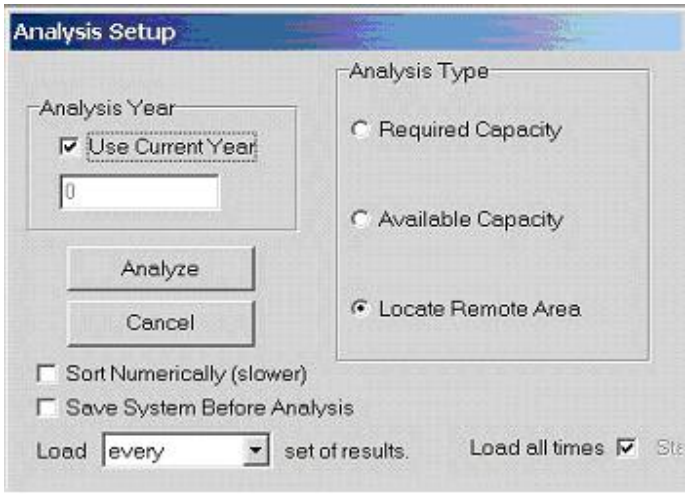
5. LC on G Box on left side of screen. Draw a box around all of the sprinklers and top of riser nipples by holding down LC, and dragging the yellow box around these nodes. Release LC and the selected nodes and pipes will be highlighted in red as shown below (make sure only the nodes you want selected are highlighted):



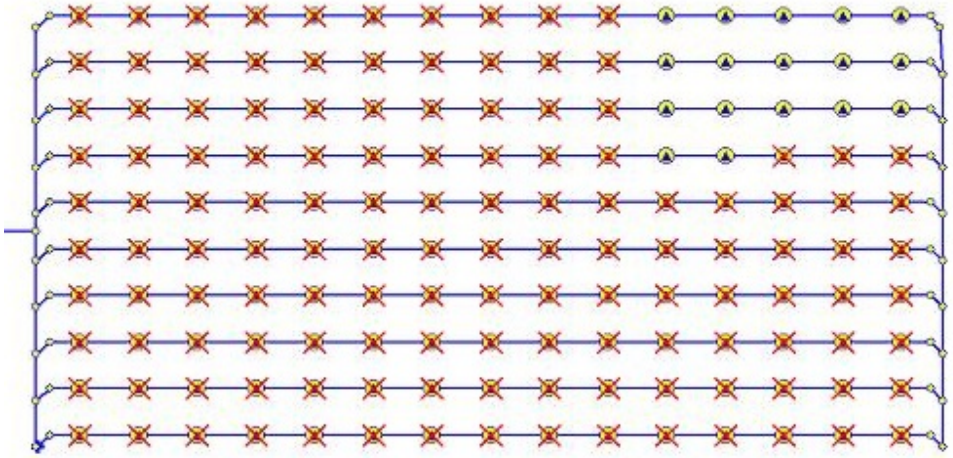
f. In the Edit Node Set window, select “Elevation” from the drop down box, enter “35” in the Value box, LC “Proceed”, click OK. This will change the elevation of all branch lines to 35 ft. Click “Layout” on left side of screen to exit the G Box mode.
g. You can quickly verify all elevations are correct by LC on “Labels” and “Node Elevation”.
h. Click on “Edit” and “Apply” to save all edits completed up this point.
i. Click on “File”, Save As...” to name and save file in desired folder.

11. Conduct error checks and analysis:

1. Click on “Analyze” and “Error Check” (a window should pop up stating “No errors”).
2. Click on “Analyze” and “Connectivity Check”; click OK and LC on any pipe in the system (a window should pop up stating “System is all connected”); click OK.
3. Click on “Analyze” and “Analyze”; the following window should pop up:




d. Select “Locate Remote Area”; click “Analyze”. This will determine the remote area of the system as follows:

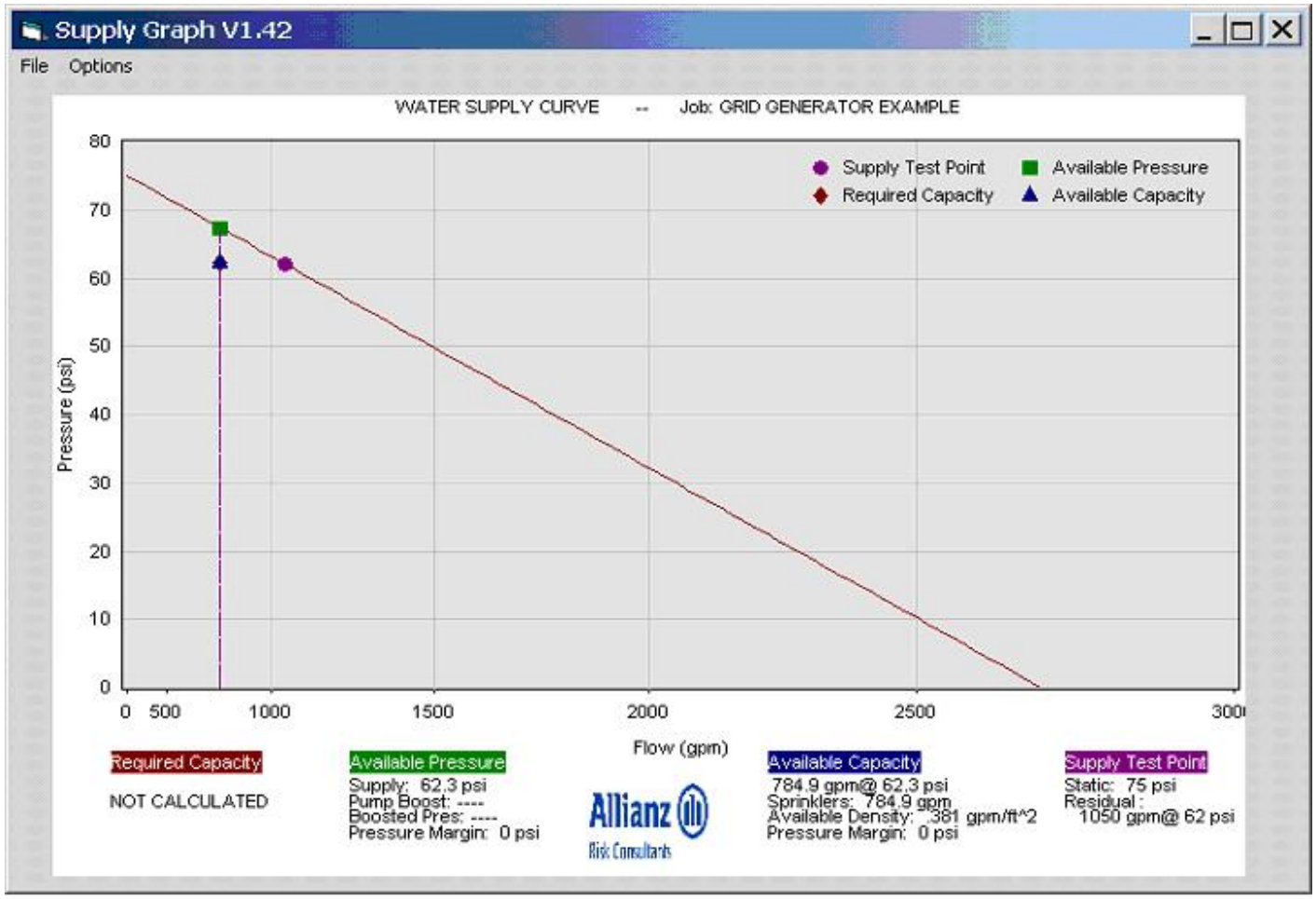


All of the sprinklers outside the remote area will be turned off to allow analysis of only the remote area sprinklers.

e. Click on “Analyze” and “Analyze”. A window will pop up; click on “Available Capacity” and “Analyze” to determine the maximum achievable density based on the water supply data entered. If you wanted to determine the water demand required to deliver the 0.32/2000, you would click on “Required Capacity” and “Analyze”.

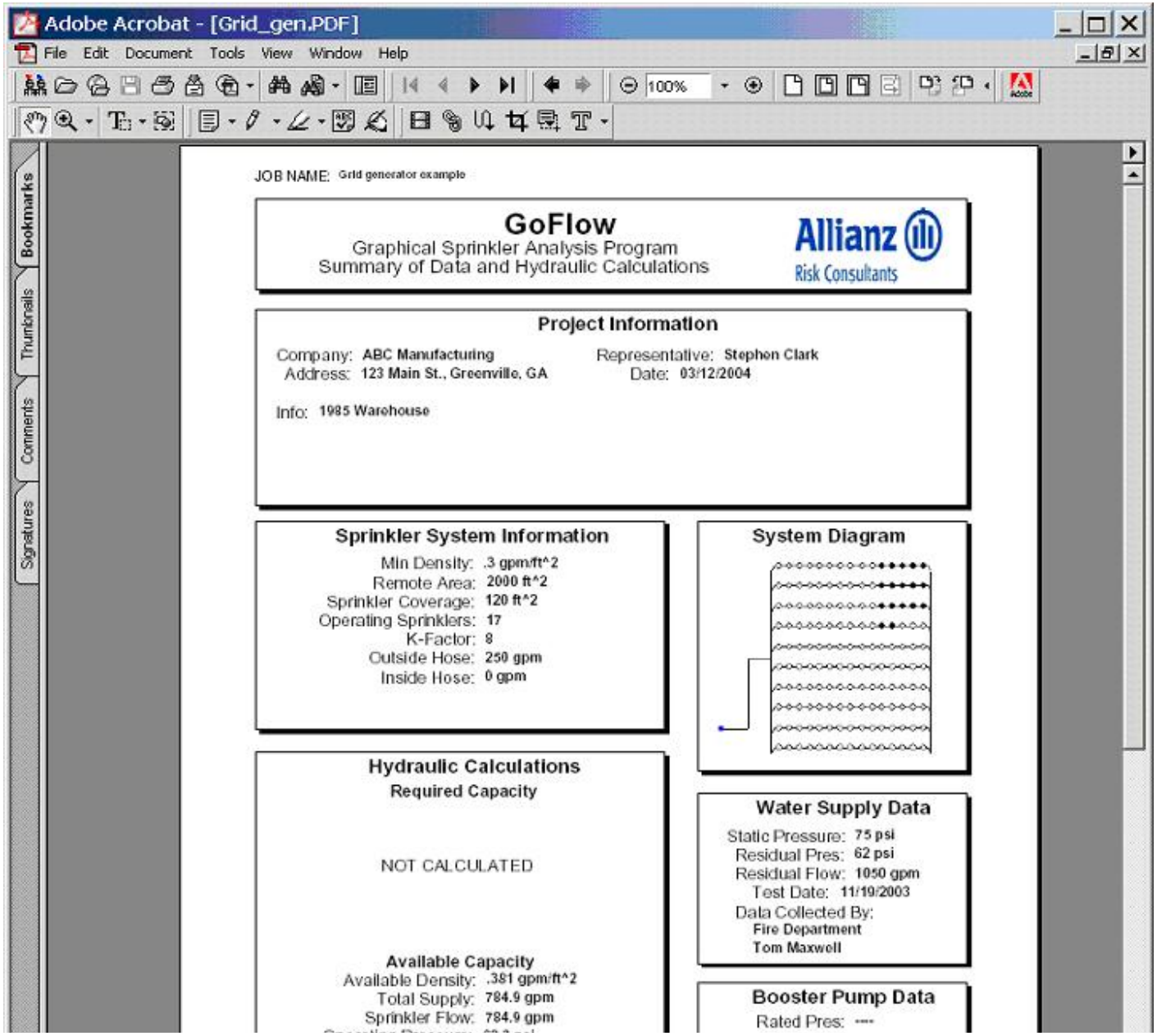
12. Obtain results:

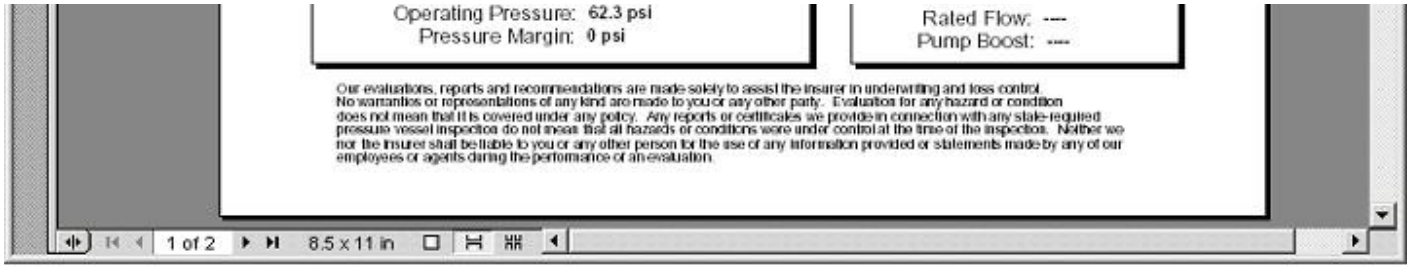
1. Click on “Report” tab to view calculations.
2. Click on “Analyze” and “Summary/Supply Plot” to view results and water supply curve as shown below:




c. Click on “Options”, “Supply Information” to enter water supply information and “Options”, “Project Information” to enter project information.

d. Results can be printed from the supply graph screen by clicking “File”, “Preview Report in PDF” as shown below:


Adobe Acrobat - [Grid_gen.PDF]
File Edit Document Tools View Window Help
100%
Bookmarks
Thumbnails
Comments
Signatures
JOB NAME: Grid generator example
GoFlow
Graphical Sprinkler Analysis Program
Summary of Data and Hydraulic Calculations
Allianz
Risk Consultants
Project Information
Company: ABC Manufacturing
Address: 123 Main St., Greenville, GA
Representative: Stephen Clark
Date: 03/12/2004
Info: 1985 Warehouse
Sprinkler System Information
Min Density: .3 gpm/ft^2
Remote Area: 2000 ft^2
Sprinkler Coverage: 120 ft^2
Operating Sprinklers: 17
K-Factor: 8
Outside Hose: 250 gpm
Inside Hose: 0 gpm
System Diagram
Hydraulic Calculations
Required Capacity
NOT CALCULATED
Available Capacity
Available Density: .381 gpm/ft^2
Total Supply: 784.9 gpm
Sprinkler Flow: 784.9 gpm
Water Supply Data
Static Pressure: 75 psi
Residual Pres: 62 psi
Residual Flow: 1050 gpm
Test Date: 11/19/2003
Data Collected By:
Fire Department
Tom Maxwell
Booster Pump Data
Rated Pres: ----




*Comments:*

Based on the results in the report (calculations or summary/supply plot), the system is capable of delivering **0.381/2000** based on the current water supplies.

## 1.6.12 GoFlow - Equivalent K-factor with a Sprinkler on a Riser Nipple

GoFlow - Equivalent K-factor with a Sprinkler on a Riser Nipple

---

**Calculating an equivalent K-factor with a sprinkler on a riser nipple:**

**1.** Determine end head pressure using the formula Q = KvP. For this example, assume an end head density is 0.25 gpm/ft², sprinkler spacing is 100 ft², and sprinkler K-factor is 5.6:

End head flow: Q = density x sprinkler spacing = 0.25 x 100 = 25 gpm

End head pressure: $P = (Q/K)^2 = (25/5.6)^2 = 19.93$ psi

1. Determine friction loss through riser nipple using Hazen-Williams formula. For this example, assume the riser nipple is 1 in. diameter and 2 ft. long with C=120 (wet system):

$$p = \frac{4.52Q^{1.85}}{C^{1.85}d^{4.87}} = \frac{4.52(25)^{1.85}}{120^{1.85}1^{4.87}} = 0.25 \text{ psi/ft based on 2 ft. length, } p = 0.25(2) = 0.5 \text{ psi}$$

1. Determine new pressure by adding friction loss through riser nipple calculated in (2) to end pressure calculated in (1):

$$p = 19.93 + 0.5 = 20.43 \text{ psi}$$

1. Determine new K-factor with new pressure calculated in (3):

$$K = Q/vP = 25/v20.43 = 5.53$$

1. Use new sprinkler K-factor (5.53) in hydraulic analysis instead of 5.6.

## 1.6.13 Locate Remote Sprinkler Area

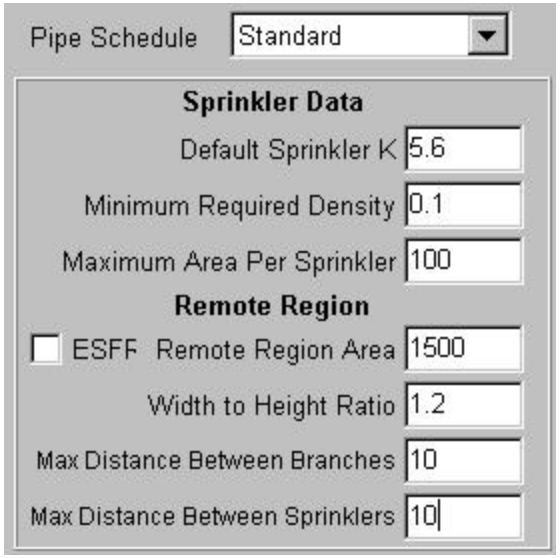
Locate Remote Sprinkler Area

---

This Help entry describes how to run a Locate Remote Area Analysis for Pipe2018 : KYPipe users. For GoFlow users, see also the GoFlow entries.

To run the analysis, go to System Data | Simulation Specs and set the relevant sprinkler data under the buttons Sprinkler 1 and Sprinkler 2. Remote Region data is required. Before running the analysis, make sure all of the sprinklers in the system are turned on. Group mode may be used to accomplish this.

Sprinkler 1 button:




**Pipe Schedule**: Select the pipe schedule to be used for the Pipe Type from the drop down list. You can add schedules to this list. Note: it is important to select or provide the appropriate schedule prior to entering data. See Pipe Type.

**Sprinkler Data**

**Default Sprinkler K**: the K factor for the principal sprinklers.

**Minimum Required Density**: the required density (in gpm/ft^2 or appropriate SI units) for the sprinklers. For Required Capacity analysis.

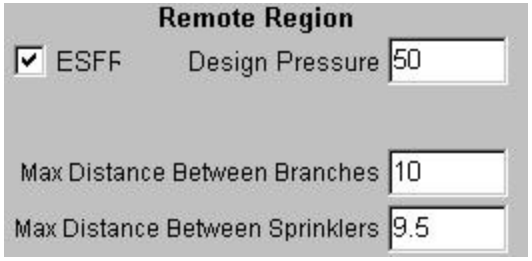
**Maximum Area Per Sprinkler**: the maximum coverage area per sprinkler. The required flow for a sprinkler is the product of this and the previous entry.

**Remote Region**

This data will be required only of the hydraulically remote area is to be determined. Most of these entries are self explanatory.

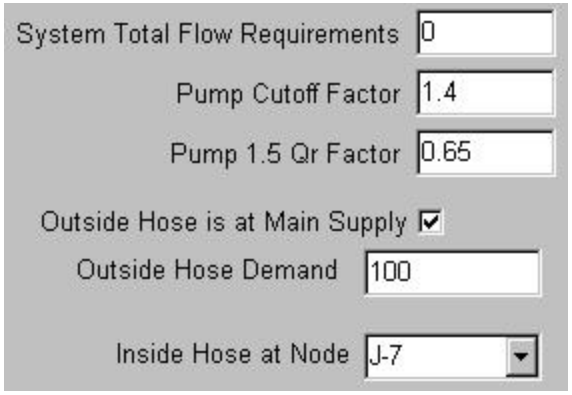
**Width to Height Ratio**: this is used to size the width and height of the remote area. For the data shown, height x 1.2 width = 1500 or height = 35.4 and width = 42.4.

**ESFR** - if this box is checked then the Remote Region parameters will be based on the following




With an ESFR (Early Suppression, Fast Response) the remote region is composed of 12 sprinklers in a 4 x 3 arrangement. A design pressure pressure is specified.

Sprinkler 2 button:




**System Total Flow Requirements**: This is an optional input for the total flow delivered to the riser. It does not affect the hydraulic calculations.

**Pump Cutoff Factor**: The pump cutoff pressure (churn) is calculated as Rated Pressure x this factor (default value is 1.4)

**Pump 1.5 Qr Factor**: This defines the pressure at 1.5 x Rated Flow as the Rated Pressure x this factor.

**Outside Hose is at Main Supply**: Check this box of the Outside Hose is located at the main supply. If this is not checked then the next box will ask you to identify the node for the Outside Hose*.

**Outside Hose Demand**: The outside hose requirement in gpm if the location is the main supply.

**Inside Hose at Node**: The node location for the primary inside hose requirement*.

*Note: the node (junction) demands will define the magnitude of the hose requirements at these nodes. These requirements can be imposed at any junction in the sprinkler system.

KYPipe users can also get a **Summary and Supply Plot.** Go to **Analyze | Summary/Supply Plot**.

To generate a **Sprinkler Report** go to **System Data | Reports** and check the box that says **Sprinkler Report**. This report will be generated and can be viewed under the Report tab.

## 1.6.14 Units GoFlow

Units GoFlow

| | English | Metric (1) | Metric (2) | Metric (3) |
|---|---|---|---|---|
| Flow Units | | | | |
| rated flow | | | | |
| hose flow | | | | |
| demand | gpm | l/s | l/min | m^3/hr |
| Pressure | | | | |
| static | | | | |
| residual | | | | |
| rated | psi | kPa | bar | bar |
| Density | gpm/ft^2 | l/s/m^2 | l/min/m^2 | m^3/hr/m^2 |
| Area | ft^2 | m^2 | m^2 | m^2 |
| Distance | | | | |
| length | ft | m | m | m |
| Elevation | | | | |
| elevation change | ft | m | m | m |
| Diameter | in | mm | mm | mm |

sprinkler constants - (flow rate in gpm (l/s))/(sprinkler pressure drop in psi (kpa))^0.5.

Example: flow = 10 gpm, pressure drop = 4 psi

K = 10 gpm/(4psi)^0.5 = 5

## 1.6.15 GoFlow

GoFlow

---

GoFlow provides a simple, fast graphical environment for designing and carrying out the hydraulic analysis of fire sprinkler systems. The pipe system can be laid out very quickly using a mouse. A number of features have been incorporated to simplify and accelerate the task of developing pipe system layouts and corresponding data files. GoFlow provides comprehensive, effective results presentation which are in accordance with NFPA 13 requirements.

GoFlow operates in the Pipe2018 graphical environment. The procedures for system layout and data entry are covered in the many audio/video tutorials. Additional tutorials are provided for GoFlow and these include

GoFlow1 - GoFlow elements

GoFlow2 - Sprinkler system layouts - 10 operations

QSP1 - QuickStart example - system layout

QSP2 - QuickStart example - data entry

QSP3 - QuickStart example - analysis and results

A QuickStart example steps you through the entire process of preparing a model and analyzing a sprinkler system. See GoFlow QuickStart.

See also GoFlow - System Data

GoFlow - Nodes

GoFlow - Generate System

GoFlow - Auto Orthogonal

## 1.6.16 Required Capacity

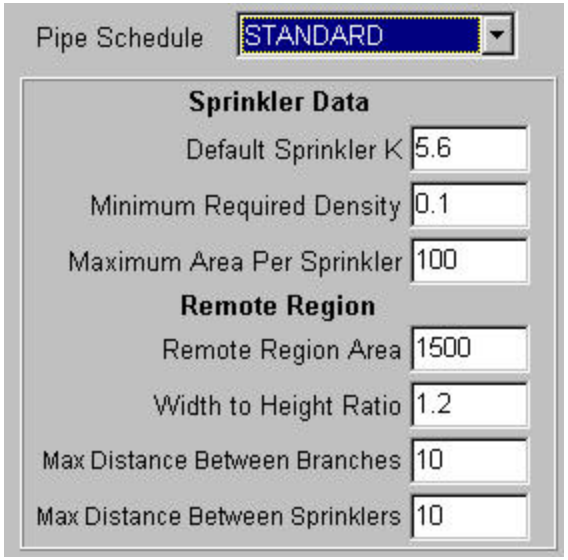
Required Capacity

This Help entry describes how to run a Required Capacity Analysis for Pipe2018 : KYPipe users. For GoFlow users, see also the GoFlow entries.

This analysis first determines which sprinkler in the system has the minimum capacity or lowest density, then analyzes the system with this element at the specified required capacity.

To run the analysis, go to System Data | Simulation Specs and set the relevant sprinkler data under the buttons Sprinkler 1 and Sprinkler 2.

Sprinkler 1 button:




**Pipe Schedule**: Select the pipe schedule to be used for the Pipe Type from the drop down list. You can add schedules to this list. Note: it is important to select or provide the appropriate schedule prior to entering data. See Pipe Type.

**Sprinkler Data**

**Default Sprinkler K**: the K factor for the principal sprinklers.

**Minimum Required Density**: the required density (in gpm/ft^2 or appropriate SI units) for the sprinklers. For Required Capacity analysis.

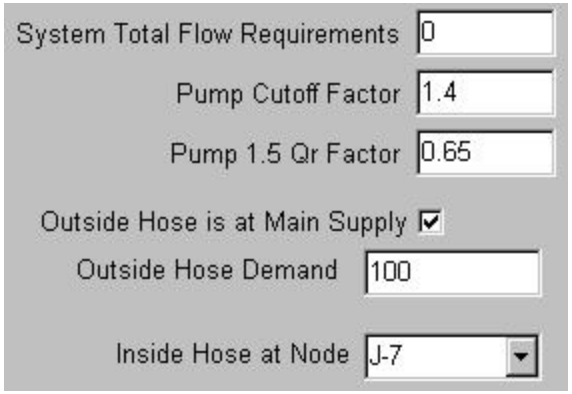
**Maximum Area Per Sprinkler**: the maximum coverage area per sprinkler. The required flow for a sprinkler is the product of this and the previous entry.

**Remote Region**

This data will be required only of the hydraulically remote area is to be determined. Most of these entries are self explanatory.

**Width to Height Ratio**: this is used to size the width and height of the remote area. For the data shown, height x 1.2 width = 1500 or height = 35.4 and width = 42.4.

Sprinkler 2 button:




**System Total Flow Requirements**: This is an optional input for the total flow delivered to the riser. It does not affect the hydraulic calculations.

**Pump Cutoff Factor**: The pump cutoff pressure (churn) is calculated as Rated Pressure x this factor (default value is 1.4)

**Pump 1.5 Qr Factor**: This defines the pressure at 1.5 x Rated Flow as the Rated Pressure x this factor.

**Outside Hose is at Main Supply**: Check this box of the Outside Hose is located at the main supply. If this is not checked then the next box will ask you to identify the node for the Outside Hose*.

**Outside Hose Demand**: The outside hose requirement in gpm if the location is the main supply.

**Inside Hose at Node**: The node location for the primary inside hose requirement*.

*Note: the node (junction) demands will define the magnitude of the hose requirements at these nodes. These requirements can be imposed at any junction in the sprinkler system.

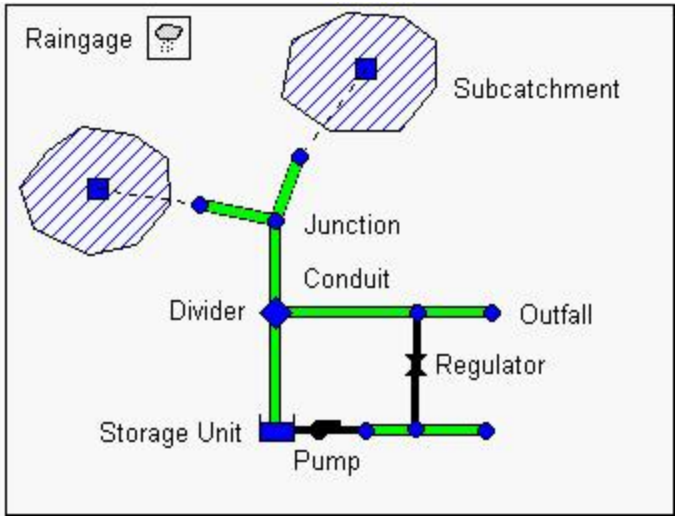
KYPipe users can also get a **Summary and Supply Plot.** Go to **Analyze | Summary/Supply Plot**.

To generate a **Sprinkler Report** go to **System Data | Reports** and check the box that says **Sprinkler Report**. This report will be generated and can be viewed under the Report tab.

## 1.8 SWMM Stormwater

### 1.7.1 SWMM (Stormwater)

# SWMM (Stormwater) Model Development




**Classic Interface only.**

Data Requirements

Collection System Data

Storm sewers, Open channels, Manholes etc.

Diversion, Storage and other Structures

Pumping stations

Outfall nodes

Inflows

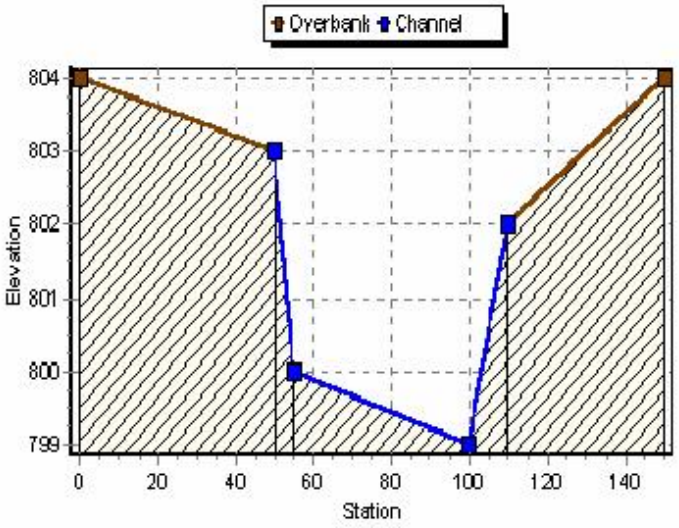
Drainage area, and Land use characteristics

Climatology - Rainfall data

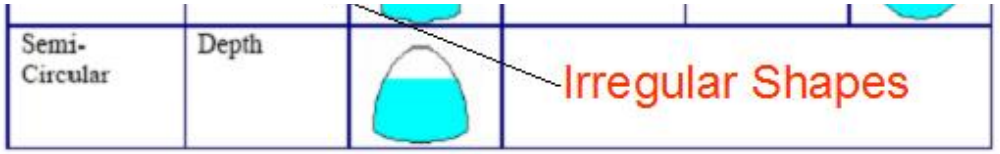
Pollutant Data

Collection System

# Storm Sewers




| Name | Parameters | Shape | Name | Parameters | Shape |
|---|---|---|---|---|---|
| Circular | Depth | | Filled Circular | Depth, Filled Depth | |
| Rectangular - Closed | Depth, Width | | Rectangular – Open | Depth, Width | |
| Trapezoidal | Depth, Base Width, Side Slopes | | Triangular | Depth, Top Width | |
| Horizontal Ellipse | Depth, Max., Width | | Vertical Ellipse | Depth, Max. Width | |
| Arch | Depth, Max. Width | | Parabolic | Depth, Top Width | |
| Power | Depth, Top Width, Exponent | | Rectangular-Triangular | Depth, Top Width, Triangle Height | |
| Rectangular-Round | Depth, Top Width, Bottom Radius | | Modified Basketehandle | Depth, Top Width | |
| Egg | Depth | | Horseshoe | Depth | |
| Gothic | Depth | | Catenary | Depth | |
| Semi-Elliptical | Depth | | Basketehandle | Depth | |

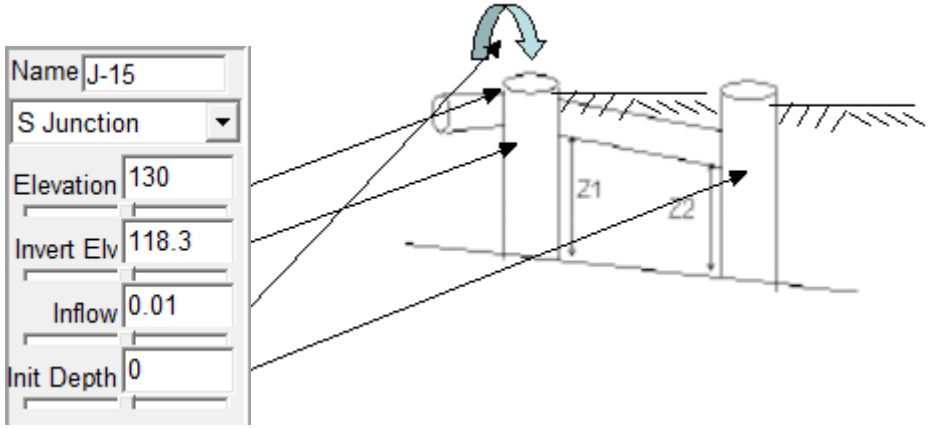



Junction Nodes - Indirect Flows

Junctions are drainage system nodes where links join together. Physically they can represent the confluence of natural surface channels, manholes in a sewer system, or pipe connection fittings. External inflows can enter the system at junctions. Excess water at a junction can become partially pressurized while connecting conduits are surcharged and can either be lost from the system or be allowed to pong atop the junction and subsequently drain back into the junction.

The principal input parameters for a junction are:

- invert elevation
- height to ground surface
- ponded surface area
- external inflow data (optional)




All elevations and depths are in feet or meters.

All areas are in sq. ft or sq. m

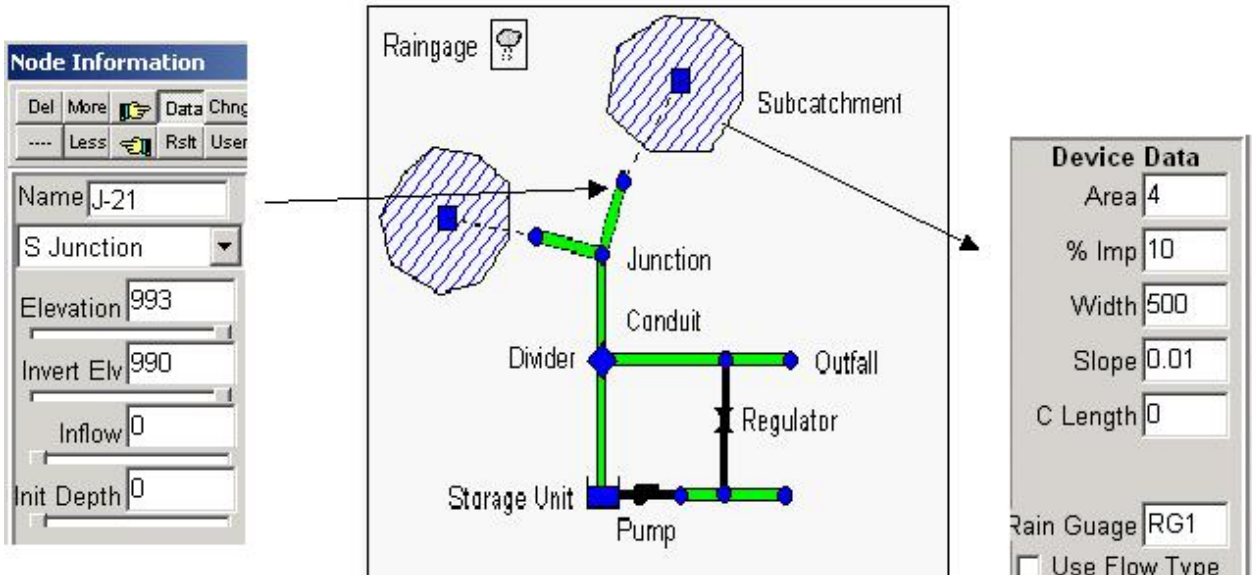
Flows are in user specified units

Inflows:

Flow Type: Should provide the associated pattern data, Direct Inflow, Dry weather Inflows, Rainfall-derived I&I

# Junction Nodes – Drainage Area

Subcatchments are hydrologic units of land whose topography and drainage system elements direct surface runoff to a single discharge point. The user is responsible for dividing a study area into an appropriate number of subcatchments, and for identifying the outlet point of each subcatchment. Discharge outlet points can be either nodes of the drainage system or other subcatchments.



# Outfall Node

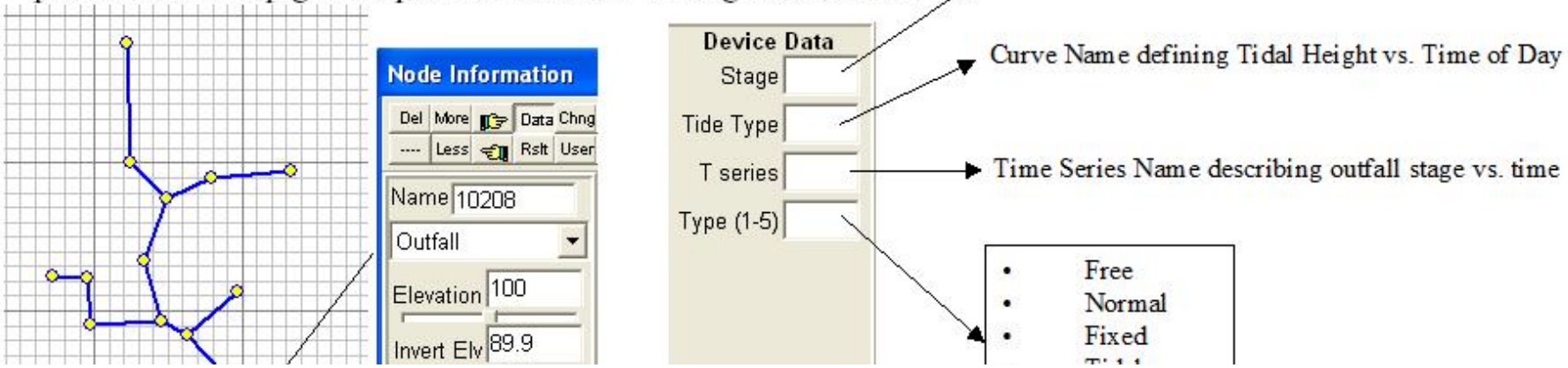
Outfalls are terminal nodes of the drainage system used to define final downstream boundaries under Dynamic Wave flow routing. For other types of flow routing they behave as a junction. Only a single link can be connected to an outfall node.

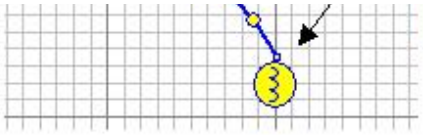
The boundary conditions at an outfall can be described by any one of the following stage relationships:

- the critical or normal flow depth in the connecting conduit
- a fixed stage elevation
- a tidal stage described in a table of tide height versus hour of the day
- a user-defined time series of stage versus time.

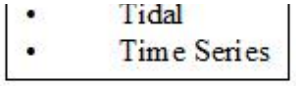
The principal input parameters for outfalls include:

- invert elevation
- boundary condition type and stage description
- presence of a flap gate to prevent backflow through the outfall.



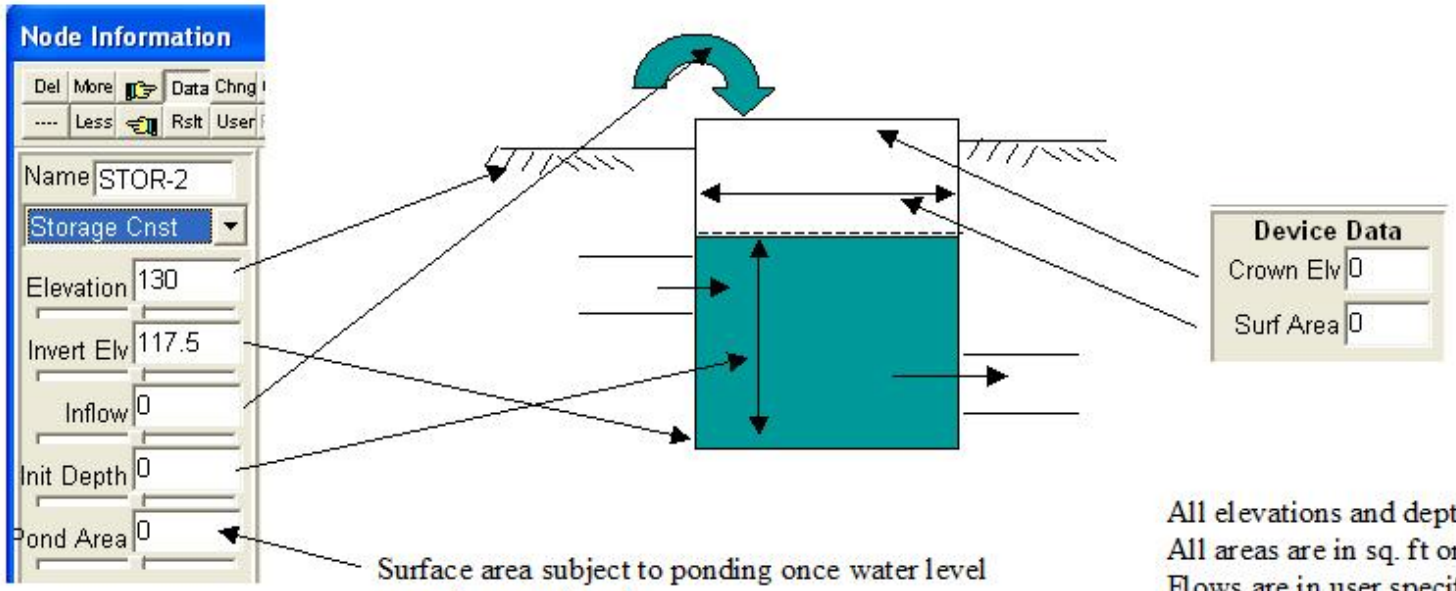


Flapgate


Tidal
Time Series

# Storage Node – Constant Surface Area

Storage Units are drainage system nodes that provide storage volume. Physically they could represent storage facilities as small as a catch basin or as large as a lake. The volumetric properties of a storage unit are described by a function or table of surface area versus height.

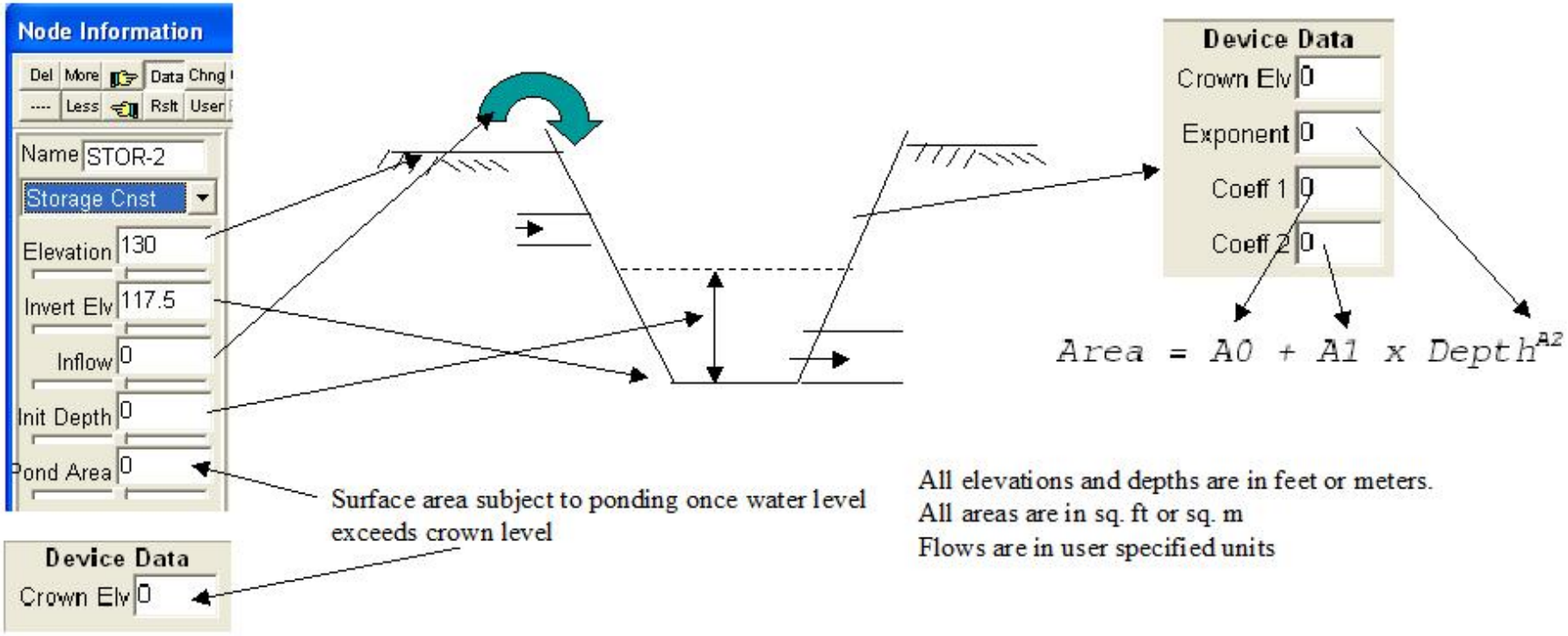



Surface area subject to ponding once water level exceeds crown level

All elevations and depths are in feet or meters.
All areas are in sq. ft or sq. m
Flows are in user specified units

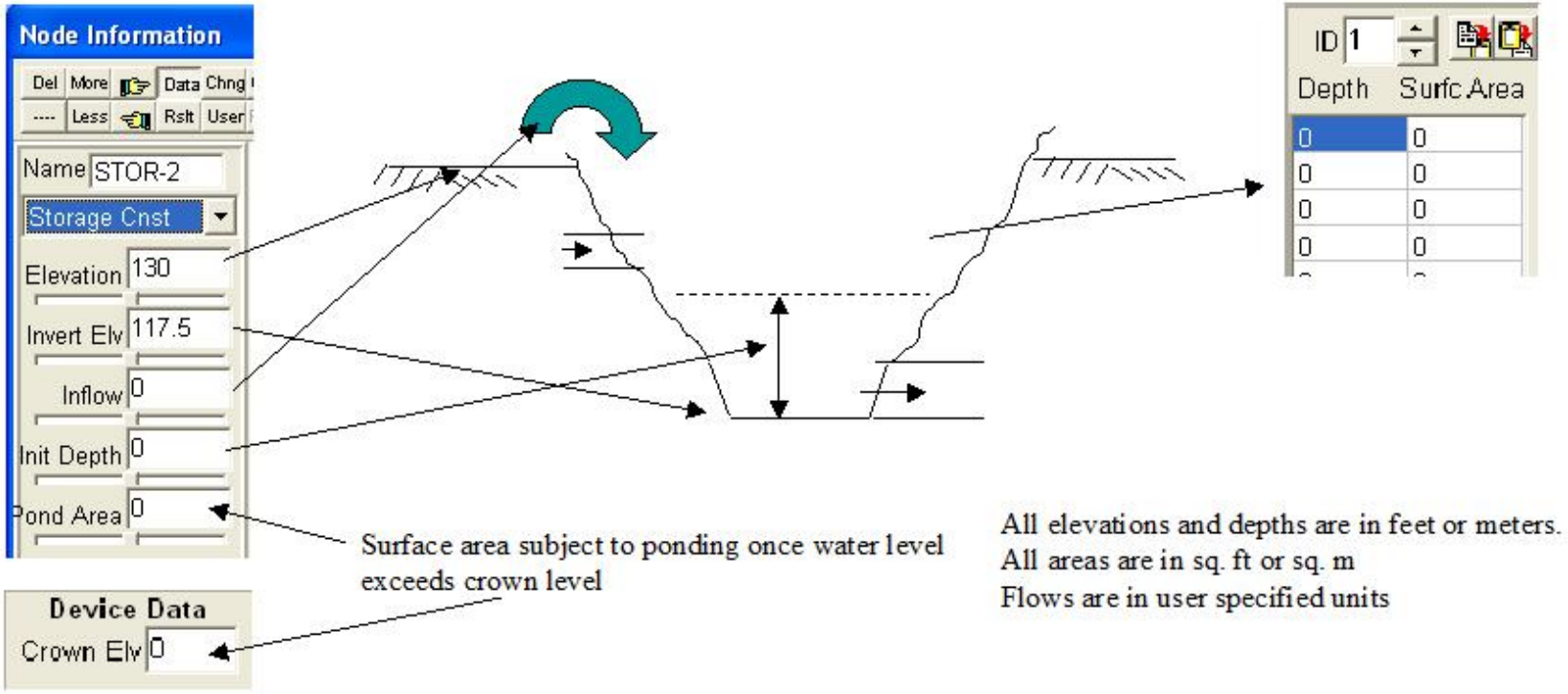
# Storage Node – Functional Depth-Surface Area

Storage Units are drainage system nodes that provide storage volume. Physically they could represent storage facilities as small as a catch basin or as large as a lake. The volumetric properties of a storage unit are described by a function or table of surface area versus height.



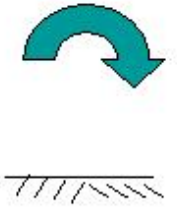
# Storage Node – Variable Surface Area

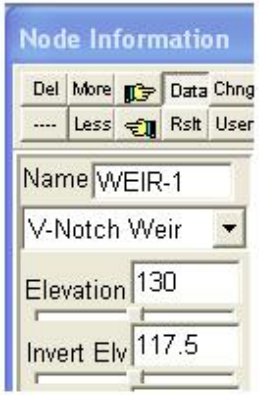
Storage Units are drainage system nodes that provide storage volume. Physically they could represent storage facilities as small as a catch basin or as large as a lake. The volumetric properties of a storage unit are described by a function or table of surface area versus height.



# Weirs – V-Notch

Weirs, like orifices, are used to model outlet and diversion structures in a drainage system. Weirs are typically located in a manhole, along the side of a channel, or within a storage unit. They are internally represented in SWMM as a link connecting two nodes, where the weir itself is placed at the upstream node. A flap gate can be included to prevent backflow.





All elevations and depths are in feet or meters

| Weir Type | Cross Section Shape | Flow Formula |
|---|---|---|
| Transverse | Rectangular | $C_w L h^{3/2}$ |
| Side flow | Rectangular | $C_w L h^{5/3}$ |
| V-notch | Triangular | $C_w S h^{5/2}$ |
| Trapezoidal | Trapezoidal | $C_w L h^{3/2}$ $C_w S h^{5/2}$ |

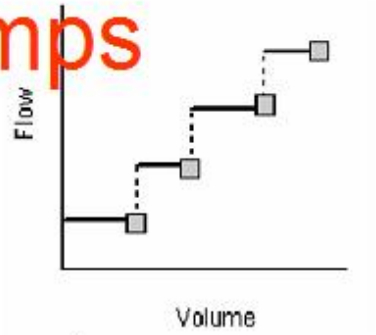
All areas are in sq. ft or sq. m
Flows are in user specified units

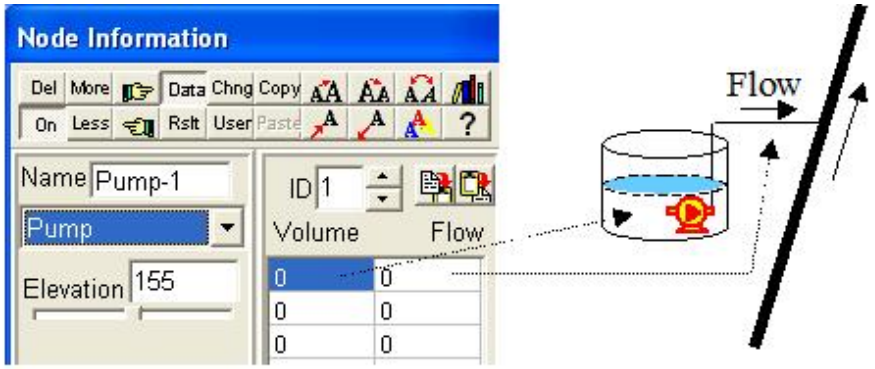
| | | |
|---|---|---|
| Trapezoidal | Trapezoidal | $C_w L h^{3/2} + C_{ws} S h^{3/2}$ |

$C_w$ = weir discharge coefficient, L = weir length, S = side slope of V-notch or trapezoidal weir, h = head difference across the weir, $C_{ws}$ = discharge coefficient through sides of trapezoidal weir.

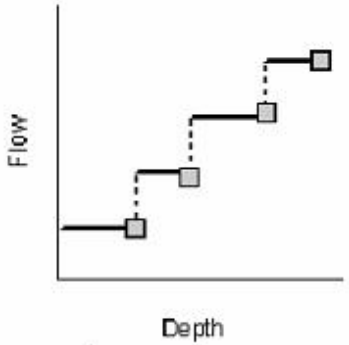
# Pumps

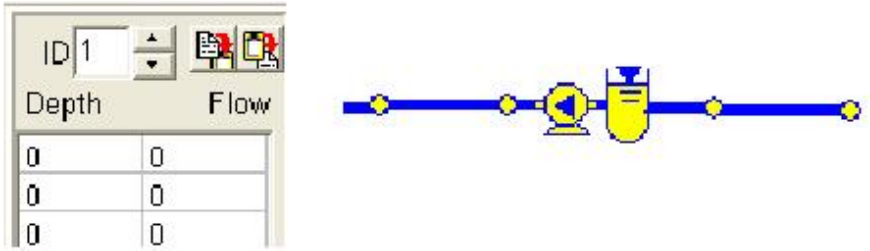
Type1
an off-line pump with a wet well where flow increases incrementally with available wet well volume



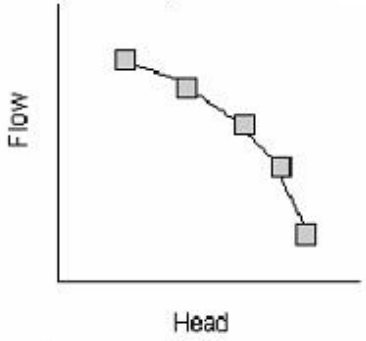



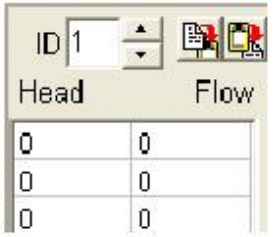

Type2
an in-line pump where flow increases incrementally with inlet node depth

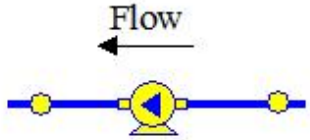


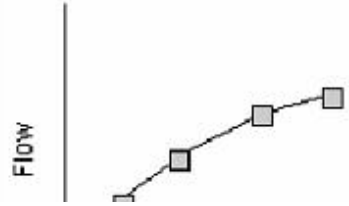

Type3
an in-line pump where flow varies continuously with head difference between the inlet and outlet nodes

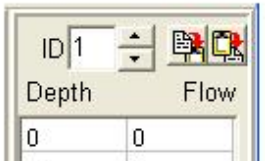


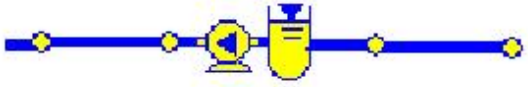




Type4
a variable speed in-line pump where flow varies continuously with inlet



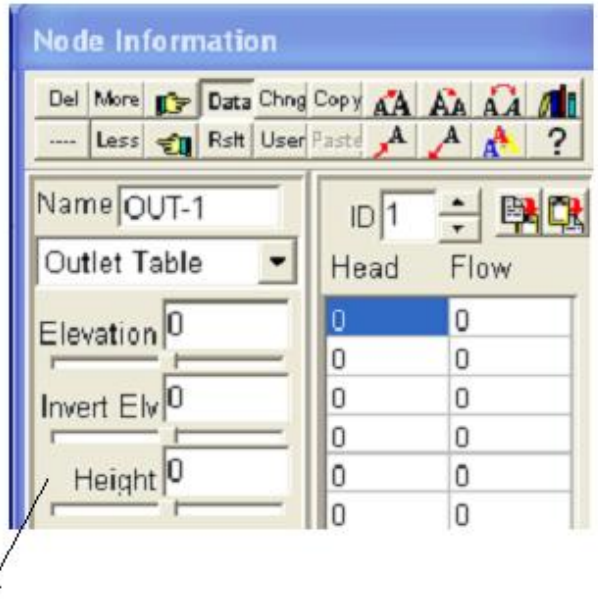



node depth

# Outlet Node – Table

Outlets are flow control devices that are typically used to control outflows from storage units. They are used to model special head-discharge relationships that cannot be characterized by pumps, orifices, or weirs. Outlets are internally represented in SWMM as a link connecting two nodes. An outlet can also have a flap gate that restricts flow to only one direction.

Outlets attached to storage units are active under all types of flow routing. If not attached to a storage unit, they can only be used in drainage networks analyzed with Dynamic Wave flow routing.



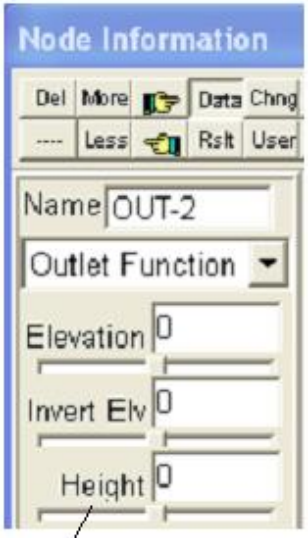

Min water depth at inlet node for outflow to occur

All elevations and depths are in feet or meters.
All areas are in sq. ft or sq. m
Flows are in user specified units

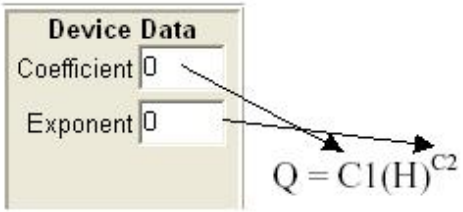
# Outlet Node – Functional

Outlets are flow control devices that are typically used to control outflows from storage units. They are used to model special head-discharge relationships that cannot be characterized by pumps, orifices, or weirs. Outlets are internally represented in SWMM as a link connecting two nodes. An outlet can also have a flap gate that restricts flow to only one direction.

Outlets attached to storage units are active under all types of flow routing. If not attached to a storage unit, they can only be used in drainage networks analyzed with Dynamic Wave flow routing.




Min water depth at inlet node for outflow to occur

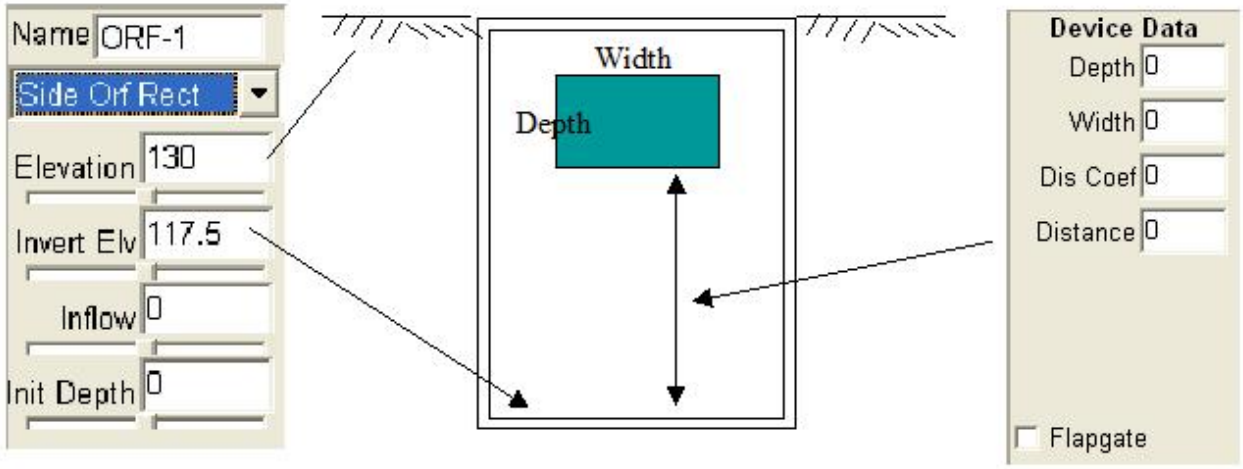



All elevations and depths are in feet or meters.
All areas are in sq. ft or sq. m
Flows (Q) are in user specified units

# Orifice Node – Rectangular

Orifices are used to model outlet and diversion structures in drainage systems, which are typically openings in the wall of a manhole, storage facility, or control gate. They are internally represented in SWMM as a link connecting two nodes. An orifice can have either a circular or rectangular shape, be located either at the bottom or along the side of the upstream node, and have a flap gate to prevent backflow.

Orifices can be used as storage unit outlets under all types of flow routing. If not attached to a storage unit node, they can only be used in drainage networks that are analyzed with Dynamic Wave flow routing.



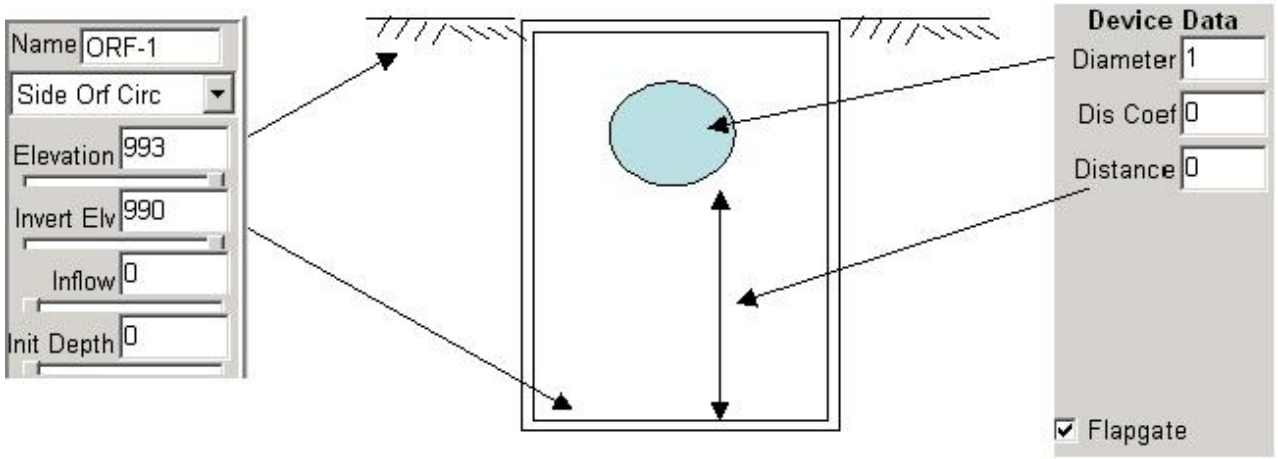

All elevations and depths are in feet or meters.
All areas are in sq. ft or sq. m

All areas are in sq. ft or sq. m
Flows are in user specified units

# Orifice Node – Circular

Orifices are used to model outlet and diversion structures in drainage systems, which are typically openings in the wall of a manhole, storage facility, or control gate. They are internally represented in SWMM as a link connecting two nodes. An orifice can have either a circular or rectangular shape, be located either at the bottom or along the side of the upstream node, and have a flap gate to prevent backflow.

Orifices can be used as storage unit outlets under all types of flow routing. If not attached to a storage unit node, they can only be used in drainage networks that are analyzed with Dynamic Wave flow routing.

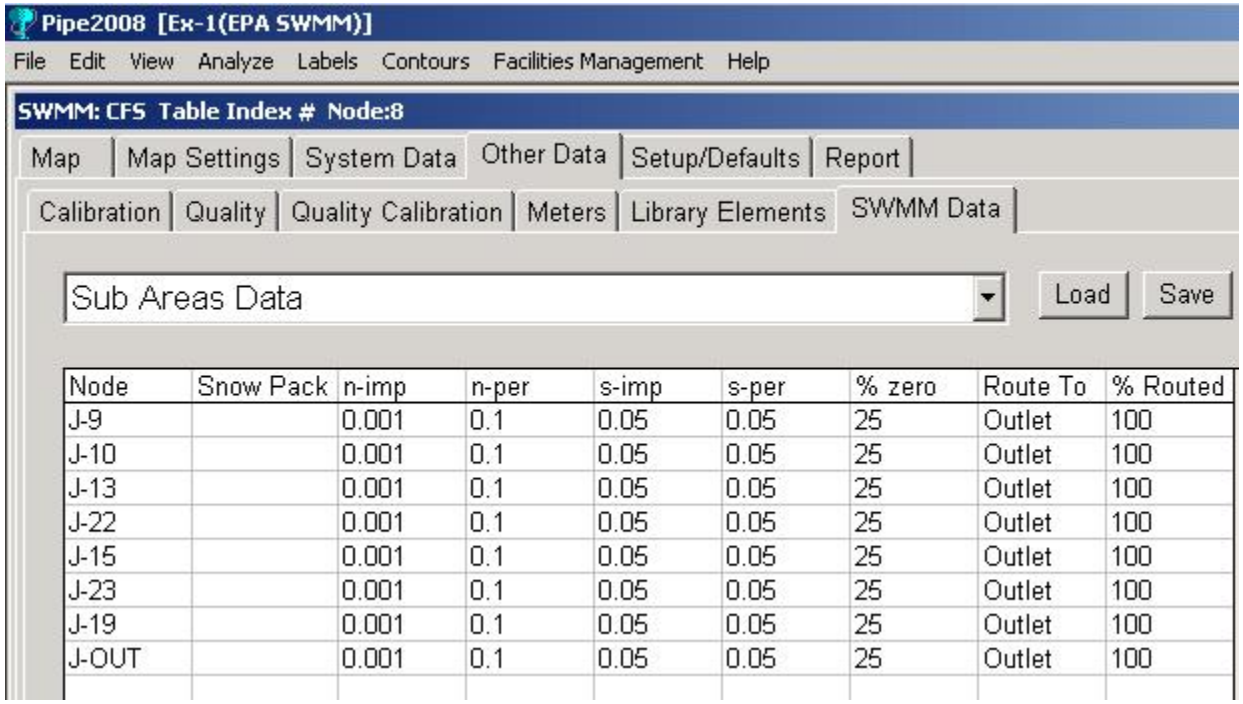



All elevations and depths are in feet or meters.
All areas are in sq. ft or sq. m

All areas are in sq. ft or sq. m
Flows are in user specified units

# INFLOWS

- Direct Inflow
  - User Defined inflows
  - Rainfall induced catchment flows
- Dry weather Inflows
- Rainfall-derived I&I

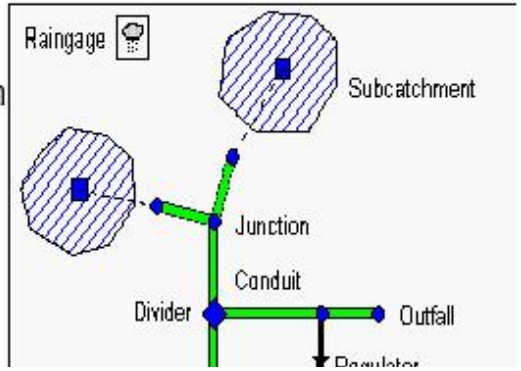


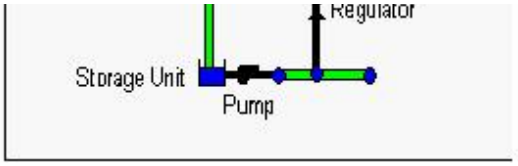

| Node | Snow Pack | n-imp | n-per | s-imp | s-per | % zero | Route To | % Routed |
|---|---|---|---|---|---|---|---|---|
| J-9 | | 0.001 | 0.1 | 0.05 | 0.05 | 25 | Outlet | 100 |
| J-10 | | 0.001 | 0.1 | 0.05 | 0.05 | 25 | Outlet | 100 |
| J-13 | | 0.001 | 0.1 | 0.05 | 0.05 | 25 | Outlet | 100 |
| J-22 | | 0.001 | 0.1 | 0.05 | 0.05 | 25 | Outlet | 100 |
| J-15 | | 0.001 | 0.1 | 0.05 | 0.05 | 25 | Outlet | 100 |
| J-23 | | 0.001 | 0.1 | 0.05 | 0.05 | 25 | Outlet | 100 |
| J-19 | | 0.001 | 0.1 | 0.05 | 0.05 | 25 | Outlet | 100 |
| J-OUT | | 0.001 | 0.1 | 0.05 | 0.05 | 25 | Outlet | 100 |

[**SUBAREAS**]

Supplies information about pervious and impervious areas for each subcatchment. Each subcatchment can consist of a pervious sub-area, an impervious sub-area with depression storage, and an impervious sub-area without depression storage.

| | |
|---|---|
| *Subcat* | subcatchment name. **(Name/Node)** |
| *Nimp* | Manning's n for overland flow over the impervious sub-area. |
| *Nperv* | Manning's n for overland flow over the pervious sub-area. |
| *Simp* | depression storage for impervious sub-area (inches or mm). |
| *Sperv* | depression storage for pervious sub-area (inches or mm). |
| *%Zero* | percent of impervious area with no depression storage. |
| *RouteTo* | Use **IMPERVIOUS** if pervious area runoff runs onto impervious area, **PERVIOUS** if impervious runoff runs onto pervious area, or **OUTLET** if both areas drain to the subcatchment's outlet (default = **OUTLET**). |
| *%Rted* | Percent of runoff routed from one type of area to another (default = 100). |
| *Spack* | name of snow pack object (from [**SNOWPACKS**] section) that characterizes snow accumulation and melting over the subcatchm |






[COVERAGES]

Specifies the percentage of a subcatchment's area that is covered by each category of land use.

*Subcat Landuse Percent Landuse Percent . . .*

*Subcat* subcatchment name.
*Landuse* land use name.
*Percent* percent of subcatchment area.

More than one pair of land use - percentage values can be entered per line. If more than one line is needed, then the subcatchment name must still be entered first on the succeeding lines.

If a land use does not pertain to a subcatchment, then it does not have to be entered.

If no land uses are associated with a subcatchment then no contaminants will appear in the runoff from the subcatchment.

# Infiltration Data

Pipe2008 [Ex-1(EPA SWMM)]

File Edit View Analyze Labels Contours Facilities Management Help

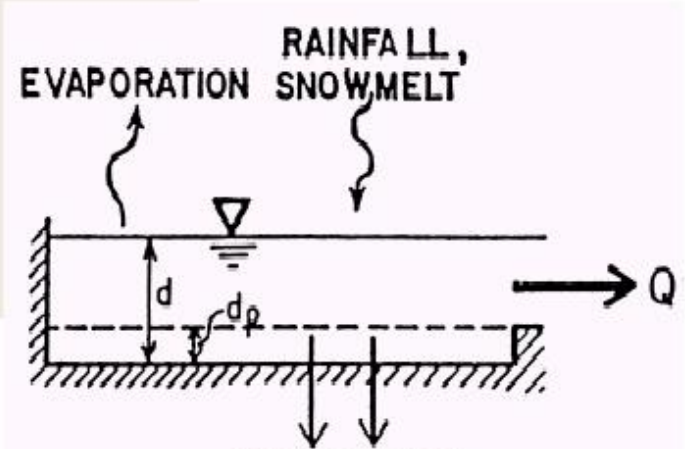
SWMM: CFS Table Index # Node:8

Map | Map Settings | System Data | Other Data | Setup/Defaults | Report

Calibration | Quality | Quality Calibration | Meters | Library Elements | SWMM Data

Infiltration Data

| Node | Data-1 | Data-2 | Data-3 | Data-4 | Data-5 |
|---|---|---|---|---|---|
| J-9 | 0.35 | 0.25 | 4.14 | 0.5 | |
| J-10 | 0.7 | 0.3 | 4.14 | 0.5 | |
| J-13 | 0.7 | 0.3 | 4.14 | 0.5 | |
| J-22 | 0.7 | 0.3 | 4.14 | 0.5 | |
| J-15 | 0.7 | 0.3 | 4.14 | 0.5 | |
| J-23 | 0.7 | 0.3 | 4.14 | 0.5 | |
| J-19 | 0.7 | 0.3 | 4.14 | 0.5 | |
| J-OUT | 0.7 | 0.3 | 4.14 | 0.5 | |



INFILTRATION

[**INFILTRATION**]

Supplies infiltration parameters for each subcatchment. Rainfall lost to infiltration only occurs over the pervious sub-area of a subcatchment.

Data-1, Data-2…. Data-5

```
Subcat   MaxRate   MinRate   Decay   DryTime   MaxInf
Subcat   Suction   Conduct   InitDef
Subcat   CurveNo   Conduct   DryTime
```

For Horton Infiltration:

*MaxRate* Maximum infiltration rate on Horton curve (in/hr or mm/hr).
*MinRate* Minimum infiltration rate on Horton curve (in/hr or mm/hr).
*Decay* Decay rate constant of Horton curve (1/hr).
*DryTime* Time it takes for fully saturated soil to dry (days).
*MaxInf* Maximum infiltration volume possible (0 if not applicable) (in or mm).

For Green-Ampt Infiltration:

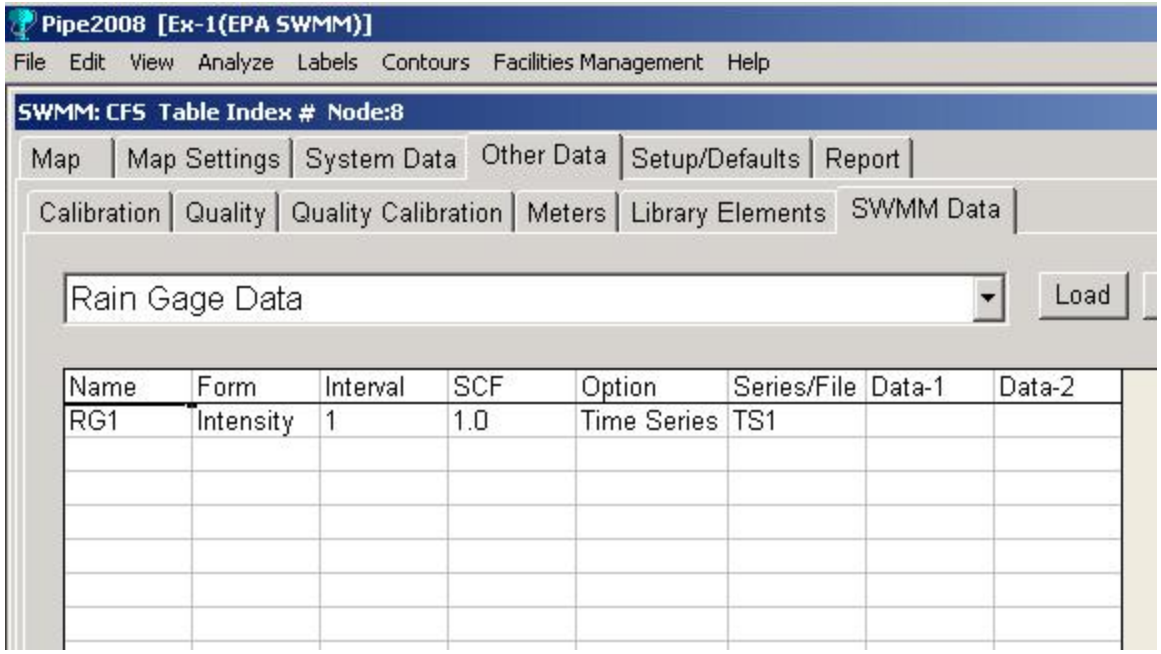
*Suction* Soil capillary suction (in or mm).
*Conduct* Soil saturated hydraulic conductivity (in/hr or mm/hr).
*InitDef* Initial soil moisture deficit (volume of voids / total volume).

For Curve-Number Infiltration:

*CurveNo* SCS Curve Number.

*Conduct* Soil saturated hydraulic conductivity (in/hr or mm/hr).
*DryTime* Time it takes for fully saturated soil to dry (days).

Rain gauge Data



## Rain Gage Data:

*Name Form Intvl SCF* **TIMESERIES** *Tseries*
*Name Form Intvl SCF* **FILE** *Fname Sta Units*

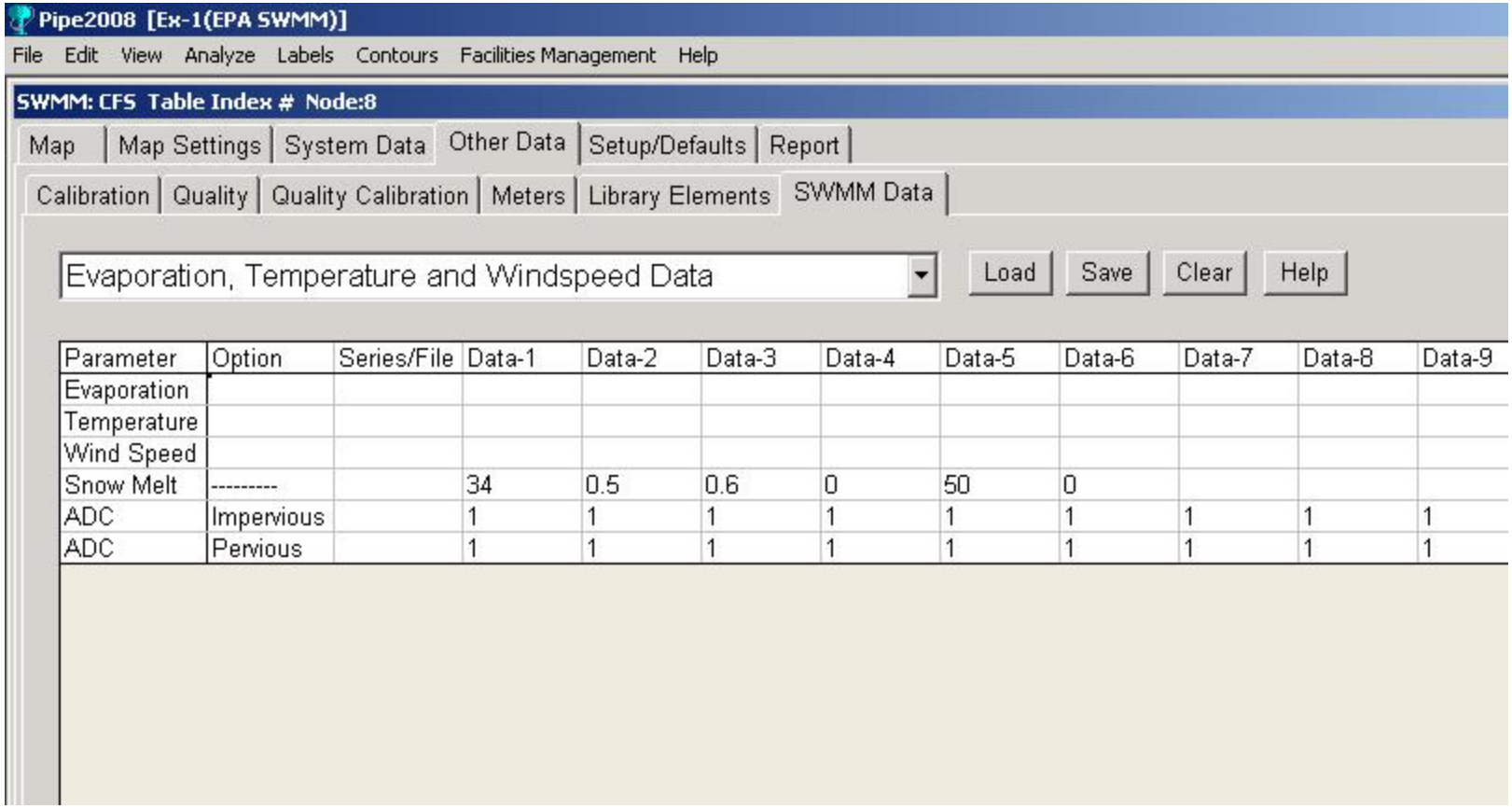
---

| | |
|---|---|
| *Name* | name assigned to rain gage. |
| *Form* | form of recorded rainfall, either **INTENSITY**, **VOLUME** or **CUMULATIVE**. |
| *Intvl* | time interval between gage readings in decimal hours or hours:minutes format (e.g., 0:15 for 15-minute readings). |
| *SCF* | snow catch deficiency correction factor (use 1.0 for no adjustment). |
| *Tseries* | name of time series in [**TIMESERIES**] section with rainfall data. |
| *Fname* | name of external file with rainfall data. Rainfall files are discussed in Section 11.3. |

Data-1 and Data-2 refer to Sta and Units respectively and are applicable only File Option in column 6

| | |
|---|---|
| *Sta* | name of recording station used in the rain file. |
| *Units* | rain depth units used in the rain file, either **IN** (inches) or **MM** (millimeters). |

Evaporation and Temperature




| Parameter | Option | Series/File | Data-1 | Data-2 | Data-3 | Data-4 | Data-5 | Data-6 | Data-7 | Data-8 | Data-9 |
|---|---|---|---|---|---|---|---|---|---|---|---|
| Evaporation | | | | | | | | | | | |
| Temperature | | | | | | | | | | | |
| Wind Speed | | | | | | | | | | | |
| Snow Melt | --------- | | 34 | 0.5 | 0.6 | 0 | 50 | 0 | | | |
| ADC | Impervious | | 1 | 1 | 1 | 1 | 1 | 1 | 1 | 1 | 1 |
| ADC | Pervious | | 1 | 1 | 1 | 1 | 1 | 1 | 1 | 1 | 1 |

[**EVAPORATION**]

Specifies how daily evaporation rates vary with time for the study area.

```
CONSTANT    evap
MONTHLY     evap1 evap2 ... evap12
TIMESERIES  Tseries
FILE        (pan1 pan2 ... pan12)
```

Data-1, Data-2… Data-12

| | |
|---|---|
| *evap* | constant evaporation rate (in/day or mm/day). |
| *evap1* | evaporation rate in January (in/day or mm/day). |
| … | |
| *evap12* | evaporation rate in December (in/day or mm/day). |
| *Tseries* | name of time series in [**TIMESERIES**] section with evaporation data. |
| *pan1* | pan coefficient for January. |
| … | |
| *pan12* | pan coefficient for December. |

Use only one of the above formats. If no [**EVAPORATION**] section appears, then evaporation is asumed to be 0.

**FILE** indicates that evaporation data will be read from the same external climate file

used for air temperatures (see below).

[**TEMPERATURE**]

Specifies daily air temperatures, monthly wind speed, and various snow melt parameters for the study area . Required only when snow melt is being modeled or when evaporation rates are read from an external climate file.

## Temperature Data

**TIMESERIES** *Tseries*
**FILE** *Fname (Start)*

| | |
|---|---|
| *Tseries* | name of time series in [**TIMESERIES**] section with temperature data. |
| *Fname* | name of external Climate file with temperature data. |
| *Start* | date to begin reading from the file in month/day/year format (default is the beginning of the file). |

Use the **TIMESERIES** line to read air temperature from a time series or the **FILE** line to read it from an external Climate file. Climate files are discussed in Section 11.4. If neither format is used, then air temperature remains constant at 70 degrees F.

[**TEMPERATURE**]

Specifies daily air temperatures, monthly wind speed, and various snow melt parameters for the study area . Required only when snow melt is being modeled or when evaporation rates are read from an external climate file.

## Wind Speed Data

**WINDSPEED MONTHLY** *s1 s2 … s11 s12*
**WINDSPEED FILE**

*s1* average wind speed in January (mph or km/hr).
…
*s12* average wind speed in December (mph or km/hr).

**Data-1, Data-2… Data-12**

*Fname* name of external Climate file with temperature data.

Wind speed can be specified either by monthly average values or by the same Climate file used for air temperature. If neither option appears, then wind speed is assumed to be 0.

[**TEMPERATURE**]

Specifies daily air temperatures, monthly wind speed, and various snow melt parameters for the study area . Required only when snow melt is being modeled or when evaporation rates are read from an external climate file.

## Snow Melt Data

**SNOWMELT** *Stemp ATIwt RNM Elev Lat DTLong*
**ADC IMPERVIOUS** *f.0 f.1 … f.8 f.9*
**ADC PERVIOUS** *f.0 f.1 … f.8 f.9*

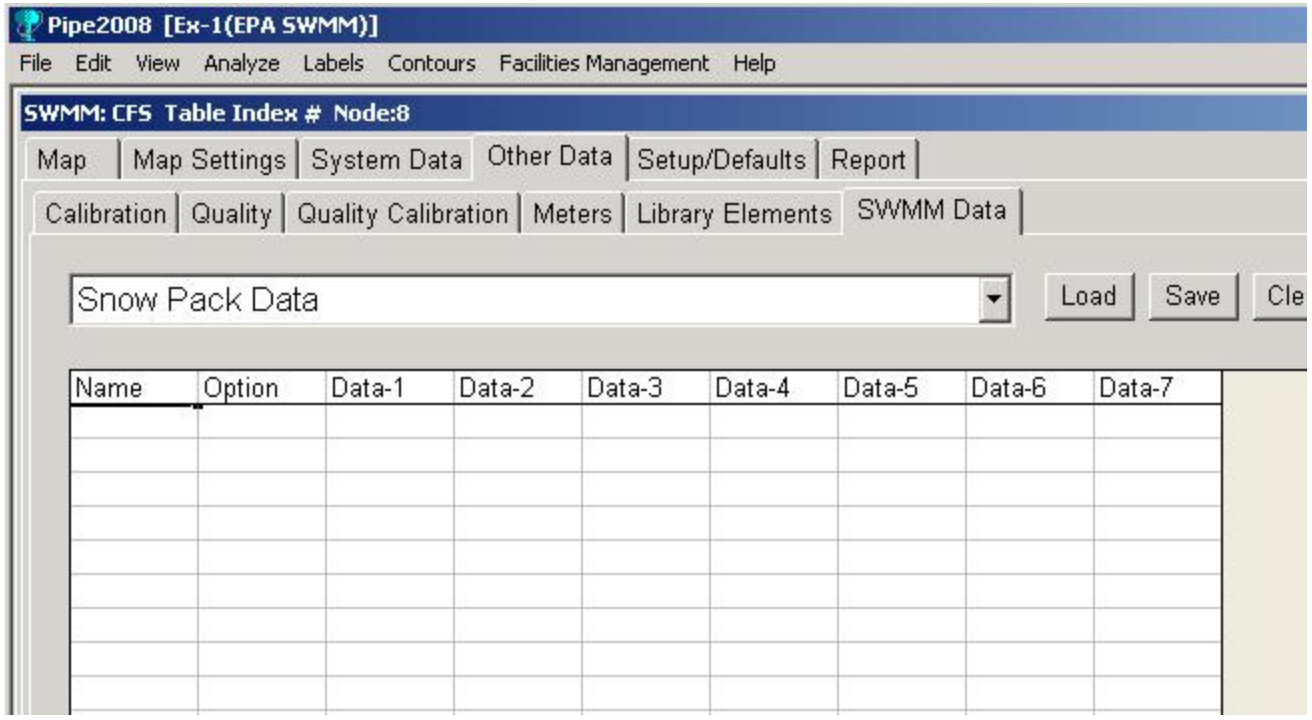
| | |
|---|---|
| *Stemp* | air temperature at which precipitation falls as snow (deg F or C). |
| *ATIwt* | antecedent temperature index weight (default is 0.5). |
| *RNM* | negative melt ratio (default is 0.6). |
| *Elev* | average elevation of study area above mean sea level (ft or m) (default is 0). |
| *Lat* | latitude of the study area in degrees North (default is 50). |
| *DTLong* | correction, in minutes of time, between true solar time and the standard clock time (default is 0). |
| *f.0* | fraction of area covered by snow when ratio of snow depth to depth at 100% cover is 0. |
| … | |
| *f.9* | fraction of area covered by snow when ratio of snow depth to depth at 100% cover is 0.9. |

Data-1,
Data-2,
…
…
Data-9

Separate Areal Depletion Curves (ADC) can be defined for impervious and pervious

sub-areas. The ADC parameters will default to 1.0 (meaning no depletion) if no data are supplied for a particular type of sub-area.

Snowpacks



[**SNOWPACKS**]

Specifies parameters that govern how snowfall accumulates and melts on the plowable, impervious and pervious surfaces of subcatchments.

```
Name   PLOWABLE    Cmin   Cmax   Tbase   FWF   SD0   FW0   SNN0
Name   IMPERVIOUS  Cmin   Cmax   Tbase   FWF   SD0   FW0   SD100
Name   PERVIOUS    Cmin   Cmax   Tbase   FWF   SD0   FW0   SD100
Name   REMOVAL  Dplow Fout Fimp Fperv Fimelt (Fsub Scatch)
```

**Data-1, Data-2… Data-7**

The **PLOWABLE** line contains parameters for the impervious area of a subcatchment that is subject to snow removal by plowing but not to areal depletion. This area is the fraction *SNN0* of the total impervious area. The **IMPERVIOUS** line contains parameter values for the remaining impervious area and the **PERVIOUS** line does the same for the entire pervious area. Both of the latter two areas are subject to areal depletion.

| | |
|---|---|
| *Name* | name assigned to snowpack parameter set . |
| *Cmin* | minimum melt coefficient (in/hr-deg F or mm/hr-deg C). |
| *Cmax* | maximum melt coefficient (in/hr-deg F or mm/hr-deg C). |
| *Tbase* | snow melt base temperature (deg F or deg C). |
| *FWF* | ratio of free water holding capacity to snow depth (fraction). |
| *SD0* | initial snow depth (in or mm water equivalent). |
| *FW0* | initial free water in pack (in or mm). |

| | |
|---|---|
| *SNN0* | fraction of impervious area that can be plowed. |
| *SD100* | snow depth above which there is 100% cover (in or mm water equivalent). |

[**SNOWPACKS**]

Specifies parameters that govern how snowfall accumulates and melts on the plowable, impervious and pervious surfaces of subcatchments.

```
Name   PLOWABLE    Cmin   Cmax   Tbase   FWF   SD0   FW0   SNN0
Name   IMPERVIOUS  Cmin   Cmax   Tbase   FWF   SD0   FW0   SD100
Name   PERVIOUS    Cmin   Cmax   Tbase   FWF   SD0   FW0   SD100
Name   REMOVAL  Dplow Fout Fimp Fperv Fimelt (Fsub Scatch)
```

**Data-1, Data-2... Data-7**

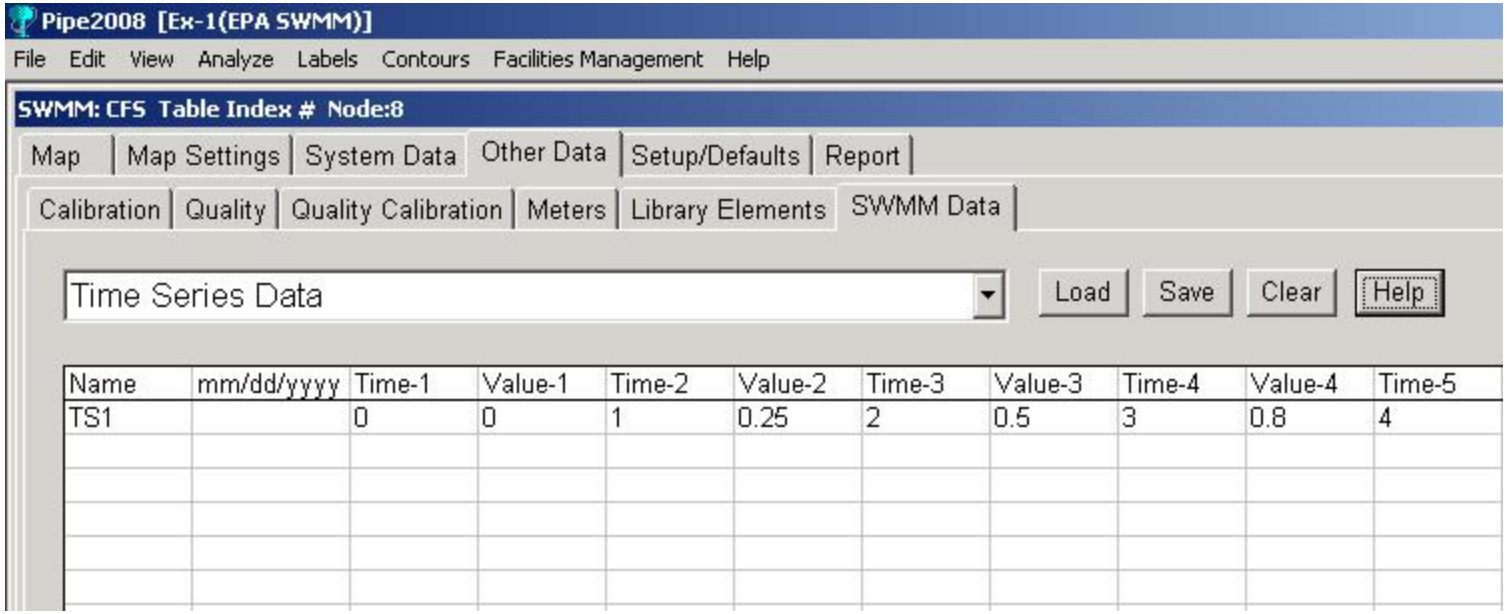
The **REMOVAL** line describes how snow removed from the plowable area is transferred onto other areas. The various transfer fractions should either sum to 1.0 or be all 0.0 to indicate that no plowing is done (or the line can simply be omitted).

| | |
|---|---|
| *SDplow* | depth of snow on plowable areas at which redistribution through plowing occurs (in or mm). |
| *Fout* | fraction of excess snow on plowable area transferred out of watershed. |
| *Fimp* | fraction of excess snow on plowable area transferred to impervious area by plowing. |
| *Fperv* | fraction of excess snow on plowable area transferred to pervious area by plowing. |
| *Fimelt* | fraction of excess snow on plowable area converted into immediate melt. |
| *Fsub* | fraction of excess snow on plowable area transferred to pervious area in |

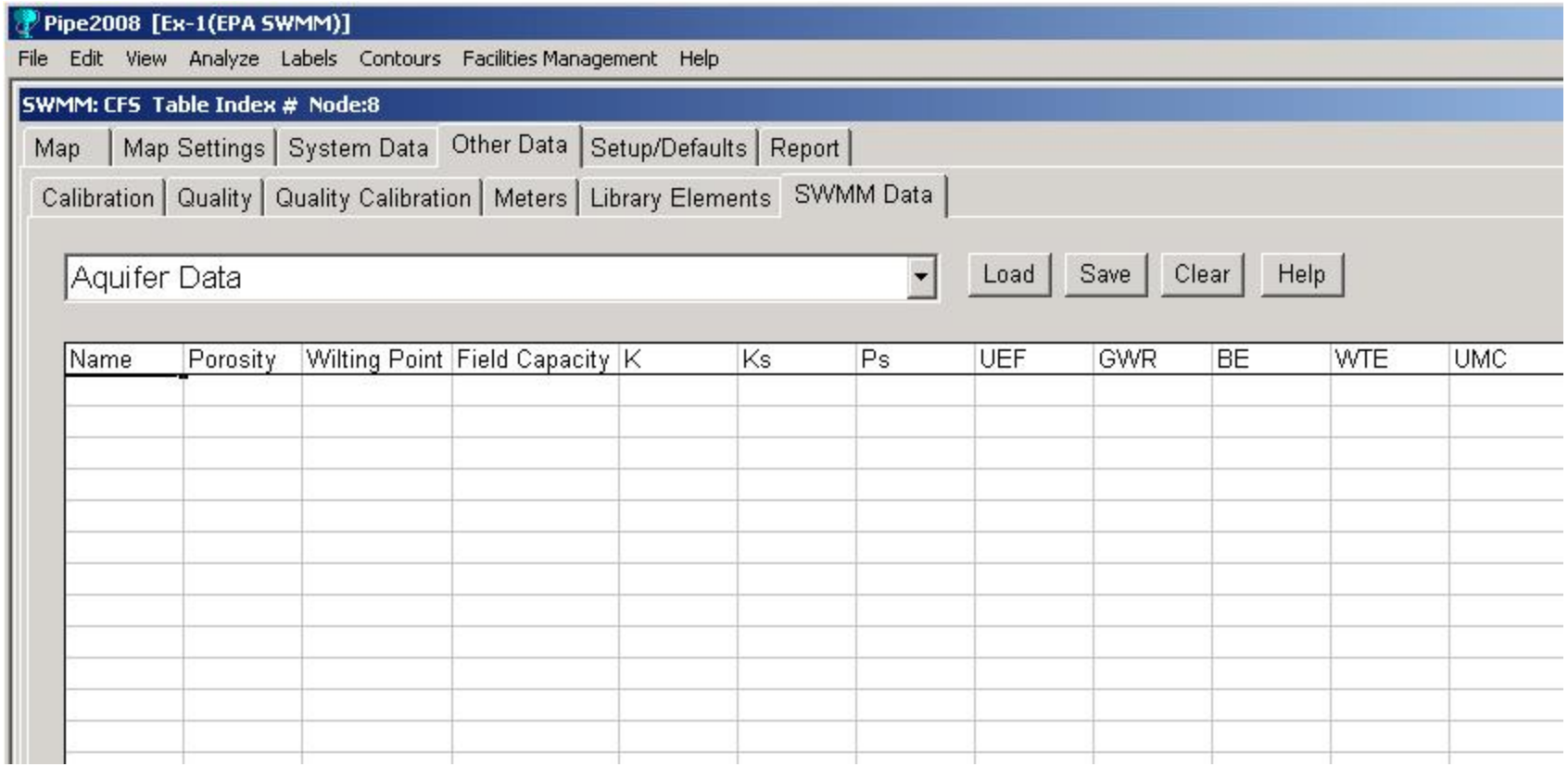
another subcatchment.

*Scatch* name of subcatchment receiving the Fsubcatch fraction of transferred snow.

Time Series Data




Aquifer Data

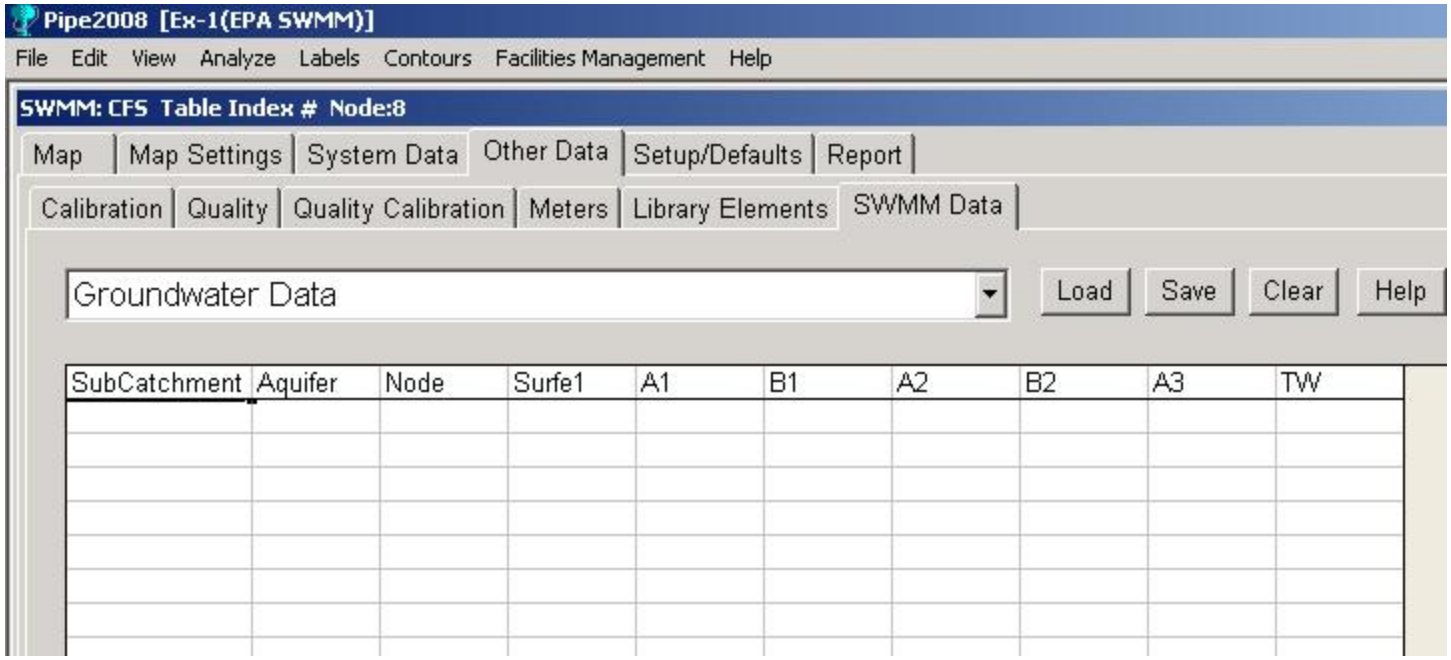

Pipe2008 [Ex-1(EPA SWMM)]
File Edit View Analyze Labels Contours Facilities Management Help
SWMM: CFS Table Index # Node:8
Map Map Settings System Data Other Data Setup/Defaults Report
Calibration Quality Quality Calibration Meters Library Elements SWMM Data
Aquifer Data
Load Save Clear Help
Name Porosity Wilting Point Field Capacity K Ks Ps UEF GWR BE WTE UMC

**[AQUIFERS]**

Supplies parameters for each unconfined groundwater aquifer in the study area. Aquifers consist of two zones – a lower saturated zone and an upper unsaturated zone.

```
Name  Por  WP  FC  K  Ks  Ps  UEF  LED  GWR  BE  WTE  UMC
```

| | |
|---|---|
| *Name* | name assigned to aquifer. |
| *Por* | soil porosity (volumetric fraction). |
| *WP* | soil wilting point (volumetric fraction). |
| *FC* | soil field capacity (volumetric fraction). |
| *K* | saturated hydraulic conductivity (in/hr or mm/hr). |
| *Ks* | slope of hydraulic conductivity versus moisture content curve (in/hr or mm/hr. |
| *Ps* | slope of soil tension versus moisture content curve (inches or mm). |
| *UEF* | fraction of total evaporation available for evapotranspiration in the upper unsaturated zone. |
| *LED* | maximum depth into the lower saturated zone over which evapotranspiration can occur (ft or m). |
| *GWR* | rate of percolation from saturated zone to deep groundwater when water table is at ground surface (in/hr or mm/hr). |
| *BE* | elevation of the bottom of the aquifer (ft or m). |
| *WTE* | water table elevation at start of simulation (ft or m). |
| *UMC* | unsaturated zone moisture content at start of simulation (volumetric fraction). |


Pipe2008 [Ex-1(EPA SWMM)]
File Edit View Analyze Labels Contours Facilities Management Help
SWMM: CFS Table Index # Node:8
Map Map Settings System Data Other Data Setup/Defaults Report
Calibration Quality Quality Calibration Meters Library Elements SWMM Data
Groundwater Data
Load Save Clear Help
SubCatchment Aquifer Node Surfe1 A1 B1 A2 B2 A3 TW

[GROUNDWATER]

Supplies parameters that determine the rate of groundwater flow between the aquifer underneath a subcatchment and a node of the conveyance system.

| | |
|---|---|
| *Subcat* | subcatchment name. |
| *Aquifer* | name of groundwater aquifer underneath the subcatchment. |
| *Node* | name of node in conveyance system exchanging groundwater with aquifer. |
| *SurfEl* | surface elevation of subcatchment (ft or m). |
| *A1* | groundwater flow coefficient (see below). |
| *B1* | groundwater flow exponent (see below). |
| *A2* | surface water flow coefficient (see below). |
| *B2* | surface water flow exponent (see below). |
| *A3* | surface water – groundwater interaction coefficient (see below). |
| *TW* | fixed depth of surface water at receiving node (ft or m) (set to zero if surface water depth will vary as computed by flow routing). |
| *E* | groundwater elevation which must be reached before any flow occurs (feet or meters). Leave blank to use the receiving node's invert elevation. |

The flow coefficients are used in the following equation that determines a groundwater flow rate based on groundwater and surface water elevations:

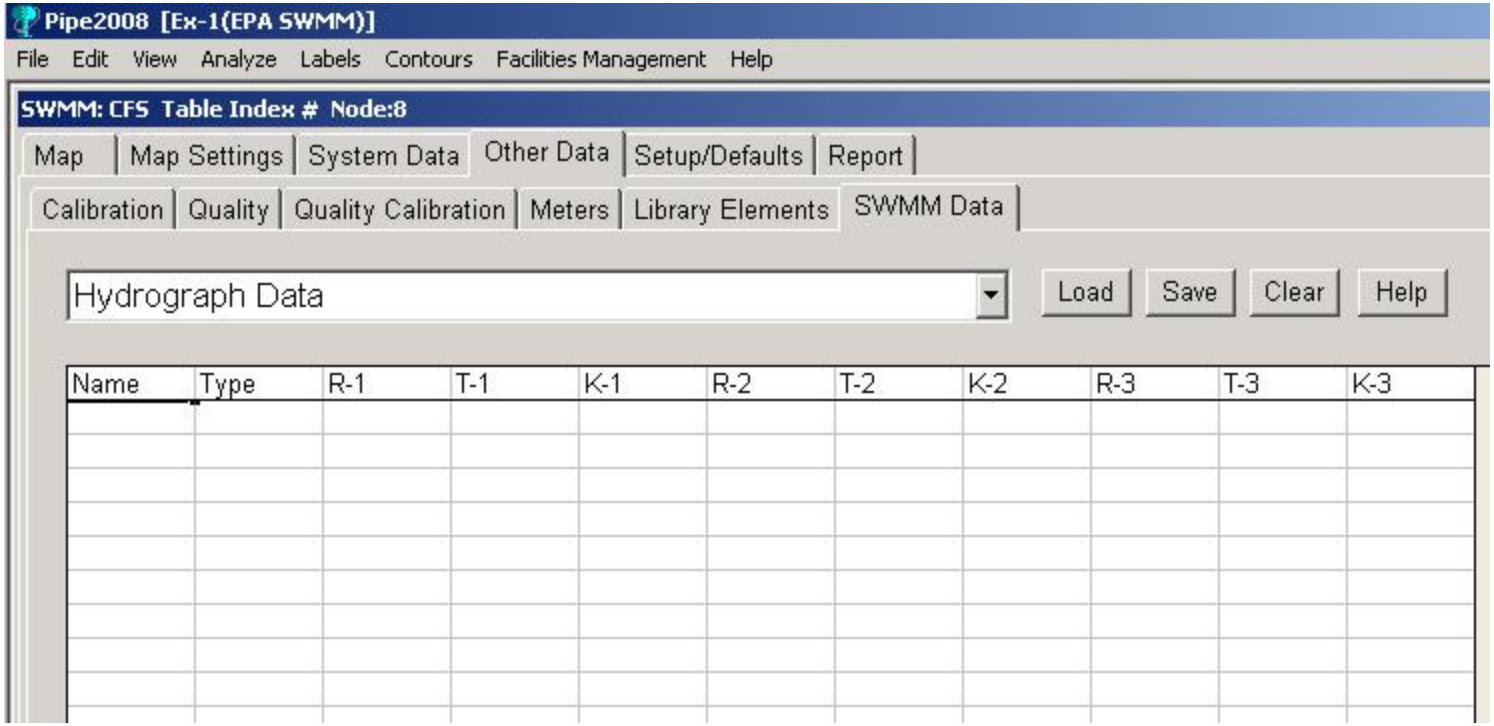
$$Q_{gw} = A1(H_{gw} - E)^{B1} - A2(H_{sw} - E)^{B2} + A3H_{gw}H_{sw}$$

where:
$Q_{gw}$ = groundwater flow (cfs per acre or cms per hectare),
$H_{gw}$ = computed elevation of groundwater table (ft or m)

$H_{gw}$ = computed elevation of groundwater table (ft or m),

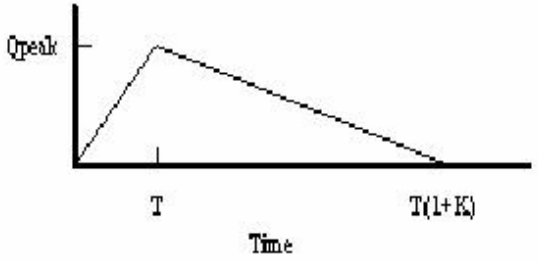
$H_{sw}$ = computed elevation of surface water at receiving node (ft or m) if $TW$ is 0 or $TW + E$ otherwise.

Hydrograph Data



[HYDROGRAPHS]

Specifies the shapes of the triangular unit hydrographs that determine the amount of rainfall-derived infiltration/inflow (RDII) entering the drainage system.




*Name Raingage*
*Name Month R1 T1 K1 R2 T2 K2 R3 T3 K3*

| | |
|---|---|
| *Name* | name assigned to a unit hydrograph (UH) group. |
| *Raingage* | name of rain gage used by UH group. |
| *Month* | month of the year (e.g., **JAN**, **FEB**, etc. or **ALL** for all months). |
| *R1, R2, R3* | response ratios for the short-term, intermediate-term, and long-term UH responses, respectively. |
| *T1, T2, T3* | time to peak (hours) for the short-term, intermediate-term, and long-term UH responses, respectively. |
| *K1, K2, K3* | recession limb ratios for short-term, intermediate-term, and long-term UH responses, respectively. |

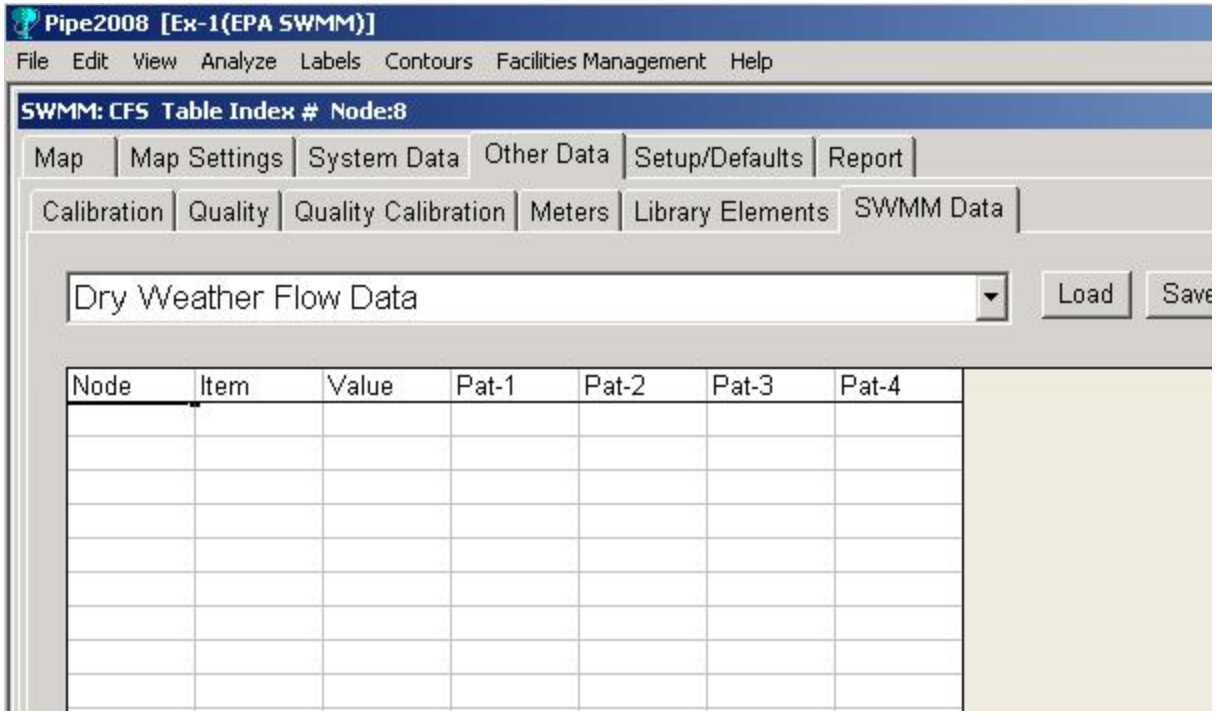
For each group of unit hydrographs, use one line to specify its rain gage followed by one or more lines containing UH shape parameters for months with RDII flow. Months not listed are assumed to have no RDII.

The response ratios (R) are the fraction of a unit of rainfall depth that becomes RDII. The sum of the ratios for the three UH's do not have to equal 1.0.

The recession limb ratios (K) are the ratio of the duration of the UH recession limb to the time to peak (T) making the UH time base T*(1+K) hours. The area under each

UH is 1 inch (or mm).

Dry Weather Flow (DWF) Data



[DWF]

Specifies dry weather flow and its quality entering the drainage system at specific nodes.

*Node Item Value (Pat1 Pat2 Pat3 Pat4)*

| | |
|---|---|
| *Node* | name of node where dry weather flow enters. |
| *Item* | keyword **FLOW** for flow or pollutant name for quality constituent. |
| *Value* | average baseline value for corresponding *Item* (flow or concentration units). |
| *Pat1*, *Pat2*, etc. | names of up to four time patterns appearing in the [**PATTERNS**] section. |

The actual dry weather input will equal the product of the baseline value and any adjustment factors supplied by the specified patterns. (If not supplied, an adjustment factor defaults to 1.0.)

**[AQUIFERS]**

Supplies parameters for each unconfined groundwater aquifer in the study area. Aquifers consist of two zones – a lower saturated zone and an upper unsaturated zone.

```
Name  Por  WP  FC  K  Ks  Ps  UEF  LED  GWR  BE  WTE  UMC
```

| | |
|---|---|
| *Name* | name assigned to aquifer. |
| *Por* | soil porosity (volumetric fraction). |
| *WP* | soil wilting point (volumetric fraction). |
| *FC* | soil field capacity (volumetric fraction). |
| *K* | saturated hydraulic conductivity (in/hr or mm/hr). |
| *Ks* | slope of hydraulic conductivity versus moisture content curve (in/hr or mm/hr. |
| *Ps* | slope of soil tension versus moisture content curve (inches or mm). |
| *UEF* | fraction of total evaporation available for evapotranspiration in the upper unsaturated zone. |
| *LED* | maximum depth into the lower saturated zone over which evapotranspiration can occur (ft or m). |
| *GWR* | rate of percolation from saturated zone to deep groundwater when water table is at ground surface (in/hr or mm/hr). |
| *BE* | elevation of the bottom of the aquifer (ft or m). |
| *WTE* | water table elevation at start of simulation (ft or m). |
| *UMC* | unsaturated zone moisture content at start of simulation (volumetric fraction). |

# Rainfall-derived Infiltration/Inflow

[RDII]

Specifies the parameters that describe rainfall-derived infiltration/inflow entering the drainage system at specific nodes.

*Node UHgroup SewerArea*

| | |
|---|---|
| *Node* | name of a node. |
| *UHgroup* | name of an RDII unit hydrograph group specified in the [HYDROGRAPHS] section. |
| *SewerArea* | area of the sewershed which contributes RDII to the node (acres or hectares). |

Useful Data

**Soil Characteristics**

| Soil Texture Class | K | Ψ | ϕ | FC | WP |
|---|---|---|---|---|---|
| Sand | 4.74 | 1.93 | 0.437 | 0.062 | 0.024 |
| Loamy Sand | 1.18 | 2.40 | 0.437 | 0.105 | 0.047 |
| Sandy Loam | 0.43 | 4.33 | 0.453 | 0.190 | 0.085 |
| Loam | 0.13 | 3.50 | 0.463 | 0.232 | 0.116 |
| Silt Loam | 0.26 | 6.69 | 0.501 | 0.284 | 0.135 |
| Sandy Clay Loam | 0.06 | 8.66 | 0.398 | 0.244 | 0.136 |
| Clay Loam | 0.04 | 8.27 | 0.464 | 0.310 | 0.187 |
| Silty Clay Loam | 0.04 | 10.63 | 0.471 | 0.342 | 0.210 |
| Sandy Clay | 0.02 | 9.45 | 0.430 | 0.321 | 0.221 |
| Silty Clay | 0.02 | 11.42 | 0.479 | 0.371 | 0.251 |
| Clay | 0.01 | 12.60 | 0.475 | 0.378 | 0.265 |

**K** = saturated hydraulic conductivity, in/hr
**Ψ** = suction head, in.
**ϕ** = porosity, fraction
**FC** = field capacity, fraction
**WP**= wilting point, fraction

Source: Rawls, W.J. et al., (1983). *J. Hyd. Engr.*, 109:1316.

**NRCS Hydrologic Soil Group Definitions**

| Group | Meaning | Saturated Hydraulic Conductivity (in/hr) |
|---|---|---|
| A | Low runoff potential. Soils having high infiltration rates even when thoroughly wetted and consisting chiefly of deep, well to excessively drained sands or gravels. | ≥ 0.45 |
| B | Soils having moderate infiltration rates when thoroughly wetted and consisting chiefly of moderately deep to deep, moderately well to well-drained soils with moderately fine to moderately coarse textures. E.g., shallow loess, sandy loam. | 0.30 - 0.15 |
| C | Soils having slow infiltration rates when thoroughly wetted and consisting chiefly of soils with a layer that impedes downward movement of water, or soils with moderately fine to fine textures. E.g., clay loams, shallow sandy loam. | 0.15 - 0.05 |
| D | High runoff potential. Soils having very slow infiltration rates when thoroughly wetted and consisting chiefly of clay soils with a high swelling potential, soils with a permanent high water table, soils with a clay-pan or clay layer at or near the surface, and shallow soils over nearly impervious material. | 0.05 - 0.00 |

**SCS Curve Numbers**[1]

| | Hydrologic Soil Group | | | |
|---|---|---|---|---|
| **Land Use Description** | **A** | **B** | **C** | **D** |
| Cultivated land | | | | |
| Without conservation treatment | 72 | 81 | 88 | 91 |
| With conservation treatment | 62 | 71 | 78 | 81 |
| Pasture or range land | | | | |
| Poor condition | 68 | 79 | 86 | 89 |
| Good condition | 39 | 61 | 74 | 80 |
| Meadow | | | | |
| Good condition | 30 | 58 | 71 | 78 |
| Wood or forest land | | | | |
| Thin stand, poor cover, no mulch | 45 | 66 | 77 | 83 |
| Good cover[2] | 25 | 55 | 70 | 77 |
| Open spaces, lawns, parks, golf courses, cemeteries, etc. | | | | |
| Good condition: grass cover on 75% or more of the area | 39 | 61 | 74 | 80 |
| Fair condition: grass cover on 50-75% of the area | 49 | 69 | 79 | 84 |
| Commercial and business areas (85% impervious) | 89 | 92 | 94 | 95 |
| Industrial districts (72% impervious) | 81 | 88 | 91 | 93 |

| | | | | |
|---|---|---|---|---|
| Residential[3] | | | | |
| Average lot size (% Impervious[4]) | | | | |
| 1/8 ac or less (65) | 77 | 85 | 90 | 92 |
| 1/4 ac (38) | 61 | 75 | 83 | 87 |
| 1/3 ac (30) | 57 | 72 | 81 | 86 |
| 1/2 ac (25) | 54 | 70 | 80 | 85 |
| 1 ac (20) | 51 | 68 | 79 | 84 |
| Paved parking lots, roofs, driveways, etc.[5] | 98 | 98 | 98 | 98 |
| Streets and roads | | | | |
| Paved with curbs and storm sewers[5] | 98 | 98 | 98 | 98 |
| Gravel | 76 | 85 | 89 | 91 |
| Dirt | 72 | 82 | 87 | 89 |

1. Antecedent moisture condition II; Source: *SCS Urban Hydrology for Small Watersheds*, 2nd Ed., (TR-55), June 1986.
2. Good cover is protected from grazing and litter and brush cover soil.
3. Curve numbers are computed assuming that the runoff from the house and driveway is directed toward the street with a minimum of roof water directed to lawns where additional infiltration could occur.
4. The remaining pervious areas (lawn) are considered to be in good pasture condition for these curve numbers.
5. In some warmer climates of the country a curve number of 95 may be used.

**Depression Storage**

| | |
|---|---|
| Impervious surfaces | 0.05 - 0.10 inches |
| Lawns | 0.10 - 0.20 inches |
| Pasture | 0.20 inches |
| Forest litter | 0.30 inches |

Source: ASCE, (1992). *Design & Construction of Urban Stormwater Management Systems,* New York, NY.

**Manning's n – Overland Flow**

| Surface | n |
|---|---|
| Smooth asphalt | 0.011 |
| Smooth concrete | 0.012 |
| Ordinary concrete lining | 0.013 |
| Good wood | 0.014 |
| Brick with cement mortar | 0.014 |
| Vitrified clay | 0.015 |
| Cast iron | 0.015 |
| Corrugated metal pipes | 0.024 |
| Cement rubble surface | 0.024 |
| Fallow soils (no residue) | 0.05 |
| Cultivated soils | |
| Residue cover < 20% | 0.06 |
| Residue cover > 20% | 0.17 |
| Range (natural) | 0.13 |
| Grass | |
| Short, prarie | 0.15 |
| Dense | 0.24 |
| Bermuda grass | 0.41 |
| Woods | |
| Light underbrush | 0.40 |
| Dense underbrush | 0.80 |

Source: McCuen, R. et al. (1996), *Hydrology*, FHWA-SA-96-067, Federal Highway Administration, Washington, DC

**Manning's n – Closed Conduits**

| Conduit Material | Manning n |
|---|---|
| Asbestos-cement pipe | 0.011 - 0.015 |
| Brick | 0.013 - 0.017 |
| Cast iron pipe<br>- Cement-lined & seal coated | <br>0.011 - 0.015 |
| Concrete (monolithic)<br>- Smooth forms<br>- Rough forms | <br>0.012 - 0.014<br>0.015 - 0.017 |
| Concrete pipe | 0.011 - 0.015 |
| Corrugated-metal pipe<br>(1/2-in. x 2-2/3-in. corrugations)<br>- Plain<br>- Paved invert<br>- Spun asphalt lined | <br><br>0.022 - 0.026<br>0.018 - 0.022<br>0.011 - 0.015 |
| Plastic pipe (smooth) | 0.011 - 0.015 |
| Vitrified clay<br>- Pipes<br>- Liner plates | <br>0.011 - 0.015<br>0.013 - 0.017 |

Source: ASCE (1982). *Gravity Sanitary Sewer Design and Construction*, ASCE Manual of Practice No. 60, New York, NY.

**Manning's n – Open Channels**

| Channel Type | Manning n |
|---|---|
| Lined Channels | |
| - Asphalt | 0.013 - 0.017 |
| - Brick | 0.012 - 0.018 |
| - Concrete | 0.011 - 0.020 |
| - Rubble or riprap | 0.020 - 0.035 |
| - Vegetal | 0.030 - 0.40 |
| Excavated or dredged | |
| - Earth, straight and uniform | 0.020 - 0.030 |
| - Earth, winding, fairly uniform | 0.025 - 0.040 |
| - Rock | 0.030 - 0.045 |
| - Unmaintained | 0.050 - 0.140 |
| Natural channels (minor streams, top width at flood stage < 100 ft) | |
| - Fairly regular section | 0.030 - 0.070 |
| - Irregular section with pools | 0.040 - 0.100 |

Source: ASCE (1982). *Gravity Sanitary Sewer Design and Construction*, ASCE Manual of Practice No. 60, New York, NY.

[POLLUTANTS]

Identifies the pollutants being analyzed.

*Name Units Crain Cgw Cii Kdecay (Sflag CoPoll CoFract)*

| | |
|---|---|
| *Name* | name assigned to pollutant. |
| *Units* | concentration units (**MG/L** for milligrams per liter, **UG/L** for micrograms per liter, or **#/L** for direct count per liter). |
| *Crain* | concentration of pollutant in rainfall (concentration units). |
| *Cgw* | concentration of pollutant in groundwater (concentration units). |
| *Cii* | concentration of pollutant in inflow/infiltration (concentration units). |
| *Kdecay* | first-order decay coefficient (1/days). |
| *Sflag* | **YES** if pollutant buildup occurs only when snowfall occurs, **NO** otherwise (default is **NO**). |
| *CoPoll* | name of co-pollutant (default is no co-pollutant). |
| *CoFract* | fraction of co-pollutant concentration (default is 0). |

**FLOW** is a reserved word and cannot be used to name a pollutant.

If pollutant buildup is not restricted to times of snowfall and there is no co-pollutant, then the last three parameters can be omitted.

When pollutant X has a co-pollutant Y, it means that fraction CoFract of pollutant Y's runoff concentration is added to pollutant X's runoff concentration when wash off from a subcatchment is computed.

[LOADINGS]

Specifies the pollutant buildup that exists on each subcatchment at the start of a simulation.

*Subcat Pollut InitBuildup Pollut InitBuildup ...*

| | |
|---|---|
| *Subcat* | name of a subcatchment. |
| *Pollut* | name of a pollutant. |
| *InitBuildup* | initial buildup of pollutant (lbs/acre or kg/hectare). |

More than one pair of pollutant - buildup values can be entered per line. If more than one line is needed, then the subcatchment name must still be entered first on the succeeding lines.

If an initial buildup is not specified for a pollutant, then its initial buildup is computed by applying the **DRY_DAYS** option (specified in the [OPTIONS] section) to the pollutant's buildup function for each land use in the subcatchment.

[LANDUSES]

Identifies the various categories of land uses within the drainage area. Each subcatchment area can be assigned a different mix of land uses. Each land use can be subjected to a different street sweeping schedule.

*Name (SweepInterval Availability LastSweep)*

| | |
|---|---|
| *Name* | land use name. |
| *SweepInterval* | days between street sweeping. |
| *Availability* | fraction of pollutant buildup available for removal by street sweeping. |
| *LastSweep* | days since last sweeping at start of the simulation. |

[WASHOFF]

Specifies the rate at which pollutants are washed off from different land uses during rain events.

*Landuse Pollutant FuncType C1 C2 SweepEffic BMPEffic*

| | |
|---|---|
| *Landuse* | land use name. |
| *Pollutant* | pollutant name. |
| *FuncType* | washoff function type: **EXP / RC / EMC**. |
| *C1, C2* | washoff function coefficients(see Table D-3). |
| *SweepEffic* | street sweeping removal efficiency (percent) . |
| *BMPEffic* | BMP removal efficiency (percent). |

| Name | Function | Equation | Units |
|---|---|---|---|
| EXP | Exponential | C1 (runoff)$^{C2}$ (buildup) | Mass/hour |
| RC | Rating Curve | C1 (runoff)$^{C2}$ | Mass/sec |
| EMC | Event Mean Concentration | C1 | Mass/Liter |

Each washoff function expresses its results in different units.

For the Exponential function the runoff variable is expressed in catchment depth per unit of time (inches per hour or millimeters per hour), while for the Rating Curve function it is in whatever flow units were specified in the [OPTIONS] section of the

input file (e.g., CFS, CMS, etc.). The buildup parameter in the Exponential function is the current buildup over the subcatchment's land use area in mass units. The units

of *C1* in the Exponential function are (in/hr) ^-1 per hour (or (mm/hr) ^-1 per hour). For the Rating Curve function, the units of *C1* depend on the flow units employed. For the EMC (event mean concentration) function, *C1* is always in concentration units.

[TREATMENT]

Specifies the degree of treatment received by pollutants at specific nodes of the drainage system.

*Node Pollut Result = Func*

| | |
|---|---|
| *Node* | Name of node where treatment occurs. |
| *Pollut* | Name of pollutant receiving treatment. |
| *Result* | Result computed by treatment function. Choices are:<br>**C** – function computes effluent concentration<br>**R** – function computes fractional removal. |
| *Func* | mathematical function expressing treatment result in terms of pollutant concentrations, pollutant removals, and other standard variables (see below). |

Treatment functions can be any well-formed mathematical expression involving:

- inlet pollutant concentrations (use the pollutant name to represent a concentration)
- removal of other pollutants (use R_ prepended to the pollutant name to represent removal)
- process variables which include:
  **FLOW** for flow rate into node (user's flow units)
  **DEPTH** for water depth above node invert (ft or m)
  **AREA** for node surface area (ft2 or m2)
  **DT** for routing time step (seconds)
  **HRT** for hydraulic residence time (hours)

[BUILDUP]

Specifies the rate at which pollutants build up over different land uses between rain events.

*Landuse Pollutant FuncType C1 C2 C3 PerUnit*

| | |
|---|---|
| *Landuse* | land use name. |
| *Pollutant* | pollutant name. |
| *FuncType* | buildup function type: ( **POW / EXP / SAT** ). |
| *C1, C2, C3* | buildup function parameters (see Table D-2). |

*PerUnit* **AREA** if buildup is per unit area, **CURBLENGTH** if per length of curb.

Buildup is measured in pounds (kilograms) per unit of area (or curb length) for pollutants whose concentration units are either mg/L or ug/L. If the concentration units are counts/L, then the buildup is expressed as counts per unit of area (or curb length).

**Table D-2. Available pollutant buildup functions (t is antecedent dry days).**

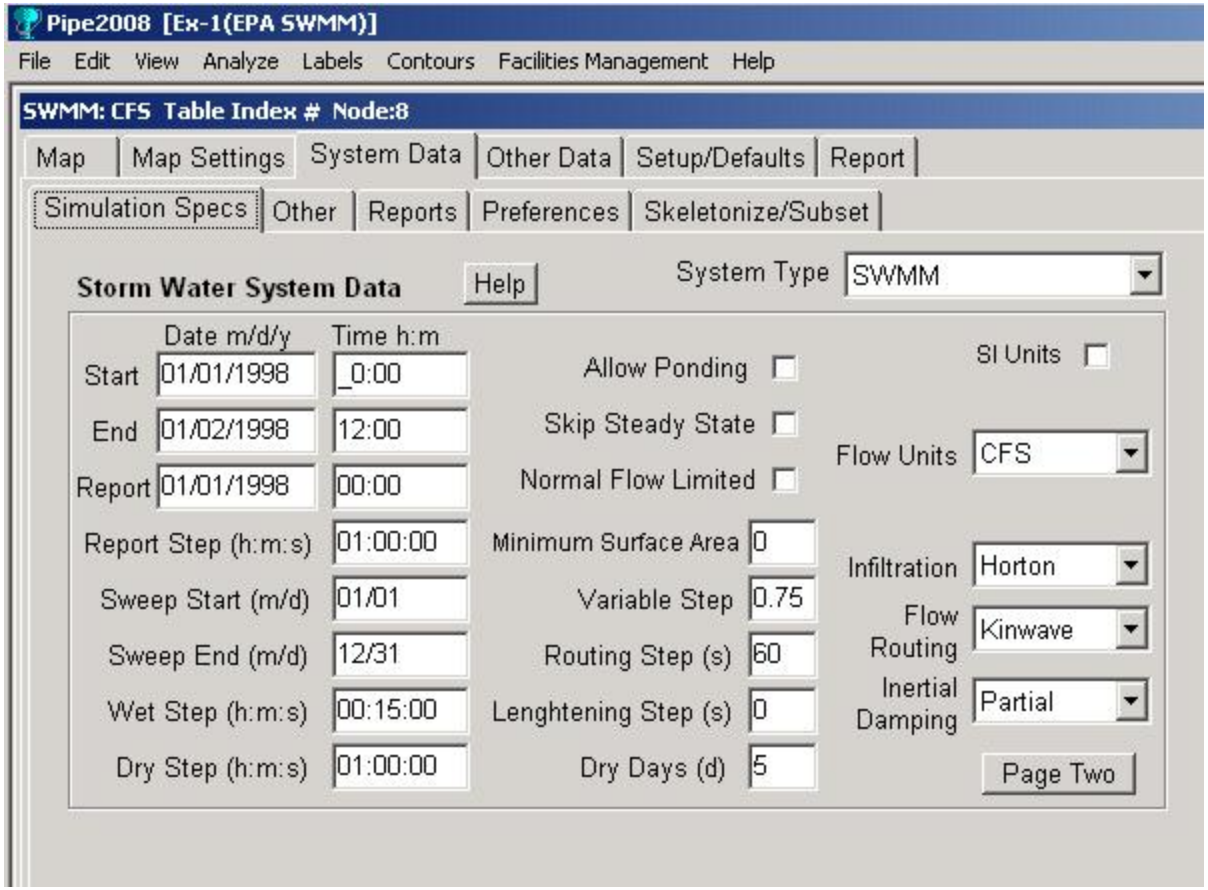
| Name | Function | Equation |
|---|---|---|
| POW | Power | Min (C1, $C2*t^{C3}$) |
| EXP | Exponential | C1*(1 – exp(-C2*t)) |

| | | |
|---|---|---|
| SAT | Saturation | C1*t / (C3 + t) |

**Water Quality Characteristics of Urban Runoff**

| Constituent | Event Mean Concentrations |
|---|---|
| TSS (mg/L) | 180 - 548 |
| BOD (mg/L) | 12 - 19 |
| COD (mg/L) | 82 - 178 |
| Total P (mg/L) | 0.42 - 0.88 |
| Soluble P (mg/L) | 0.15 - 0.28 |
| TKN (mg/L) | 1.90 - 4.18 |
| NO2/NO3-N (mg/L) | 0.86 - 2.2 |
| Total Cu (ug/L) | 43 - 118 |
| Total Pb (ug/L) | 182 - 443 |
| Total Zn (ug/L) | 202 - 633 |

Source: U.S. Environmental Protection Agency. (1983). *Results of the Nationwide Urban Runoff Program (NURP),* Vol. 1, NTIS PB 84-185552), Water Planning Division, Washington, DC.


Pipe2008 [Ex-1(EPA SWMM)]
File Edit View Analyze Labels Contours Facilities Management Help
SWMM: CFS Table Index # Node:8
Map Map Settings System Data Other Data Setup/Defaults Report
Simulation Specs Other Reports Preferences Skeletonize/Subset
Storm Water System Data
Help
System Type SWMM
Date m/d/y Time h:m
Start 01/01/1998 _0:00
End 01/02/1998 12:00
Report 01/01/1998 00:00
Report Step (h:m:s) 01:00:00
Sweep Start (m/d) 01/01
Sweep End (m/d) 12/31
Wet Step (h:m:s) 00:15:00
Dry Step (h:m:s) 01:00:00
Allow Ponding
Skip Steady State
Normal Flow Limited
Minimum Surface Area 0
Variable Step 0.75
Routing Step (s) 60
Lenghtening Step (s) 0
Dry Days (d) 5
SI Units
Flow Units CFS
Infiltration Horton
Flow Routing Kinwave
Inertial Damping Partial
Page Two

**REPORT_STEP** is the time interval for reporting of computed results. The default is 0:15:00.

**SWEEP_START** is the day of the year (month/day) when street sweeping operations begin. The default is 1/1.

**SWEEP_END** is the day of the year (month/day) when street sweeping operations end. The default is 12/31.

**WET_STEP** is the time step length used to compute runoff from subcatchments during periods of rainfall or when ponded water still remains on the surface. The default is 0:05:00.

**DRY_STEP** is the time step length used for runoff computations (consisting essentially of pollutant buildup) during periods when there is no rainfall and no ponded water. The default is 1:00:00.

**ALLOW_PONDING** determines whether excess water is allowed to collect atop nodes and be re-introduced into the system as conditions permit. The default is NO ponding. In order for ponding to actually occur at a particular node, a non-zero value for its Ponded Area attribute must be used.

**SKIP_STEADY_STATE** should be set to **YES** if flow routing computations should be skipped during steady state periods of a simulation during which the last set of computed flows will be used. A time step is considered to be in steady state if there has been no significant change in external inflows, storage volumes, and either node water depths (for dynamic wave routing) or conduit flows (for other forms of routing). The default for this option is **NO**.

**NORMAL_FLOW_LIMITED** should be set to **YES** if the criterion used to limit flow in a conduit to normal flow is that the Froude number exceeed 1.0. If set to **NO** (the default) then the normal flow limitation applies only when the water surface slope is less than the conduit invert slope.

**MIN_SURFAREA** is a minimum surface area used at nodes when computing changes in water depth under dynamic wave routing. If 0 is entered, then the default value of 12.566 ft2 (i.e., the area of a 4-ft diameter manhole) is used.

**VARIABLE_STEP** is a safety factor applied to a variable time step computed for each time period under dynamic wave flow routing. The variable time step is computed so as to satisfy the Courant stability criterion for each conduit, to prevent an excessive change in water depth at each node, and yet not exceed the **ROUTING_STEP** value. If the safety factor is 0 (the default), then no variable time step is used.

**ROUTING_STEP** is the time step length in secinds used for routing flows and water

quality constituents through the conveyance system. The default is 600 sec (5 minutes) which should be reduced if using dynamic wave routing. Fractional values (e.g., 2.5) are permissible as are values entered in hours:minutes:seconds format.

**LENGTHENING_STEP** is a time step, in seconds, used to lengthen conduits under dynamic wave routing, so that they meet the Courant stability criterion under full-flow conditions (i.e., the travel time of a wave will not be smaller than the specified conduit lengthening time step). As this value is decreased, fewer conduits will require lengthening. A value of 0 (the default) means that no conduits will be lengthened.

**DRY_DAYS** is the number of days with no rainfall prior to the start of the simulation. The default is 0.

**INFILTRATION** selects a model for computing infiltration of rainfall into the upper soil zone of subcatchments. The default model is **HORTON**.

**FLOW_ROUTING** determines which method is used to route flows through the drainage system. **STEADY** refers to sequential steady state routing (i.e. hydrograph translation), **KINWAVE** to kinematic wave routing, and **DYNWAVE** to dynamic wave routing. The default routing method is **KINWAVE**.

**INERTIAL_DAMPING** indicates how the inertial terms in the Saint Venant momentum equation will be handled under dynamic wave flow routing. Choosing **NONE** maintains these terms at their full value under all conditions. Selecting **PARTIAL** will reduce the terms as flow comes closer to being critical (and ignores them when flow is supercritical). Choosing **FULL** will drop the terms altogether.

[PATTERNS]

Specifies time pattern of dry weather flow or quality in the form of adjustment factors applied as multipliers to baseline values.

```
Name   MONTHLY    Factor1  Factor2  ...  Factor12
Name   DAILY      Factor1  Factor2  ...  Factor7
Name   HOURLY     Factor1  Factor2  ...  Factor24
Name   WEEKEND    Factor1  Factor2  ...  Factor24
```

*Name* name used to identify the pattern.
*Factor1*,
*Factor2*,
etc. multiplier values.

The **MONTHLY** format is used to set monthly pattern factors for dry weather flow constituents.

The **DAILY** format is used to set dry weather pattern factors for each day of the week, where Sunday is day 1.

The **HOURLY** format is used to set dry weather factors for each hour of the of the day starting from midnight. If these factors are different for weekend days than for weekday days then the **WEEKEND** format can be used to specify hourly adjustment factors just for weekends.

The pattern factors are applied as multipliers to any baseline dry weather flows or quality concentrations supplied in the [DWF] section.

## 1.7.2 Storm Example

Stormwater Example

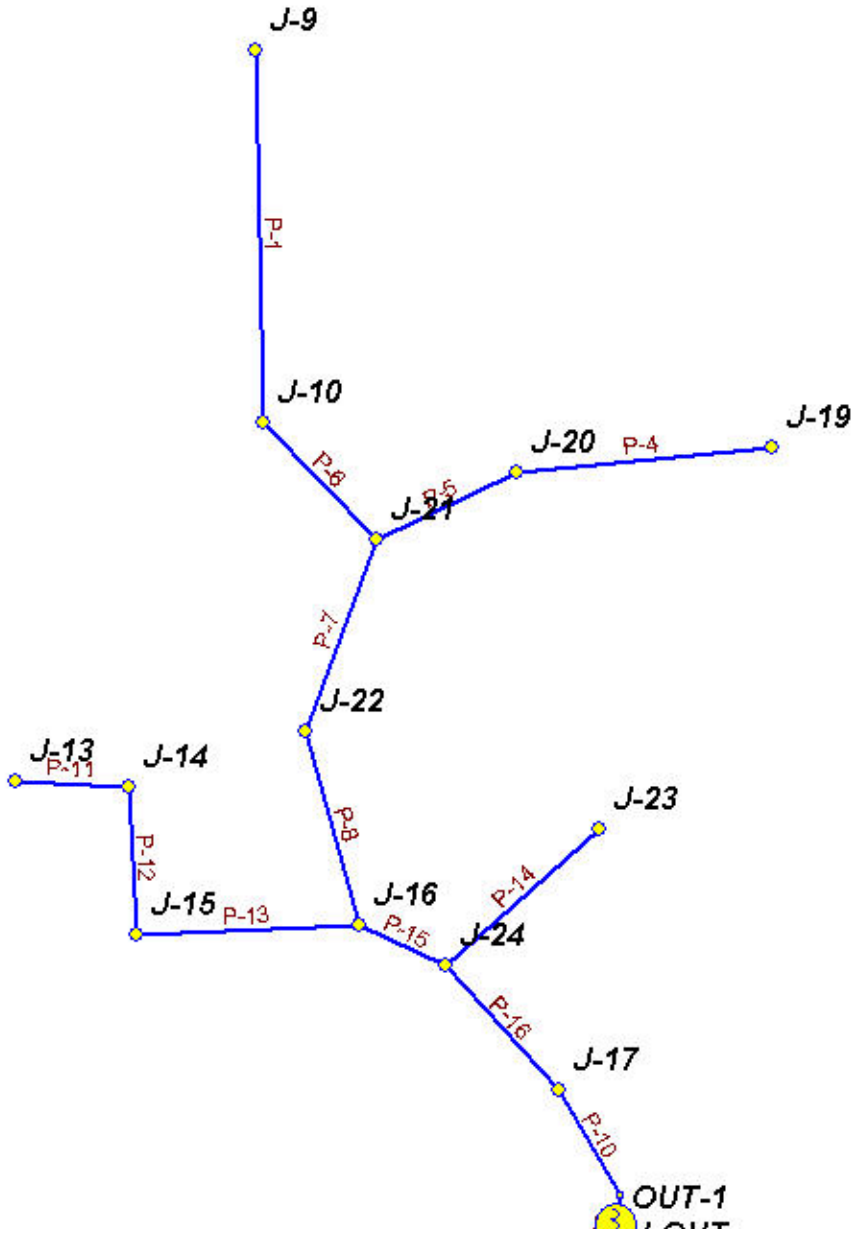
---

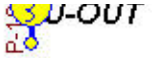
### Step-by-step Instructions - Example 1

This example is from the EPA SWMM 5 program

## Overview

The objective of this exercise is to develop a Pipe2018 : SWMM model for the "Example 1" stormwater network from EPA SWMM5 Manual and compare the Pipe2018 : SWMM results with those from EPA SWMM model results.






## Model Description and Suggestions from EPA SWMM Documentation.

Example1
========

This example models runoff quantity and quality in a small hypothetical drainage area. It consists of 8 subcatchments connected to a system of 1 to 2 foot diameter storm drains. There are two pollutants (TSS and lead) and two land uses (residential and undeveloped). Rainfall data for both a short-term 36-hour period and a long-term 2 year period are provided.

Here are some suggestions for running this example:

1. The model is set up to run in single event mode for a period of 36 hours using Kinematic Wave routing. Run the simulation.

2. View the Status Report for the run, noting the Continuity error.

3. Select some different output variables (e.g. subcatchment runoff, node flooding, link flow) for viewing on the study area map. Use the Day/Time controls on the Map Browser to move back and forth through time.

4. Try animating the map by using the Animator Toolbar (make the toolbar visible by selecting View | Toolbars | Animator from the Main Menu).

5. Try generating some other types of reports, such as a time series plot and a profile plot.

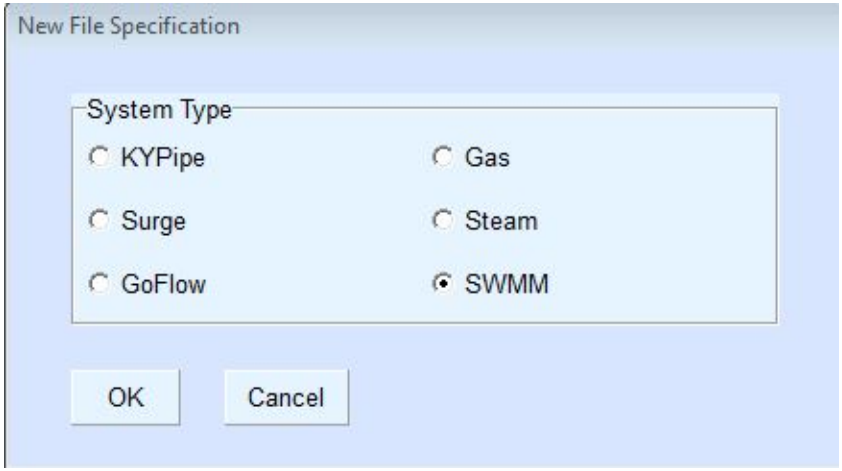
6. Change the Simulation Options to use Dynamic Wave routing and re-run the analysis.

7. Modify the model to run a continuous simulation as follows:
   1. Edit the rain gauge and change its Data Source to File, click on the File Name field and specify the rain data file named sta310301.dat, and enter 310301 for the Station Number.
   2. Edit the Simulation Options as follows:
      i. change the Routing Method back to Kinematic Wave
      ii. make the End Date 01-02-2000 (for a 2-year simulation)
      iii. change the Routing Time Step to 5 minutes.

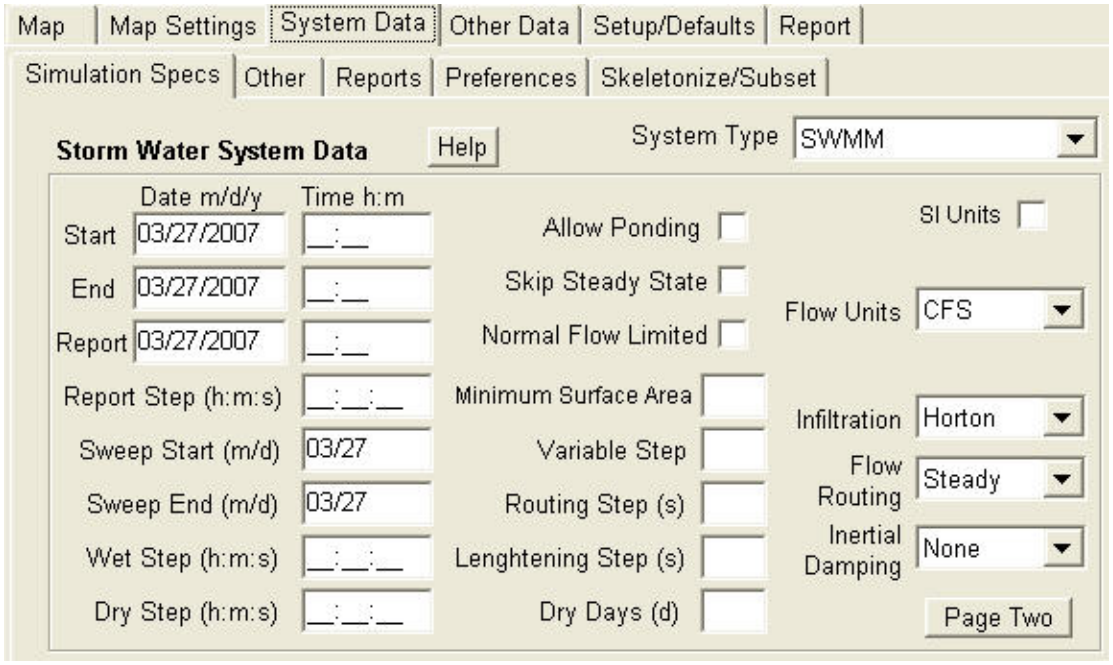
8. Run the simulation (it may take several minutes). Try generating a Statistics Report to see how the magnitude of various kinds of events (rainfall, runoff, quality, etc.) are distributed over time.

## Initial Preparation and Model Layout

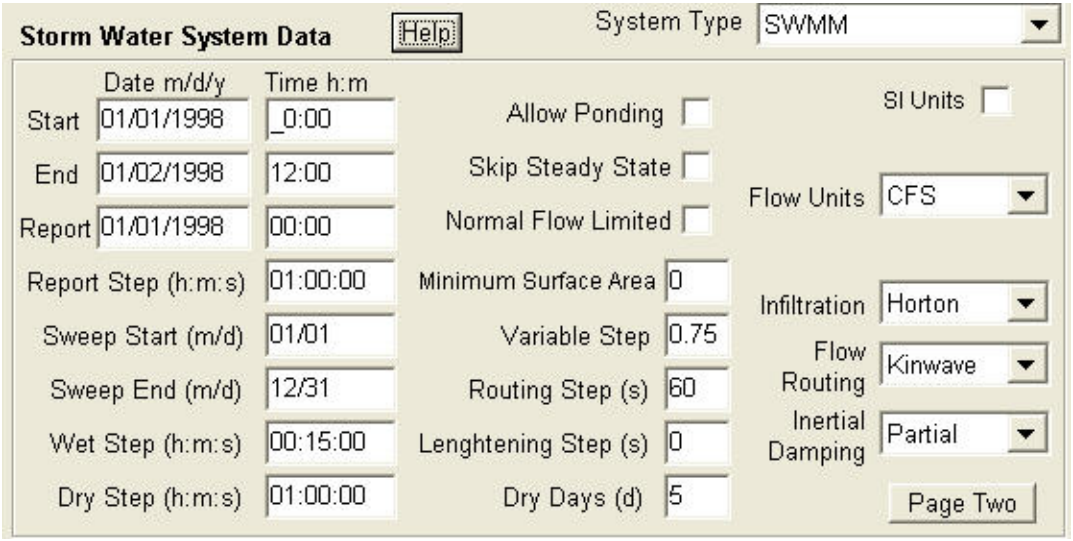
### Opening Screen: New File Specification in Storm




Under System Type, choose 'SWMM'. Click Okay. This will take you to the Simulation Specifications page.

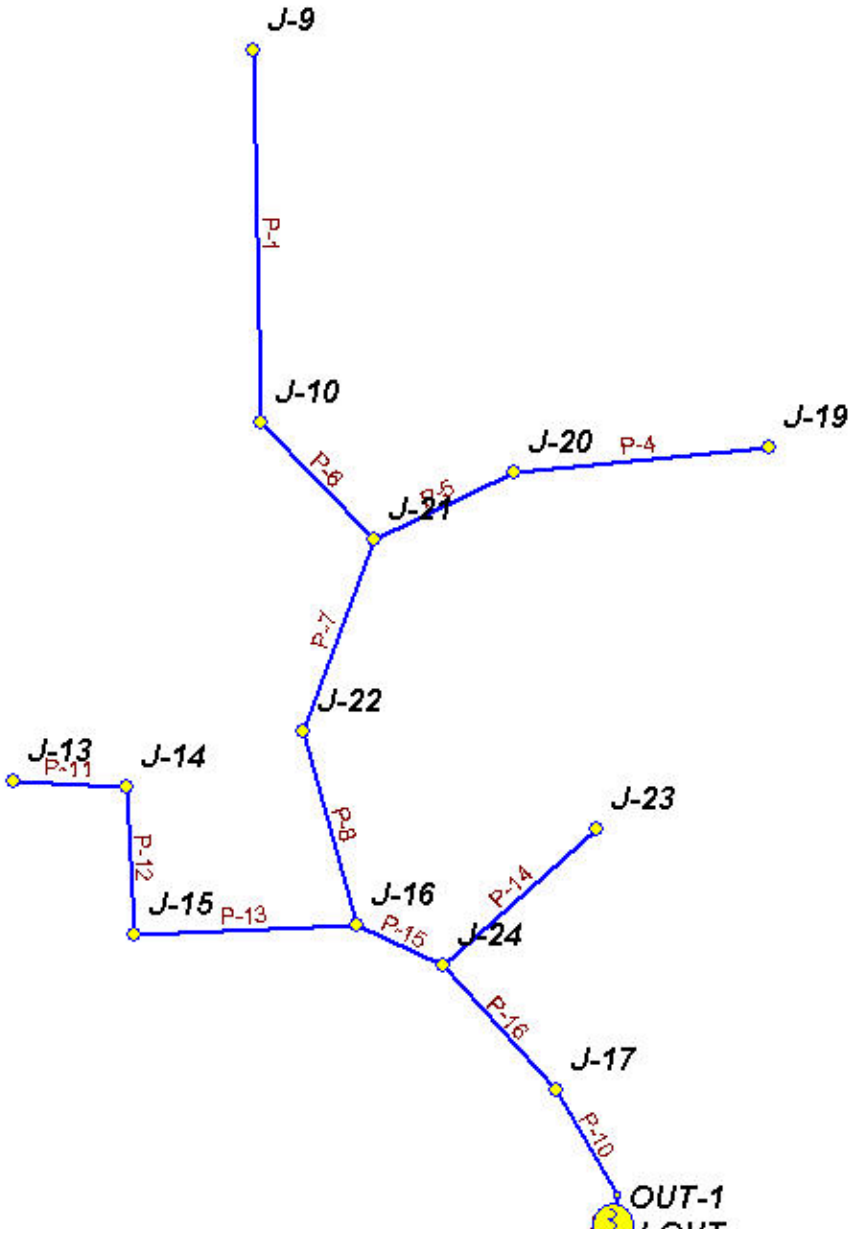



Provide Simulations Specifications Data as shown and click on Map Tab.

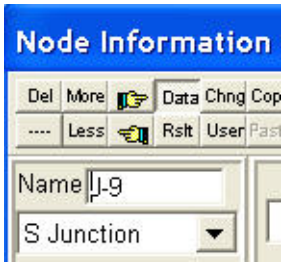



Network Schematic with Node and Pipe Names

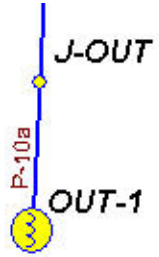
Lay out the entire network similar to what's shown in this slide. **Node and pipe names should be exactly as shown**. Required data will be provided later.




Junction Node




Outfall Node



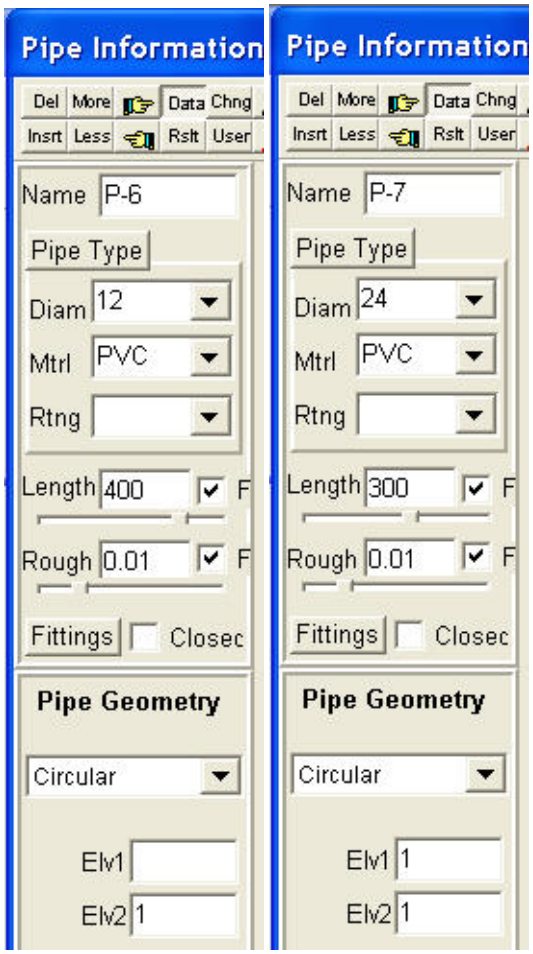
After laying out the file, go to File | Save As. Save the file as Ex-1.

## Provide Data

### Conduit Data:

After laying out the network model, click on each pipeline and provide the appropriate data for each pipe.

| Conduit Geometry | | | | | | | | |
|---|---|---|---|---|---|---|---|---|
| Name | Start Node | End Node | Shape | Length (ft) | Diameter (in) | Manning Roughness | Elv1 | Elv2 |
| P-1 | J-9 | J-10 | CIRCULAR | 400 | 18 | 0.01 | 0 | 0 |
| P-10 | J-17 | J-OUT | CIRCULAR | 400 | 24 | 0.01 | 0 | 0 |
| P-10a | J-OUT | OUT-1 | CIRCULAR | 10 | 24 | 0.01 | 0 | 0 |
| P-11 | J-13 | J-14 | CIRCULAR | 400 | 18 | 0.01 | 0 | 0 |
| P-12 | J-14 | J-15 | CIRCULAR | 400 | 18 | 0.01 | 0 | 0 |
| P-13 | J-15 | J-16 | CIRCULAR | 400 | 18 | 0.01 | 0 | 0 |
| P-14 | J-23 | J-24 | CIRCULAR | 400 | 12 | 0.01 | 0 | 0 |
| P-15 | J-16 | J-24 | CIRCULAR | 100 | 24 | 0.01 | 0 | 0 |
| P-16 | J-24 | J-17 | CIRCULAR | 400 | 24 | 0.01 | 0 | 0 |
| P-4 | J-19 | J-20 | CIRCULAR | 200 | 12 | 0.01 | 0 | 0 |
| P-5 | J-20 | J-21 | CIRCULAR | 200 | 12 | 0.01 | 0 | 0 |
| P-6 | J-10 | J-21 | CIRCULAR | 400 | 12 | 0.01 | 0 | 1 |
| P-7 | J-21 | J-22 | CIRCULAR | 300 | 24 | 0.01 | 1 | 1 |
| P-8 | J-22 | J-16 | CIRCULAR | 300 | 24 | 0.01 | 0 | 0 |




**Data for Pipe-6 and Pipe-7**

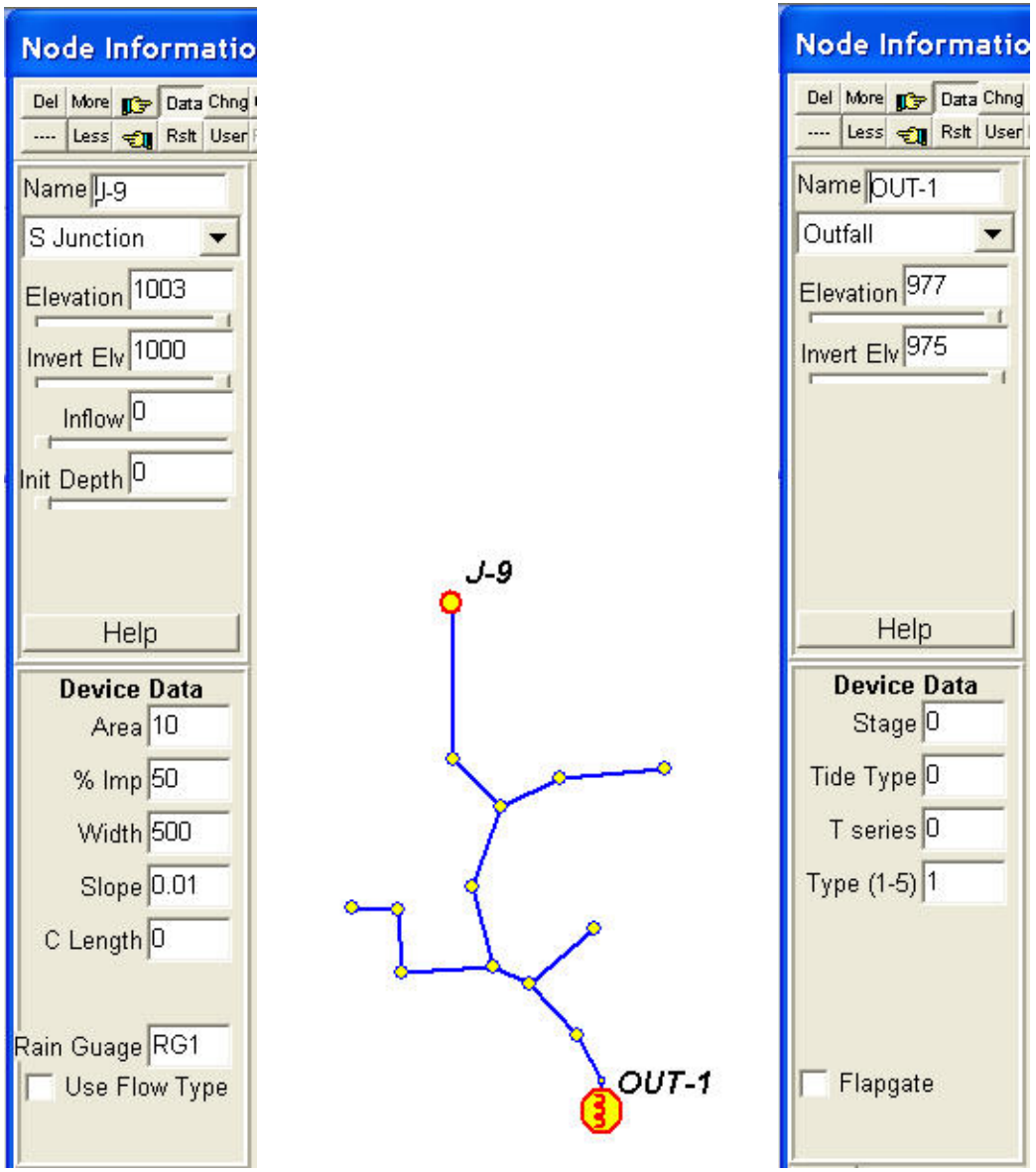
## Hydrological and other node data:

In Storm program, the catchment data is provided directly at the junction node.

Click on a junction node and provide the appropriate catchment and other node data. Repeat this for all nodes in the Network, including the outfall node OUT-1

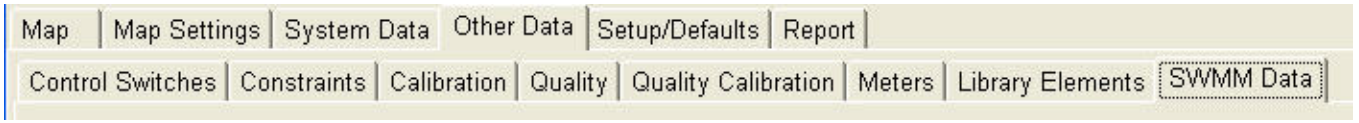
| Node Data | | | | | |
|---|---|---|---|---|---|
| Node | Type | Elevation (ground) ft | Invert Elevation (ft) | Inflow (cfs) | Initial Depth (ft) |
| J-10 | JUNCTION | 998 | 995 | 0 | 0 |
| J-13 | JUNCTION | 998 | 995 | 0 | 0 |
| J-14 | JUNCTION | 993 | 990 | 0 | 0 |
| J-15 | JUNCTION | 990 | 987 | 0 | 0 |
| J-16 | JUNCTION | 988 | 985 | 0 | 0 |
| J-17 | JUNCTION | 983 | 980 | 0 | 0 |
| J-19 | JUNCTION | 1013 | 1010 | 0 | 0 |
| J-20 | JUNCTION | 1008 | 1005 | 0 | 0 |
| J-21 | JUNCTION | 993 | 990 | 0 | 0 |
| J-22 | JUNCTION | 990 | 987 | 0 | 0 |
| J-23 | JUNCTION | 993 | 990 | 0 | 0 |
| J-24 | JUNCTION | 987 | 984 | 0 | 0 |
| J-9 | JUNCTION | 1003 | 1000 | 0 | 0 |
| J-OUT | JUNCTION | 978 | 975 | 0 | 0 |
| OUT-1 | OUTFALL | 977 | 975 | 0 | 0 |

| Hydrology: Catchment Data | | | | | |
|---|---|---|---|---|---|
| Node/subcatchment Name | Area (acres) | Width (ft) | %Imperv | %Slope | Rain Gage |
| J-10 | 10 | 500 | 50 | 0.01 | RG1 |
| J-13 | 5 | 500 | 50 | 0.01 | RG1 |
| J-15 | 15 | 500 | 50 | 0.01 | RG1 |
| J-19 | 4 | 500 | 10 | 0.01 | RG1 |
| J-22 | 5 | 500 | 50 | 0.01 | RG1 |
| J-23 | 12 | 500 | 10 | 0.01 | RG1 |
| J-9 | 10 | 500 | 50 | 0.01 | RG1 |
| J-OUT | 10 | 500 | 10 | 0.01 | RG1 |

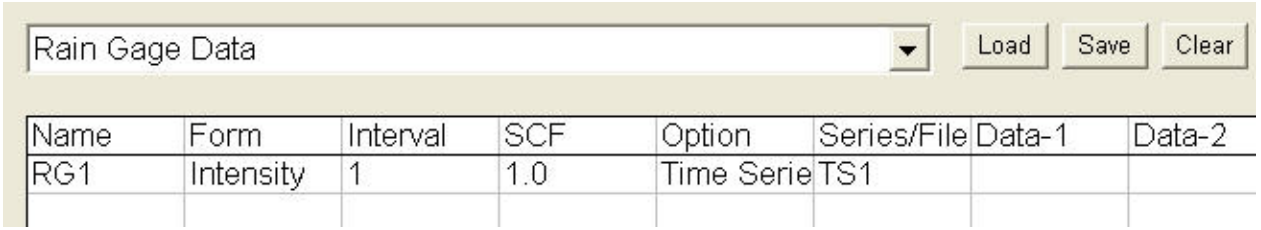



The next step is to provide meteorological and pollutant data.

Click on Other Data | SWMM Data.

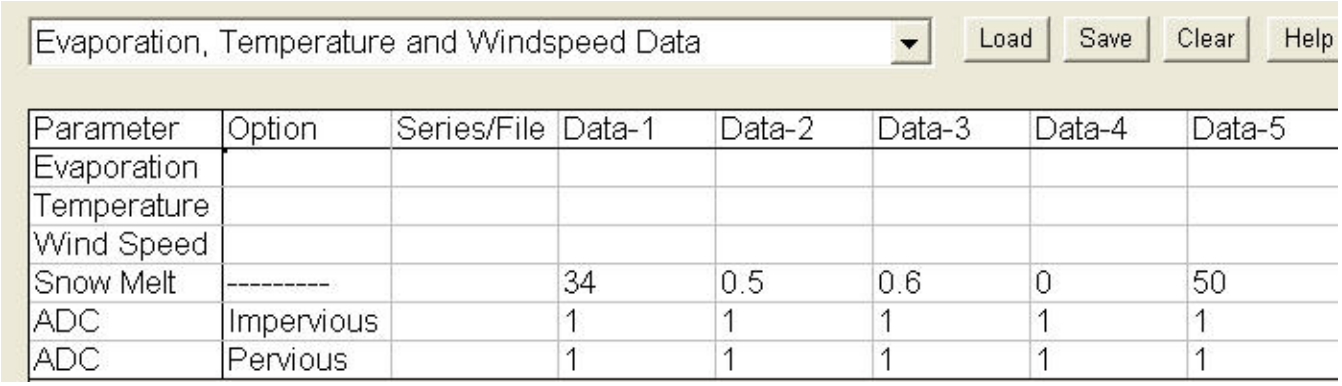



Fill in Rain gauge Data as shown….




| Name | Form | Interval | SCF | Option | Series/File | Data-1 | Data-2 |
|---|---|---|---|---|---|---|---|
| RG1 | Intensity | 1 | 1.0 | Time Serie | TS1 | | |
| | | | | | | | |

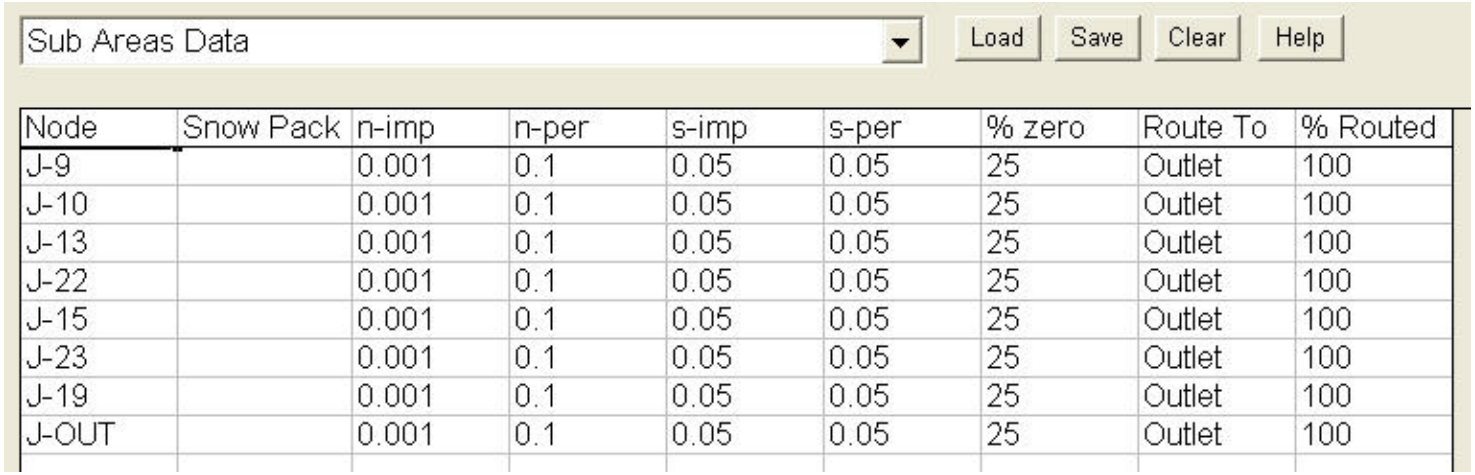
Then access “Evaporation….” Tab and fill in the data as shown….




| Parameter | Option | Series/File | Data-1 | Data-2 | Data-3 | Data-4 | Data-5 |
|---|---|---|---|---|---|---|---|
| Evaporation | | | | | | | |
| Temperature | | | | | | | |
| Wind Speed | | | | | | | |
| Snow Melt | --------- | | 34 | 0.5 | 0.6 | 0 | 50 |
| ADC | Impervious | | 1 | 1 | 1 | 1 | 1 |
| ADC | Pervious | | 1 | 1 | 1 | 1 | 1 |

| Parameter | Data-6 | Data-7 | Data-8 | Data-9 | Data-10 | Data-11 | Data-12 |
|---|---|---|---|---|---|---|---|
| Evaporation | | | | | | | |
| Temperature | | | | | | | |
| Wind Speed | | | | | | | |
| Snow Melt | 0 | | | | | | |
| ADC | 1 | 1 | 1 | 1 | 1 | 1 | 1 |
| ADC | 1 | 1 | 1 | 1 | 1 | 1 | 1 |

Similarly, fill in “Sub Areas Data” and “Infiltration Data”




| Node | Snow Pack | n-imp | n-per | s-imp | s-per | % zero | Route To | % Routed |
|---|---|---|---|---|---|---|---|---|
| J-9 | | 0.001 | 0.1 | 0.05 | 0.05 | 25 | Outlet | 100 |
| J-10 | | 0.001 | 0.1 | 0.05 | 0.05 | 25 | Outlet | 100 |
| J-13 | | 0.001 | 0.1 | 0.05 | 0.05 | 25 | Outlet | 100 |
| J-22 | | 0.001 | 0.1 | 0.05 | 0.05 | 25 | Outlet | 100 |
| J-15 | | 0.001 | 0.1 | 0.05 | 0.05 | 25 | Outlet | 100 |
| J-23 | | 0.001 | 0.1 | 0.05 | 0.05 | 25 | Outlet | 100 |
| J-19 | | 0.001 | 0.1 | 0.05 | 0.05 | 25 | Outlet | 100 |
| J-OUT | | 0.001 | 0.1 | 0.05 | 0.05 | 25 | Outlet | 100 |

Infiltration Data

| Node | Data-1 | Data-2 | Data-3 | Data-4 | Data-5 |
|---|---|---|---|---|---|
| J-9 | 0.35 | 0.25 | 4.14 | 0.5 | |
| J-10 | 0.7 | 0.3 | 4.14 | 0.5 | |
| J-13 | 0.7 | 0.3 | 4.14 | 0.5 | |
| J-22 | 0.7 | 0.3 | 4.14 | 0.5 | |
| J-15 | 0.7 | 0.3 | 4.14 | 0.5 | |
| J-23 | 0.7 | 0.3 | 4.14 | 0.5 | |
| J-19 | 0.7 | 0.3 | 4.14 | 0.5 | |
| J-OUT | 0.7 | 0.3 | 4.14 | 0.5 | |

Fill in Time Series Data as shown… Use data from the adjoining table for complete list (NOTE: TS1 data goes from time 0 through 30 hours and time values are entered consecutively, not necessarily correlating with column heading).

Time Series Data

Load Save Clear Help

| Name | mm/dd/yyyy | Time-1 | Value-1 | Time-2 | Value-2 | Time-3 | Value-3 | Time-4 | Value-4 | Time-5 | Value-5 |
|---|---|---|---|---|---|---|---|---|---|---|---|
| TS1 | | 0 | 0 | 1 | 0.25 | 2 | 0.5 | 3 | 0.8 | 4 | 0.4 |

| Time-6 | Value-6 | Time-7 | Value-7 | Time-8 | Value-8 | Time-9 | Value-9 | Time-10 | Value-10 | Time-11 | Value-11 | Time-12 |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 5 | 0.1 | 6 | 0 | 27 | 0 | 28 | 0.4 | 29 | 0.2 | 30 | 0 | |

| Time Series TS-1 | |
|---|---|
| Time | Value |
| 0 | 0 |
| 1 | 0.25 |
| 2 | 0.5 |
| 3 | 0.8 |
| 4 | 0.4 |
| 5 | 0.1 |
| 6 | 0 |
| 27 | 0 |
| 28 | 0.4 |
| 29 | 0.2 |
| 30 | 0 |

Then fill in pollutant and land use data….

Pollutants Data

Load Save Clear Help

| Name | Units | Conc(Rain) | Conc(GW) | Conc(i/i) | K(Decay) | Snow Flag | Co-Pollutant | Co-P Fraction |
|---|---|---|---|---|---|---|---|---|
| TSS | MG/L | 0 | 0 | 0 | 0 | No | | |
| Lead | UG/L | 0 | 0 | 0 | 0 | No | TSS | 0.2 |

Land Uses Data

| Name | Sweep Interv | Availability | Last Sweep |
|---|---|---|---|
| Residential | 0 | 0 | 0 |
| Undeveloped | 0 | 0 | 0 |

Fill in “Coverages Data”, “Buildup Data”, and “Washoff Data” as shown.

Coverages Data

| SubCatchment | Land Use-1 | Percent-1 | Land Use-2 | Percent-2 | La |
|---|---|---|---|---|---|
| J-9 | Residential | 100 | | | |
| J-10 | Residential | 50 | Undeveloped | 50 | |
| J-13 | Residential | 100 | | | |
| J-22 | Residential | 50 | Undeveloped | 50 | |
| J-15 | Residential | 100 | | | |
| J-23 | Residential | 0 | Undeveloped | 100 | |
| J-19 | Residential | 0 | Undeveloped | 100 | |
| J-OUT | Residential | 0 | Undeveloped | 100 | |

Buildup Data Load Save Cl

| Name | Pollutant | Func Type | C1 | C2 | C3 | Per Unit |
|---|---|---|---|---|---|---|
| Undeveloped | Lead | | | | | Area |
| Undeveloped | TSS | SAT | 100 | 0 | 3 | Area |
| Residential | Lead | | | | | Area |
| Residential | TSS | SAT | 50 | 0 | 2 | Area |

Washoff Data Load Save Clea

| Land Use | Pollutant | Func Type | C1 | C2 | Sweep Effic | BMP Effic |
|---|---|---|---|---|---|---|
| Residential | TSS | EXP | 0.1 | 1 | 0 | 0 |
| Residential | Lead | EMC | 0 | 0 | 0 | 0 |
| Undeveloped | TSS | EXP | 0.1 | 0.7 | 0 | 0 |
| Undeveloped | Lead | EMC | 0 | 0 | 0 | 0 |

Save the model…

## Analyze the Model

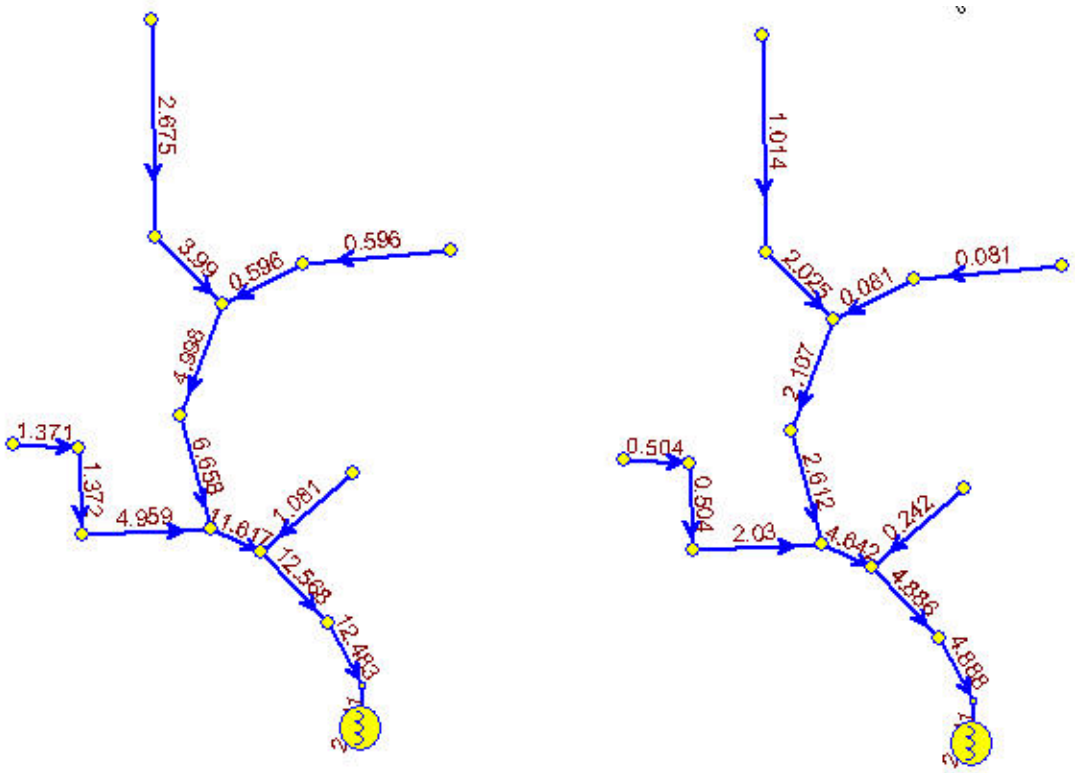
Analyze the model by clicking Analyze | Analysis | Analyze. This step generates the data file in EPA SWMM format, analyzes the model, and imports the results back into Pipe2018 : SWMM.

## Review Results

To view the tabulated (text) results generated by EPA SWMM, click on the Report tab.

To display results as labels on the map screen do the following:

For pipe results, select Labels in the main menu, then Pipe Results | Pipe Result A. Make sure the results are set to 'flow' in the Results Selector (next to "P" bottom right of screen). The pipe labels displayed are flowrates at time zero. To see the flowrates at different time periods, click on the right arrow button next to the "A" in the Results Selector area.

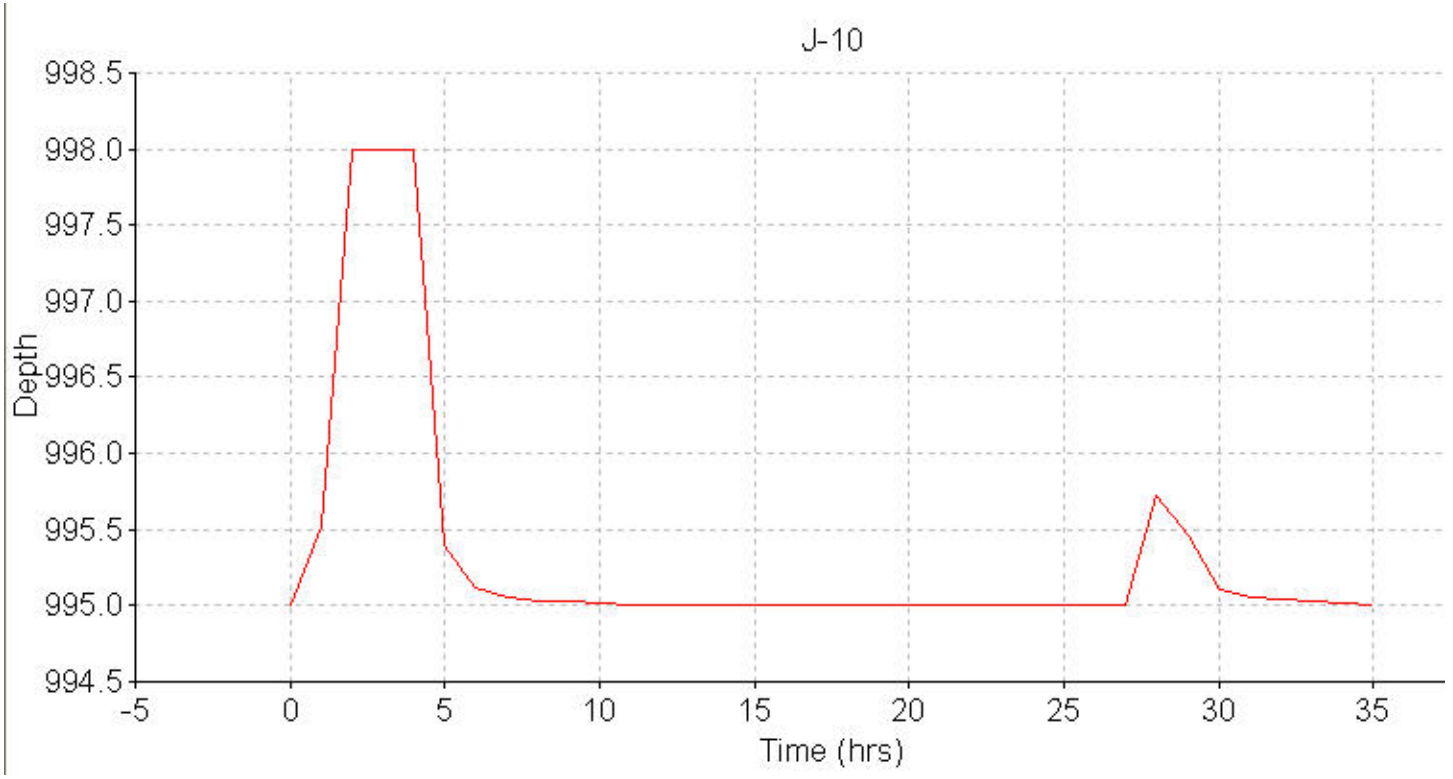



**Flowrates at hour 4 Flowrates at hour 29**

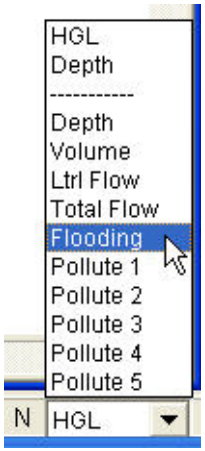
To view variation in depth at node J-10….

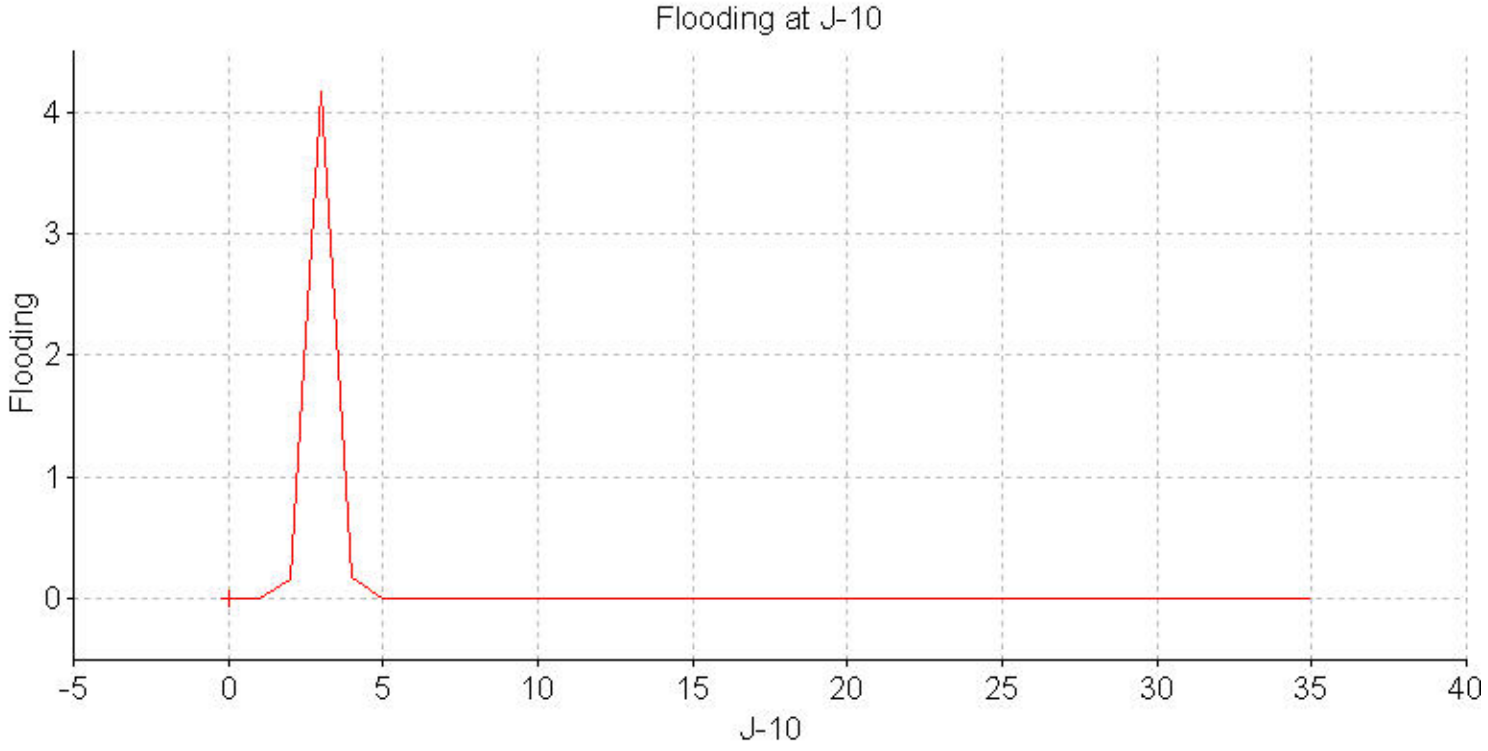
- Click on Node J-10 while in the layout mode.
- Click on “Rslt” button under Node Information Window
- Click on the node results pull down menu as shown
- Select “Depth” from the pull down menu
- These actions result in a node graph as shown below.

- Click on “Full” button to enlarge the depth graph

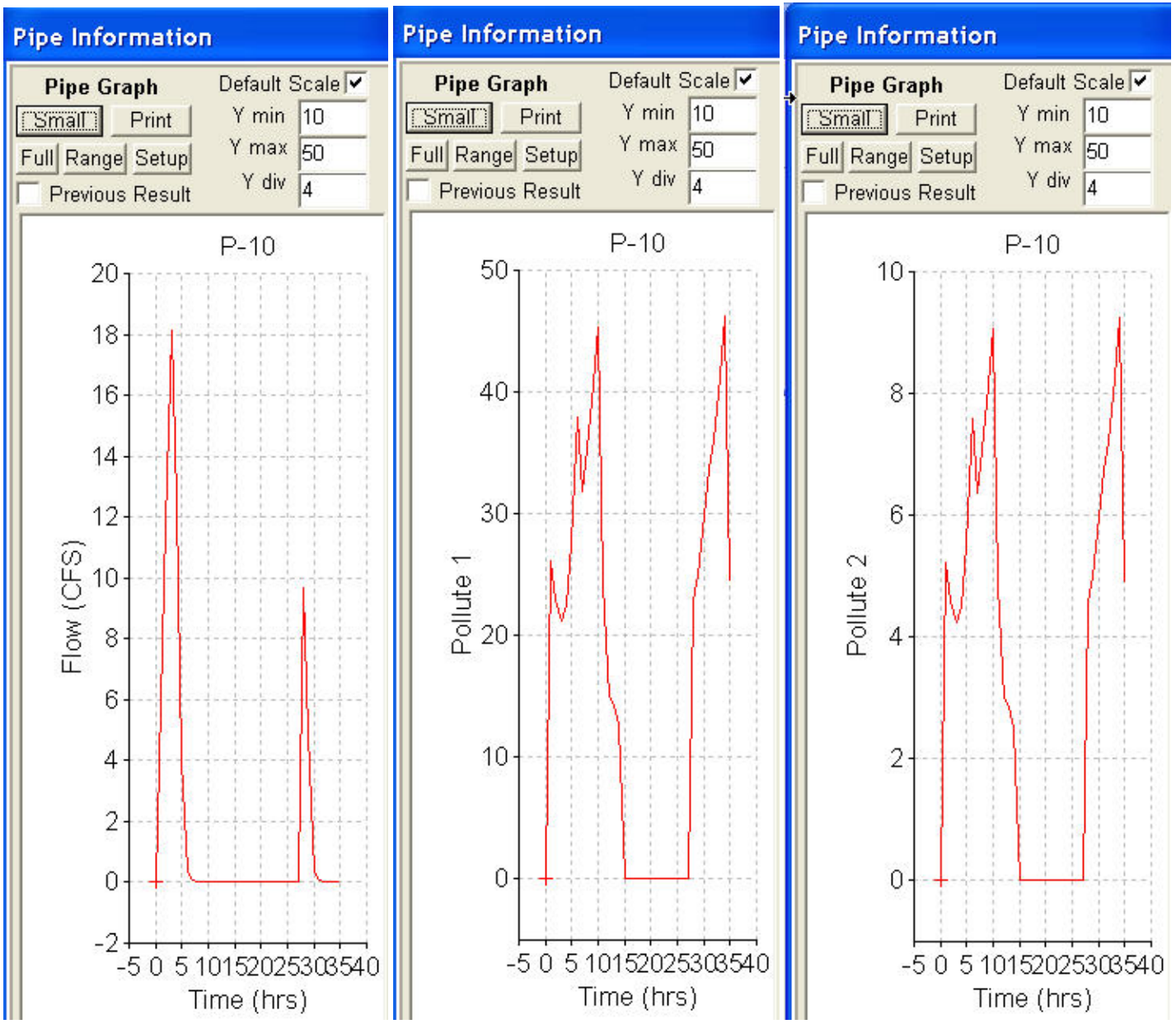



To display the amount of flooding at node J-10, simply change the node results parameter to “Flooding” under the pull down menu as shown.


HGL
Depth
---
Depth
Volume
Ltrl Flow
Total Flow
Flooding
Pollute 1
Pollute 2
Pollute 3
Pollute 4
Pollute 5
N
HGL


Flooding at J-10
Flooding
0
1
2
3
4
-5
0
5
10
15
20
25
30
35
40
J-10

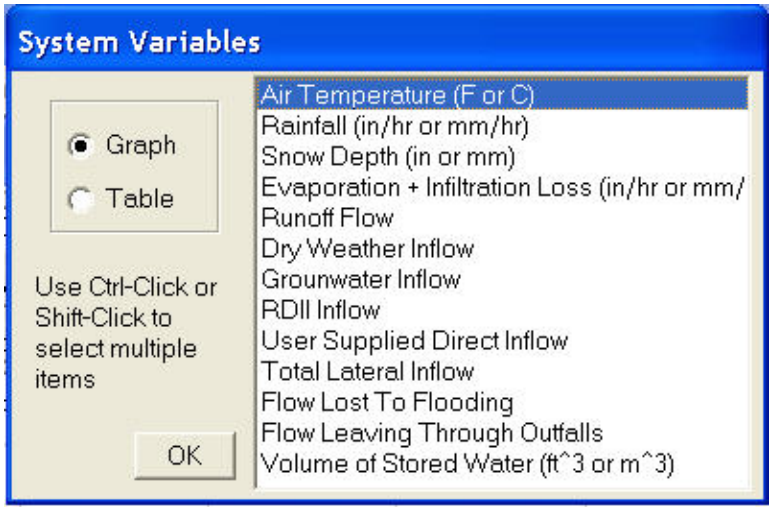
Similarly, several pipe parameters can be graphed….


Pipe Information
Pipe Graph
Default Scale
Small
Print
Y min 10
Y max 50
Full
Range
Setup
Y div 4
Previous Result
P-10
Flow (CFS)
Time (hrs)
Pollute 1
Pollute 2

**Flowrate variation in pipe P-10 Pollutant 1 (TSS) in P-10 Pollutant 2 (Lead) in P-10**

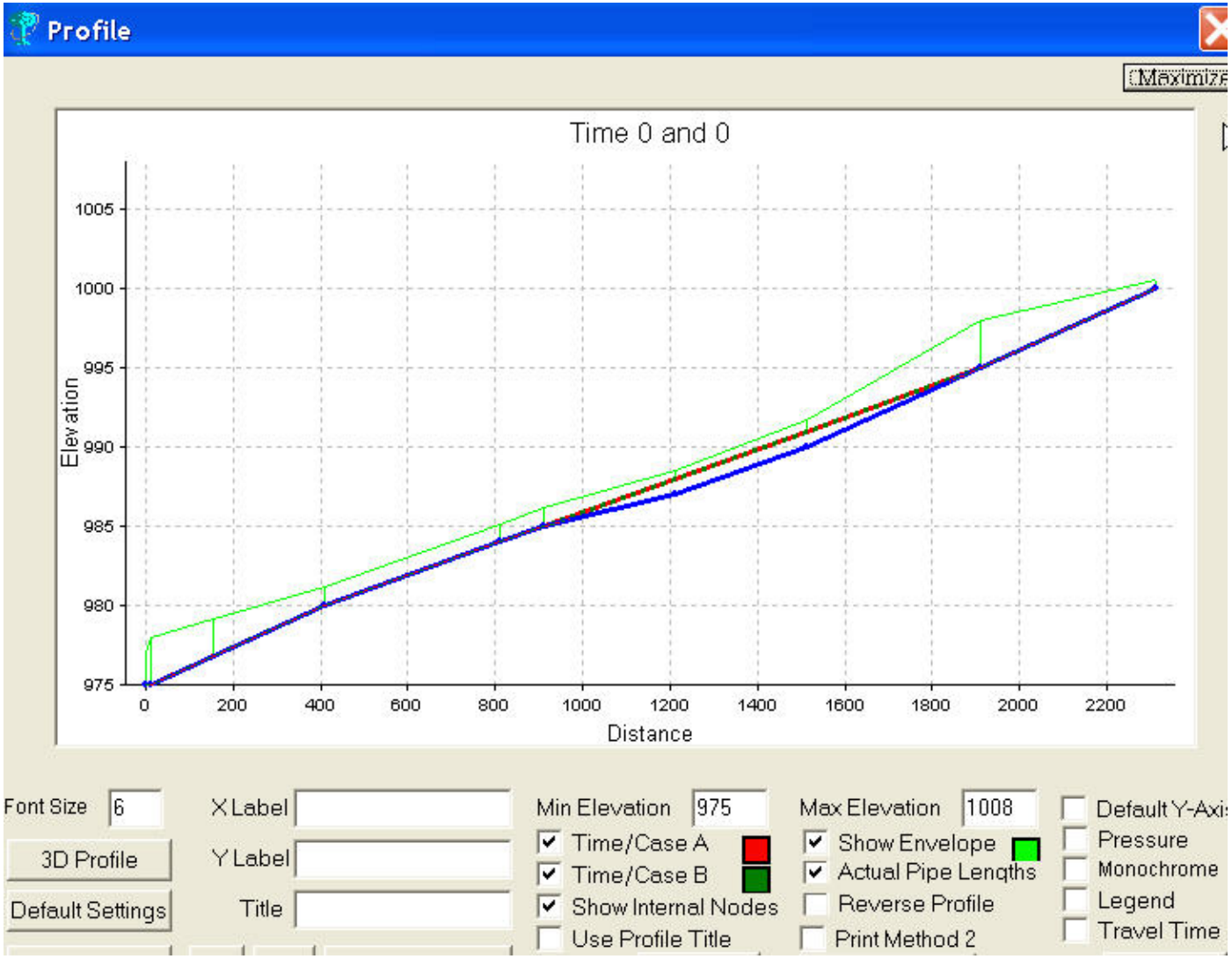
System variables such as rainfall, flow lost to flooding may be plotted as follows:

In main menu, click on View | System Variables Graph. A node graph along with the following menu will be displayed. Select the item to be plotted or tabulated.




To generate a profile plot along a path:

- Click on “Group” button and then on “Clear” button
- Click on the start node for the profile (e.g. J-9)
- Click on the end node for the profile (e.g. OUT-1)
- Click on “Analyze” and then on “Profile”
- Select “Create Profile from Last Selected Node”




## 1.7.3 SWMM Error Messages

The error messages listed below may appear in the SWMM-generated output report. This is found in the Report after an analysis has been run.

ERROR MESSAGES

ERROR 101: memory allocation error.
There is not enough physical memory in the computer to analyze the study area.

ERROR 103: cannot solve KW equations for Link xxx.
The internal solver for Kinematic Wave routing failed to converge for the specified link at some stage of the simulation.

ERROR 105: cannot open ODE solver.
The system could not open its Ordinary Differential Equation solver.

ERROR 107: cannot compute a valid time step.
A valid time step for runoff or flow routing calculations (i.e., a number greater than 0) could not be computed at some stage of the simulation.

ERROR 108: ambiguous outlet ID name for Subcatchment xxx.
The name of the element identified as the outlet of a subcatchment belongs to both a node and a subcatchment in the project's data base.

ERROR 109: invalid parameter values for Aquifer xxx.
The properties entered for an aquifer object were either invalid numbers or were inconsistent with one another (e.g., the soil field capacity was higher than the porosity).

ERROR 111: invalid length for Conduit xxx.
Conduits cannot have zero or negative lengths.

ERROR 113: invalid roughness for Conduit xxx.
Conduits cannot have zero or negative roughness values.

ERROR 114: invalid number of barrels for Conduit xxx.
Conduits must consist of one or more barrels.

ERROR 115: adverse slope for Conduit xxx.
Under Steady or Kinematic Wave routing, all conduits must have positive slopes. This can usually be corrected by reversing the inlet and outlet nodes of the conduit (i.e., right click on the conduit and select Reverse from the popup menu

that appears). Adverse slopes are permitted under Dynamic Wave routing.

ERROR 117: no cross section defined for Link xxx.
Cross section geometry was never defined for the specified link.

ERROR 119: invalid cross section for Link xxx.
Either an invalid shape or invalid set of dimensions was specified for a link's cross section.

ERROR 121: missing or invalid pump curve assigned to Pump xxx.
Either no pump curve or an invalid type of curve w-as specified for a pump.

ERROR 131: the following links form cyclic loops in the drainage system.
The Steady and Kinematic Wave flow routing methods cannot be applied to systems where a cyclic loop exists (i.e., a directed path along a set of links that begins and ends at the same node). Most often the cyclic nature of the loop can be eliminated by reversing the direction of one of its links (i.e.. switching the inlet and outlet nodes of the link). The names of the links that form the loop will be listed following this message.

ERROR 133: Node xxx has more than one outlet link.
Under Steady and Kinematic Wave flow routing, a junction node can have only a single outlet link.

ERROR 134: Node xxx has more than one DUMMY outlet link.
Only a single conduit with a DUMMY cross-section can be directed out of a node.

ERROR 135: Divider xxx does not have two outlet links.
Flow divider nodes must have two outlet links connected to them.

ERROR 136: Divider xxx has invalid diversion link.
The link specified as being the one carrying the diverted flow from a flow divider node was defined with a different inlet node.

ERROR 137: Weir Divider xxx has invalid parameters.
The parameters of a Weir-type divider node either are non-positive numbers or are inconsistent (i.e.. the value of the discharge coefficient times the weir height raised to the 3/2 power must be greater than the minimum flow parameter).

ERROR 138: Node xxx has initial depth greater than maximum depth.

Self-explanatory.

ERROR 139: Regulator xxx is the outlet of a non-storage node.
Under Steady or Kinematic Wave flow routing, orifices, weirs, and outlet links can only be used as outflow links from storage nodes.

ERROR 141: Outfall xxx has more than 1 inlet link or an outlet link.
An outfall node is only permitted to have one link attached to it.

ERROR 143: Regulator xxx has invalid cross-section shape.
An orifice must have either a CIRCULAR or RECT_CLOSED shape, while a weir must have either a RECT_OPEN, TRAPEZOIDAL, or TRIANGULAR shape.

ERROR 145: Drainage system has no acceptable outlet nodes.
Under Dynamic Wave flow routing, there must be at least one node designated as an outfall.

ERROR 151: a Unit Hydrograph in set xxx has invalid time base.
The time base of a Unit hydrograph must be greater than 0.

ERROR 153: a Unit Hydrograph in set xxx has invalid response ratios.
The response ratios for a set of Unit Hydrographs (the short-, medium-, and long-term response hydrographs) must be between 0 and 1.0 and cannot add up to a value greater than 1.0

ERROR 155: invalid sewer area for RDII at Node xxx.
The sewer area contributing RDII inflow to a node cannot be a negative number.

ERROR 161: cyclic dependency in treatment functions at Node xxx.
An example would be where the removal of pollutant 1 is defined as a function of the removal of pollutant 2 while the removal of pollutant 2 is defined as a function of the removal of pollutant 1.

ERROR 171: Curve xxx has its data out of sequence.
The X-values of a curve object must be entered in increasing order.

ERROR 173: Time Series xxx has its data out of sequence.
The time (or date/time) values of a time series must be entered in sequential order.

ERROR 181: invalid Snow Melt Climatology parameters.

The ATI Weight or Negative Melt Ratio parameters are not between 0 and 1 or the site latitude is not between -60 and +60 degrees.

ERROR 182: invalid parameters for Snow Pack xxx.
A snow pack's minimum melt coefficient is greater than its maximum coefficient; the fractions of free water capacity or impervious plowable area are not between 0 and 1; or the snow removal fractions sum to more than 1.0.

ERROR 191: simulation start date comes after ending date.
Self-explanatory.

ERROR 193: report start date comes after ending date.
Self-explanatory.

ERROR 195: reporting time step is less than routing time step.
Self-explanatory.

ERROR 200: one or more errors in input file.
This message appears when one or more input file parsing errors (the 200-series errors) occur.

ERROR 201: too many characters in input line.
A line in the input file cannot exceed 1024 characters.

ERROR 203: too few items at line n of input file.
Not enough data items were supplied on a line of the input file.

ERROR 205: invalid keyword at line n of input file.
An unrecognized keyword was encountered when parsing a line of the input file.

ERROR 207: duplicate ID name at line n of input file.
An ID name used for an object was already assigned to an object of the same category.

ERROR 209: undefined object xxx at line n of input file.
A reference was made to an object mat was never defined. An example would be if node 123 were designated as the outlet point of a subcatchment. yet no such node was ever defined in the study area.

ERROR 211: invalid number xxx at line n of input file.
Either a non-numeric character was encountered where a numerical value was

expected or an invalid number (e.g., a negative value) was supplied.

ERROR 213: invalid date/time xxx at line n of input file.
An invalid format for a date or time was encountered. Dates must be entered as month/day/year and times as either decimal hours or as hour:minute:second.

ERROR 217: control rule clause out of sequence at line n of input file.
Errors of this nature can occur when the format for writing control rules is not followed correctly (see Section C.3).

ERROR 219: data provided for unidentified transect at line n of input file.
A GR line with Station-Elevation data was encountered in the [TRANSECTS] section of the input file after an NC line but before any XI line that contains the transect"s ID name.

ERROR 221: transect station out of sequence at line n of input file.
The station distances specified for the transect of an irregular cross section must be in increasing numerical order starting from the left bank.

ERROR 223: Transect xxx has too few stations.
A transect for an irregular cross section must have at least 2 stations defined for it.

ERROR 225: Transect xxx has too many stations.
A transect cannot have more than 1500 stations defined for it.

ERROR 227: Transect xxx has no Manning's N.
No Manning's N was specified for a transect (i.e., there was no NC line in the [TRANSECTS] section of the input file.

ERROR 229: Transect xxx has invalid overbank locations.
The distance values specified for either the left or right overbank locations of a transect do not match any of the distances listed for the transect's stations.

ERROR 231: Transect xxx has no depth.
All of the stations for a transect were assigned the same elevation.

ERROR 233: invalid treatment function expression at line n of input file.
treatment function supplied for a pollutant at a specific node is either not a correctly formed mathematical expression or refers to unknown pollutants, process variables, or math functions.

ERROR 301: files share same names.
The input, report, and binary output files specified on the command line cannot have the same names.

ERROR 303: cannot open input file.
The input file either does not exist or cannot be opened (e.g.. it might be in use by another program).

ERROR 305: cannot open report file.
The report file cannot be opened (e.g., it might reside in a directory to which the user does not have write privileges).

ERROR 307: cannot open binary results file.
The binary output file cannot be opened (e.g., it might reside in a directory to which the user does not have write privileges).

ERROR 309: error writing to binary results file.
There was an error in trying to write results to the binary output file (e.g., the disk might be full or the file size exceeds the limit imposed by the operating system).

ERROR 311: error reading from binary results file.
The command line version of SWMM could not read results saved to the binary output file when writing results to the report file.

ERROR 313: cannot open scratch rainfall interface file.
SWMM could not open the temporary file it uses to collate data together from external rainfall files.

ERROR 315: cannot open rainfall interface file xxx.
SWMM could not open the specified rainfall interface file, possibly because it does not exist or because the user does not have write privileges to its directory.

ERROR 317: cannot open rainfall data file xxx.
An external rainfall data file could not be opened, most likely because it does not exist.

ERROR 319: invalid format for rainfall interface file.
SWMM was trying to read data from a designated rainfall interface file with the wrong format (i.e., it may have been created for some other project or actually be some other type of file).

ERROR 321: no data in rainfall interface file for gauge xxx.
This message occurs when a project wants to use a previously saved rainfall interface file, but cannot find any data for one of its rain gauges in the interface file.

ERROR 323: cannot open runoff interface file xxx.
A runoff interface file could not be opened, possibly because it does not exist or because the user does not have write privileges to its directory.

ERROR 325: incompatible data found in runoff interface file.
SWMM was trying to read data from a designated runoff interface file with the wrong format (i.e., it may have been created for some other project or actually be some other type of file).

ERROR 327: attempting to read beyond end of runoff interface file.
This error can occur when a previously saved runoff interface file is being used in a simulation with a longer duration than the one that created the interface file.

ERROR 329: error in reading from runoff interface file.
A format error was encountered while trying to read data from a previously saved runoff interface file.

ERROR 331: cannot open hotstart interface file xxx.
A hotstart interface file could not be opened, possibly because it does not exist or because the user does not have write privileges to its directory.

ERROR 333: incompatible data found in hotstart interface file.
SWMM was trying to read data from a designated hotstart interface file with the wrong format (i.e., it may have been created for some other project or actually be some other type of file).

ERROR 335: error in reading from hotstart interface file.
A format error was encountered while trying to read data from a previously saved hotstart interface file.

ERROR 336: no climate file specified for evaporation and/or wind speed.
This error occurs when the user specifies that evaporation or wind speed data will be read from an external climate file, but no name is supplied for the file.

ERROR 337: cannot open climate file xxx.
An external climate data file could not be opened, most likely because it does not exist.

ERROR 338: error in reading from climate file xxx.
SWMM was trying to read data from an external climate file with the wrong format.

ERROR 339: attempt to read beyond end of climate file xxx.
The specified external climate does not include data for the period of time being simulated.

ERROR 341: cannot open scratch RDII interface file.
SWMM could not open the temporary file it uses to store RDII flow data.

ERROR 343: cannot open RDII interface file xxx.
An RDII interface file could not be opened, possibly because it does not exist or because the user does not have write privileges to its directory.

ERROR 345: invalid format for RDII interface file.
SWMM was trying to read data from a designated RDII interface file with the wrong format (i.e.. it may have been created for some other project or actually be some other type of file).

ERROR 351: cannot open routing interface file xxx.
A routing interface file could not be opened, possibly because it does not exist or because the user does not have write privileges to its directory.

ERROR 353: invalid format for routing interface file xxx.
SWMM was trying to read data from a designated routing interface file with the wrong format (i.e., it may have been created for some other project or actually be some other type of file).

ERROR 355: mismatched names in routing interface file xxx.
The names of pollutants found in a designated routing interface file do not match the names used in the current project.

ERROR 357: inflows and outflows interface files have same name.
In cases where a run uses one routing interface file to provide inflows for a set of locations and another to save outflow results, the two files cannot both have the same name.

## 1.9 Gas and Steam

## 1.8.1 Gas

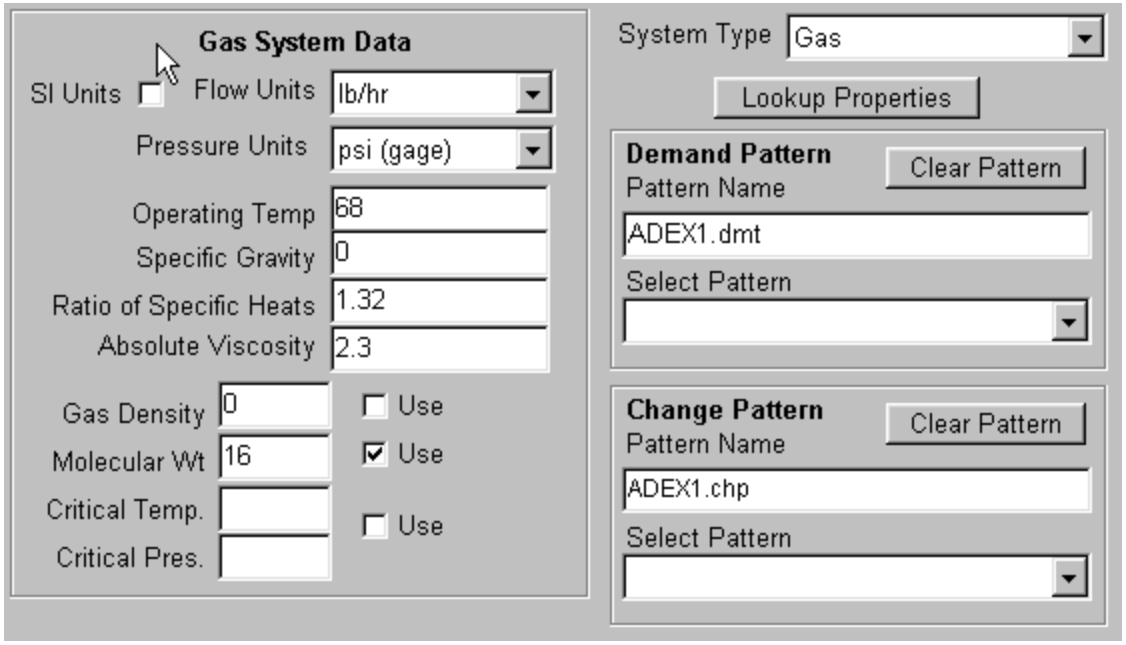
Gas

---

Gas is the Pipe2018 compressible flow model.

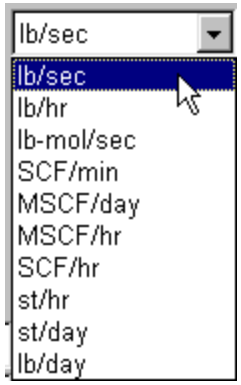
**System Data**

To analyze a gas model, go to the System Data | Simulation Specs tab or in the KYNetic interface. There is a drop-down selector box called System Type. Choose Gas. Once Gas has been chosen, relevant System Data fields will appear to the left, as shown below. There are three options for Gas. Each will require some (but not all) of the data shown.




All options require the following data:
SI Units - This allows the user to toggle between SI (metric) and English units for the analysis and results. A check in this box means SI units are selected.

Flow Units - This allows the user to select the flow units in which the flow results will be displayed.




lb/sec (pounds/second)

lb/hr (pounds/hour)

lb-mol/sec (pound-moles/second)

SCF/min (standard cubic feet/minute)

MSCF/day (thousand standard cubic feet/day)

MSCF/hr (thousand standard cubic feet/hour)

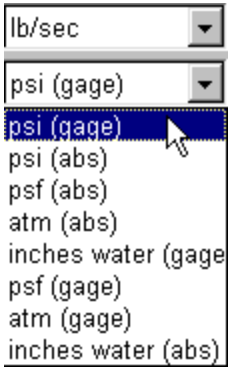
SCF/hr (standard cubic feet/hour)

st/hr (standard tons/hour)

st/day (standard tons/hour)

lb/day (pounds/day)

Pressure Units - This allows the user to select the pressure units in which the pressure results will be displayed.




**Some of the following data is required depending on which option is chosen:**

Operating Temp - Enter the operating temperature in degrees Fahrenheit (English) or degrees Celsius (SI).

Specific Gravity (this data or Molecular Weight required) * - Enter the specific gravity (ratio to air density at standard (atmospheric - 68 degrees F or 20 degrees C) conditions). Alternately, molecular weight may be entered, see below.

Ratio of Specific Heats * - Enter the ratio of specific heats (must be > 1).

Absolute Viscosity * - Enter the absolute viscosity (lb sec./ft*ft or Newton sec/m*m).

Gas Density - Enter the density here (in lb sec/ft*ft*ft or N/m*m*m). The Use box must checked if the constant gas density feature is to be used in the analysis. This entry will key the use of constant density for the analysis. Otherwise, no data is required in this field.

Molecular Wt.(or Specific Gravity required) * - To use molecular weight instead of specific gravity, click on the Use box. Then enter the molecular weight.

Critical Temp. (non-ideal gas) * - (degrees Rankine or Kelvin) If this data is entered, the analysis of the system will be conducted as a non-ideal gas.

Critical Pres. (non-ideal gas) . * - (psia or MPa (megaPascals)) If this data is entered, the analysis of the system will be conducted as a non-ideal gas.

* A lookup module for gas properties is provided.

Demand Patterns and Change Patterns - see Simulation Specs

## The Gas Model

Gas is written to analyze for the steady state flows and pressures in gas distribution networks. The ideal gas law is used to describe the pressure-temperature-density relationships. Flow in a Gas model is steady, one-dimensional, and isothermal. A procedure is a available to account for different temperatures in various lines.

There are three options for a gas analysis. Each option is specified by the data entered into the System Data box. Examples of the system data required to specify

each analysis type is provided.

1. Ideal Gas
2. Non-Ideal Gas
3. Constant Density Gas

Gas uses variable density (density that changes with pressure during the iterative process and at different locations in the network) unless the user checks either Gas Density option or Mol. Weight option. If any of these options are checked, the program uses fixed density values (computes the density if mol. weight is specified) irrespective of pressure conditions. In all cases, a constant viscosity value is used.

Elements

A gas analysis supports the following elements: supply points (reservoirs and pressure supplies), compressors or fans (pumps), pressure regulating valves, loss elements, valves and check valves, meters, and fittings. Below the data requirements are summarized.

Input (minimum required data in red)

No initial flow or pressure assumptions are required. Any number of parameter changes may be made within the same analysis. At least one Fixed Pressure Node or Reservoir is required for each system.

Pipes - length, diameter (inside), Darcy Weisbach roughness, fittings (minor loss).

Junctions - demand, elevation.

Reservoirs (Fixed Pressure Supply) - static pressure, elevation.

Pumps (Compressor Nodes)- useful power or operating curve (three pressure/flow data points), elevation.

Regulators (Pressure Regulating Valves) - downstream pressure setting, elevation.

Active Valves (standard valves) - Flow Coefficient, Cv, or Minor Loss, K, value for 100% open valve, open ratio, elevation, pressure if one end is not connected to a pipe. When modeling an Active Valve, use a minor loss coefficient, K, or flow coefficient, Cv, calculated from one of the available tools in order to obtain an appropriate equivalent value. Gas properties, flowrate, and pressure drop must be considered when determining the appropriate Cv or K for compressible fluid models

Loss Elements - three pressure drop/flow data points.

Pressure Supply (Variable Pressure Supply) - 1. static pressure, one or two data point for residual pressure and residual flow , elevation, one additional pressure/flow data point optional.

Check Valve - elevation

Valve - elevation

Metered Connection - elevation, meter ID and demand

Intermediate Nodes - elevation

Device 1 - elevation, user data

Device 2 - elevation, user data

Output

The output may be customized either using the Customize report feature or the System Data | Report tab.

Nodes - density, absolute and gauge pressure

Pipes - flow, pressure loss, mean flow velocity

System - summary of system inflows and loads, tables of maximum and minimum pressures, velocities, and losses per length, choking conditions (maximum Mach number), and minimum flow areas.

## 1.8.2 Constant Density Gas

Constant Density Gas

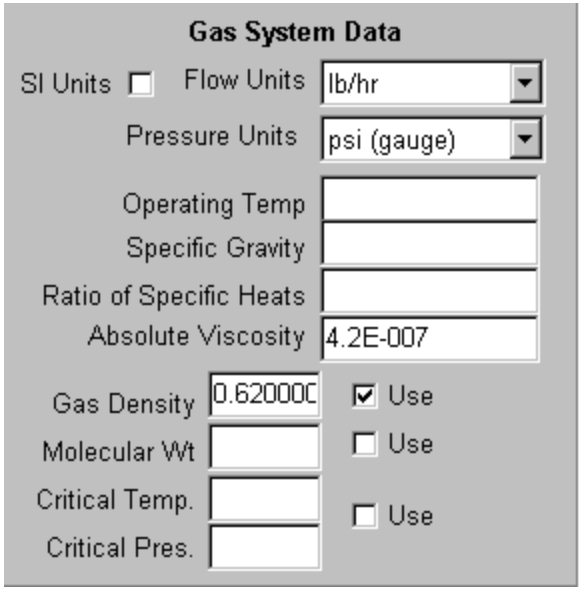
---

For a Constant Density Gas analysis the following data is required:

Absolute Viscosity

Gas Density

The Use box next to Gas Density must be checked, as shown. This will cause a Constant Density Gas analysis to be performed and the gas will be treated as incompressible.




Return to Gas

## 1.8.3 Ideal Gas

Ideal Gas

---

The Lookup Properties program may be used to determine the relevant properties of the gas used in the model.

For an Ideal Gas analysis the following data is required:

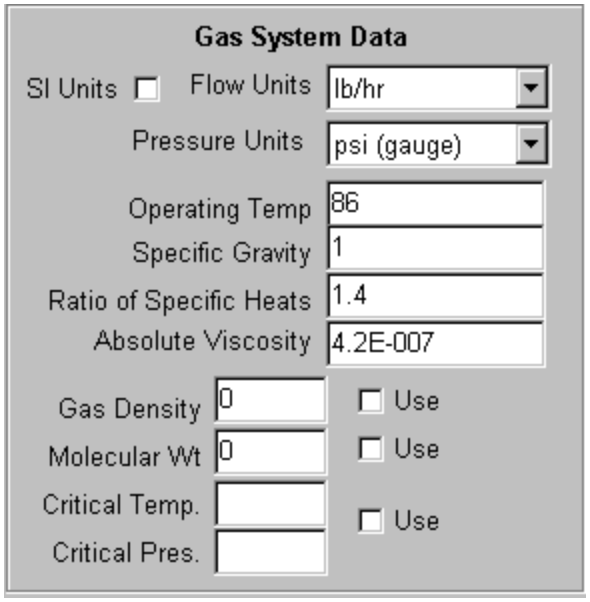
Operating Temperature

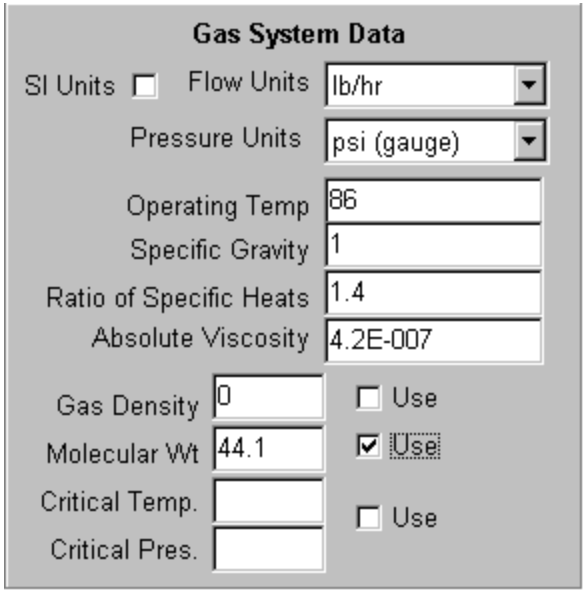
Ratio of Specific Heats

Absolute Viscosity, and one of the following:

Specific Gravity or Molecular Weight

The two combinations are demonstrated as below. When Molecular Weight is to be used, the Use box must be checked, as shown. This will cause data in the Specific Gravity field to be ignored.


Gas System Data
SI Units
Flow Units
lb/hr
Pressure Units
psi (gauge)
Operating Temp
86
Specific Gravity
1
Ratio of Specific Heats
1.4
Absolute Viscosity
4.2E-007
Gas Density
0
Use
Molecular Wt
0
Use
Critical Temp.
Use
Critical Pres.




Return to Gas

## 1.8.4 Non-Ideal Gas

Non-Ideal Gas

---

The Lookup Properties program may be used to determine the relevant properties of the gas used in the model.

For a Non-Ideal Gas analysis the following data is required:

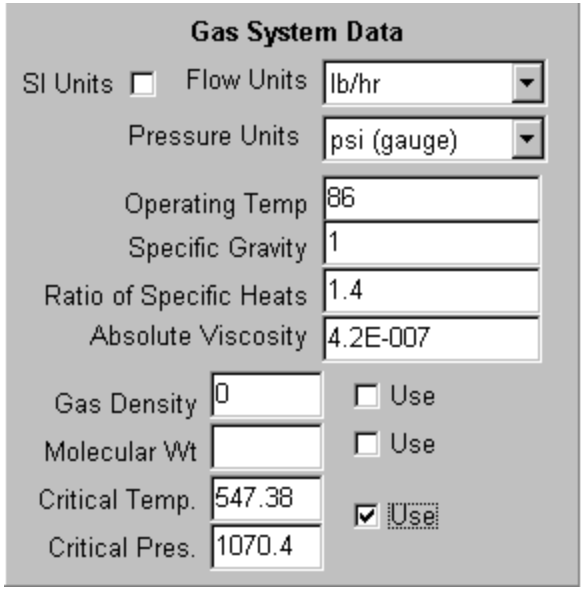
- Operating Temperature
- Ratio of Specific Heats
- Absolute Viscosity

Specific Gravity **or** Molecular Weight

Critical Temperature

Critical Pressure

When a non-ideal gas analysis is to be done, the Use box must be checked next to the Critical Temp, and Critical Pres. fields, as shown. When Molecular Weight is to be used, the Use box must be checked. This will cause data in the Specific Gravity field to be ignored.




For example, a $CO_2$ at a pressure of 2200 psig and a temperature of 90 deg. F is in a critical state and therefore acts as a non-ideal gas. Ratio of Specific Heats does not typically change in this gaseous state.

Return to Gas

## 1.8.5 Lookup Properties for Gas

Lookup Properties for Gas

---

In the System Data screen, click the Lookup Properties button to access the program. Click Begin and the following screen appears.

The user selects the Gas, the Units and the Temperature and the Gas properties appear.

Return to Gas

## 1.8.6 Active Valve for GAS

An Active Valve is a valve which may be opened, throttled, or closed for modeling purposes. Unlike other on/off valves, the minor loss for an Active Valve is based on the valve type and the open ratio is calculated and incorporated into the analysis. The open ratio can be changed by the user during a simulation using change data.

**Active Valve Data**

Elevation - This is the elevation (ft. or m) of the valve.

K Value (Cv 100%) - This accounts for the pressure drop across the valve while it is 100% open. Use a minor loss coefficient, K, orflow coefficient, Cv, calculated from one of the available tools in order to obtain an appropriate equivalent value. Gas properties, flowrate, and pressure drop must be considered when determining the appropriate Cv or K for compressible fluid models

Init Ratio - the is the ratio of the initial valve stem position to the fully opened stem position (0 = closed and 1 = wide open). The minor loss coefficient based on this number is calculated and used in the analysis.

Pressure - This appears only if no pipe links are attached to one side of the active valve so that the valve is at the end of a pipe. It is the pressure (in selected pressure units) of the connecting supply/delivery point. To model a discharge to atmospheric pressure, make this value equal zero (gauge pressure).

Valve Type - The type of valve is chosen from the drop-down selections. For all valve types (except **Other**), the open ratio refers to the ratio of the stem position to the fully open position. A ratio of 0.4 means the stem has moved 40% of the range from fully closed to fully open. For the **Other** designation, the open ratio means the ratio of the open area to the fully open area for the valve.

## 1.8.7 Steam

Steam

---

Steam is the Pipe2018 series saturated steam flow model. For information on Superheat analysis, see Superheat

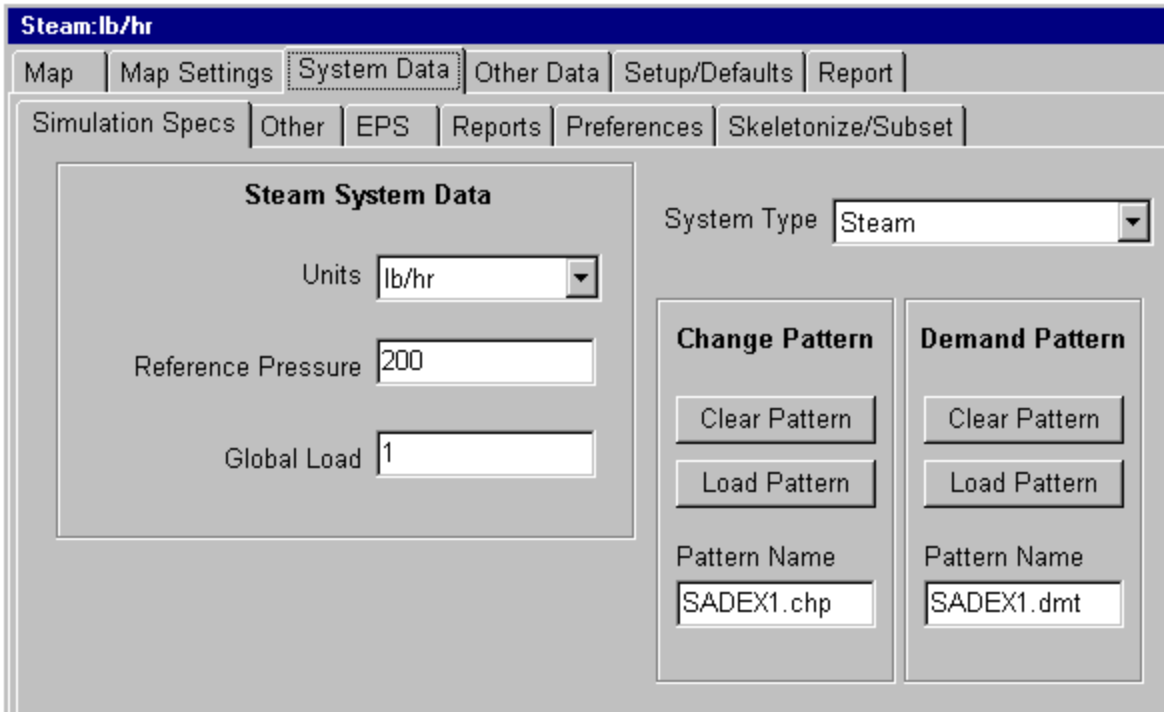
Steam uses the latest equations from steam tables to relate density-viscosity-pressure. In the case of super heated steam the effect of temperature is on these three parameters is also related. As we are aware, steam behaves like a real (or ideal) gas only under super heated conditions (a condition that assures no moisture conditions). The superheat feature computes temperature changes along the pipeline by balancing the heatloss across the pipeline (important mainly for pipelines carrying superheated steam).

System Data

Note: Steam cannot calculate loss due to condensation. However, steam loads may be input at junction nodes which can be used to simulate condensate loss.

To analyze a steam model, go to the System Data | Simulation Specs tab. There is a drop-down selector box called System Type. Choose Steam. Once Steam has been chosen, relevant System Data fields will appear to the left, as shown below.

In KYnetic use the System Data icon:




(red indicates required data)

Units - User may select the flow units from the options in this drop-down selector box (lb/hr or MBtu/hr).

Reference Pressure - Enter a reference pressure in psi here. This should be the highest supply pressure for your system.

Global Load - This represents a factor which multiplies all the assigned loads (demands). The default is one.

Change Pattern and Demand Pattern - See Simulation Specs.

## The Steam Model

Steam is written to analyze for the steady state flows and pressures in steam distribution networks. Flow in a Steam model is steady, one-dimensional, and saturated. A procedure is available to account for superheat in various lines.

### Elements

A steam analysis supports the following elements: supply points (reservoirs), compressors or fans (pumps), pressure regulating valves, valves and check valves, meters, and fittings.

Input (minimum required data in red)

No initial flow or pressure assumptions are required. Any number of parameter changes may be made within the same analysis. At least one Fixed Pressure Node or Reservoir is required for each system. Below the data requirements are summarized.

Pipes - length, diameter (inside), Darcy Weisbach roughness, fittings (minor loss).

Junctions - demand, elevation.

Reservoirs (Fixed Pressure Supply) - static pressure, elevation.

Pumps (Compressor Nodes)- useful power or operating curve (three pressure/flow data points), elevation.

Regulators (Pressure Regulating Valves) - downstream pressure setting, elevation.

Loss Elements - three pressure drop/flow data points.

Pressure Supply (Variable Pressure Supply) - 1. static pressure, residual pressure, residual flow or 2. three pressure/flow data points, elevation, one additional pressure flow data point optional.

Check Valve - elevation

Valve - elevation

Metered Connection - elevation, meter ID and demand

Intermediate Nodes - elevation

Device 1 - elevation, user data

Device 2 - elevation, user data

Output

Nodes - density, absolute and gauge pressure

Pipes - flow, pressure loss, mean flow velocity

System - summary of system inflows and loads, tables of maximum and minimum pressures, velocities, and losses per length, choking conditions (maximum Mach number), and minimum flow areas.

## 1.8.8 Temperature Sensitive Analysis

Superheat for Pipe2018: Steam

## Temperature Sensitive Analysis

About Superheat for Pipe2018: Steam
Using Superheat

## Superheat for Pipe2018: Steam

Superheat for Pipe2018: Steam is an advanced feature available with the Pipe2018: Steam program and was developed specifically for modeling distribution networks handling superheated steam. The program can handle complex steam distribution networks with multiple supply sources generating either superheated or saturated steam, multiple compressors, and pressure reducing valves. The program employs data from latest steam tables (IAPWS97) for various thermodynamic properties of the steam. The powerful Pipe2018 graphical interface allows for rapid model building, fast and accurate computation of flows, pressures, temperatures, and heat losses, and visual presentation of results. The following summarizes the important features of the program.

The Superheat for Pipe2018: Steam program allows for

- multiple compressors defined by pressure-flow characteristic data
- multiple pressure reducing valves (PRVs)
- multiple steam generating sources (both high and low pressure stations)
- user specified steam loads
- mass flow units in kg/hr, lb(mass)/hr, or MBTU/hr
- temperature specification at known locations in the network
- user specified heat transfer coefficients for pipes
- user specified ambient temperature

The powerful analysis engine

- calculates heat loss for each pipe line based on flow and pressure drop across the pipe line, pipe heat transfer coefficient, etc.
- keeps track of saturated temperatures for all nodes and sets the temperatures to saturated temperatures if the calculations force them to go below saturated values.
- uses appropriate data from the latest steam tables to calculate the required state parameters.
- automatically balances mass, energy and other thermodynamic equations irrespective of the model complexity
- incorporates a temperature mixing model
  - temperature at any node is equal to the volumetric average of the temperature of all the pipes feeding the node
- allows for multiple scenario simulations in one model run
  - for example, what if scenarios, such as,
    - the effect of increasing or decreasing of load factors may be studied in a single run

- changing certain model parameters/characteristics (eg. pipe diameter, pipe roughness, on/off status of a valve, specific load change at any node etc)

Note: Steam cannot calculate loss due to condensation. However, steam loads may be input at junction nodes which can be used to simulate condensate loss.

The graphical user interface (GUI) allows for efficient visual processing of results. Some of the important features include:

- displaying node data and results (load, elevation, pressure, density, and temperature) directly on the network map
  - for all nodes or just a few selected nodes
- displaying pipe data and results (length, diameter, roughness, minorloss coefficient, flowrate, velocity, headloss, etc)
  - for all pipes or just a few selected pipes
- display node/pipe results for any two different scenarios (multiple scenario runs) simultaneously for visual comparison
- compare current set of results with the results from previous run
- query and display node/pipe results meeting certain criteria
  - for example, display nodes with pressures less than 40psig, or nodes with superheat greater than 50F
- color code pipes/nodes based on data/results
  - For example, color code pipes by diameters, velocities, flowrate, headloss etc. Color code nodes by loads, elevations, pressures etc.
- plot contours of pressures, densities, temperatures etc.
- plot variation of pressure for a node (or a group of nodes) under multiple scenarios. For example, plot variation of pressure at a node with increasing/decreasing global load factor.

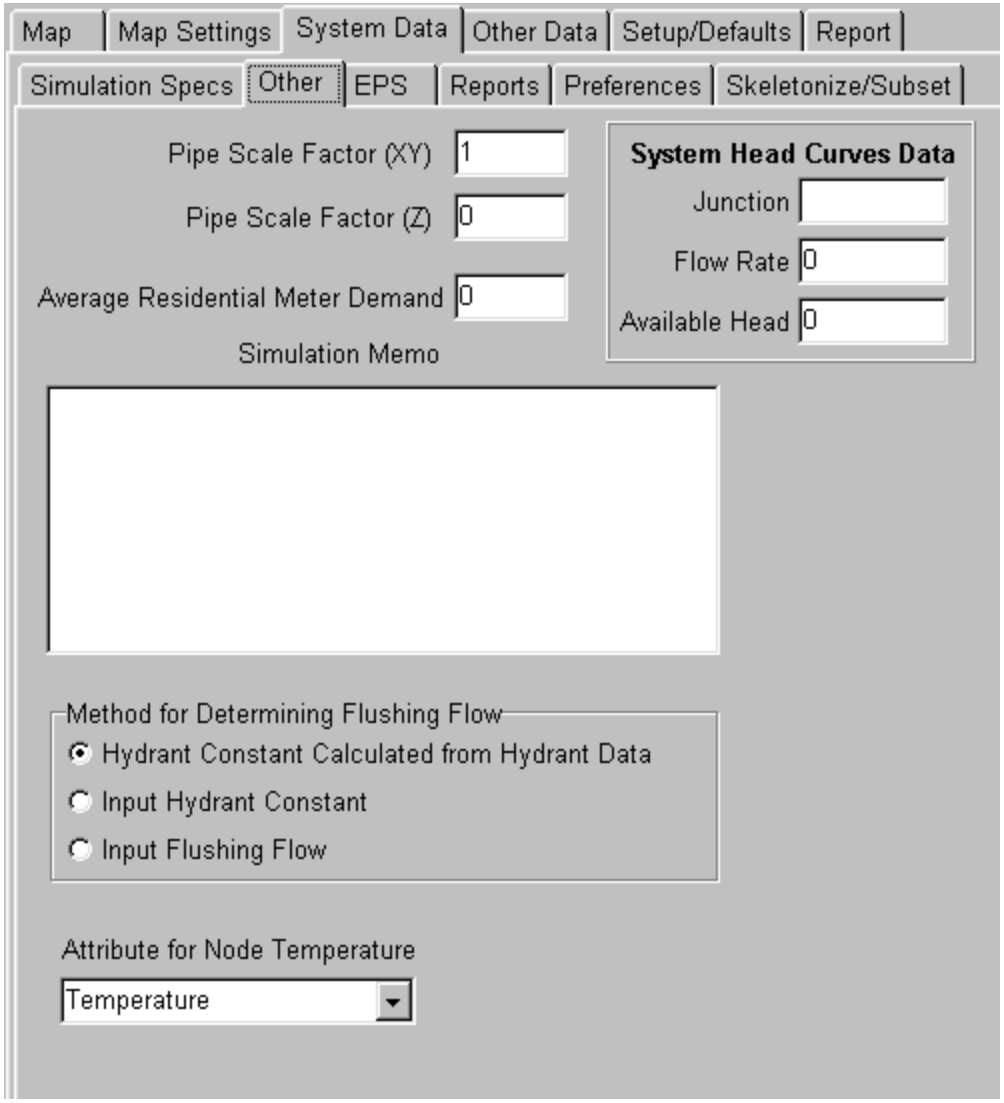
The program generates a detailed tabulated report in the ASCII (text) format. The following lists some of the important features of the report.

- Complete list of input data
  - Pipeline data – pipe length, diameter, roughness, minor loss coefficients, heat transfer coefficients etc.
  - Node data – steam loads, etc…
- Detailed list of pipe and node results
  - Pipe results – pipe flow rate, head loss, heat loss, velocity, average viscosity, average density etc.
  - Node results – loads in lbm/hr (or kg/hr), loads in MBTU/hr, enthalpy, pressure in absolute units as well as gauge units, density, temperature, saturate temperature, superheat (if any).
- Low pressure areas are highlighted
- Highest Mach number (indication of choking condition) and the pipeline in which it occurred are listed
- Summary of results including total load in the system
- Flows in/out of each supply/receiving node.
- Detailed report on pressure reducing valves (PRVs)

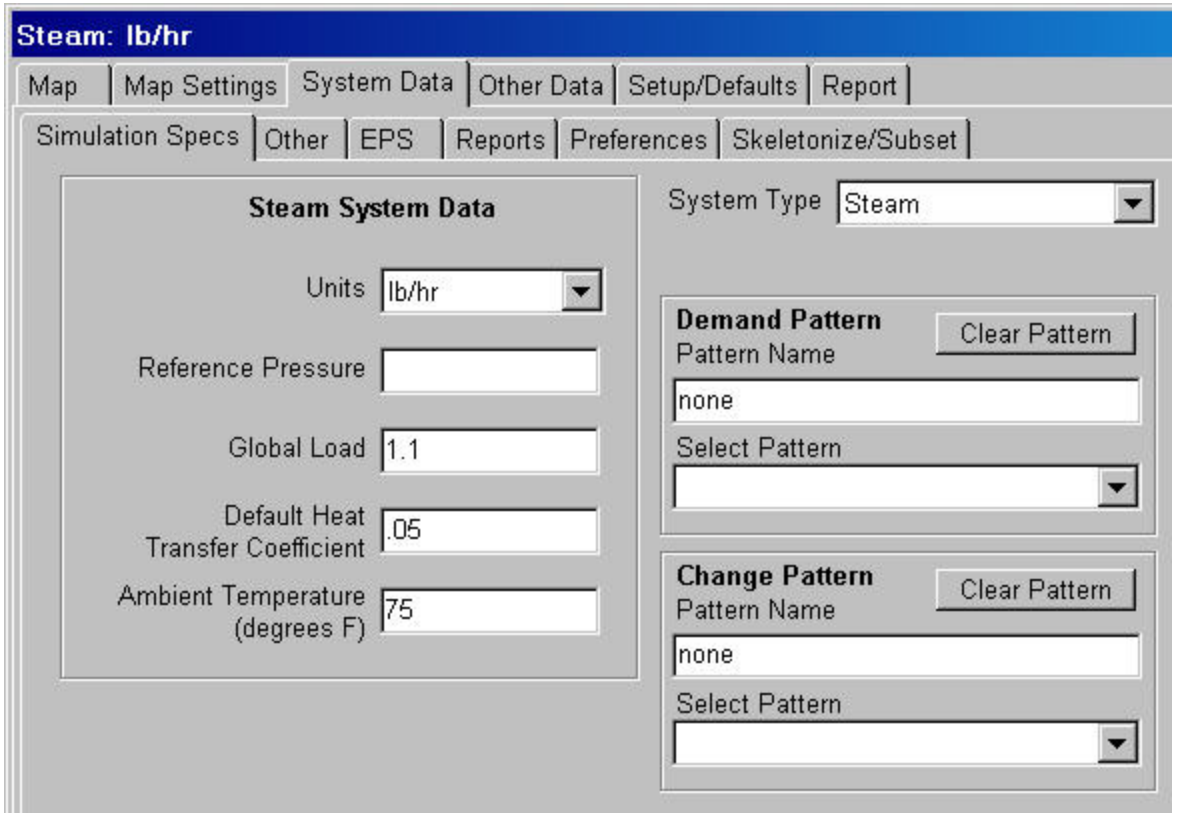
    - Flowrates, specified pressure setting, pressure drop across the PRV
- Detailed report on compressor operation

### Using Superheat

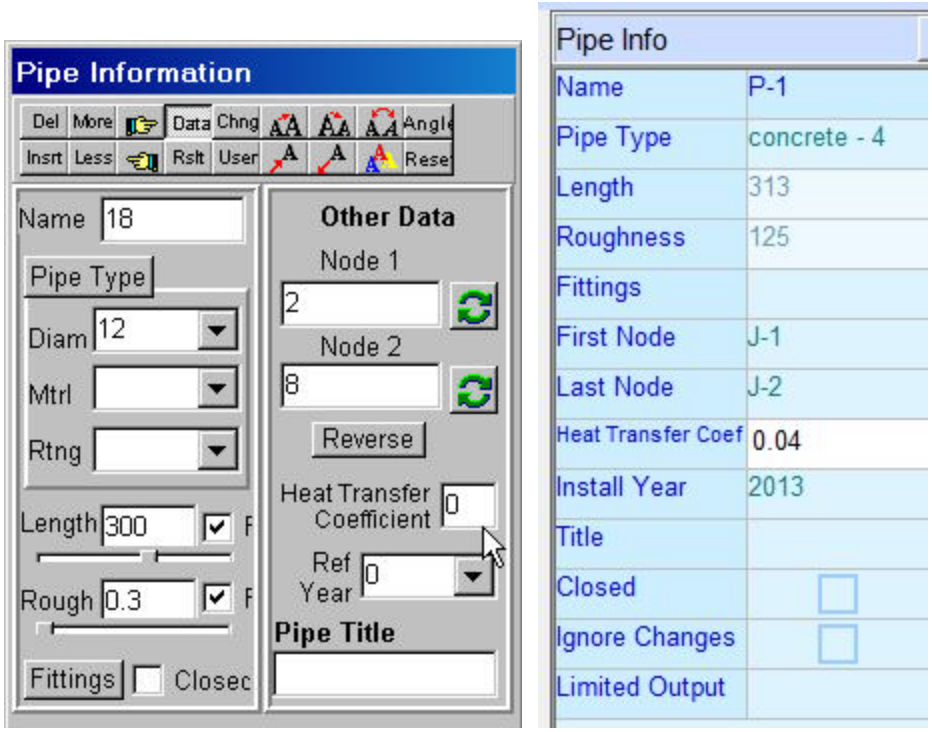
If a user data item called **Temperature** does not already exist, create one now (see User Data). Next go to the Other Data icon (KYnetic) or System Data | Other menu and specify the Attribute for Node Temperature box as shown below. See User Data.




Provide a default **Heat Transfer Coefficient** (Btu/(hr-ft^2-F) or Btu/(hr-m^2-C)) and **Ambient Temperature** (F or C) in the System Data menu (KYnetic).



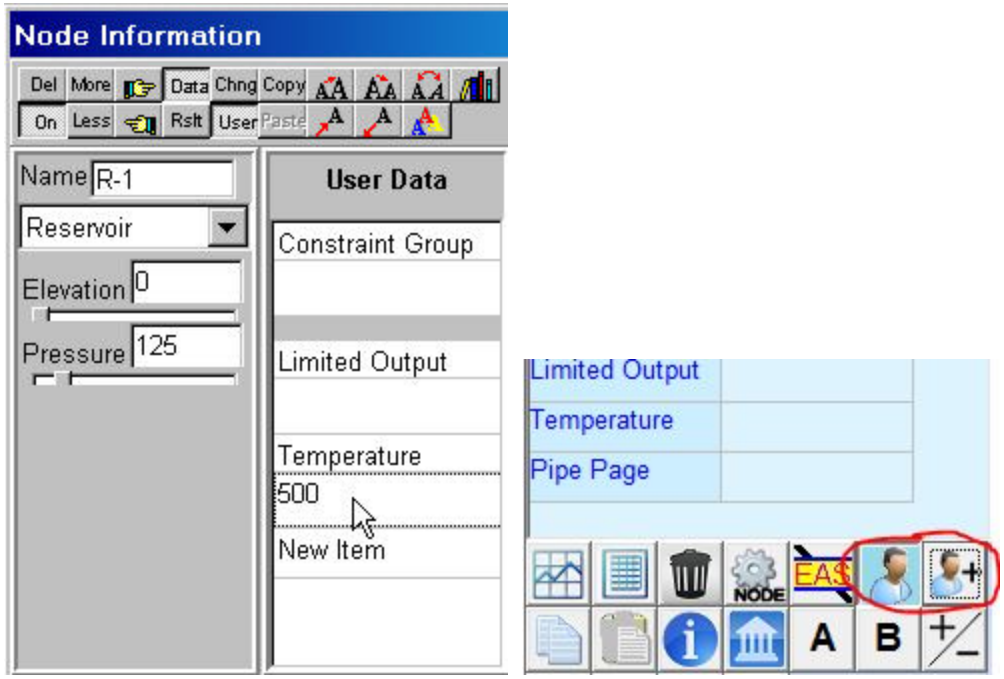

Once the model is created and all necessary information has been provided, select a pipe and provide the **Heat Transfer Coefficient** if different from the default value.







At all supply points, provide temperatures. To do this, click on the supply node, make sure the **User** button is depressed. Use the item called **Temperature** to assign a temperature value.




Save the model. Run an **Error Check** (**Analyze | Error Check**), then analyze (**Analyze | Analyze**, select **Temperature Sensitive Analysis**, then click **Analyze**). To view the results, click the **Report** Tab. In the summary of pipeline data, note the heat transfer coefficients. In the summary of node data, note the temperatures. Then view the pipeline and node results. The last three columns of node results report superheat analysis results. Results, including **Superheat**, can also be viewed in the map screen as labels.

## 1.8.9 Units Gas and Steam

Units Gas and Steam

| Flow | length | diameter | roughness (Darcy Weisbach) | viscosity (absolute - Gas) | demand | elevations | pressure | velocity | pressure loss | pump pressure | density | pump flow |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| English - | ft | in | mft | lbs-s/ft*ft | same as | ft | Gas-choose | ft/s | same as | same as | lbs/ft*ft*ft | same |

| | | | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| choose units | | | | | flow | | Steam-psi | | pressure | pressure | | as flow |
| SI - choose units | m | mm | mm | Newton-s/m*m | same as flow | m | Gas-choose<br>Steam-kPa | m/s | same as pressure | same as pressure | kg/m*m*m | same as flow |

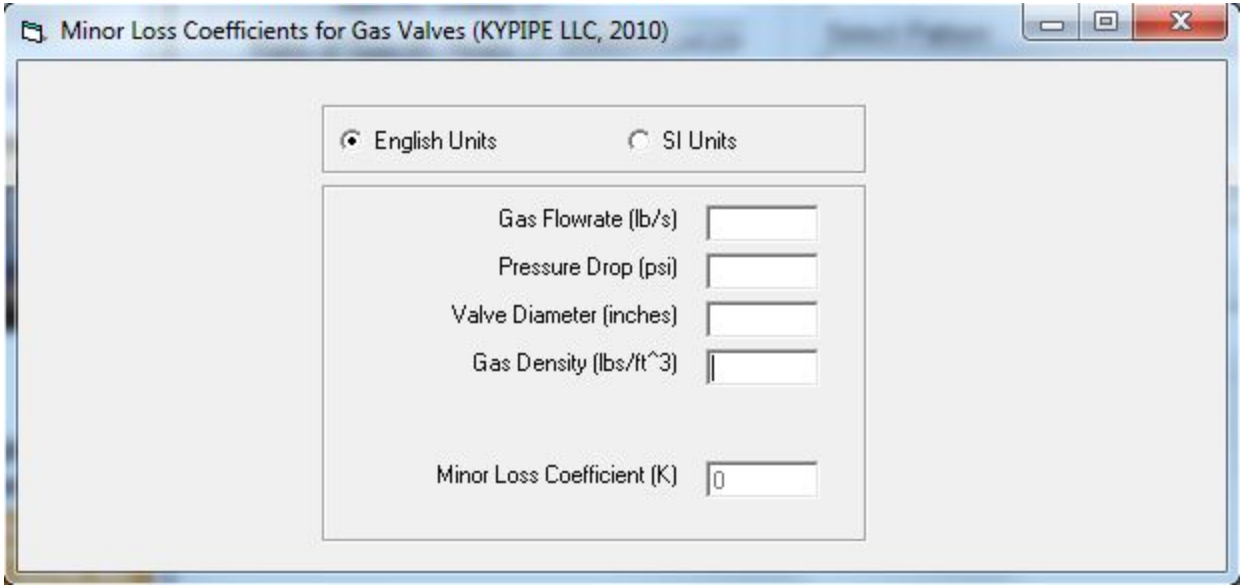
loss element - table of pressure drop (ft or m) versus flow in defined units.

regulator setting - PRV, PSV - same as pressure (psi or kpa)

FCV - flow in defined units

## 1.8.10 Tools - Gas and Steam

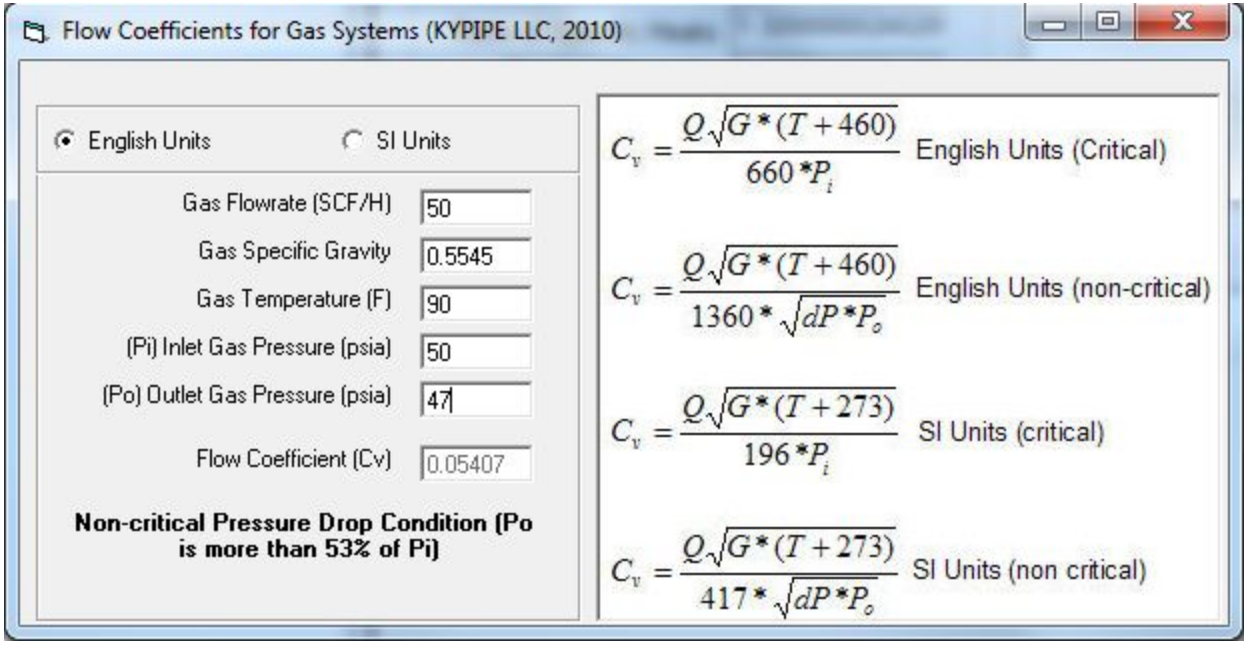
### 1.8.10.1 Gas K Calculations Tool



Enter listed data to obtain a minor loss K value for an Active Valve.

When modeling an Active Valve, use a minor loss coefficient, K, or flow coefficient, Cv, calculated from one of the available tools in order to obtain an appropriate equivalent value. Gas properties, flowrate, and pressure drop must be considered when determining the appropriate Cv or K for compressible fluid models

## 1.8.10.2 Gas Cv Calculations Tool




Enter relevant gas data to get flow coefficient value for Active Valves in a Gas model.

When modeling an Active Valve, use a minor loss coefficient, K, or flow coefficient, Cv, calculated from one of the available tools in order to obtain an appropriate equivalent value. Gas properties, flowrate, and pressure drop must be considered when determining the appropriate Cv or K for compressible fluid models

## 1.10 GoPlot

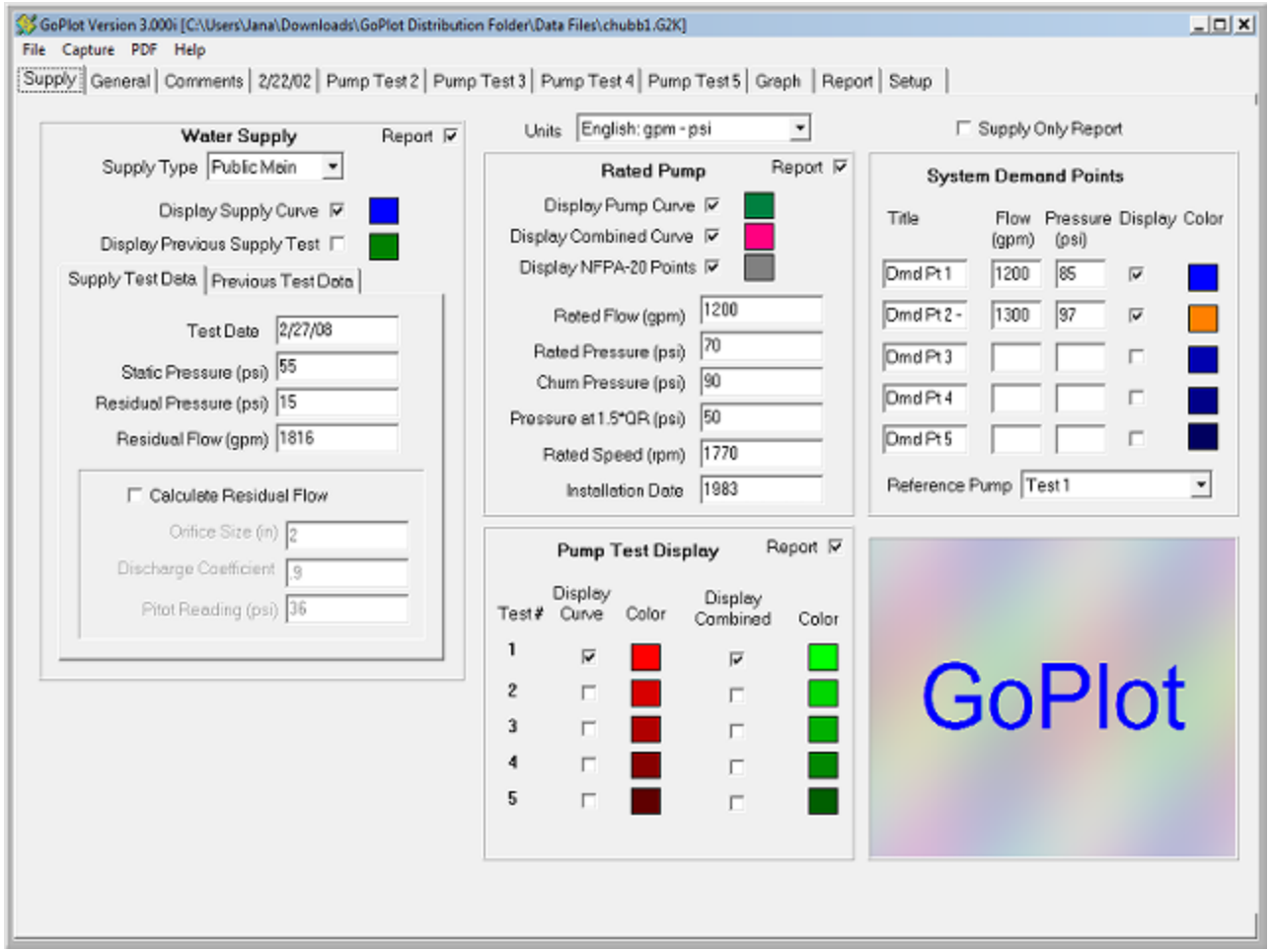
## 1.9.1 GoPlot

GoPlot Users Manual

# *GoPlot Users Manual*

GoPlot is designed for Loss Control Engineers to prepare a report and supporting graphs to evaluate supplies and fire pumps for fire protection systems. GoPlot allows the engineer to enter a variety of information about the water supply, rated and test pump performance data and a variety of auxiliary information about clients, controller and driver data, fire pumps, alarms and other pertinent information. The engineer can produce a variety of plots showing the individual performance of supplies and pumps and the combined performance of these critical elements. GoPlot also will produce a report which includes all information the engineer selects.

The GoPlot Supply Screen is shown below. This program is designed to provide user friendly access to data entry screens and graph and report generating. The engineer chooses which data they want to enter, which plots they wish to generate (Graph) and what information they want to be in the report (Report). The GoPlot Manual will refer to this screen as the features are illustrated.




# GoPlot – Main Menu & Tabs

The Main Menu has the following selections

**File** has the following selections:

New – open new project

Open – open existing project

Save – save the current project using same name and location

Save As – save the current project with a different name and/or different location

Recover Last File –if a crash occurs while GoPlot is running it may be possible to recover the current file. Start GoPlot and **immediately** select this option.

Error Check – check data entries for errors

Print – print report or graph (This option may not appear in some versions of GoPlot. Use the PDF options instead.)

Exit

**Capture** will capture a jpg file or files of the report or graph

**PDF** has the following options:

PDF Save – Create a PDF of the report or graph using the default file name.

PDF Save As – Create a PDF of the report or graph with an assigned name.

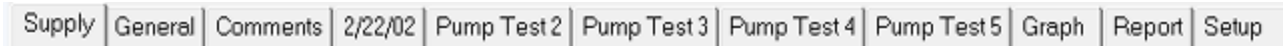
For both PDF options if a PDF already exists with that name you will have the option to Insert the new pages at the beginning of the PDF, Append the pages to the end of the PDF, or Overwrite the old pages with the new ones.

**Help** has the following selections:

About will show the version information

Content will display the help file

The following Tabs access the various Data Entry and Results Presentation Screens




**Supply** – data entry screen for system requirements and features

**General** – data entry screen for client information, pump accessories and alarms

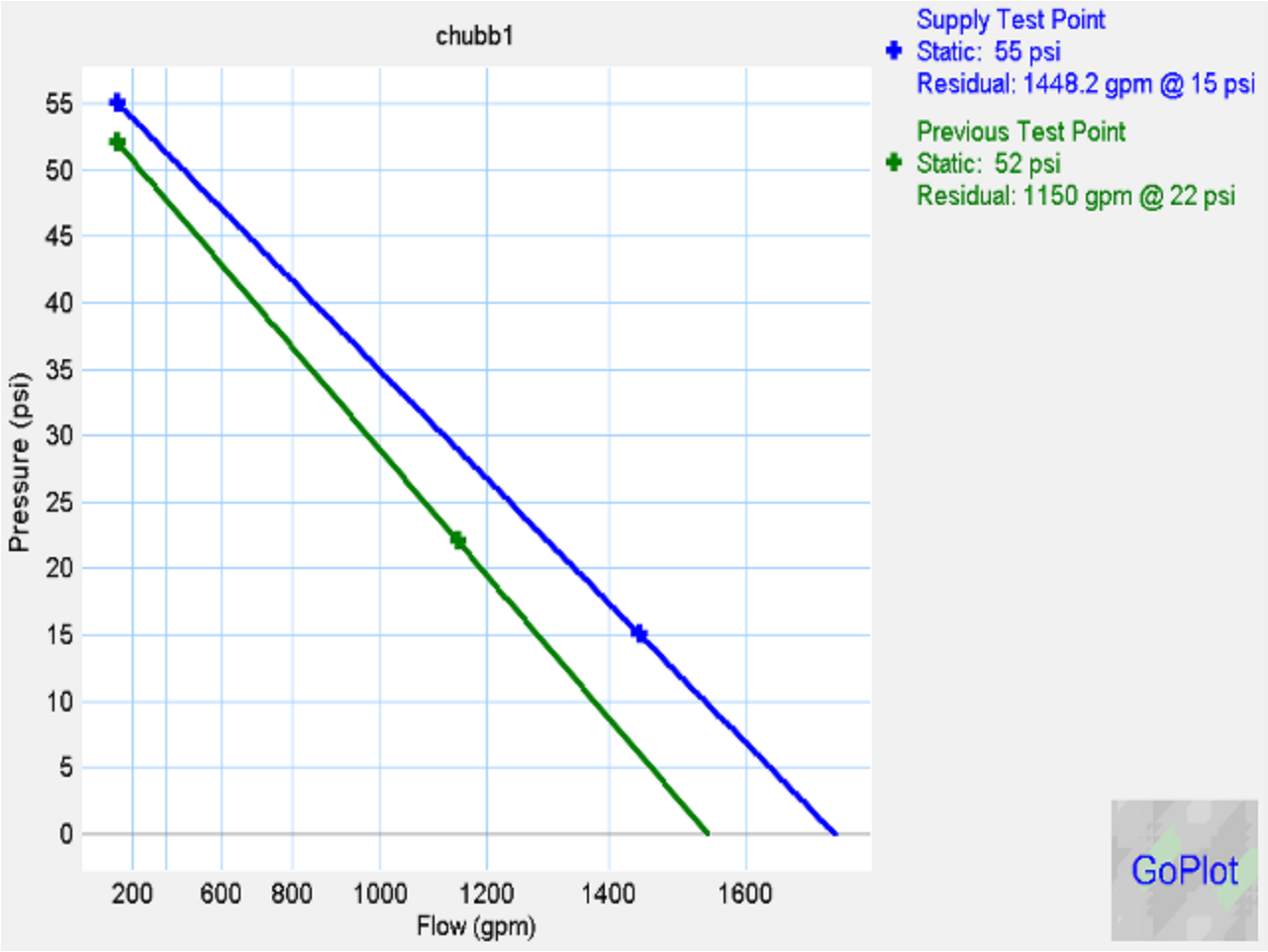
**Comments** – enter any general comments for the report

**Pump Test 1 (2/22/02)** to **Pump Test 5** – screens for up to 5 pump test data. Once entered the test date appears as shown above

**Graph** – show the plot with current plot selections

**Report** – show report

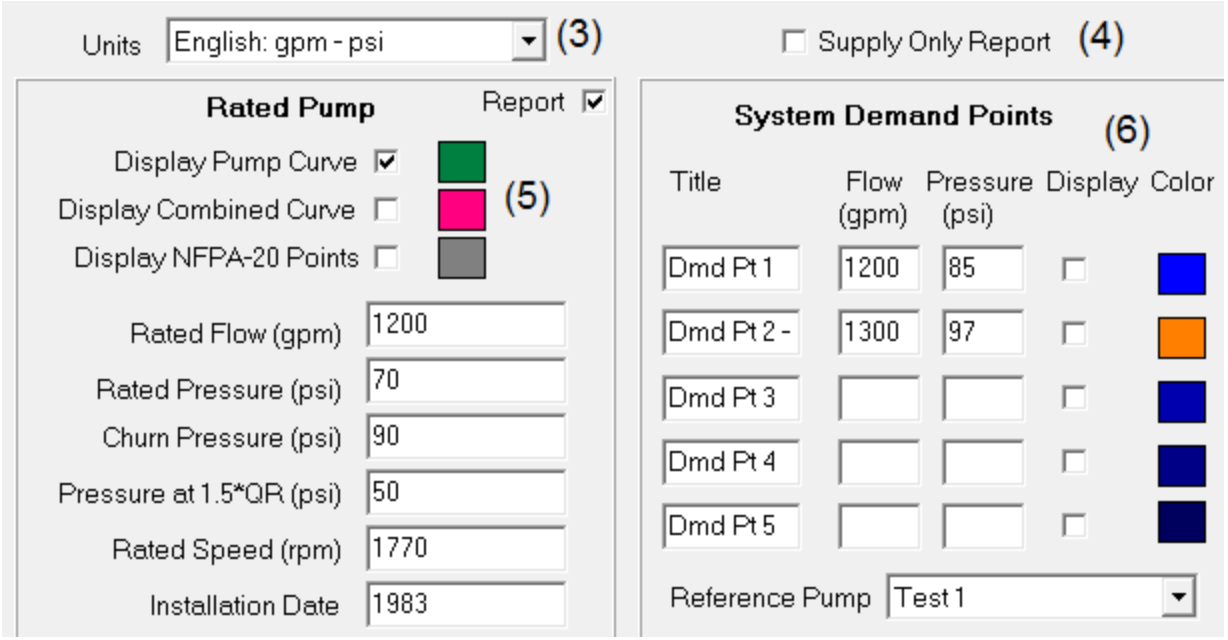
# Supply Screen Tab – Water Supply Data


chubb1
Supply Test Point
Static: 55 psi
Residual: 1448.2 gpm @ 15 psi
Previous Test Point
Static: 52 psi
Residual: 1150 gpm @ 22 psi
Pressure (psi)
55
50
45
40
35
30
25
20
15
10
5
0
200
600
800
1000
1200
1400
1600
Flow (gpm)
GoPlot

These screens provide data and set up plots for the water supply. Several options are available for the Supply Type:

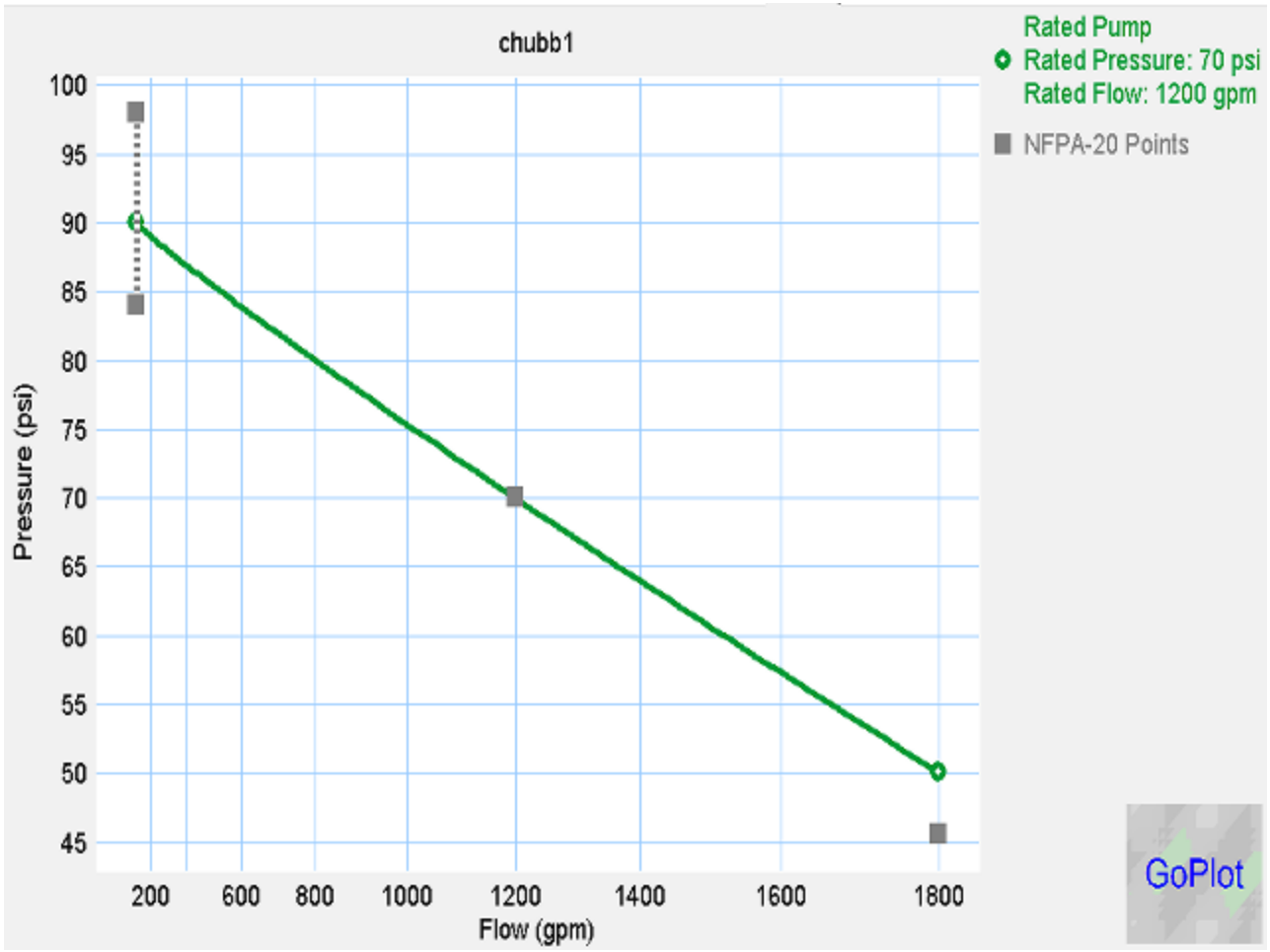
There is an option to provide and plot one or two (previous) sets of Supply data. The plots shown above show the supply plot(s) for both options (1). There is also an option to calculate the Residual Flow using a pitot reading as shown on the upper left screen (2). The Supply Plots can be plotted separately as shown above or combined with other plots which will be illustrated in this manual. Note that there is a choice for the flow scale (X) of a linear of 1.85 power (Setup). The plots shown in this manual use the power option.

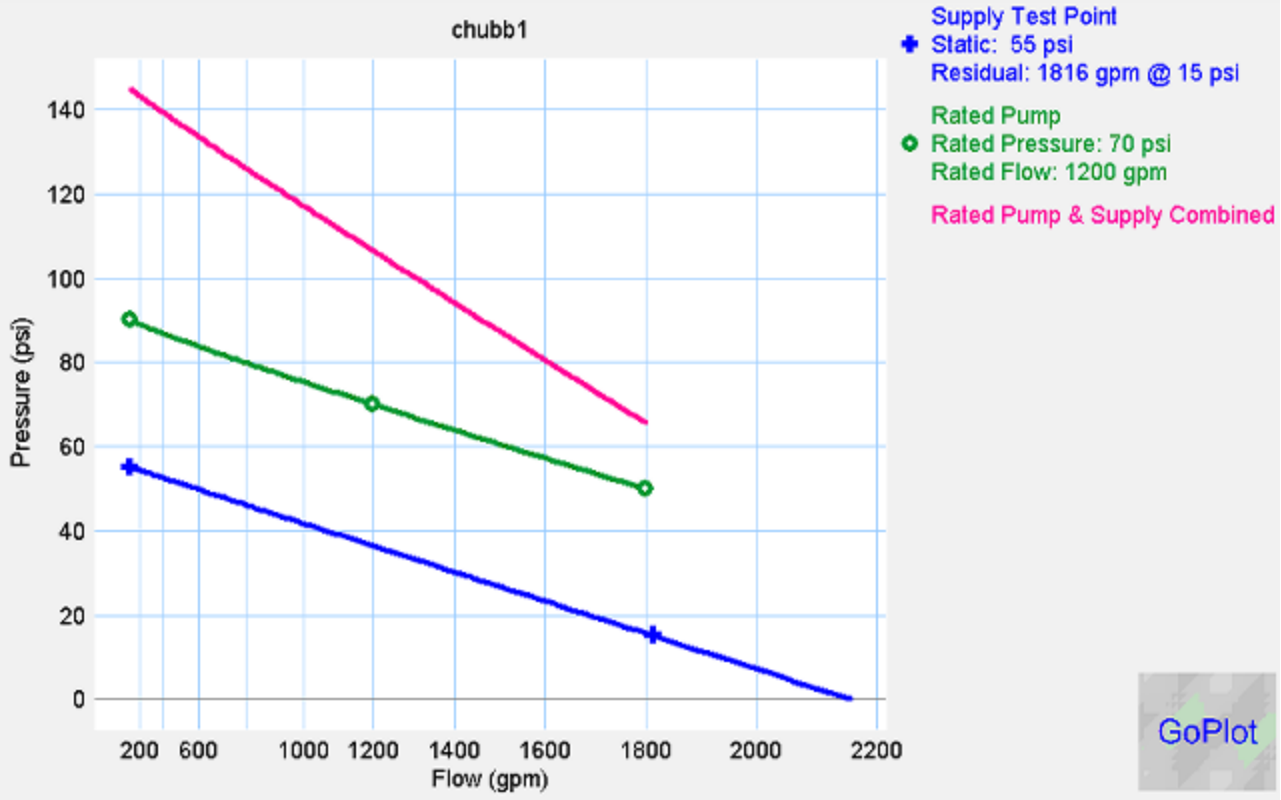
## Supply Screen Tab – Rated Pump & Demand Points Data



This area of the Supply Screen includes a drop-down list to select units (3). A variety of metric choices are provided. A check box to provide a report for the Supply Only (4) is also shown.

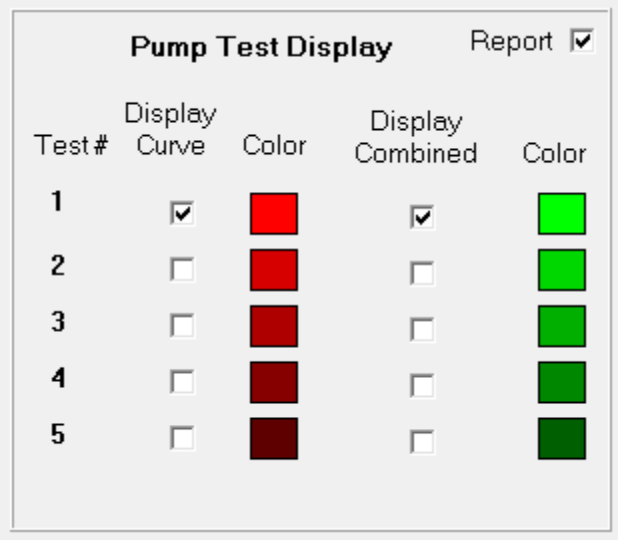
The section titled Rated Pump allows the user to enter the rated conditions for the pump and select options for plots (5). The plots below show one with the Display Pump Curve and Display NFPA-20 Points selected (left). The other one shows the Display Supply Curve(1) and Display Combined Curve checked.

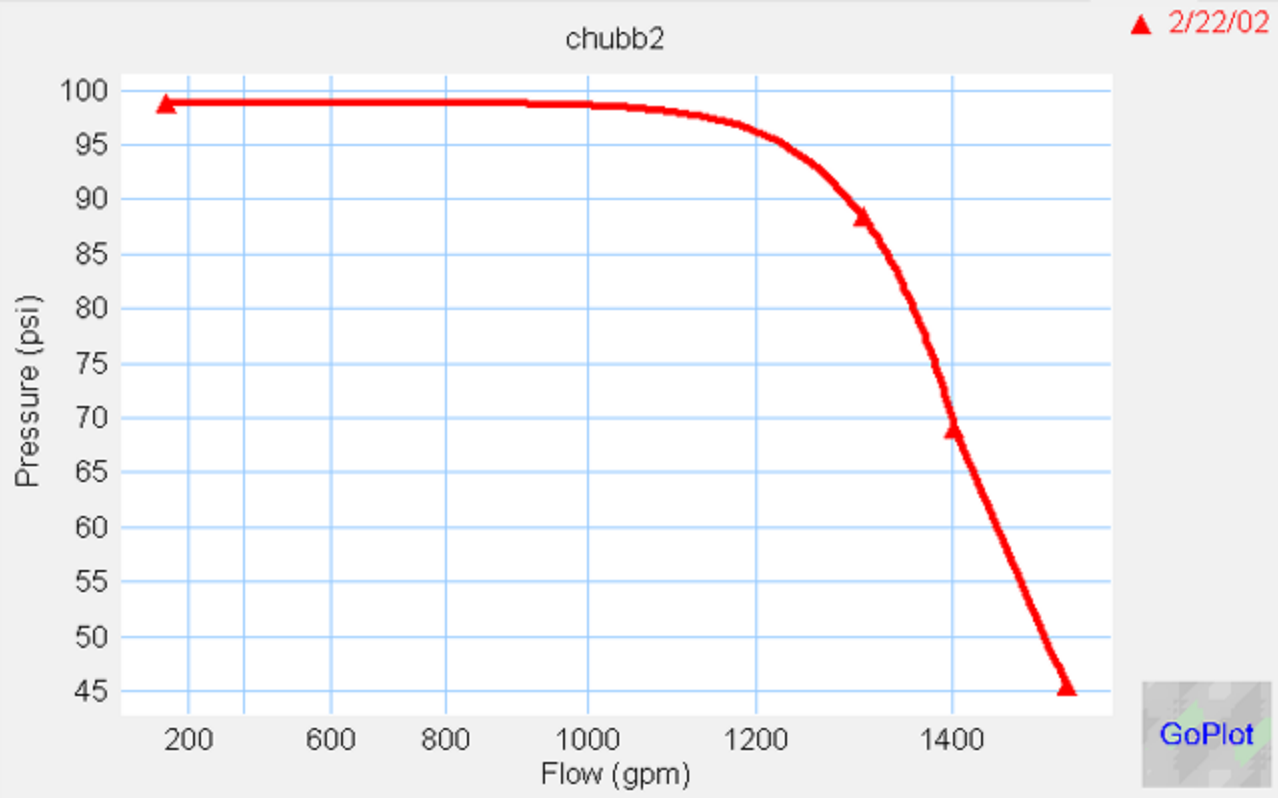


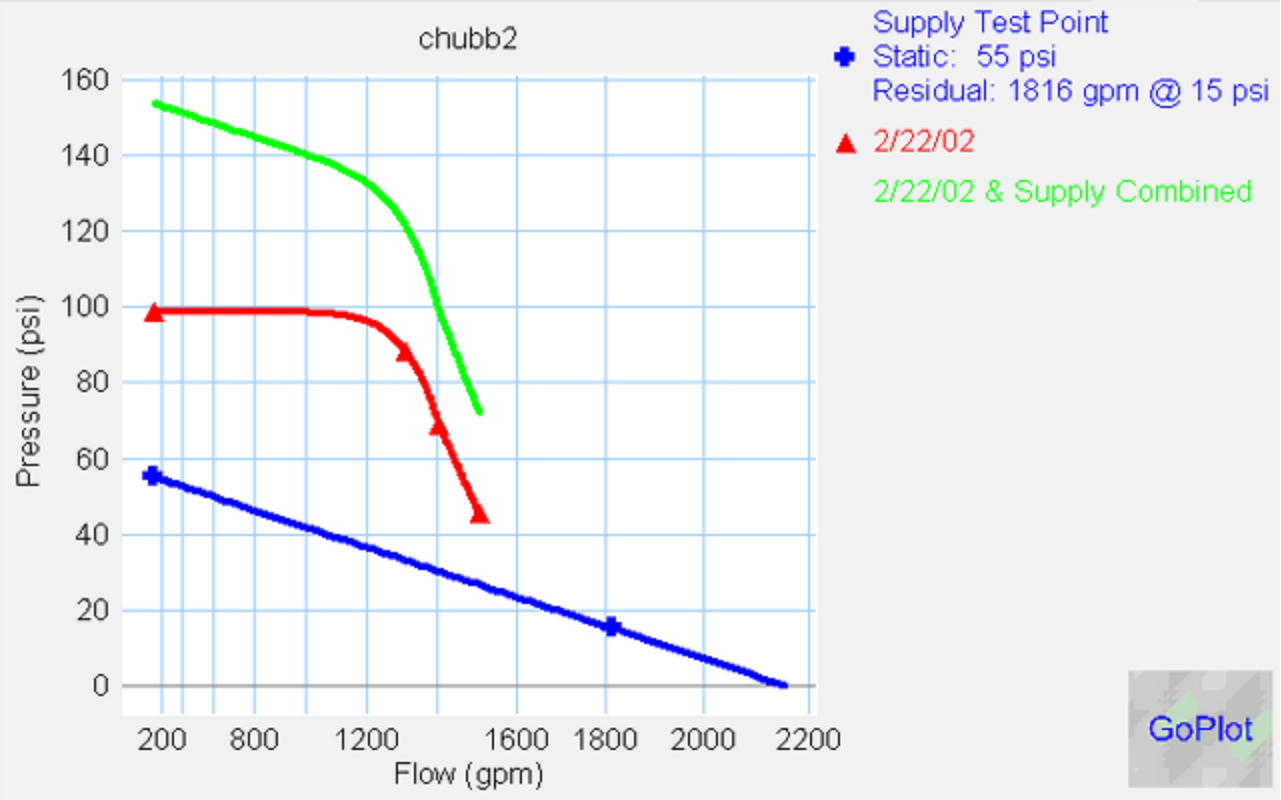



The section titled **System Demand Points** (6) allows the user to enter up to 5 Demand Points (pressure/flow) and select them for display on the plot. The user can customize the title (click on) and designate a pump (or None) for calculating the pressure margin between the Demand Point and the combined supply and pump (if any) curve.

## Supply Screen Tab – Pump Test Display Data

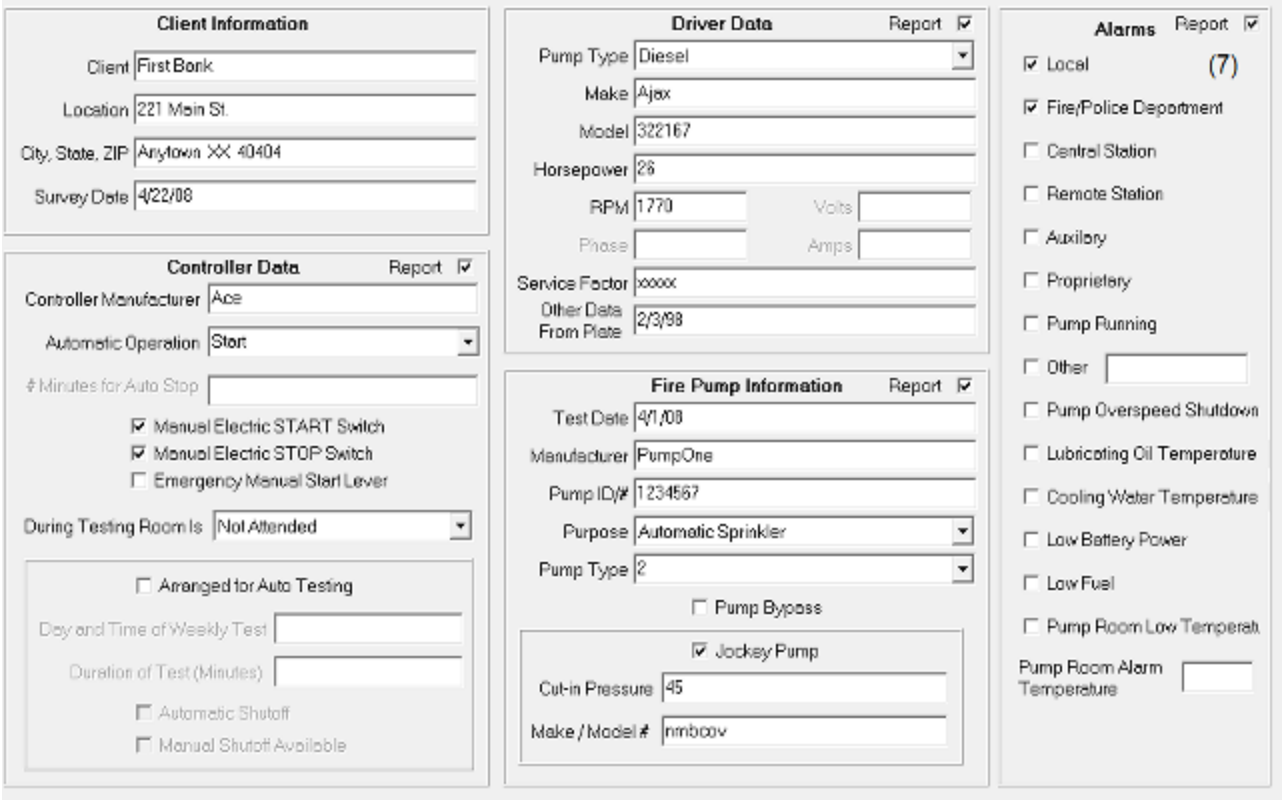

Pump Test Display
Report
Test #
Display Curve
Color
Display Combined
Color
1
2
3
4
5


chubb2
2/22/02
100
95
90
85
80
75
70
65
60
55
50
45
Pressure (psi)
200
600
800
1000
1200
1400
Flow (gpm)
GoPlot




The **Supply Screen** includes all the graph selections. The **Pump Test Display** area shown above allows users to select graphing options for up to 5 sets of pump data. This data is entered using the Pump Test 1 to Pump Test 5 tabs. Example plots are shown above. The plot on the left shows only the pump data and head/flow curve for one pump. The curve on the right shows the Supply Curve (1), a selected Pump Test and the Combined (supply & pump) curve.

## General Data Screen Tab



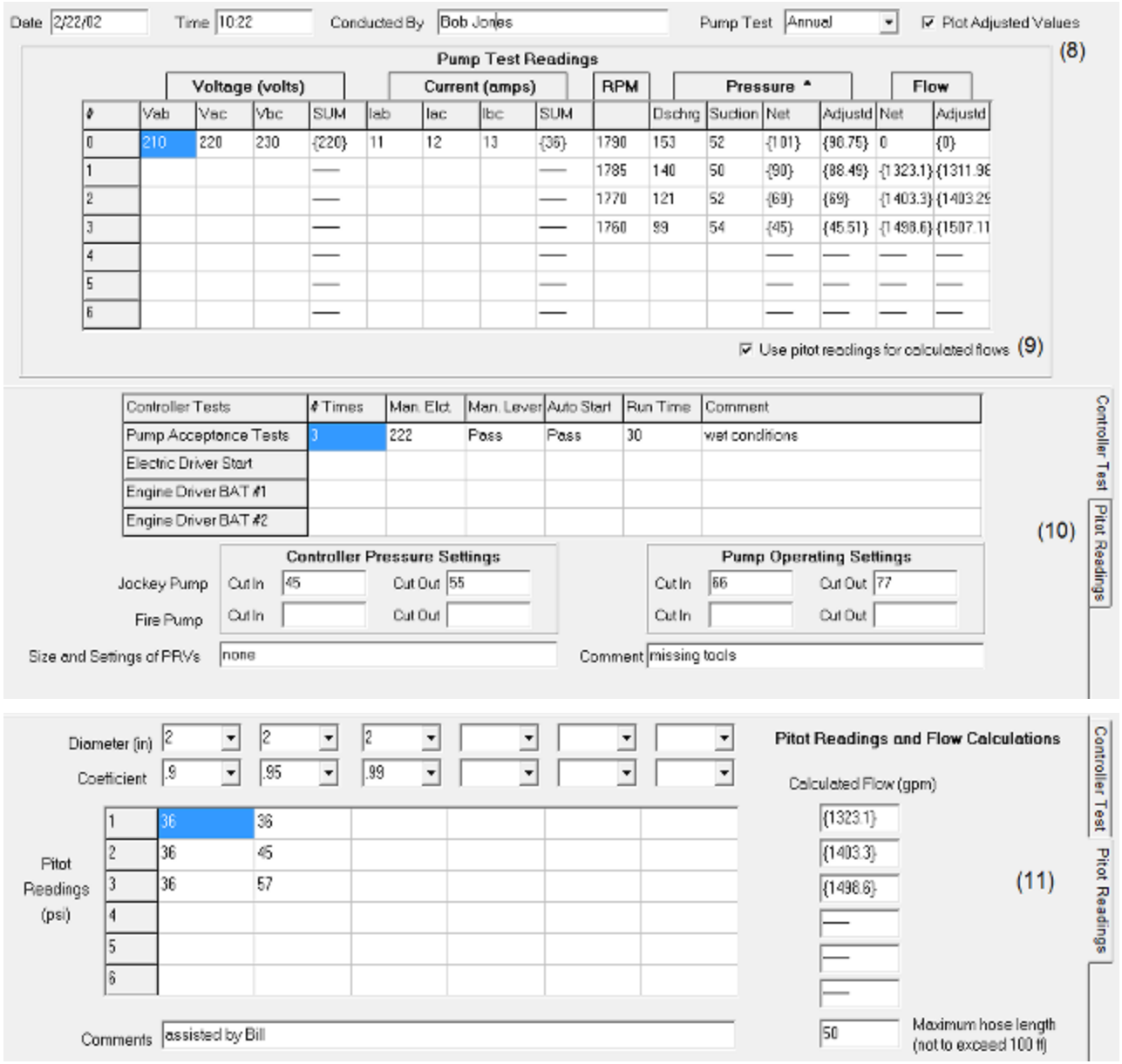

The **General Data** screen includes comprehensive auxiliary information for this system. Information that the user wants included in the GoPlot Report can be selected using the Report check boxes (7).

**Pre-filled Data**

In some cases when GoPlot is launched from another application a new file will be created with some of the data pre-filled. This data may include the Client Information on the General Data Screen and the contents of the Comment Screen.

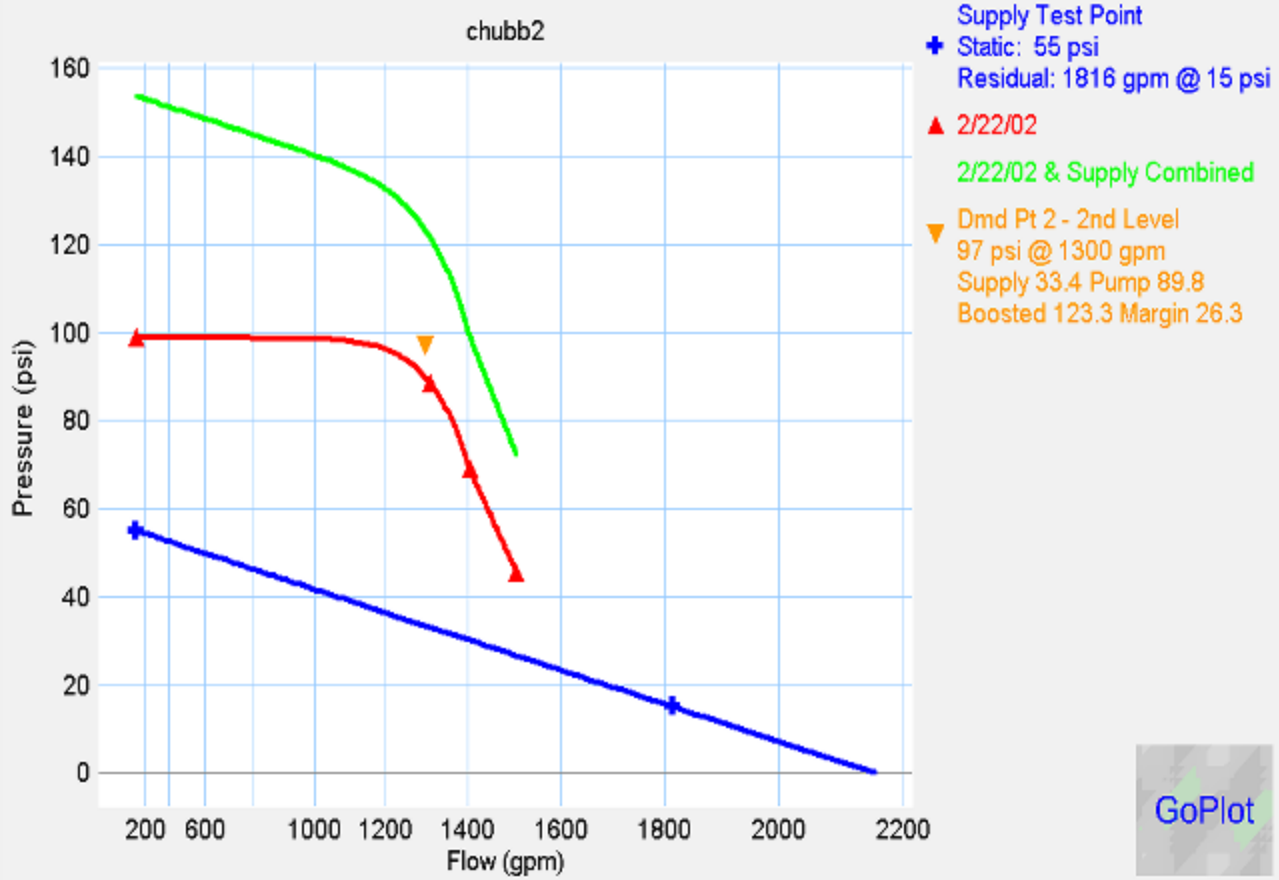
This can be accomplished if the application sends GoPlot the name of an .ini file in the command line parameter. Call Tech Support for information on the required format of this file.

# Pump Test Data Screen Tabs


Date 2/22/02
Time 10:22
Conducted By Bob Jones
Pump Test Annual
Plot Adjusted Values
(8)
Pump Test Readings
Voltage (volts)
Current (amps)
RPM
Pressure
Flow
# Vab Vac Vbc SUM Iab Iac Ibc SUM Dschrg Suction Net Adjustd Net Adjustd
0 210 220 230 {220} 11 12 13 {36} 1790 153 52 {101} {90.75} 0 {0}
1 1785 140 50 {90} {88.49} {1323.1} {1311.98
2 1770 121 52 {69} {69} {1403.3} {1403.29
3 1760 99 54 {45} {45.51} {1498.6} {1507.11
4
5
6
Use pitot readings for calculated flows
(9)
Controller Tests # Times Man. Elct. Man. Lever Auto Start Run Time Comment
Pump Acceptance Tests 3 222 Pass Pass 30 wet conditions
Electric Driver Start
Engine Driver BAT #1
Engine Driver BAT #2
(10)
Controller Test
Pitot Readings
Controller Pressure Settings
Jockey Pump Cut In 45 Cut Out 55
Fire Pump Cut In Cut Out
Pump Operating Settings
Cut In 66 Cut Out 77
Cut In Cut Out
Size and Settings of PRVs none
Comment missing tools
Diameter (in) 2 2 2
Coefficient .9 .95 .99
Pitot Readings (psi)
1 36 36
2 36 45
3 36 57
4
5
6
Comments assisted by Bill
Pitot Readings and Flow Calculations
Calculated Flow (gpm)
{1323.1}
{1403.3}
{1498.6}
(11)
50
Maximum hose length (not to exceed 100 ft)
Controller Test
Pitot Readings

This screen allows the user to enter comprehensive Pump Test data. Note the pressure/flow data will be adjusted for changes in the pump speed and the adjusted values may be used for the graphs (8). An option to make flow calculations using pitot readings is also provided (9). If this option is selected then the user should click on Pitot Readings (10) to access the pitot data entry screen (11).

## Graph Tab



The Graph tab will produce a graph showing the items selected on the Supply Data Screen (12). As shown on the above Supply Data Screen the Supply plot, the Pump Test 1 plot and the Combined plot along with Demand Point 2 are shown on the graph (12)

# Report Tab

A comprehensive Report is generated when this Tab is selected. The first page of the Report consists of a number of sections:

CLIENT INFORMATION

FIRE PUMP INFORMATION*

PUMP DATA (read from pump data plate)*

DRIVER DATA (read from driver data plate)*

CONTROLLER DATA*

WATER SUPPLY*

ALARMS*

The user can choose not to include one or more sections designated with an * by not checking the Report Box included with the data entry screens.

An additional comprehensive report can be generated for each Pump Test for which data is provided. This report appears on one page for each set of PUMP Test data selected and includes the following sections:

PUMP TEST READINGS

CONTROLLER TEST READINGS

JOCKEY AND FIRE PUMP SETTINGS

ADDITIONAL INFORMATION

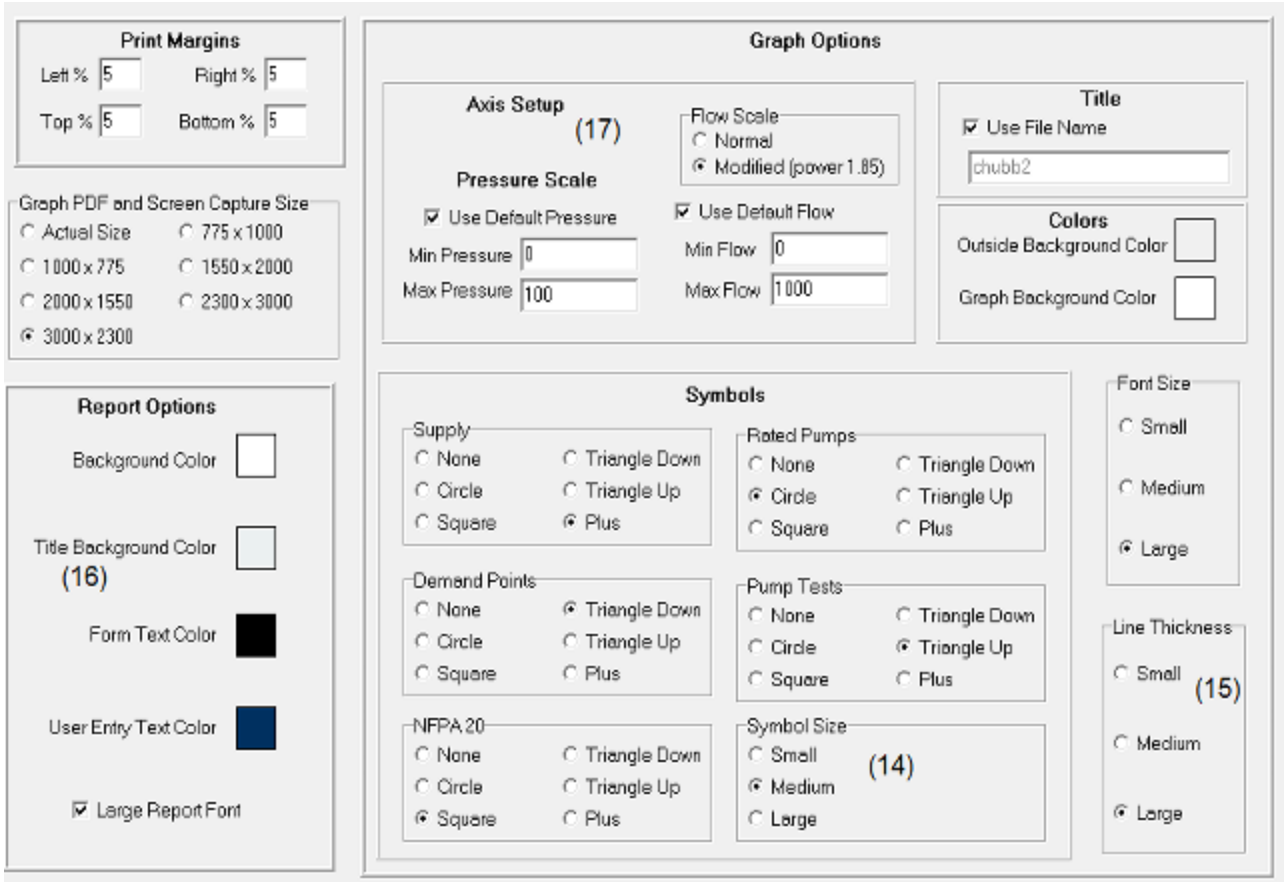
COMMENTS

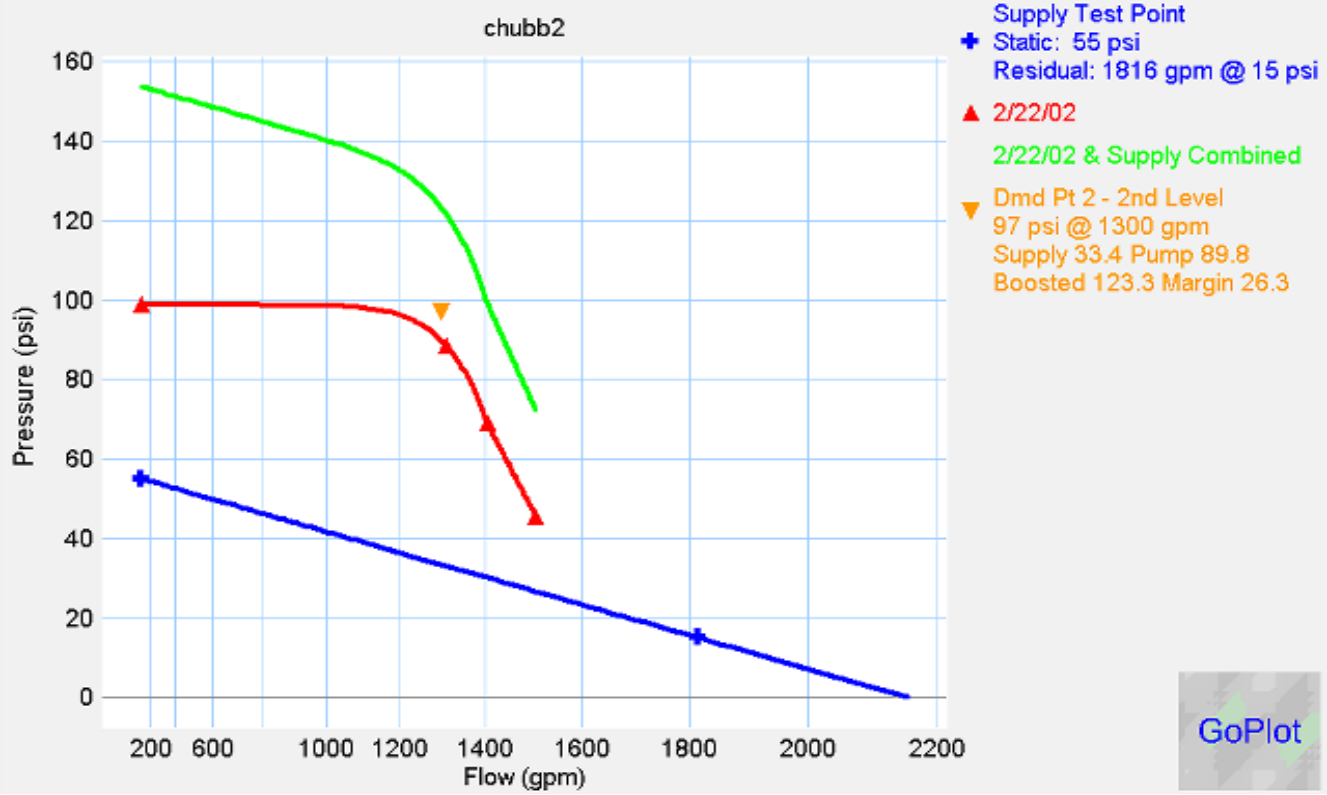
The Report Screen has the buttons shown below for easy access to view the report.

1 2 3 4 5 6

Button 1 displays the layout of all of the pages in the report. Button 2 shows a single page. Use buttons 5 and 6 to move between pages. Button 3 shows a page zoomed to the page width level. Button 4 shows a page at maximum zoom.

# Setup Screen Tab


Print Margins
Left % 5
Right % 5
Top % 5
Bottom % 5
Graph PDF and Screen Capture Size
Actual Size
775 x 1000
1000 x 775
1550 x 2000
2000 x 1550
2300 x 3000
3000 x 2300
Report Options
Background Color
Title Background Color
(16)
Form Text Color
User Entry Text Color
Large Report Font
Graph Options
Axis Setup
(17)
Pressure Scale
Use Default Pressure
Min Pressure 0
Max Pressure 100
Flow Scale
Normal
Modified (power 1.85)
Use Default Flow
Min Flow 0
Max Flow 1000
Title
Use File Name
chubb2
Colors
Outside Background Color
Graph Background Color
Symbols
Supply
None
Triangle Down
Circle
Triangle Up
Square
Plus
Rated Pumps
None
Triangle Down
Circle
Triangle Up
Square
Plus
Demand Points
None
Triangle Down
Circle
Triangle Up
Square
Plus
Pump Tests
None
Triangle Down
Circle
Triangle Up
Square
Plus
NFPA 20
None
Triangle Down
Circle
Triangle Up
Square
Plus
Symbol Size
Small
Medium
Large
(14)
Font Size
Small
Medium
Large
Line Thickness
Small
(15)
Medium
Large




GoPlot includes many features to allow the user to customize their plots . Most lines for plots and symbols have a color button next to the various Display checks. By clicking on the color button the user can select the colors for each line or symbol and the accompanying legend that appears on the right. This is illustrated on the above graph (13). The Supply Plot, for example) is blue and the accompanying information in the legend is also blue. Symbol Type and size can be customized on this screen ( 14). Three selections are provided for Plot Font size (15). Several options for the Report are provided (16). Plot scales and type Normal (linear) or Power can be selected (17). Note that the Power (1.85) is usually used for supply curves since this selection produces a straight line.

# 2 Index

**~Type, 242-247**
**10 Year HW Coefficient , 839**
**10 year roughness, 1034-1042**
**24 Hour Simulation, 901-907**
**3d Symbols, 405-413**
**About the Online Help , 37-38**
**Absolute Viscosity, 1599-1600 , 1594-1599 , 1603-1604**
**Absolute Viscosity Ideal gas, 1600-1602**
**accuracy, 276-281**
**accuracy was not attained, 721-737**
**Active Valve, 315-318**
**Active Valve for GAS, 1604-1605**
**Active Valve K in Gas systems, 1615-1616**
**active valve surge, 1058-1120**
**active valves surge, 1203-1209**
**actual diameter, 264**
**Additional Data Surge, 1284**
**Age Based Roughness, 1034-1042**
**air in pipeline for surge, 1298-1299**
**air in pipes, 401-404**
**Air Slam Pressure Surge Tool, 1321-1322**
**Air Vacuum Library, 347-349 , 351-352**
**air vacuum valve surge, 1058-1120**
**air valve locations export, 1356-1357**
**Air Valve Orifice Size Tool, 1322-1323**
**Air Valve Placement Tool, 1323-1332**
**Air Valves, 1209-1220**
**Algorithm, 1056-1058**
**All about nodes, 319-323**

**All About Pipes, 318-319**
**Alt Key Commands, 110-113**
**Altitude valve, 909-912**
**altitude valves, 912-914**
**Analysis method former versus current, 744-745**
**Analysis Setup, 116-120**
**Analyze (Main Menu), 116-120**
**analyze scenarios, 664-669**
**Analyzing for liquids other than water, 299**
**Animate map, 162-165 , 471-473**
**Animated Profile, 706-717**
**Annulus Pipe Flow, 323-324**
**Anticipation Valve, 1209-1220**
**anticipation valve surge, 1058-1120**
**Apply / Undo to Last Apply, 261-264**
**ArcView Export Utility, 804-807**
**ArcView Import Utility, 793-804**
**ARI-CAD Profile Import, 833-835**
**Arrow for Flow, 501-504**
**arrow panning, 518-520**
**Atmospheric Discharge, 324-325**
**audio video tutorials, 89-92**
**Auto Orthongonalize GoFlow, 1440-1442**
**AutoCad Exchange, 807-812**
**AutoCAD files as Backgrounds, 480-481**
**Automatic Demand Distribution, 228-232**
**Automatic on/off, 217-220**
**automatically insert nodes into pipe, 120-127**
**Back Flow Preventor Library, 347-349**
**Back Flow Preventors, 351-352**
**Background file types supported, 507-511**

**Background Images, 473-475**
**Background maps from internet, 418-466**
**background position, 507-511**
**background reference file, 507-511**
**background scaling, 525-532**
**Backgrounds, 475-480**
**Backgrounds Grids, 498-501**
**Backup Files, 206-207**
**Ball Valve, 315-318**
**BFP Library, 347-349 , 351-352**
**Bladder Precharge Tool, 1332-1353**
**bladder surge tank, 1058-1120**
**Bladder Tanks, 1209-1220**
**blowoff constant, 849-850**
**Blowoff Hydrant, 324-325**
**bmp file for nodes, 513-518**
**Booster Pump Sprinkler System, 1442-1444**
**boxes node information, 174-180**
**Branch diameter calculation for LPS systems, 986-1028**
**Branch Line Diameters, 959-961**
**branch lines - removing, 738-740**
**Branched Rural Systems, 914-957 , 958-959**
**break in pipe with rupture disk in Surge, 1058-1120**
**breaker, 401-404**
**Bulk Rate**
Calibrating, 878-880
**Butterfly Valve, 315-318**
**buttons, 261-264**
**buttons node information, 174-180**
**buttons on map, 146-152**
**bypass lines surge, 1203-1209**

**C Factor Calculator Tool, 840**
**CAD files as Backgrounds, 480-481**
**CAD Import Export, 807-812**
**Calculate branch diameter applicaton, 986-1028**
**Calculate Branch Diameters, 959-961**
**Calculate Intrusion, 1052-1055**
**Calculator Tool, 839-840**
**Calibration, 575-576 , 619-634**
Water Quality, 878-880
**Calibration Demonstration, 47-89**
**Calibration Examples, 595-618**
**Calibration of Hydraulic Networks, 543-575**
**Calibration setup, 619-634**
**Calibration Wizard, 576-595**
**Case Time numbers, 221-223**
**case/time report, 152-156**
**Cavitation Head , 1052-1055**
**Cavity Collapse Surge, 1052-1055 , 1247-1261**
**CD tutorials, 89-92**
**Change Data, 213-214**
**Change Pattern Automatic, 217-220**
**Change Patterns, 214-217**
**Changes for nodes, 169-170**
**Changes for pipes, 184-185**
**changes in surge limit, 1055-1056**
**Check Valve, 325**
**Check Valve action Surge, 1247-1261**
**Check Valve Closure Time, 1137-1151**
**Check Valve Dynamics Example, 1137-1151**
**Check Valve Dynamics in Surge, 1127-1137**
**Check Valve Node, 1123-1127**

**Check Valve Settings Surge, 1052-1055**
**Check Valves - Zero Velocity, 1191-1193**
**Check valves surge, 1203-1209**
**Chinese, 162-165**
**Classic Menu in KYnetic Interface, 114-116**
**Closed Loop Systems, 325-326**
**Closed Pipes, 105-110**
**closed surge tank, 1058-1120**
**Closed Surge Tanks, 1209-1220**
**Closed Tank Hybrid, 1166-1175**
**Colors, 481-482**
**Colors contours emphasis, 486-495**
**Column Headings, 235-242**
**Complex CAD Formats Background Method, 480-481**
**Compressors, 326**
**Computational Period EPS, 907-909**
**Connect Crisscross Pipes, 232-234**
**Connection to Main, 369-373**
**Connection to tank resistance, 847-849**
**Connectivity Check, 116-120**
**Constant Density Gas, 1599-1600**
**Constant for Hydrants Flushing, 974-985**
**Constant for sprinkler or leak, 394-396**
**Constant power pumps, 376-383**
**Constraints Data, 640-643**
**Contact Us, 22-23**
**Contents, 19-22**
**Contour animation, 162-165**
**Contours, 486-495 , 777-790**
**Contours - Main Menu, 482-484**
**control switches, 912-914**

**Control Switches Data, 909-912**
**conventions surge, 1056-1058**
**convergence issues, 721-737**
**Convert Surge 5 files, 835-836**
**Convert WaterCAD, 812**
**Converting files**
WaterCAD, 838-839
**Copy and Paste, 224-228**
**Copy Paste, 120-127**
**Cost and Inventory Calculations, 643-652**
**creating a transient surge, 1058-1120**
**Creating Custom Pump Files, 1151-1156**
**Creating Profiles, 717-721**
**Critical Pres. Gas, 1594-1599**
**Critical Temp. Gas, 1594-1599**
**Critical temperature and pressure, 1603-1604**
**Crossover, 190-196**
**crossover pipes connecting, 232-234**
**csv file profile import, 833-835**
**cumulative flow, 338**
**Cursor Notation, 484-486**
**curve graphing, 721-737**
**curves for pumps, 721-737**
**Customized Reporting, 748-753**
**customized reports, 777-790**
**Cv 100%, 315-318**
**CV Dynamics Example, 1137-1151**
**CV Setting for Inertial Effects, 1052-1055**
**CV Time Tool, 1137-1151**
**Cv valve coefficient , 146-152**
**Cybernet Import, 816**

**Cylindrical Tank, 396-401**
**Darcy-Weisbach Table, 299-301**
**DAT Import, 816-819**
**Data conversion**
WaterCAD, 838-839
**data editing in groups, 267-276**
**Data Entry in Tables, 242-247**
**Data Exchange, 819-820**
**Data Files, 207-208**
**Data Files Combining, 224-228**
**data for nodes, 170-174**
**Data Records, 208-209**
**Data Requirements, 234-235**
**Data Table - Quickstart Example, 242-247**
**data table button, 261-264**
**data table set up, 209-210**
**Data Tables, 235-242 , 825-827**
**DBP Rule, 851-853**
**Deceleration in Check Valve Dynamics Example, 1137-1151**
**decimal place in labels, 791-793**
**decimal seperator, 36-37**
**deep wells, 390-393**
**Default Hydrant and Hose Constant, 961-974**
**Default Power Cost EPS, 907-909**
**Default Sprinkler K, 1432-1435**
**Default Wave Speed, 1052-1055**
**Deleting Intermediate Nodes, 247-248**
**DEM Elevation alternative, 418-466**
**DEM Elevations Import Tool, 820-821**
**Demand**
Meters, 355-358

**Demand Allocations/Meters, 220-221**
**demand change surge, 1058-1120 , 1220-1244**
**Demand Factors, 221-223**
**Demand Patterns, 221-223**
**demand residential connections, 141-145**
**Demand Specification, 223-224**
**Demands**
Meters, 358-359 , 393-394
**Demands in Surge, 1052-1055**
**demands multiple types at nodes, 146-152**
**demands surge, 1058-1120**
**demo kypipe, 40-47**
**Demos, 47-89**
**Density and Viscosity Table Tool, 1033-1034**
**density changes, 1028-1033**
**Design Tool Parameter Calculator, 640-643**
**diameter nominal and actual, 264**
**Diameters of Branch Lines, 959-961**
**Digital Elevation Maps, 820-821**
**Dip Tube Surge Tank, 1166-1175**
**Direct Parameter Calculation, 640-643 , 652-662**
**Discharge to atmosphere, 324-325**
**disconnected hydrants valves or meters, 281-284**
**Disconnection, 105-110**
**distance of flow through piping, 740-744**
**Distance Scale, 504-507**
**DOS versions, 833**
**Driving Directions - Internet Maps, 466-471**
**DT2 Import, 832-833**
**DXF Import Export, 807-812**
**Dynamic Air Valves, 1156-1160**

**Dynamic Characteristic of Check Valves, 1127-1137**
**Dynamic Friction Method, 1284-1285**
**Edit (Main Menu), 120-127**
**edit pipes and nodes, 210-213**
**editing groups of pipes or nodes, 267-276**
**Electrically Actuated Surge Anticipation Valve, 1160-1164**
**EleSAV, 1160-1164**
**Elevation Contours, 486-495**
**elevation datum, 141-145**
**Elevation Import from DEM, 820-821**
**Elevation Retrieval, 418-466**
**Emphasis, 777-790**
**Emphasis / Contours - Nodes, 486-495**
**Emphasize Pipes as Nodes, 495-498**
**End Nodes, 326-328 , 361-364**
**Envelope pressure hgl, 706-717**
**EPA Stage 2 DBP Rule, 851-853**
**EPA Surge, 1319-1321**
**EPANET, 853-878**
**EPANET Conversion, 821-824**
**EPANET Export, 824-825**
**EPS, 901-907**
**EPS (Extended Period Simulation), 907-909**
**EPS demonstration, 47-89**
**EPS2Surge, 1353**
**equations for flow, 276-281**
**Equations KYPipe analysis, 94-95**
**Equivalent Surge Tank Converter Tool, 1353-1354**
**Errors in Report, 105-110**
**ESFR Remote Region sprinklers, 1475-1478**
**ESFR Systems, 1442-1444**

**example model kypipe, 40-47**
**Example Quality Analysis, 853-878**
**Example Surge Model, 1261-1274**
**excel file profile import, 833-835**
**Excel Import and Export, 825-827**
**Execute GenFile, 827**
**Exit Head Surge, 1052-1055**
**Explicit Calibration, 595-618**
**export air valve locations, 1356-1357**
**Export EPANET, 824-825**
**Extended Period Simulation, 907-909**
**Extended Period Simulations (EPS), 901-907**
**Extract Elevations from DEM Files, 820-821**
**Facilities Management (Main Menu), 127-129**
**FCV, 328-329**
**Features, 28-36**
**Feed Pipe, 396-401**
**feed tank surge, 1058-1120**
**File (Main Menu), 129-135**
**File Conversion, WaterCAD, 838-839**
**File format Surge 5 SIC, 835-836**
**File printing, 202-206**
**File Pump - Custom, 1151-1156**
**File Recovery, 206-207**
**File size, 146-152**
**file types for background, 507-511**
**filename display, 146-152**
**Files Backup, 206-207**
**fill empty pipeline in Surge, 1298-1299**
**Find Pressure Zone, 127-129**
**find/purge parallel pipes, 116-120**

**Fire Flow Graphs, 961-974**
**Fire Flows, 985-986**
**Fire Flows (Calculated), 961-974**
**Fire pump evalutation, 1616-1633**
**Fire Pump for Sprinkler System, 1442-1444**
**Fire Sprinkler demo, 1407-1429**
**Fire Sprinkler System Generator, 1444-1458**
**Fire Sprinkler System Generator., 1435-1440 , 1458-1474**
**Fittings, 248-254**
**fittings losses, 305-309**
**Fittings table, 186-190**
**Fixed Demands Surge, 1052-1055**
**fixed mode, 261-264**
**Flow Change Surge, 1220-1244**
**flow coefficient resistance, 847-849**
**Flow Control Valve, 388-390**
**Flow Control Valve (FCV), 328-329**
**flow control valves surge, 1203-1209**
**Flow cumulative, 338**
**Flow Direction Arrow, 501-504**
**Flow equations, 276-281**
**Flow path, travel time, and distance, 740-744**
**Flow specifications, 102-103**
**Flow Units, 311-313**
**flows negative, 116-120**
**Fluids not water, 299**
**Flush Flows, 331-332**
**Flushing, 985-986**
**flushing flow method, 141-145**
**Flushing Pipes, 974-985**
**Font for Labels, 501-504**

**font size labels, 174-180**
**Force Calculations Tool, 1354-1356**
**Force main modeling, 986-1028**
**Force Main Pumping, 352-355**
**Former Analysis Method, 744-745**
**Friction Method, 146-152 , 1284-1285**
**Full Menu, 162-165**
**Gas, 1594-1599**
Units, 1614-1615
**Gas Analysis non-ideal, 1602-1603**
**Gas Constant Density, 1599-1600**
**Gas Cv Calculations Tool, 1616**
**Gas Density, 1599-1600 , 1594-1599**
**Gas Ideal, 1600-1602**
**Gas K Calculations Tool, 1615-1616**
**Gas properties, 1603-1604**
**Gate Valve, 315-318**
**Generate Air Valvel Locations Tool, 1356-1357**
**Generate Change Pattern, 217-220**
**Generate Fire Sprinkler System, 1444-1458 , 1458-1474**
**Generate Intermediate Pump File Tool, 1357-1358**
**Generate System GoFlow, 1435-1440**
**Generator, 1175-1191**
**GIS data from Google Earth, 828-832**
**GIS Export, 804-807**
**GIS files as background images, 473-475**
**GIS Import, 793-804**
**GIS import tips, 281-284**
**Global Demand Factors, 221-223**
**Globe Valve, 315-318**
**Glyphs, 405-413**

**Go Past Graph Automatically, 1284-1285**
**GoFlow, 1407**
Units, 1478-1479
**GoFlow - Auto Orthogonalize, 1440-1442**
**GoFlow - Booster or Fire Pump Water Supply, 1442-1444**
**GoFlow - Equivalent K-factor with a Sprinkler on a Riser Nipple, 1474-1475**
**GoFlow - Generate System, 1435-1440**
**GoFlow - Grid Generator Example, 1458-1474**
**GoFlow - Nodes, 1429-1432**
**GoFlow - Quickstart Example, 1407-1429**
**GoFlow - System Data, 1432-1435**
**GoFlow - Tree Generator Example, 1444-1458**
**GoFlow remote area, 1475-1478**
**Google Earth Import and Export, 828-832**
**Google Maps, 418-466**
**GoPlot, 1616-1633**
**grade, 390-393**
**Grade Change Surge, 1220-1244**
**Graph during Surge analysis, 1244-1245**
**Graph fire pumps, 1616-1633**
**Graph pipes, 196-199**
**Graph results setup, 180-183**
**graphic files for nodes, 513-518**
**Graphs, 777-790**
**Graphs Surge, 1247-1261**
**grid snap, 146-152**
**Grids, 498-501**
**group editing, 267-276**
**Group mode buttons, 261-264**
**Group Operations, 254**
**Groups**

User Data, 284-299
**Groups, Subsets, Minimizing, 254-255**
**Guage Difference, 369-373**
**Hazen Williams factor calculator tool, 840**
**Hazen-Williams Table, 301-304**
**Head drop element, 349-351**
**Head envelope, 706-717**
**head flow data resistance, 847-849**
**Headloss Calculator Tool, 840-842**
**Headloss Factor, 276-281**
**heat loss in pipes, 1607-1614**
**Help (Main Menu), 135-137**
**Hill Charts, 1175-1191**
**History and Development, 23-24**
**Holding Tank, 329-331**
**Holding tank element, 352-355**
**Hose and Nozzle Constants Tool, 843-844**
**Hose connections, 332-334**
**Hotkeys and Shortcuts, 110-113**
**hover data, 484-486**
**hybrid surge tank, 1058-1120**
**Hybrid Surge Tanks, 1166-1175**
**Hydrant and Hose Constant, 961-974**
**Hydrant Color Coding, 961-974**
**hydrant constant, 849-850**
**Hydrant Data, 331-332**
**Hydrant Fire Flows, 961-974**
**Hydrant flow equation for pressure supply, 369-373**
**Hydrant flushing, 974-985**
**Hydrant Monitor, 332-334**
**Hydrant Report, 961-974**

**hydrant/monitor data, 141-145**
**Hydrants, Fire Flows, and Flushing Pipes, 985-986**
**hydraulic grade, 390-393**
**Hydraullically Actuated Surge Anticipation Valve (HydSAV), 1164-1166**
**Hydropneumatic Tank, 336-338**
**Hydropneumatic Tank Element, 334-336**
**Hydrotank, 334-336**
**Hydrotank Using Tank Element, 336-338**
**HydSAV, 1164-1166**
**hyperlink for nodes, 538-542**
**Ideal Gas, 1600-1602**
**image files for background, 507-511**
**images for nodes, 513-518**
**Implicit Calibration, 595-618**
**Import , 837-838**
**Import DT2, 129-135**
**Import DT2 File, 832-833**
**Import Elevations from DEM, 820-821**
**import inp EPANET, 821-824**
**Import KY, 833**
**Import WaterCAD, 812**
**Improve Loop Analysis Rural, 914-957**
**inertia calculation, 1360-1362**
**Inertia for pumps, 1360-1362**
**Inertial effects Surge, 1052-1055 , 1247-1261**
**Inf error, 146-152**
**Inflow to Tank, 396-401**
**Information Window pipes, 190-196**
**Information Windows, 168-169**
**infows demand pattern, 141-145**
**Init Ratio, 315-318**

**Initial Gas Volume, 1285-1298**
**Initial Level - Bladder Tank, 1058-1120**
**Inline Meter, 338**
**inp export, 824-825**
**inp import, 821-824**
**Input and Editing Shortcuts, 255-257**
**Input Table, 113-114**
**insert nodes into pipe automatically, 120-127**
**Inside Hose, 1432-1435**
**Installation Instructions, 25-28**
**Instantaneous Peak Demand Curve, 914-957**
**Intermediate Nodes Deleting, 247-248**
**Intermediate pump file, 1357-1358**
**Internal Node Basic Data, 340-341**
**Internal Node Data, 341-344**
**Internal Nodes, 338-340 , 361-364**
**International Decimal Settings, 36-37**
**Internet Background Maps and Automatic Elevation Retrieval, 418-466**
**Internet link for nodes, 538-542**
**Interpolate Elevations, 257-259**
**Intersection of pipe - crossover, 190-196**
**Introduction to Modeling, 92-94**
**Intrusion Calculator, 1042-1044**
**Intrusion Surge, 1052-1055**
**Inventory, 643-652**
**Inventory Cost, 116-120**
**IPD Curves, 914-957 , 958-959**
**ISDE Initial Distribution System Evaluation, 851-853**
**Isolation Zones, 414-418**
**Junction Data, 344-346**
**junction data surge, 1058-1120**

**K Factor Riser Nipple, 1474-1475**
**K, Loss Coefficients, 248-254**
**Key for map, 504-507**
**Keyboard shortcuts, 110-113**
**Kinematic Viscosity, 276-281**
**KMZ export import, 828-832**
**Korean, 162-165**
**Ks sprinkler constant, 394-396**
**KY files, 833**
**KYnetic Configuration Menu, 113-114**
**KYNetic Features, 754-776**
**KYPipe and Surge Demo files , 47-89**
**KYPipe equations, 94-95**
**Label font settings, 501-504**
**Labels, 501-504**
Text, 534-537 , 532-534
User Data, 284-299
**Labels (Main Menu), 137-141**
**labels decimal place, 791-793**
**Labels editing, 174-180**
**Labels on Map, selecting results, 776-777**
**Labels won't display (Selected Labels Only), 501-504**
**Language**
Korean, 162-165
**layout mode, 261-264**
**Leak Element, 394-396**
**leak surge, 1058-1120**
**Leakage Coefficient, 1052-1055**
**Legend, 504-507**
**Length Accuracy (Tolerance), 1052-1055**
**Libraries, 347-349**

**Library elements, 351-352**
**License Agreement, 24-25**
**limited output, 791**
**link for nodes, 538-542**
**liquids not water, 299**
**Liquified Natural Gas Bulk Modulus, 1193-1203**
**LNG Bulk Modulus, 1193-1203**
**Load Rejection, 1175-1191**
**Locate Remote Sprinkler Area, 1475-1478**
**lock button, 261-264**
**Lock Zoom Window, 518-520**
**Logical Text Inputs, 259-260**
**Logo on map, 504-507**
**Lookup Properties for Gas, 1603-1604**
**Loop Analysis Rural, 914-957**
**Loss Coefficients, 248-254**
**Loss Element Data, 349-351**
**Loss Element Library, 351-352**
**losses for fittings, 305-309**
**Low Pressure Sewer element, 352-355**
**Low Pressure Sewer Holding Tank, 329-331**
**Low Pressure Sewer Models, 986-1028**
**LPS Tank, 352-355**
**Macro Calibration, 595-618**
**Main Connection, 369-373**
**Main Menu, 141**
**Main Supply, 369-373**
**Map animation, 162-165**
**map buttons, 146-152**
**Map label editing, 174-180**
**map labels, 777-790**

**Map Legend, 504-507**
**Map Link, 507-511**
**map notes, 523-525**
**map panning, 518-520**
**Map printing, 202-206**
**Map results, 137-141**
**map scaling, 525-532**
**Map Screen, 511-513**
**map screen crashes, 513**
**MAP SETTINGS, 141**
**Map view**
saved, 162-165
**map views, 261-264**
**Mapquest Maps, 418-466**
**Maps from internet, 418-466**
**Material Cost, 643-652**
**Material Roughness table HW, 301-304**
**Materials and Rating, 260**
**Max Distance Between Branches, 1432-1435**
**Max Distance Between Sprinklers, 1432-1435**
**max/min output values, 156-159**
**Maximum Area Per Sprinkler, 1432-1435**
**Maximum number of trials, 276-281**
**maximum number of trials executed, 721-737**
**Memo, 162-165**
**Menu Customize, 114-116**
**Menu, missing items, 162-165**
**Merging Data Files, 224-228**
**Meter Based Demands, 260-261**
**Meter inline for EPS, 338**
**Meter Record Data, 355-358**

**Metered Connection Data, 355-358**
**Meters, 358-359**
**Methane Hydrate Bulk Modulus, 1193-1203**
**Method of Analysis, 94-95**
**Minimize During Analysis, 1284-1285**
**minimum capacity for sprinkler systems, 1479-1481**
**Minimum Pressure for Fire Flows, 961-974**
**Minimum Required Density, 1432-1435**
**Minimum View Scale, 513**
**Minor Loss Coefficients Table, 305-309**
**minor loss components, 349-351 , 359-360**
**minor loss resistance, 847-849**
**Minor Losses, 248-254**
**Model combining, 224-228**
**Model Simplification, 95-96**
**Modeling Air in Pipeline for Surge, 1298-1299**
**Modeling intro, 92-94**
**modeling primer, 40-47**
**Modelling wells, 390-393**
**Modified Hazen Williams Formula, 304-305**
**Modulating (Regulating) Valve Tool, 1358-1359**
**modulating regulator valve, 1058-1120**
**molecular weight, 1603-1604**
**Molecular Wt., 1594-1599**
**Monitor, 332-334**
**monitor data, 141-145**
**mouse pointer data, 484-486**
**Move System, 120-127**
**Movie, 706-717**
**Multiple Scenarios - Changes, 103-104**
**Name Label Off, 501-504**

**names for nodes and pipes, 146-152**
**Needle Valve, 315-318**
**negative flows, 116-120**
**Network Analysis, 662-664**
**Network Elements, 360-361**
**New Features, 28-36**
**New File Specifications, 129-135**
**NFPA 15 Report Content, 745-747**
**NFPA Hydrant Color Coding, 961-974**
**NFPA Standards in KYPipe and GoFlow, 1369-1405**
**Nipple Riser K factor, 1474-1475**
**Node ~Type, 242-247**
**Node and Pipe Glyphs, 405-413**
**Node Change Box, 169-170**
**Node Changes, 214-217**
**Node Color, 481-482**
**Node Data Boxes, 170-174**
**Node Emphasis, 486-495**
**Node Images and Text Nodes, 513-518**
**Node Info window, 113-114**
**Node Information Window, 174-180**
**Node memo, 523-525 , 162-165**
**Node Results Boxes, 180-183**
**Node Results for Pipes, 495-498**
**Node Size, 481-482**
**Node Symbols, 405-413**
**Node types, 361-364**
**Node Types and Properties, 361-364**
**Node User Box, 183-184**
**Nodes, 326-328 , 360-361**
**Nodes GoFlow, 1429-1432**

**Nodes Internal, 341-344 , 340-341 , 338-340**
**Nodes Too Many, 247-248**
**nominal diameter, 264**
**Non Return Valves, 1191-1193**
**Non-Ideal Gas, 1602-1603**
**Non-Spilling Tanks, 1209-1220**
**north arrow, editing, 261-264**
**notes on map, 523-525 , 162-165**
**nozzle constant, 843-844**
**NRVs, 1191-1193**
**number formats in Windows, 36-37**
**OCS Screen, 664-669**
**on/off pumps and valves, 217-220**
**On/Off Valves, 364-365**
**one way surge tank, 1058-1120**
**open surge tank, 1058-1120**
**Open Surge Tanks, 1209-1220**
**Operating Modes, 261-264**
**Operating Temp, 1594-1599**
**Operating Temperature gas, 1600-1602**
**Operational Control Settings, 664-669**
**Optimal Design, 669-671**
**Optimal Pump Scheduling, 671-676**
**Optimal Pump Scheduling Algorithms, 676-697**
**Optimized Calibration, 618-619**
**Optimized Calibration Data, 619-634**
**Orifice constant, 394-396**
**orifice resistance, 847-849**
**orifice size air valve, 1322-1323**
**Orthogonalize pipes, 255-257**
**Other, 141-145**

**OTHER DATA, 145-146**
**output - reports, 156-159**
**output selected, 791**
**Output Surge, 1245-1247**
**Outside Hose Demand, 1432-1435**
**Overview and Getting Started, 38-40**
**Panning Controls, 518-520**
**panning crashes, 513**
**parallel pipes, 116-120**
**Parallel pump graph, 721-737**
**partial pipe flow, 401-404**
**path of flow through network, 740-744**
**pathname display, 146-152**
**PCX background files, 833**
**PCX backgrounds, 836-837**
**PDD Curves, 914-957 , 958-959**
**Peak Demand, 221-223**
**Peak Demand Diversity Analysis, 914-957**
**Peak Demand Requirements, 958-959**
**Peak Factors, 959-961**
**Penstock, 1175-1191**
**periodic variation, 1058-1120**
**PID Symbols, 405-413**
**Pipe Aging, 1034-1042**
**Pipe Break, 697-701**
**Pipe break in surge using rupture disk, 1058-1120**
**Pipe Break Volume, 697-701**
**Pipe Change Box, 184-185**
**Pipe Changes, 214-217**
**Pipe Color, 481-482**
**Pipe connections, 393-394**

**Pipe Copy Paste, 224-228**
**Pipe Cost, 643-652**
**Pipe crossover, 190-196**
**Pipe Data Boxes, 186-190**
**pipe data surge, 1058-1120**
**pipe design, 669-671**
**Pipe Diameter, 264**
**pipe direction, 186-190**
**Pipe Emphasis, 520-523**
**Pipe Emphsis, 486-495**
**Pipe Fittings, 248-254**
**Pipe fittings losses, 305-309**
**pipe flow partial, 401-404**
**Pipe Flushing, 974-985**
**Pipe Info window, 113-114**
**Pipe Information Window, 190-196**
**pipe length minimum, 120-127**
**Pipe losses, 349-351**
**Pipe Material HW roughness table, 301-304**
**Pipe Material Roughnesses DW, 299-301**
**pipe meter demand, 141-145**
**Pipe meters, 393-394**
**Pipe Minor Losses, 248-254**
**Pipe Results Boxes, 196-199**
**pipe scale factor, 141-145**
**Pipe schedules, 264-267**
**Pipe Size , 481-482**
**pipe sizing, 669-671**
**Pipe Startup Surge, 1220-1244**
**Pipe System Components, 101-102**
**Pipe System Geometry, 96-101**

**Pipe System Layout and Modification, 210-213**
**Pipe Thickness, 405-413**
**Pipe Type Data, 264-267**
**Pipe Type Table**
User Data, 284-299
**Pipe User Box, 199-202**
**PIPE2000 Tutorial (Audio/Video) , 89-92**
**Pipe2010 Features , 706-717**
**Pipe2012 Features, 28-36**
**Pipe2014 Features, 754-776 , 28-36**
**Pipe2016 Features , 28-36**
**Pipeline profile, 706-717**
**Pipeline profile Pipe2008 , 701-706**
**Pipes, 360-361**
**Pipes number in model, 135-137**
**pipes parallel, 116-120**
**pipes wave speed, 1193-1203**
**Piping and Instrumentation Diagram Symbols, 405-413**
**piping equivalent resistance, 847-849**
**Plan View-Profile View, 413-414**
**Plots fire pumps, 1616-1633**
**pointer data, 484-486**
**Positive Displacement Pump in Surge, 365-368**
**Power (HP or KW) Calculations Tool, 844**
**Power Cost, 643-652**
**Power Cost EPS, 907-909**
**Precharge Bladder Tanks, 1332-1353**
**precision, 791-793**
**Preferences, 146-152 , 1284-1285**
**Prefixes node and pipe names, 146-152**
**Presentation Generator, 754-776**

**Presenting results, 777-790**
**Pressure and Flow Specifications, 102-103**
**Pressure Change Surge, 1220-1244**
**Pressure Contours, 486-495**
**Pressure graph during surge analysis, 1244-1245**
**Pressure Reducing Valve, 388-390**
**Pressure Regulating Valve (PRV), 374**
**pressure regulating valves surge, 1203-1209**
**Pressure Relief Valve in KYPipe, 375-376**
**pressure relief valve surge, 1058-1120**
**Pressure Relief Valves, 1209-1220**
**Pressure Sensitive Analysis, 635-640**
**Pressure Sensitive Demands Surge, 1052-1055**
**Pressure spiking Surge, 1247-1261**
**Pressure Supplies GoFlow, 1429-1432**
**Pressure Supply Data, 369-373**
**pressure supply surge, 1058-1120**
**Pressure Sustaining Valve, 388-390**
**Pressure Sustaining Valve (PSV), 374-375**
**pressure sustaining valves surge, 1203-1209**
**Pressure Switch, 912-914**
**Pressure Switches, 909-912**
**Pressure Zone, 127-129**
**pressurized sewer modeling, 986-1028**
**Pressurized Tank, 334-336**
**Printing, 202-206**
**Printing reports, 152-156**
**Profile, 701-706 , 706-717**
**Profile Animated, 706-717**
**Profile Data Import, 833-835**
**PRV, 374**

**PSV, 374-375**
**pump - well, 390-393**
**Pump 1.5 Qr Factor, 1432-1435**
**Pump and System Curves, 721-737**
**Pump Characteristic Curves Tool, 844-846**
**Pump Control, 909-912**
**pump curve equations, 383-388**
**Pump curves, 721-737**
**Pump Cutoff Factor, 1432-1435**
**Pump Data, 376-383**
**Pump Design, 721-737**
**Pump File - Custom, 1151-1156**
**Pump File Characteristics Tool, 1359-1360**
**Pump file intermediate, 1357-1358**
**Pump Library, 347-349 , 351-352**
**Pump Minimum Flow Valve, 383**
**pump on/off surge, 1058-1120**
**Pump Optimizing, 671-676**
**Pump plotting, 1616-1633**
**pump power, 844**
**Pump Scheduling, 671-676**
**Pump Selection Tool, 846**
**Pump shut down Surge, 1220-1244**
**Pump Sizing, 721-737**
**pump trip surge, 1058-1120 , 1220-1244**
**Pumping Cost, 643-652 , 671-676**
**Pumps, 383-388**
**Pumps - Positive Displacement Pumps in Surge, 365-368**
**pumps surge, 1058-1120 , 1203-1209**
**purge parallel pipes, 116-120**
**Pushpin, 523-525**

**Pushpin notes, 162-165**
**Quality modeling, 853-878**
**Quick Guide to Running Rural Analysis, 957-958**
**Quick Start, 40-47**
**R 100%, 315-318**
**R Value, 847-849**
**R valve coefficient , 146-152**
**Rack Sprinkler, 1429-1432**
**Rapid Pressure or Flow Change Surge, 1220-1244**
**Ratchet Pump Trip, 1058-1120**
**Rated Pressure Supply Curve Equation, 369-373**
**Rated pump curves, 721-737**
**Rated Pumps, 376-383**
**rating pipes, 260**
**Ratio of Specific Heats, 1600-1602 , 1603-1604**
**Ratio of Specific Heats Gas, 1594-1599**
**Reaction Rates**
Calibration, 878-880
**Records**
User Data, 284-299
**reference elevation, 141-145**
**reference file for background, 507-511**
**Reference Manual, 92**
**refresh, 261-264**
**regional settings, 36-37**
**Regulating Valves, 388**
**regulating valves surge, 1203-1209**
**Regulator Data, 388-390**
**Regulator Flow Control Valve, 328-329**
**Regulator pressure sustaining, 374-375**
**regulators surge, 1058-1120**

**Relief Valve in KYPipe, 375-376**
**Relief Valves Surge, 1209-1220**
**Remote area sprinkler, 1475-1478**
**Remote Region Area, 1432-1435**
**remote region for sprinklers, 1479-1481**
**REPORT, 152-156**
**Report - Adding Custom Information, 745-747**
**report - limiting output, 791**
**Report Customized, 748-753**
**report notes, 141-145**
**Report Period EPS, 907-909**
**Report Surge, 1245-1247 , 1247-1261**
**report table sorting, 146-152**
**Reports, 777-790**
**Reports (System Data), 156-159**
**Required Capacity, 1479-1481**
**Reservoir Data, 390-393**
**reservoir grade change surge, 1058-1120**
**reservoirs surge, 1058-1120**
**residential meter demand, 141-145**
**Residential Meters, 393-394**
**Residual Pressure Adjustment Tool, 846-847**
**Resistance Calculations Tool, 847-849**
**Resistance for Active Valve in Gas models, 1615-1616**
**resistance for active valves, 146-152**
**Resistance R, 847-849**
**Result Selector Bar, 776-777**
**results - limiting output, 791**
**Results animation, 162-165**
**Results Contours Emphasis, 486-495**
**Results in Surge, 1247-1261**

**Results Labels, selecting,** 776-777
**results map labels inlet versus outlet,** 146-152
**Results Nodes,** 180-183
**results on map,** 137-141
**results pipes,** 196-199
**results precision,** 791-793
**Results Presentations,** 777-790
**Results previous,** 180-183
**Results Selected Surge,** 1245-1247
**Riser elevation for sprinkler,** 394-396
**Riser Nipple K factor,** 1474-1475
**Role of Pump Inertia Values in Transient Analysis of Pipe Line System,** 1302-1319
**Roughness aging,** 1034-1042
**Roughness calculator,** 840
**Roughness Darcy Weisbach,** 299-301
**Roughness Table HW,** 301-304
**Ruler on map,** 504-507
**Rupture Disk,** 1209-1220
**rupture disk surge,** 1058-1120
**Rural Data Attribute,** 141-145
**Rural Looped Systems,** 914-957
**Rural Water Systems,** 914-957
**Rural Water Systems (Peak Demand Requirements),** 958-959
**Satic Pressure Limit,** 961-974
**Saved Results,** 747-748
**saved views,** 261-264
**SCADA,** 664-669
**SCADA for nodes,** 538-542
**scale pipes,** 141-145
**Scale System,** 120-127
**Scaling Background Maps,** 525-532

**Scenario Management, 224**
**Scenarios creating, 214-217**
**Scenarios for nodes, 169-170**
**schedules for pipes, 264-267**
**Screen Capture, 120-127**
**Screen Plot Data, 1244-1245**
**Scroll Bar, 532**
**Select Pump File Calculate Inertia Tool, 1360-1362**
**Selected Output, 791**
**series pump graph, 721-737**
**Sets and Group Mode, 267-276**
**SETUP / DEFAULTS, 159**
**Sewer holding tank, 329-331**
**sewer modeling pressurized, 986-1028**
**Shape Data, 396-401**
**shapefiles as background images, 473-475**
**Shortcuts, 110-113**
**SHP Export, 804-807**
**SHP Import, 793-804**
**SIC files, 835-836**
**side orifice surge, 1058-1120**
**sign conventions surge, 1056-1058**
**Simplified Menu option, 162-165**
**simulation memo, 141-145**
**Simulation Specs, 276-281**
**Siphon Breaker, 1120-1123**
**sizing pipes, 669-671**
**Skeletonize, 738-740**
**Skeletonize/Subset, 738**
**Sliders/Precision, 791-793**
**Snap Hydrants Valves and Meters, 281-284**

**Snap to Grid, 146-152**
**Some Special Features, 104-105**
**sorting table in report, 146-152**
**Specific Gravity, 276-281**
**Specific Gravity Gas, 1594-1599**
**Specific Gravity of a Gas, 1603-1604**
**Spike Track Tool, 1362-1366**
**spiking Surge, 1247-1261**
**spilling surge tank, 1058-1120**
**Spilling Tanks, 1209-1220**
**Spreadsheet editor, 825-827**
**Spreadsheets, 242-247 , 235-242**
**sprinker area, 1479-1481**
**sprinkler analysis lowest density, 1479-1481**
**Sprinkler Data, 394-396**
**Sprinkler demo, 1407-1429**
**Sprinkler K Default, 1432-1435**
**Sprinkler or Blowoff Constant Tool, 849-850**
**sprinkler surge, 1058-1120**
**Sprinkler System Generator, 1435-1440 , 1444-1458 , 1458-1474**
**Sprinklers remote area, 1475-1478**
**Stabilization Settings for Surge, 1284**
**Stabilization Settings Surge, 1052-1055**
**Stage 2 Rule, 851-853**
**static head elevation, 141-145**
**Steady State Demonstration, 47-89**
**Steam, 1605-1607**
Units, 1614-1615
**Storm Example , 1563-1586**
**Stormwater Modeling , 1481-1563**
**submersible pump, 390-393**

**Subsets, 254 , 254-255**
**Suction Pressure at Fire Pump, 961-974**
**Superheated Steam, 1607-1614**
**Supply from Main, 369-373**
**Supply plot fire pumps, 1616-1633**
**Surge - Reviewing and Presenting Surge Results, 1247-1261**
**Surge - System Data / Other, 1244-1245**
**Surge - System Data / Reports, 1245-1247**
**Surge - System Data / Simulation Specs, 1052-1055**
**Surge 5 conversion, 835-836**
**surge anticipation valve, 1058-1120**
**Surge anticipation valve hydraulically actuated, 1164-1166**
**Surge Anticipation Valves, 1209-1220**
**Surge Calculations and Sign Conventions, 1056-1058**
**surge changes limit, 1055-1056**
**Surge Check Valve Dynamics, 1127-1137**
**Surge Components, 1203-1209**
**Surge Control and Control Devices, 1209-1220**
**Surge Control Device examples, 1209-1220**
**Surge Demonstration, 47-89**
**Surge Elements and Data Requirements, 1058-1120**
**surge events, 1058-1120**
**Surge Friction Method, 146-152 , 1284-1285**
**Surge Graph during analysis, 1244-1245**
**Surge Modeling Example, 1261-1274**
**Surge Modeling Requirements, 1055-1056**
**Surge Pipe Break Analysis, 1274-1278**
**Surge pipe break with rupture disk, 1058-1120**
**Surge Pipe Data (Wave Speed), 1193-1203**
**Surge Preferences, 146-152 , 1284-1285**
**Surge QuickStart Example, 1261-1274**

**Surge Report Definitions, 1301-1302**
**Surge Stabilization Settings, 1284**
**Surge Tank - Hybrid, 1166-1175**
**Surge Tank Data, 1285-1298**
**surge tanks, 1058-1120**
**Surge Transient Setup Examples, 1220-1244**
**Surge5 Conversion, 835-836**
**Suter Diagram - custom, 1151-1156**
**Suter file, 1175-1191**
**SWMM, 1481-1563**
**SWMM (Stormwater) , 1481-1563**
**SWMM Error Messages, 1586-1594**
**SWMM Example, 1563-1586**
**Symbols, 405-413**
**System Curve for Surge, 1044-1052**
**System Curves, 721-737**
**SYSTEM DATA, 159-160**
**System Data Gas, 1594-1599**
**System Data GoFlow, 1432-1435**
**System Data Preferences, 1284-1285**
**System Data Surge, 1052-1055**
**system head curves data, 141-145**
**System Total Flow Requirements, 1432-1435**
**system type, 276-281**
**System Variable Graph, 162-165**
**Table of Contents, 19-22**
**table pipes, 196-199**
**Table pumps, 376-383**
**Table results setup, 180-183**
**Table Setup, 209-210**
**table sorting, 146-152**

**Tables,** 777-790
**Tables Surge,** 1247-1261
**Tabs,** 159-160
**Tabular Data Entry,** 242-247 , 235-242
**Tabulated report,** 777-790
**Tailrace,** 1175-1191
**tank connection resistance,** 847-849
**Tank Data,** 396-401
**tank inflows demand pattern,** 141-145
**tanks surge,** 1058-1120
**Temperature change in pipes,** 1607-1614
**Temperature Dependent Liquid,** 1028-1033
**Temperature Sensitive Analysis,** 1607-1614
**Text and Notes menu,** 537-538
**text mode,** 261-264
**Text Node Data,** 534-537
**text nodes,** 513-518 , 532-534
**Thickness of pipes,** 405-413
**Time Case numbers,** 221-223
**time of flow through piping,** 740-744
**Time Simulation,** 901-907
**Time Step in Surge,** 1299-1301
**Time Step Increment,** 1052-1055
**Time/case report,** 152-156
**Title for map,** 504-507
**Titles**
Map, 534-537 , 532-534
**To TIFF,** 836-837
**Too Many Nodes,** 247-248
**toolbar,** 146-152
**Tools (Main Menu),** 160-162

**Torque, 1175-1191**
**total flow, 338**
**Total Time EPS, 907-909**
**Trace Analysis, 853-878**
**training videos, 89-92**
**Transient data Surge, 1220-1244**
**transients setting up, 1058-1120**
**Travel Time Path and Distance, 740-744**
**trials, number of, 276-281**
**trip pump surge, 1058-1120**
**Troubleshoot, 105-110**
**turbine data convertor tool, 1175-1191**
**Turbines, 1175-1191**
**tutorial videos, 89-92**
**unconnected hydrants valves or meters, 281-284**
**undo / redo, 261-264**
**Units, 311-313 , 313-315**
**Units Converter Tool, 850**
**Units Gas and Steam, 1614-1615**
**Units GoFlow, 1478-1479**
**Units Help - Hover Buttons, 313**
**Units Pipe2000 : KYPipe / Surge , 313-315**
**URL and SCADA for nodes, 538-542**
**User Data, 284-299**
**User Data configuration, 209-210**
**User data nodes, 183-184**
**User data pipes, 199-202**
**USER Flow Units, 311-313**
**User Information in Report, 745-747**
**Utility Programs, 837-838**
**Vacuum Breaker element, 401-404**

**vacuum valve surge, 1058-1120**
**Valve Closure Surge, 1220-1244**
**valve open/close surge, 1058-1120**
**Valve pressure regulating, 374**
**Valve pressure sustaining, 374-375**
**Valve resistance calculation, 847-849**
**Valve Stroking Tool, 1367-1368**
**Valves, 315-318 , 404-405**
**valves on/off, 364-365**
**Variable Level Storage Tank, 396-401**
**Variable Pressure Supply, 369-373**
**Version Features, 28-36**
**Version number, 135-137**
**Versions previous, 833**
**View (Main Menu), 162-165**
**view controls, 518-520**
**viscosity changes, 1028-1033**
**Viscosity Table, 1033-1034**
**Volume - Bladder Tank, 1058-1120**
**Volume Pipe Break, 697-701**
**Wall Rate**
Calibrating, 878-880
**Warnings and Error, 105-110**
**Water Age Analysis, 853-878**
**Water Quality Calibration, 878-880**
**Water Quality Demonstration, 47-89**
**Water Quality Modeling, 853-878**
**Water Quality Sensor Placement Tool, 880-901**
**WaterCAD Conversion, 812 , 838-839**
**WaterCAD import, 821-824**
**Wave Speed, 1052-1055 , 1193-1203**

**Wave Speed Adjustment, 146-152 , 1284-1285**
**Wave Speed Tool, 1368-1369**
**Web link for nodes, 538-542**
**Wells, 390-393**
**Wicket Gate, 1175-1191**
**Width to Height Ratio GoFlow, 1432-1435**
**Zero Velocity Valves, 1191-1193**
**Zoom Lock, 518-520**
**Zoom save, 162-165**
**zoom saved views, 261-264**
**zooming, 518-520**
**zooming crashes, 513**