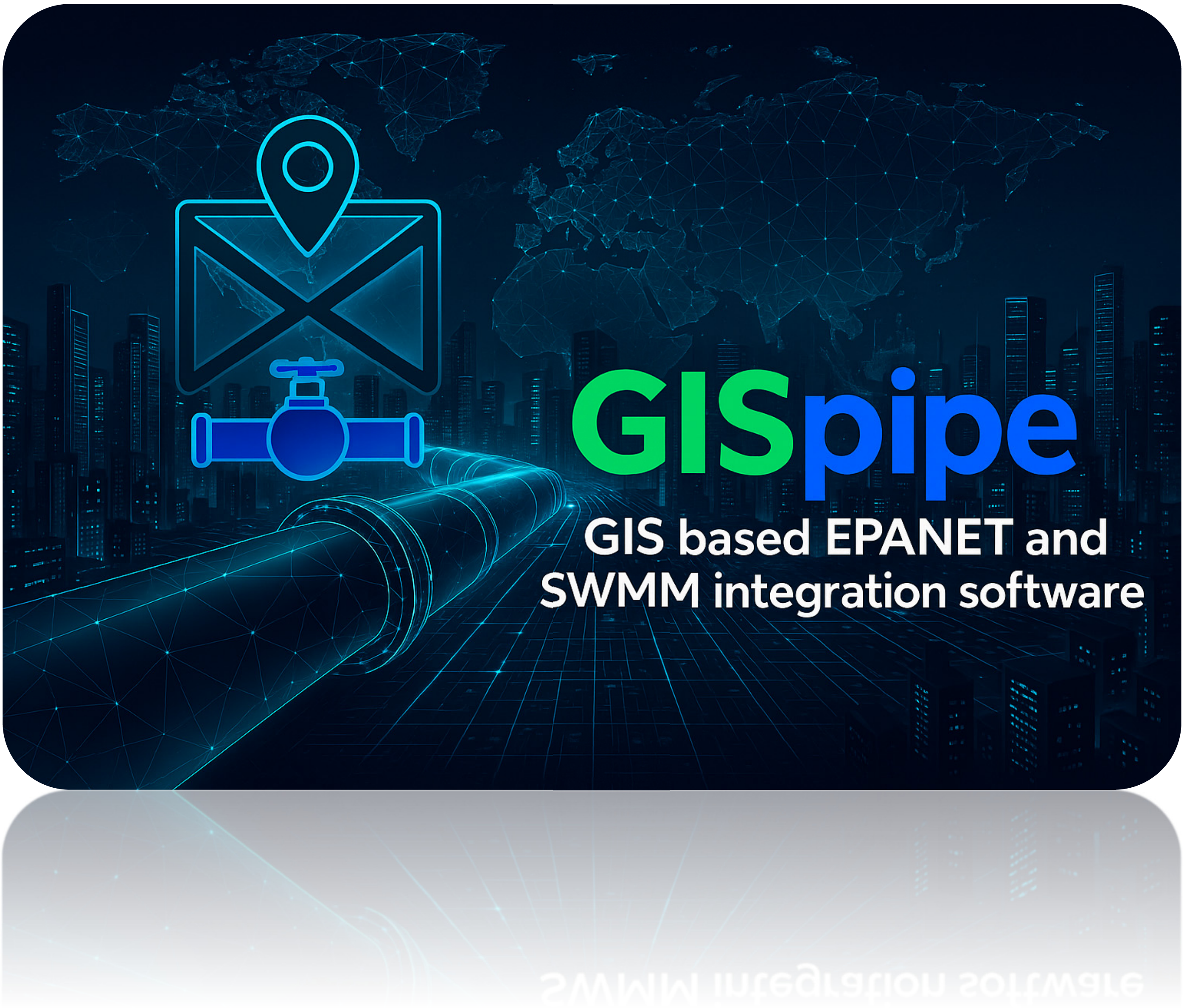


User manual

Version 1.0
September 2025

GISpipe

User Manual
Version 1.0

TABLE OF CONTENTS

Page

ACKNOWLEDGEMENT

GISpipe has been developed to enhance the planning, analysis, and management of water distribution and stormwater systems by integrating the powerful simulation capabilities of two publicly available software packages: EPANET and EPA SWMM. We gratefully acknowledge the significant contributions of the U.S. Environmental Protection Agency (USEPA) and Dr. Lewis A. Rossman for their development and ongoing maintenance of these essential tools.

The EPANET 2.2 engine, which simulates hydraulic and water quality behaviour within pressurized pipe networks, and the EPA SWMM 5.2 engine, which models the quantity and quality of runoff in urban drainage systems, are both incorporated into GISpipe to provide robust and reliable simulation results.

Full credit is given to the following official manuals, which have also been referenced and incorporated into the GISpipe documentation to ensure technical consistency and user clarity:

EPANET 2.2 User Manual
EPA/600/R-20/133
Rossman, L.A. and U.S. Environmental Protection Agency (2020)
Available at: www.epa.gov/water-research/epanet

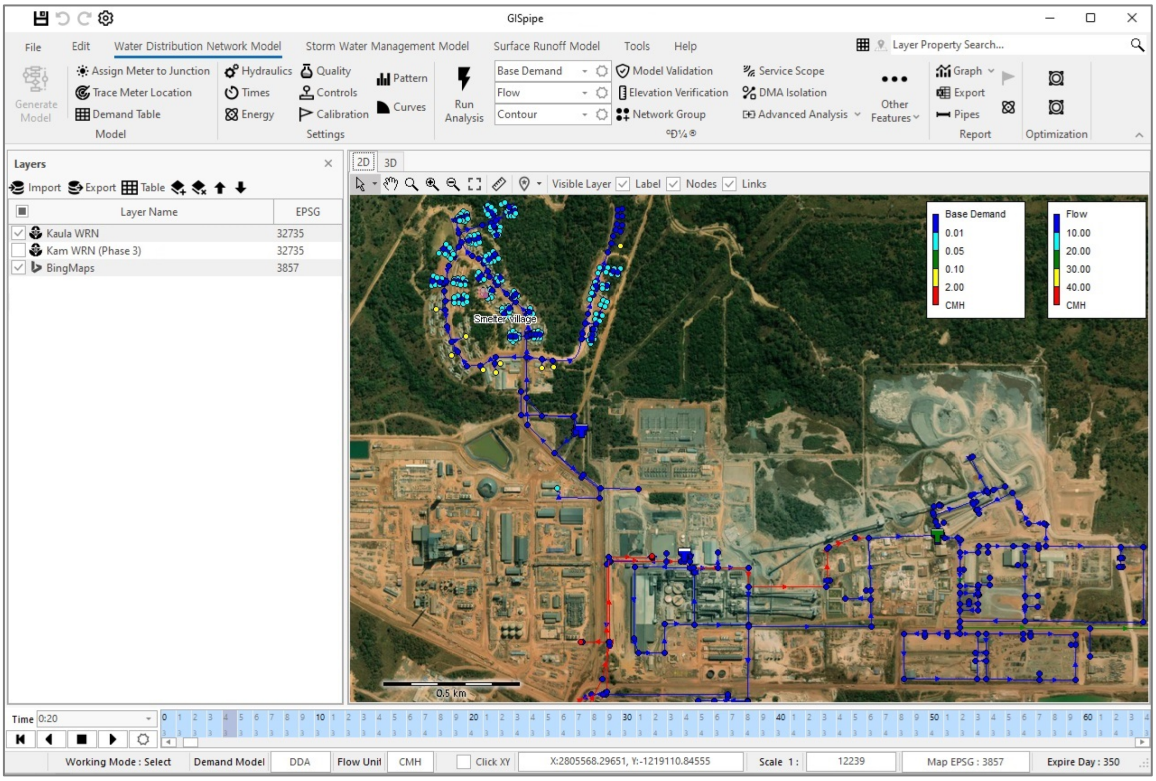
Storm Water Management Model (SWMM) User's Manual Version 5.2
EPA/600/R-22/195
U.S. Environmental Protection Agency (USEPA) (2022).
Available at: www.epa.gov/water-research/storm-water-management-model-swmm

We sincerely thank the USEPA and contributors for making these tools publicly available, enabling broader innovation, education, and practical application across the water sector.

1 ABOUT GISPIPE

GISpipe is an integrated GIS-Based Water and Stormwater/Sewer Network Analysis Software program. **GISpipe** is an advanced, user-friendly software solution that seamlessly integrates GIS, EPANET, and SWMM, making it ideal for the design, analysis, and operation of water supply and stormwater/sewerage systems. Developed based on Geographic Information Systems (GIS), the software includes key components such as the water distribution network modelling tool EPANET, the stormwater modelling tool SWMM, and surface runoff models.

JinboSoft's mission is, based on the extensive experience in information technology and geospatial systems, to deliver world-class tools for water distribution modelling and analysis. The goal is to develop industry-leading software that empowers engineers, utilities, and planners to design and manage efficient, reliable, and sustainable water distribution networks.



Some of the key features of **GISpipe** include:

1. Integrated GIS + hydraulic modelling

GISpipe combines Geographic Information Systems (GIS) with the proven analytical engines of EPANET (for water distribution) and SWMM (for sewerage and stormwater) into a single platform.

- No need for third-party GIS software such as AutoCAD or ArcView
- Native spatial data integration streamlines your workflow
- Map-based modelling interface simplifies network visualization and updates

2. Intuitive graphical user interface (GUI)

GISpipe is designed for engineers and planners, not just programmers.

- Drag-and-drop design with real-time map feedback
- Simple toolbars and logical workflows reduce the learning curve
- Visual error-checking and result interpretation directly on the map

3. Efficient data management and editing tools

Save time with smart data management:

- Built-in tools for data import/export, cleaning, and validation
- Features like undo/redo, batch editing, pipe splitting/merging, and node adjustment
- Seamless integration of attribute data (e.g., demand, elevation, pipe material)

4. Fast and accurate simulation

Leverage powerful hydraulic and hydrologic simulations:

- High-speed performance even for large networks
- Accurate hydraulic calculations using industry-standard algorithms
- Visual results display: pressures, flows, velocities, headlosses, water age, and more

5. One-stop solution

GISpipe eliminates the need for multiple tools:

- Design, analyse, and optimize water and stormwater/sewer systems in a single environment
- Easily switch between EPANET and SWMM workflows depending on your project
- Ideal for municipalities, consultants, and utility operators

6. Project planning and lifecycle management

Plan beyond hydraulics:

- GISpipe helps assess design alternatives, compare scenarios, and support asset management
- Export results and reports for documentation and decision-making
- Useful for both greenfield designs and rehabilitation projects

---

GISpipe is built with a user-centric design philosophy. Its modern and intuitive graphical user interface (GUI) minimizes the learning curve, allowing users to begin working productively with minimal training. Tasks such as network layout, data entry, and model execution are presented in a logical and accessible manner, significantly reducing project setup time. Whether you're a seasoned engineer or a first-time user, the software ensures a smooth and guided experience throughout.

GISpipe is optimized to handle large and complex datasets with speed and precision. By integrating GIS data directly into the modelling workflow, users can quickly validate existing infrastructure layouts, perform hydraulic and hydrologic analyses, and calibrate models using real-world data. The platform supports batch operations and high-speed simulations, making it possible to complete detailed pipe network plans and scenario analyses faster than ever, while maintaining the highest level of accuracy.

Engineers often require the ability to adjust and refine models quickly, and **GISpipe** delivers with a suite of advanced editing tools. Features such as undo/redo, copy and paste, pipe splitting, and point merging enable users to manipulate network geometry effortlessly. The software also includes live feedback on changes, allowing users to immediately visualize the impact of edits. After running a simulation, results such as pressure, flow, and surcharge levels can be displayed directly on the map, making it easy to assess system performance and identify issues.

GISpipe eliminates the need for external CAD or GIS software like AutoCAD, ArcView, or third-party plugins. All essential functions, from spatial data handling and network modelling to result visualization, are fully integrated into a single environment. This not only simplifies the software ecosystem but also ensures compatibility and seamless performance across the entire workflow. With **GISpipe**, users can manage the entire lifecycle of a water or stormwater/sewer system model, plan, design, analyse, and report, without leaving the application.

1.1 Relationship with EPANET

GISpipe uses EPANET 2.2 as its underlying computational engine. All hydraulic and water quality simulations are executed through the EPANET framework, ensuring compatibility with industry-standard modelling approaches.

While EPANET uses a schematic (non-georeferenced) map, **GISpipe** incorporates true spatial referencing (e.g., UTM zones, EPSG codes), allowing the user to model real-world pipe networks over topographic or cadastral base maps. GISpipe also extends EPANET's basic interface by offering:

- Enhanced data table interfaces for rapid inspection and editing
- GIS-based topology management and attribute-driven controls
- Integrated tools for batch editing, validation, and visual filtering

Users familiar with EPANET will find **GISpipe** intuitive, with the added benefit of real-world spatial context and streamlined workflows.

1.2 Relationship with EPAWMM

GISpipe integrates EPA SWMM 5.2 as its computational engine for stormwater and surface runoff simulations. All hydrologic and hydraulic routing calculations, including runoff generation, infiltration, and flow through stormwater networks, are processed using SWMM's proven modelling framework.

Whereas SWMM traditionally relies on a schematic interface, **GISpipe** provides a fully georeferenced modelling environment, enabling the representation of stormwater elements (e.g., subcatchments, junctions, conduits, storage units) over high-resolution GIS maps and layers.

GISpipe enhances the standard SWMM interface through:

- Spatially-linked subcatchment delineation tools and automatic network connectivity.
- Rainfall and land use data integration with batch import/export from standard formats.
- Embedded time-series and climatology assignment per node or subcatchment.
- Thematic mapping of simulation results (e.g., runoff, ponding, depth profiles) using intuitive colour legends.

Users experienced with SWMM will find the transition to **GISpipe** seamless, while benefiting from an enriched spatial interface and improved workflow for scenario creation, validation, and reporting.

GISpipe

2 SYSTEM REQUIREMENTS

System requirements for installing and running **GISpipe**.

Operating system

- Windows - Windows 10 or later (64-bit recommended)
- Linux - Ubuntu 20.04 or compatible distribution (for advanced users)
- macOS - Not natively supported (may work using emulation/virtualisation, but not officially recommended)

Hardware requirements

Component	Minimum requirement	Recommended requirement
Processor	Dual-core 2.0 GHz	Quad-core 3.0 GHz or higher
RAM	4 GB	8–16 GB
Storage	500 MB for software + project files	SSD with 2 GB free space
Graphics	Integrated graphics supported	Dedicated GPU for large datasets
Display	1280×768 resolution	1920×1080 or higher

Software dependencies and additional requirements

- .NET Framework 4.8 or later (for Windows users)
- QGIS 3.x (optional, for visualisation and export)
- Administrator rights for installation
- Internet connection for software activation, updates, and cloud integration features (if used)

Licensing and activation

- Requires a valid license key from the developer or distributor.
- Offline activation options may be available upon request.

3 INSTALLING THE PROGRAM

Step 1: Obtain the installation package

- Download the latest version of the **GISpipe** installer from the official website or from your software provider.
- Ensure the downloaded .exe is from a trusted source.

Step 2: System preparation

- Verify that the system meets the minimum requirements (see previous paragraph).
- Ensure to have:
  - Administrative rights to install software.
  - .NET Framework 4.8 or later installed (usually comes with Windows 10+).

Step 3: Run the installer

- Double-click the GISpipe_Setup.exe file.
- Follow the on-screen instructions:
  - Accept the license agreement.
  - Choose the installation directory (default is usually C:\Program Files\GISpipe).
  - Select any optional components if prompted (e.g., database integration, examples).

Step 4: Activate the software

- After installation, launch **GISpipe**.
- Under the *Help* menu click on *Request for use*
  - Enter your email address
  - Enter your company name
  - Enter your phone number
  - Click on the *Okay* button
- The request will be sent to activate the software once confirmation of payment has been received.

Step 5: Verify installation

- Open a test project or import sample pipeline shapefiles (downloadable from the website).
- Ensure tools and features load without error.
- Confirm output generation (e.g., reports, network layout, hydraulic calculations).

4 PURCHASING THE SOFTWARE

The software will function in demo version but once the Software protection dongle is detected by the software it changes to a full working program. The demo version is limited i.e., not allowing saving, analysing and some advance functions.

Full license

- One-time single license activation fee
- Unlocks all core features and modules:
    - Full GIS-based pipe network design tools
    - EPANET and SWMM integration for hydraulic and hydrological modelling
    - Advanced editing and analysis tools
    - Full project save/export capabilities
- Includes 12 months of:
    - Software updates
    - Email-based technical support

Annual subscription fee
Paid from year 2 onward ensures:

- Continued access to updates and improvements
- Ongoing technical support
- Access to new features and user-requested enhancements

If you would like to purchase the **GISpipe** software please contact us. Please visit **www.gispipe.co.za** for more information and to download the order form or send the developers an e-mail, **sales@gispipe.co.za**.

5 DISCLAIMER

The software program was developed for the convenience of its users. The GISpipe software is provided "as is" without any expressed or implied warranties, including but not limited to warranties of merchantability, fitness for a particular purpose or non-infringement.

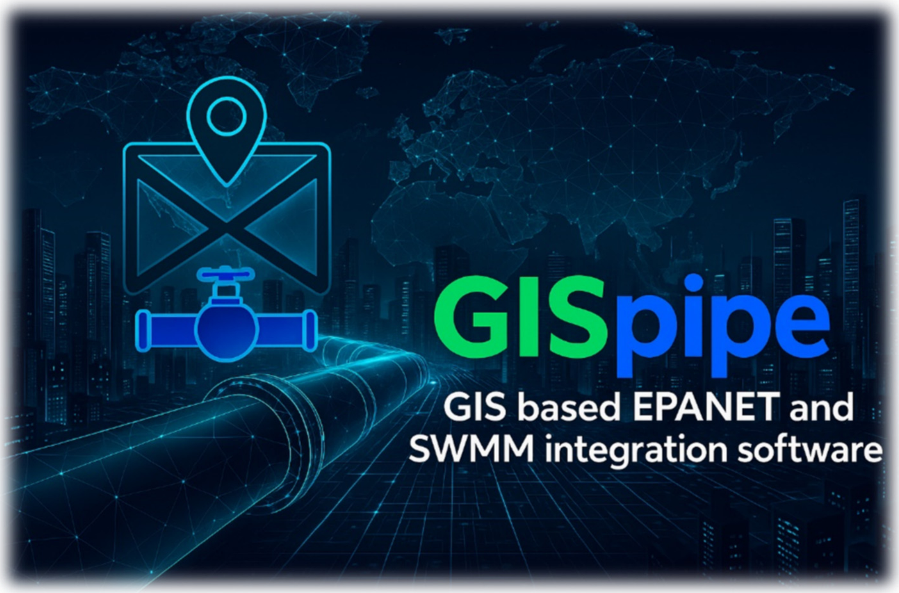
Neither JinboSoft nor Green Hippo Innovations (Pty) Ltd shall be held liable for any direct, indirect, incidental, consequential, or special damages arising out of or in connection with the installation, use, performance, results obtained or interpretation thereof from the software. This includes, but is not limited to, data loss, operational disruptions, incorrect analysis outcomes, or infrastructure planning decisions based on the software outputs.

Users are solely responsible for verifying the accuracy, adequacy, and applicability of the results for their specific needs. Any reliance placed on the software outputs is strictly at the user’s own risk.

By installing or using the software, the user agrees to these terms and acknowledges that they have read and understood this disclaimer.

6 GETTING STARTED

Welcome to **GISpipe** – a powerful software tool designed for the hydraulic analysis and management of pipe networks using geographic information system (GIS) integration. This section will guide the user through the initial setup and provide the foundational steps needed to begin using the program effectively.



GISpipe can be run by double clicking on the program icon in the Windows Program Manager.

The main screen will be shown as depicted in **Figure 6-1** below.

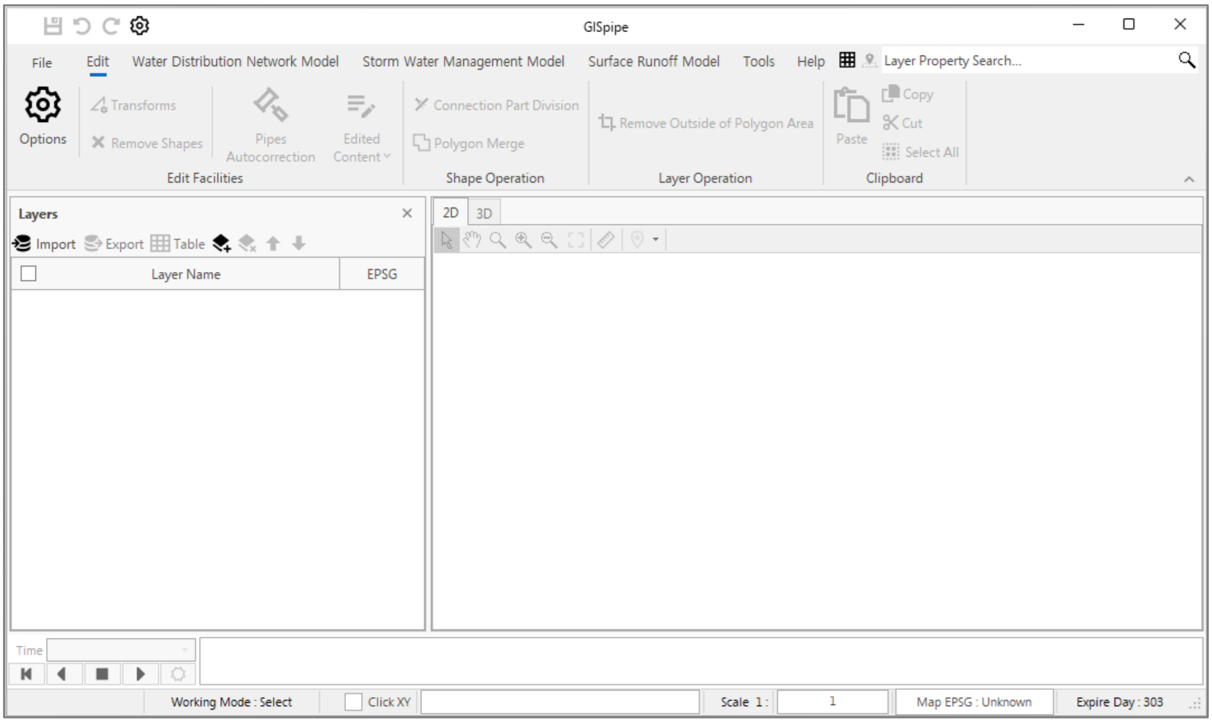


Figure 6-1: GISpipe main screen

7 GISPIPE'S WORKSPACE

This section discusses the essential features of GISpipe's workspace. It describes the various areas of the program (Main Menu Area, the Layer Control Panel, Map Viewer, Time Control Area and Status Bar). It also shows how to set program options.

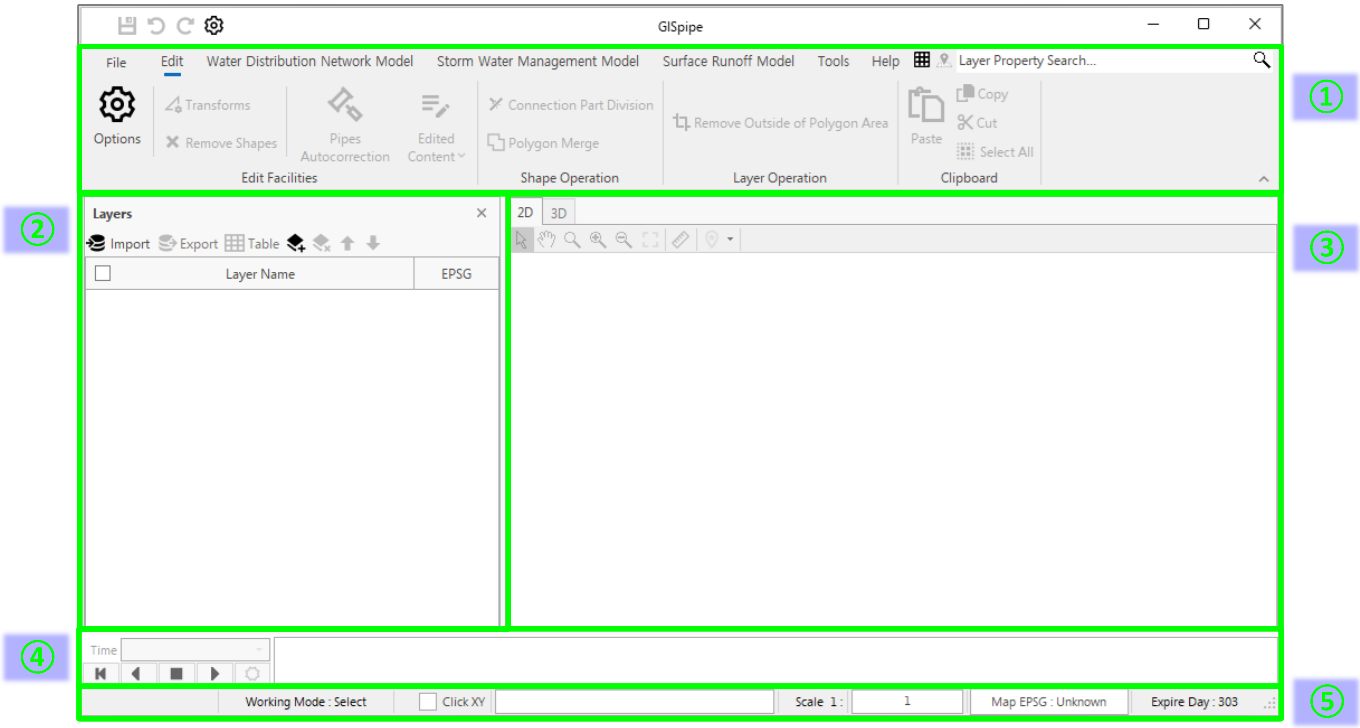


Figure 7-1: Main screen layout and features

① Main Menu Area

It consists of the menu bar and toolbar for management of the software.

- Menu Bar (Top-most row)
  Contains standard menus: *File, Edit, Tools, Help*
  Plus, domain-specific modules: *Water Distribution Network Model, Storm Water Management Model* and *Surface Runoff Model*
- Toolbar (Below Menu Bar)

This area of the Main Menu Area provides quick-access icons for key operations depending on the choice of the selected menu.

② Layer Control Panel (Left Pane)

This is an area that controls the layers that performs functions such as switching layers on and off or importing or exporting layers.

- Import / Export / Table to assist in managing GIS layers
- Layer List displays all loaded spatial layers with their EPSG coordinate system.
- Users can toggle visibility, manage attributes, or apply symbology here.

③ Map Viewer (Right Pane)

2D / 3D tabs allowing user to toggle between 2D and 3D spatial views.

- Navigation tools such as Pan, zoom, select, and edit features.
- Displays the visual representation of the imported GIS data.

④ Time Control Area (Bottom-Left)

Used for simulations involving time (Extended Period Simulations) (e.g., events, animations). Controls include play, pause, stop, and time step navigation.

⑤ Status Bar (Bottom-Right)

This area shows the current coordinates, scale, expiration date

- Working Mode (e.g., Select, Edit)
- Click XY (cursor coordinate tracking)
- Scale and Map EPSG (projection and zoom control)

This provides real-time feedback on spatial positioning and tool status.

File menu

The *File* menu contains commands for opening and saving data files and for setting the preferred language. The commands for the *File* menu are shown in **Figure 7-2** below. It also shows a list of recently worked on projects.

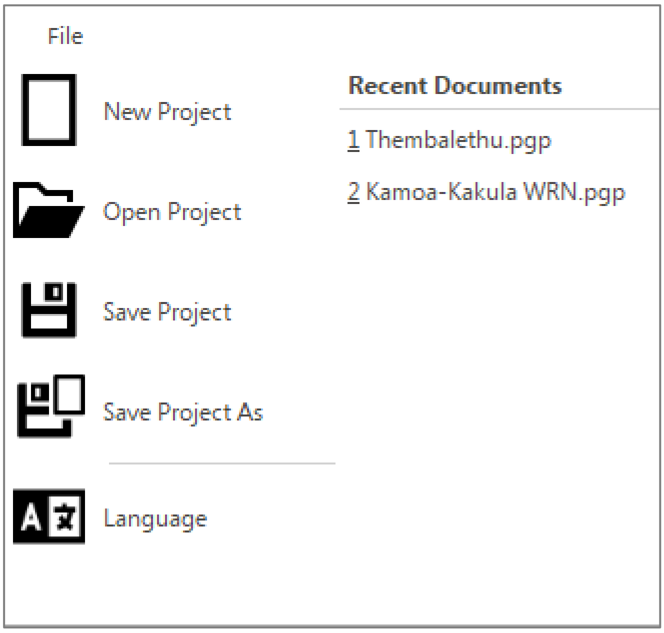


Figure 7-2: File menu

- New Project - Creates a new GISpipe project (file extension *.pgp)
- Open Project - Opens an existing project
- Save Project - Saves the current project
- Save Project As - Saves the current project under a different name
- Language - Set the language to preferred language (English or Korean)

Edit menu

Selecting the *Edit* menu will show the relevant toolbar which depends on what type of layer the user is working on i.e., the screen GUI changes according to the selected layer.

When the selection is on a layer (other than Water Distribution Network Model, Storm Water Management Model or Surface Runoff Model) the toolbar options that will be available are as shown in **Figure 7-3**.

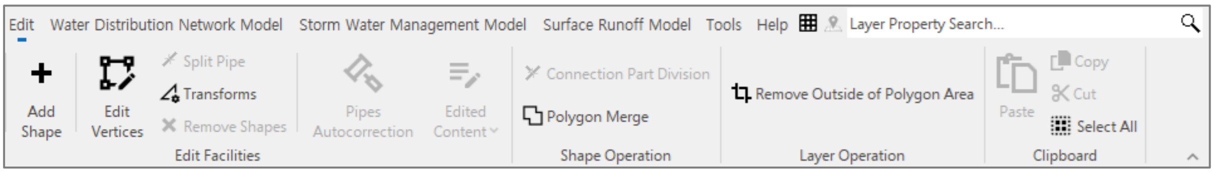


Figure 7-3: Edit toolbar options

- Add Shape - Simply select the layer on which a shape wants to be placed, press the *Add Shape* button, and add the shape to the map. For a line or polygon, click right mouse button to finish. Pressing the ESC key while adding, the drawing is cancelled.
- Edit Vertices - Click the *Edit Vertices* button and select a shape to edit on the map. It will change to a dotted line and the shape can be changed by moving the green dot.
  Clicking the green dot removes the clicked vertex and clicking between the solid lines adds the vertex.
- Split Pipe - A function to divide a pipe.
- Transforms - Moving of axis or scales.
- Remove Shapes - This function removes selected shapes.

Water Distribution Network Model

When working on the Water Distribution Network Model selecting *Edit* menu will show the editing functions associated with the water distribution model as shown in **Figure 7-4**.

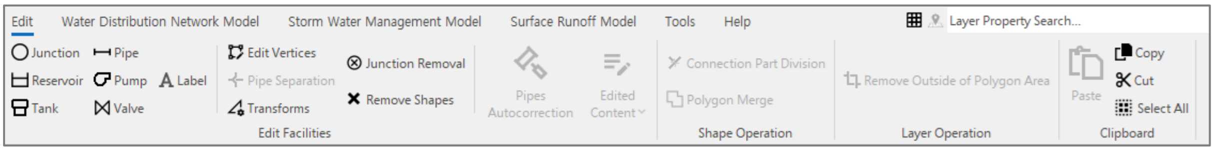


Figure 7-4: Water Distribution Network Model edit functions

These functions are described in more detail in Section 9.2.

Storm Water Management Model

When working on the Storm Water Management Model selecting Edit menu will show the editing functions associated with the storm water management model as shown in **Figure 7-5**.

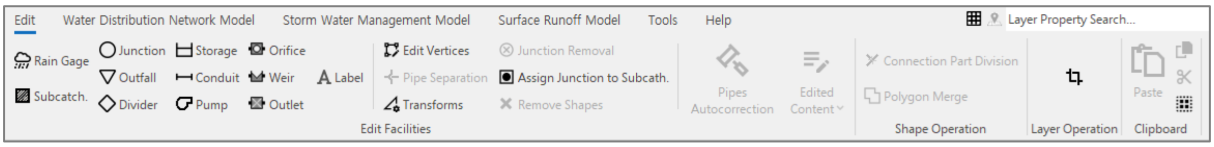


Figure 7-5: Storm Water Management Model edit functions

These functions are described in more detail in Section 10.4.

Surface Runoff Model

When working on the Surface Runoff Model selecting *Edit* menu will show the editing functions associated with the surface runoff model as shown in **Figure 7-6**.

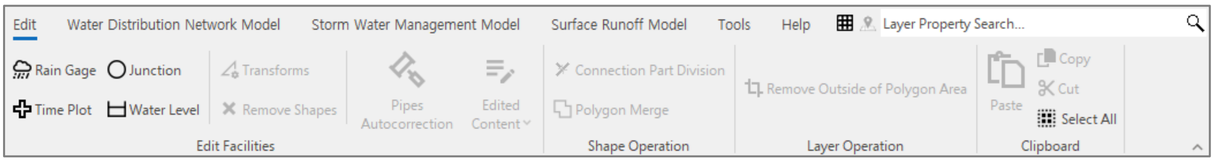


Figure 7-6: Surface Runoff Model edit functions

These functions are described in more detail in Section 11.

Water Distribution Network Model menu

When working on Water Distribution Network Models selecting *Water Distribution Network Model* Menu will show the analysis and viewing functions associated with the water distribution model as shown in **Figure 7-4**.

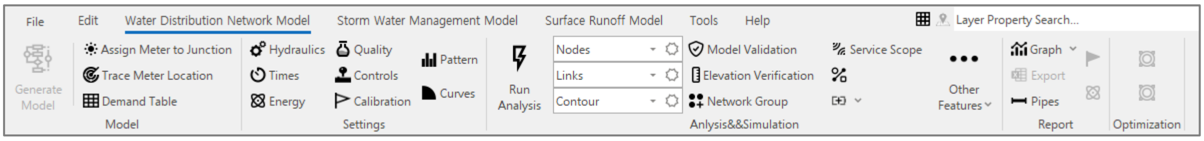


Figure 7-7: Tool bar options when selecting the Water Distribution Network Model Menu

These analyses and viewing functions are described in more detail in Section 9.

Storm Water Management Model menu

When working on Storm Water Management Models selecting *Storm Water Management Model* Menu will show the analysis and viewing functions associated with the water distribution model as shown in **Figure 7-8**.

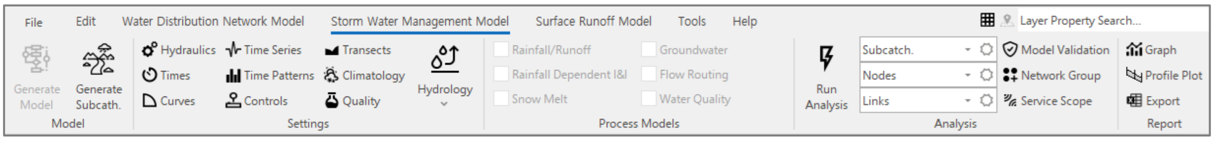


Figure 7-8: Tool bar options when selecting the Storm Water Management Model Menu

These analyses and viewing functions are described in more detail in Sections 10.6 and 10.6.

Surface Runoff Model menu

When working on Surface Runoff Models selecting *Surface Runoff Model* Menu will show the analysis and viewing functions associated with the surface runoff model as shown in **Figure 7-8**.

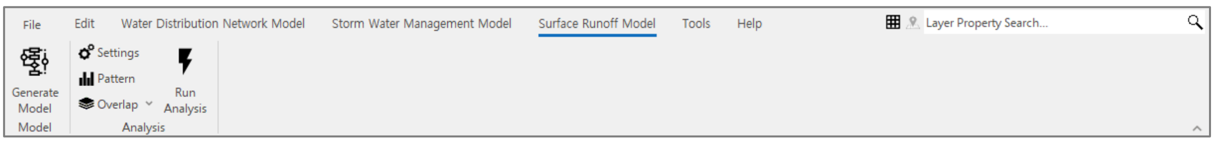


Figure 7-9: Tool bar options when selecting the Surface Runoff Model Menu

These analyses and viewing functions are described in more detail in Section 11.

Tools menu

Selecting the *Tools* menu will provide the user with specific tools related to the DEM data, Shapefiles and generating of basins or streams as shown in **Figure 7-10**.

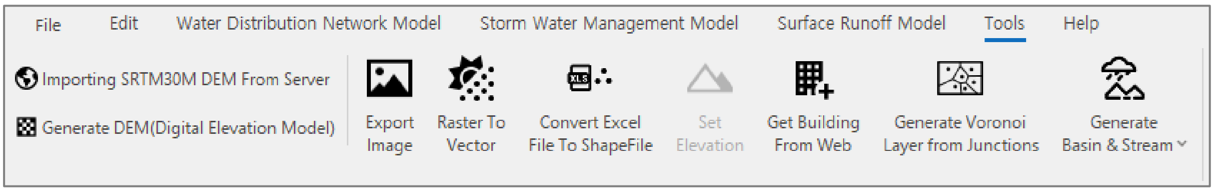


Figure 7-10: Tool bar options when selecting the Tools Menu

Help menu

Selecting the *Help* menu will provide the user with toolbar options related to the use of the software as well as the licensing thereof as shown in **Figure 7-11**.

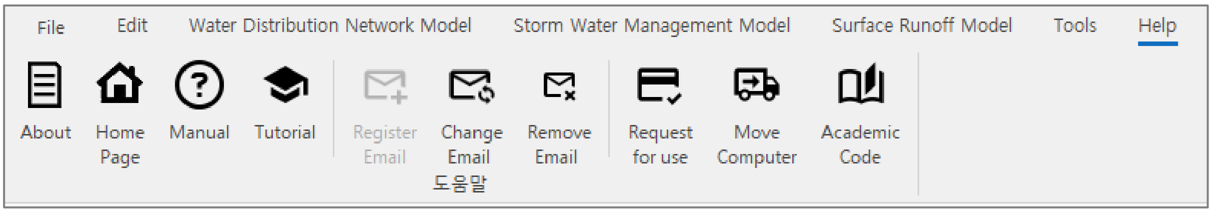


Figure 7-11: Help menu tool bar options

Clicking the *Options* toolbar button, provides the user with a set of preference that can be set as shown in **Figure 7-12**. The options relate to what the user preference is for scrolling with the mouse, the setting of Auto length on or off, cursor, map background colour and keyboard short cut keys.

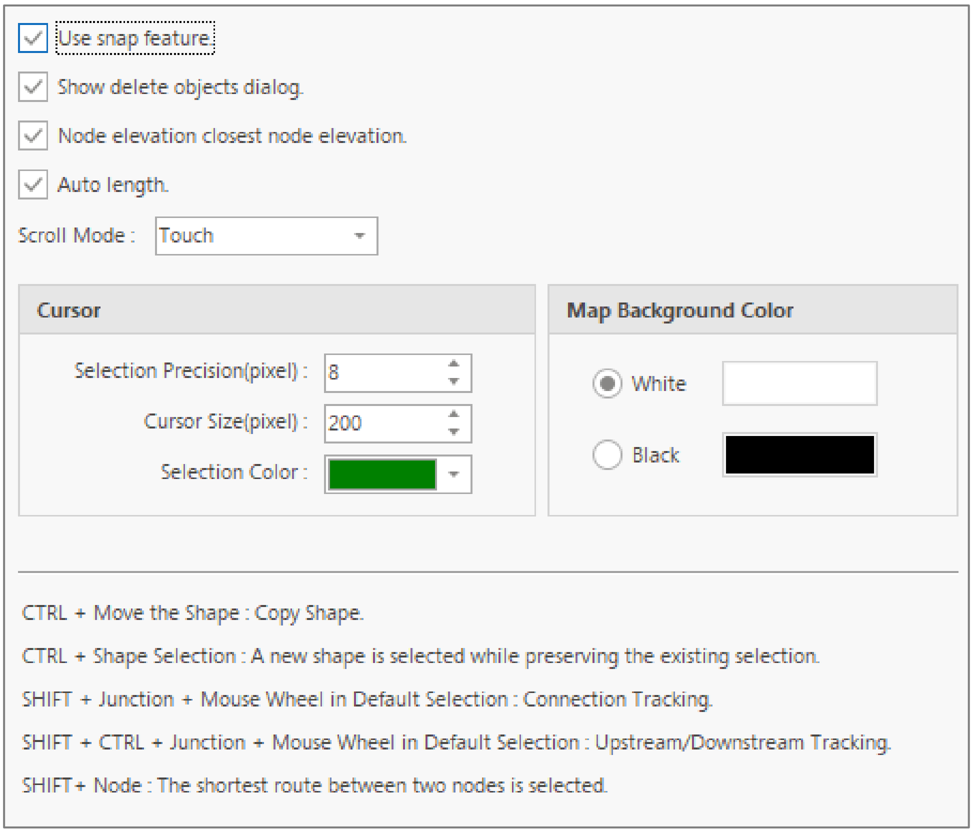


Figure 7-12: Options screen

8 GRAPHICAL INFORMATION SYSTEM

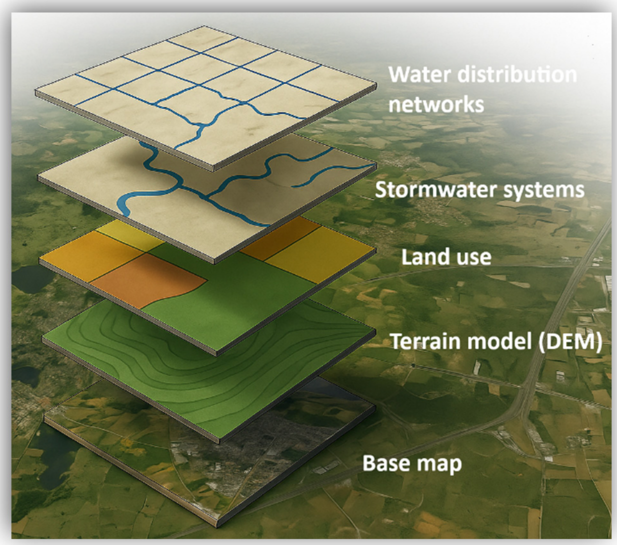
8.1 What is a Geographic Information System (GIS)?

A Geographic Information System (GIS) is a tool that allows users to collect, manage, analyse, and display spatial or geographic data. In the context of GISpipe, GIS is fundamental for creating and working with water distribution networks and stormwater systems that are geographically referenced. It links data to a specific location, whether a pipe junction, reservoir, valve, catch basin, or terrain feature, allowing users to visualize and analyse how infrastructure interacts with the environment.

In water-related modelling, most spatial data, such as pipe routes, ground elevation, land use, rainfall zones, and service boundaries, have a geographic component. GISpipe uses this data to enable tasks such as locating leaking pipelines, assessing stormwater flow paths, sizing detention basins, or planning upgrades to distribution systems. For example, a GIS layer showing high-resolution elevation data can be combined with vector-based storm drain networks to identify flow accumulation zones or undersized culverts.

GIS technology allows all these different types of information, raster data like digital elevation models (DEMs), and vector data like pipelines and land parcels, to be overlaid on top of one another on a single map. By treating location as the key index variable, GISpipe enables users to perform advanced spatial analysis, such as calculating pressure zones, estimating runoff contributions from impervious surfaces, or assessing the hydraulic connectivity between subsystems.

GIS goes far beyond just mapping. It serves as a system of record, integrates data from diverse sources, enables rich spatial analysis, and facilitates data communication through intuitive visualizations such as thematic maps and dashboards.



These capabilities make GIS an indispensable tool in hydrology, environmental engineering, stormwater management, and municipal water services.

8.2 Coordinate Systems

Accurate spatial representation in GISpipe depends on the correct use of coordinate systems. A coordinate system provides a framework for defining locations on the Earth's surface and is fundamental to integrating and analysing spatial data.

At a broad level, there are two main types of coordinate systems used in GIS:

- **Geographic Coordinate Systems (GCS)** - These define locations using a spherical model of the Earth, typically in degrees of latitude and longitude.
- **Projected Coordinate Systems (PCS) -** These "flatten" the Earth's surface onto a two-dimensional plane, using mathematical transformations to express positions in linear units such as meters. This is essential for engineering design and spatial analysis.

GISpipe allows the user to select and specify the coordinate system of any layer (**Figure 8-1**).

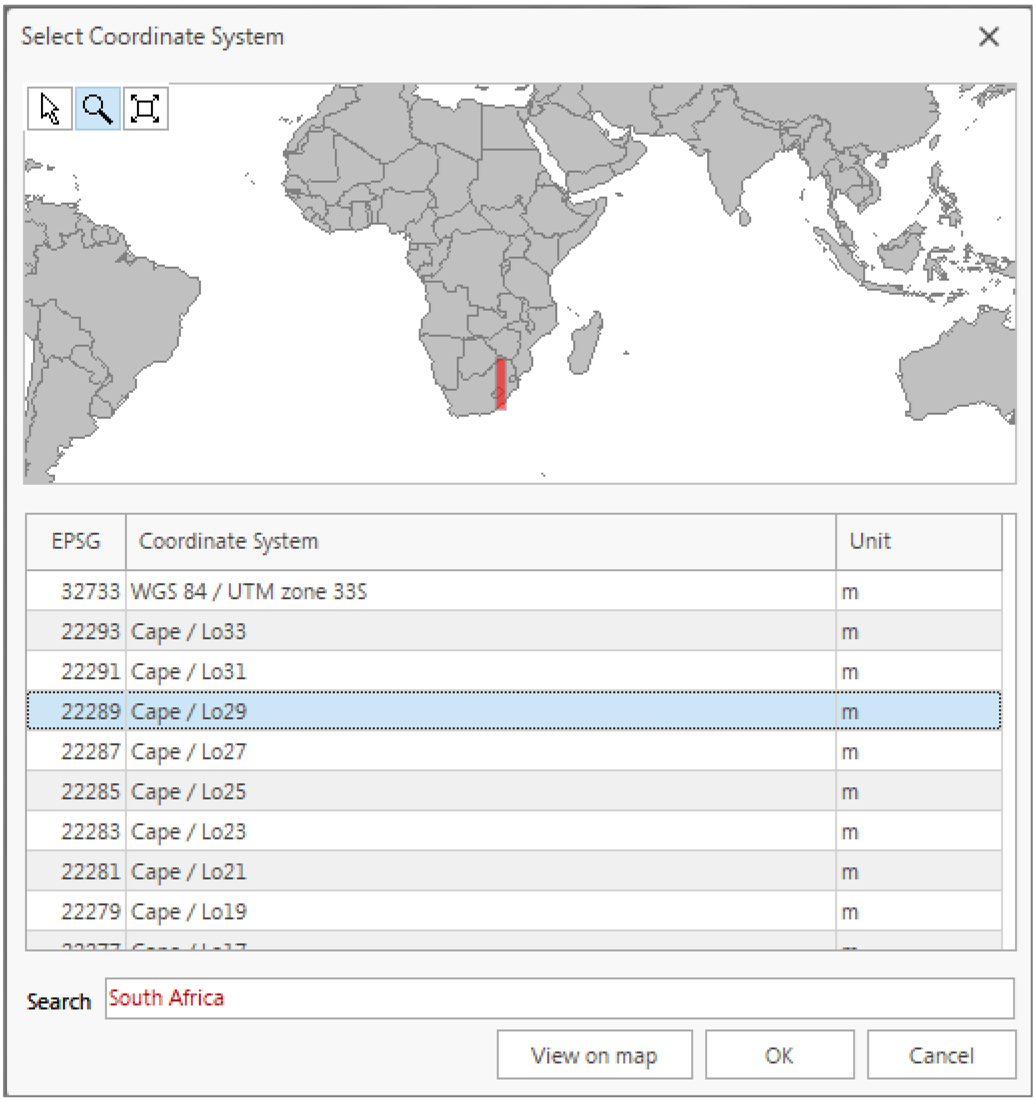


Figure 8-1: Coordinate System

8.3 Understanding Map Projections

To represent the Earth's curved surface on a flat map or screen, a mathematical transformation called a map projection is required. All map projections introduce some distortion, whether of area, distance, shape, or direction. Selecting the correct projection is critical in GISpipe to ensure spatial accuracy, especially for tasks like calculating pipe lengths, analysing flow directions, or determining catchment boundaries.

Some projections, like the Mercator, are useful for navigation because they preserve direction, but they distort area, particularly near the poles. Others, like equal-area or equidistant projections, preserve area or distances along specific lines but may distort shape. For localized hydraulic or engineering modelling, Transverse Mercator projections are often preferred because they minimize distortion over small regions.
Selecting an appropriate projection is critical for ensuring spatial accuracy, especially in hydraulic modelling tasks such as pipe length calculations, catchment analysis, or flow direction assessment.

For example:

- The Mercator projection preserves direction but significantly distorts area near the poles.
- Equal-area projections maintain true area relationships, making them suitable for land use analysis.
- Transverse Mercator projections minimize distortion in small regions and are commonly used for engineering and cadastral mapping.

GISpipe supports projection-aware workflows and provides tools for assigning and transforming coordinate systems of spatial layers. This is particularly important when integrating data from multiple sources or conducting precise hydraulic simulations. For example, a terrain dataset in a projected CRS can be aligned with a pipeline network in another CRS using built-in transformation tools.

In South Africa, the modern standard is the Hartebeesthoek94 (HB94) coordinate system, based on the WGS84 ellipsoid. It is divided into 2-degree longitude zones with corresponding EPSG codes (e.g., EPSG:2053 for Lo29). Accurate CRS selection ensures that all components of a model, e.g., pipe alignments, terrain elevations, and rainfall catchment, fit together seamlessly.

When combining spatial data from multiple sources, such as survey drawings, satellite imagery, and scanned topographic maps, differences in their underlying coordinate systems and projections can cause misalignment. GIS software addresses this by reprojecting layers to a common coordinate system, ensuring spatial consistency.

This process is essential for water distribution modelling in GISpipe, where overlaying layers such as elevation (from DEMs), pipe networks (vector), and land use zoning must align precisely for accurate analysis.

8.4 Spatial Data Formats, Capturing and Scales

GIS data is generally stored in two primary formats:

- **Raster** - Gridded data made up of cells or pixels. Commonly used for continuous data like elevation, land cover, or temperature.
- **Vector** - Uses discrete geometries, points, lines, and polygons, to represent features like junctions, pipelines, tanks, and pressure zones.

Regardless of format, spatial datasets must be georeferenced—linked to a coordinate system—to ensure spatial consistency and enable meaningful analysis.

Incorporating data into GIS involves data capture. Native digital datasets (e.g., GPS, shapefiles, CAD drawings) can be directly imported. Analogue sources, such as paper maps or scanned plans, require digitising and georeferencing before use. Proper scale management is also crucial. Scale refers to the ratio between a map's distance and the actual ground distance. In engineering workflows, maintaining consistent scale ensures reliable measurements, particularly when assessing distances, elevations, or hydraulic gradients.

GISpipe facilitates consistent data integration through coordinate system management, scale control, and projection handling, enabling engineers to build robust, accurate hydraulic models and carry out effective spatial analysis for water infrastructure planning and design.

9 WATER DISTRIBUTION NETWORK MODEL

A typical GISpipe modelling workflow for a water distribution network model includes the following steps:

① Create or import base layers (pipes, junctions, reservoirs, tanks)

This step involves setting up the foundational components of the network either manually or by importing from external GIS, CAD, or database files. Accurate spatial and structural representation at this stage ensures that the network topology is valid and complete.

② Assign attributes such as elevation, diameter, demand, and status

Each network element must be defined with hydraulic and operational parameters to reflect real-world conditions. This includes node elevations, pipe diameters, expected demand at junctions, and operational status (open/closed) of valves and pumps.

③ Edit and validate the network using map tools and tables

Visual editing allows users to reposition elements and correct connectivity or attribute errors using a map-based interface. Validation tools then check for data inconsistencies such as unconnected nodes, zero-length pipes, or missing attributes.

④ Set simulation parameters, including time patterns and controls

Simulation settings such as duration, hydraulic time steps, and water quality parameters are configured in this step. Control rules and time patterns (e.g., diurnal demand curves or pump schedules) are also assigned to reflect temporal variations.

⑤ Run the simulation (hydraulic or water quality using the EPANET engine)

The hydraulic and/or water quality simulation is executed using EPANET's solver. It computes flow rates, pressures, tank levels, and concentrations across the network over time using iterative numerical techniques like the Gradient Algorithm

⑥ View results graphically (e.g., contours, graphs, tables)

Post-processing tools enable the analysis of results using colour-coded contours, interactive graphs, and summary tables. These visualizations help identify problem areas such as low-pressure zones or high-water age regions.

⑦ Export simulation output or INP files for documentation or further analysis

The results and model configuration can be saved or exported in standard formats (e.g., *.INP, *.PGP, *.CSV) for reporting or sharing with other software programs. This ensures reproducibility, and supports collaborative workflows.

9.1 Hydraulic and quality analysis model

9.1.1 Hydraulic Simulation Model

GISpipe utilises EPANET's hydraulic simulation model which computes junction heads and link flows for a fixed set of reservoir levels, tank levels, and water demands over a succession of points in time. From one time step to the next reservoir levels and junction demands are updated according to their prescribed time patterns while tank levels are updated using the current flow solution. The solution for heads and flows at a particular point in time involves solving simultaneously the conservation of flow equation for each junction and the headloss relationship across each link in the network.

This process, known as "hydraulically balancing" the network, requires using an iterative technique to solve the nonlinear equations involved. EPANET employs the "Gradient Algorithm" for this purpose. The hydraulic time step used for extended period simulation (EPS) can be set by the user. A typical value is 1 hour. Shorter time steps than normal will occur automatically whenever one of the following events occurs:

- The next output reporting time period occurs
- The next time pattern period occurs
- A tank becomes empty or full
- A simple control or rule-based control is activated

9.1.2 Water Quality Simulation Model

Basic transport

GISpipe utilises the water quality engine of EPANET which uses a Lagrangian time-based method to simulate how discrete parcels of water move and mix through the pipe network. Each pipe is filled with a series of non-overlapping segments whose quality is updated at every water quality time step. These steps are shorter than hydraulic steps to capture dynamic behaviour, and involve applying reactions, updating mass balances at nodes, and creating new segments when inflow concentrations change significantly. Flow reversals are handled by reordering segments, and water quality at nodes is computed based on cumulative inflow from all contributing segments.

Mixing in storage tanks

GISpipe utilises the water quality engine of EPANET which provides four tank mixing models:

- Complete Mixing – Assumes instant and uniform mixing throughout the tank.
- Two-Compartment Mixing – Splits the tank into two completely mixed zones to simulate short-circuiting and dead zones.
- FIFO Plug Flow – Assumes water flows in and out in the same order, like a pipeline; no mixing occurs.
- LIFO Plug Flow – Water enters and exits from the bottom in reverse order; useful for standpipes.

These models allow flexibility to represent various physical tank behaviours more accurately.

Water quality reactions

Reactions can occur in the bulk flow or along the pipe wall:

- Bulk Reactions follow n^{th} order kinetics and can include growth, decay, or reactions with limiting concentrations. GISpipe allows users to input the rate coefficient, reaction order, and limiting concentration as needed.

- Wall Reactions depend on the concentration in the bulk and the surface-to-volume ratio of the pipe. Reaction rates can be zero- or first-order and are influenced by pipe roughness, material, and age. The wall reaction coefficients can be related to the pipe's head loss characteristics using empirical relationships.

Water age and source tracing

Water age is modelled as a zero-order growth process, where each second adds one second to the age of water, providing a general indicator of water freshness. For source tracing, the model tracks the proportion of water at each node originating from a designated source, treated as a 100% concentration of a non-reactive constituent. This helps evaluate blending patterns and source contributions throughout the network over time.

9.2 Building a Water Network Model

This section outlines the step-by-step process for constructing a hydraulic water distribution network within GISpipe, from establishing the base layout to configuring simulation parameters for accurate system analysis.

9.2.1 Working with Layers

GISpipe offers robust layer management tools to support hydraulic, stormwater, and runoff modelling. This section explains how to work with layers, including importing data, assigning coordinate systems, managing geometry, and accessing layer-specific properties.

Importing Layers and Data

When starting a new project, users can either:

- Digitize features manually using map-based drawing tools, or
- Import existing GIS datasets, such as:
    - Shapefiles (.shp)
    - GeoJSON (.geojson)
    - CSV files with coordinate columns
    - Raster files for DEMs (e.g., .tif)
    - INP files for EPANET or SWMM network import

      See **Appendix A** for a list of supported formats
- To import a shapefile, navigate to the Layer Control Panel (left pane) and click on the *Import* button, .
- Select and open the file and choose the geometry type (Waterworks, Sewer or Other), as shown in **Figure 9-1**.
- Assign it to a GISpipe object category (e.g., Junctions, Pipes, Sewer Lines, etc.).
- Define field mappings such as diameter, elevation, or roughness.

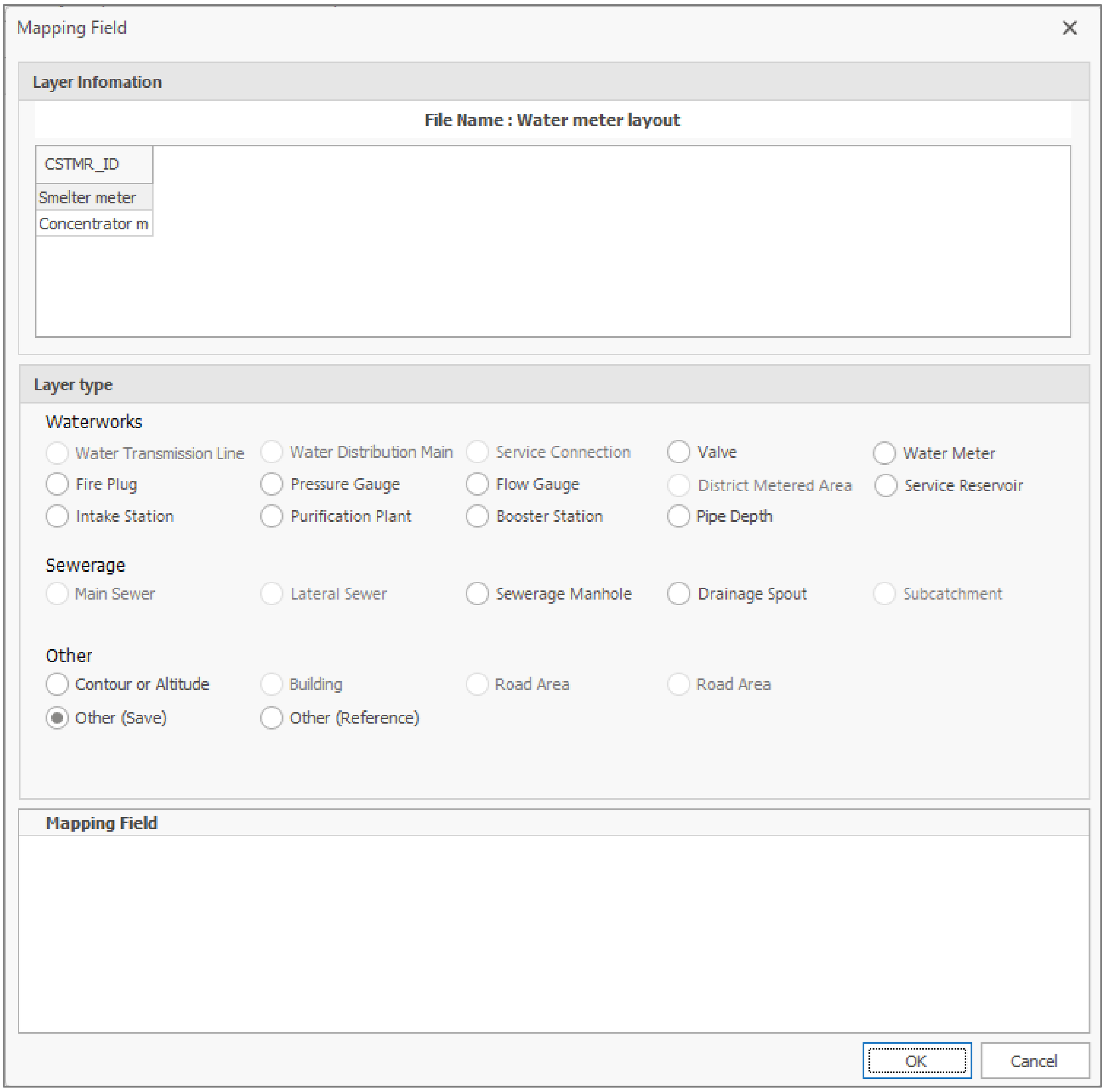


Figure 9-1: Mapping field type selection

Layer Types and Geometry

Each object type has a predefined geometry:

Table 9-1: Layer types and geometry

Layer	Geometry	Description
Junctions	Point	Model demand points and connection nodes
Reservoirs	Point	Represents fixed-head sources
Tanks	Point	Model storage with variable water level
Pipes	Line	Convey water between nodes
Pumps/Valves	Line	Represent special hydraulic links
Labels	Point	Optional annotation layer
Catchments	Polygon	Used in stormwater and runoff models
Streams/Basins	Line/Poly	For DEM and terrain-based analysis
SWMM Elements	Various	Sewer elements like manholes and laterals
Background Maps	Raster	Google Maps, Bing, OpenStreetMap

Layer Toolbar Functions

Icons at the top of the layer panel, as shown in **Figure 9-2**, provides various functionalities, as listed in **Table 9-2** to integrate different layers into the project.

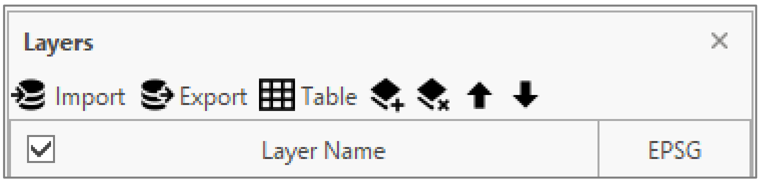


Figure 9-2: Layer toolbar

Table 9-2: Layer toolbar functions

Option	Description
Import	Add shapefiles, INP files, raster data and many other file formats. A list of the different files that can be imported is provided in **Appendix A**.
Export	Depending on the selected layer type, the layer can be exported to the following file formats. • Water network: EPAnet 2 INP, EPAnet 2 NET, EPAnet 3 INP, Shapefile (see **Figure 9-3**) and DXF • Stormwater/Sewer network: SWMM INP and Shapefile • Any other (polygon, line, point, subcatchment, DMA, Water meter etc.): Shapefile
Table	View and edit values in the attribute tables see **Figure 9-4**.
Add	Add a new layer. Various options will be provided as depicted in **Figure 9-5**.
Delete	Delete the selected layer from the map.
Move up	Move selected layer up in the display order.
Move down	Move selected layer down in the display order.
Visibility toggle	Enable or disable a layer's visibility.

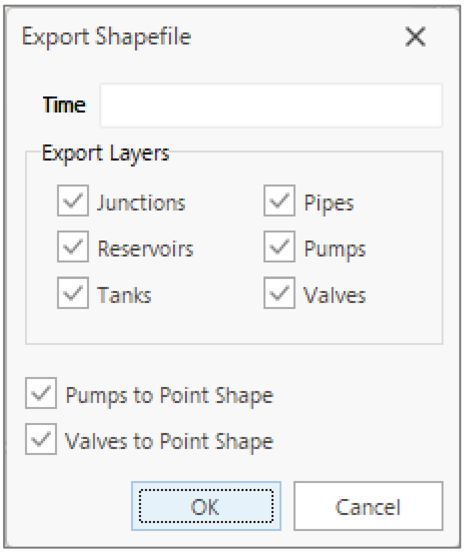


Figure 9-3: Exporting to shape file (example of water network)

As an example, the user has options of the information of the components of a water distribution network that should be included in the compiled shape file see **Figure 9-3**.

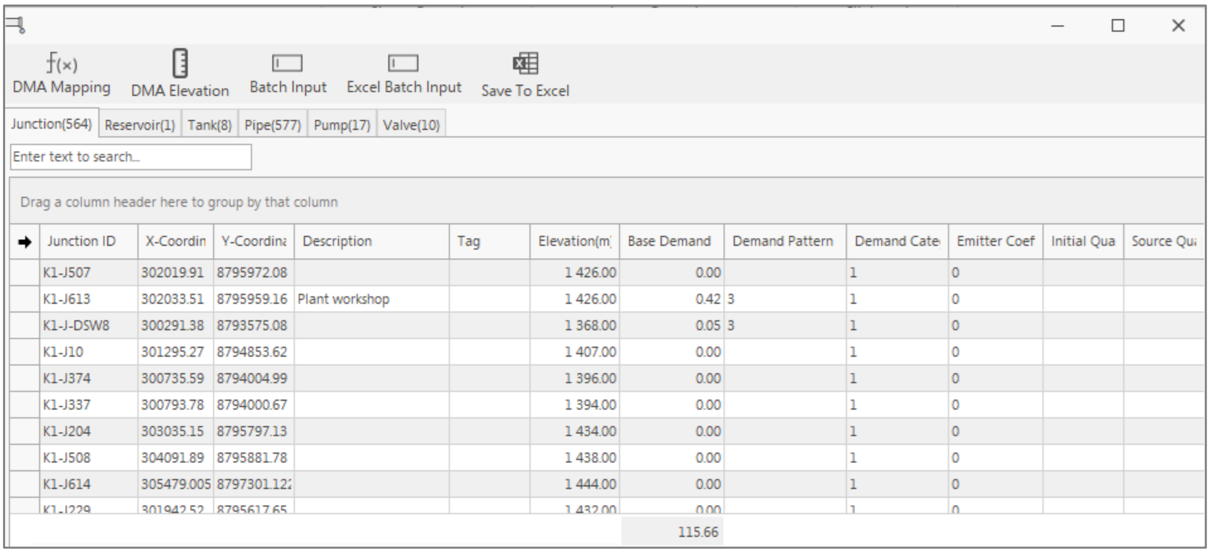


Figure 9-4: Attribute table

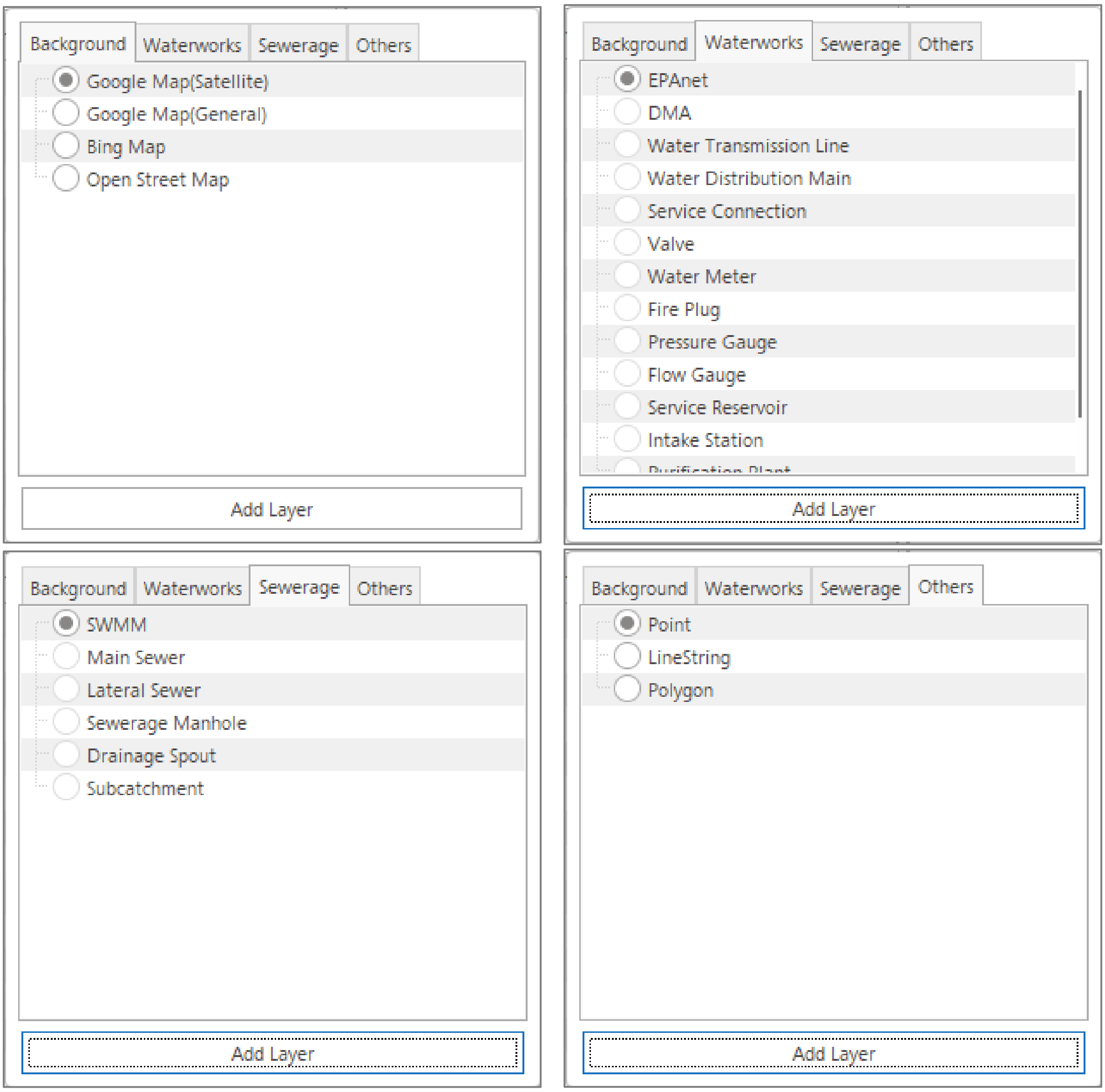


Figure 9-5: Adding of layers (Background, Waterworks, Sewerage, Others)

Attribute Tables and Data Extraction

Each vector layer can be opened in a tabular view where there are functionalities (**Figure 9-6**), where users can:

- View, filter, and group data
- Edit values (e.g., elevation, demand, roughness)
- Export to Excel or CSV
- Perform batch input or batch editing
- Use tools such as DMA Mapping, Valve ID assignment, or Contour tagging.

The attribute table can be viewed by clicking on the *Table* button on the Layer toolbar.

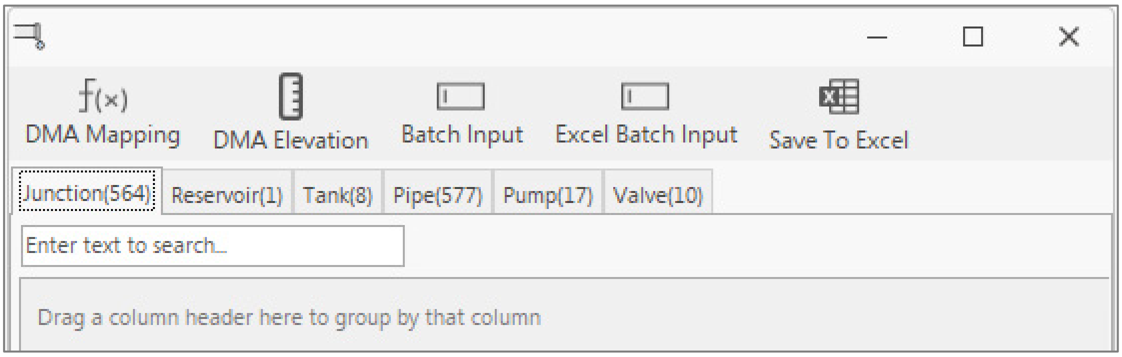


Figure 9-6: Attribute table functionalities

Creating New Layers

Users can create new editable layers by clicking on the *Add* button on the Layer toolbar. Various options will be available on 4 different tabs as depicted in **Figure 9-5**

Tabs:

- Background: Add base maps like Google Maps, Bing Map or Open Street Map.
- Waterworks (EPANET layers): A new EPANET network layer with junctions, pipes, tanks, valves, meters, etc., or specific layers related to the water network model such as Water Meter, Flow gauges etc.
- Sewerage (SWMM layers): A new SWMM network with junctions, outfalls, conduits, weirs etc., or specific layers such as Main/Lateral sewers, manholes, subcatchments etc.
- Others: Generic vector types like Points, LineStrings and Polygons.

Click the *Add Layer* button to include the new layer in your workspace.

Layer Context Menu Options

Right-clicking on a layer opens the context menu as depicted in **Figure 9-7**.

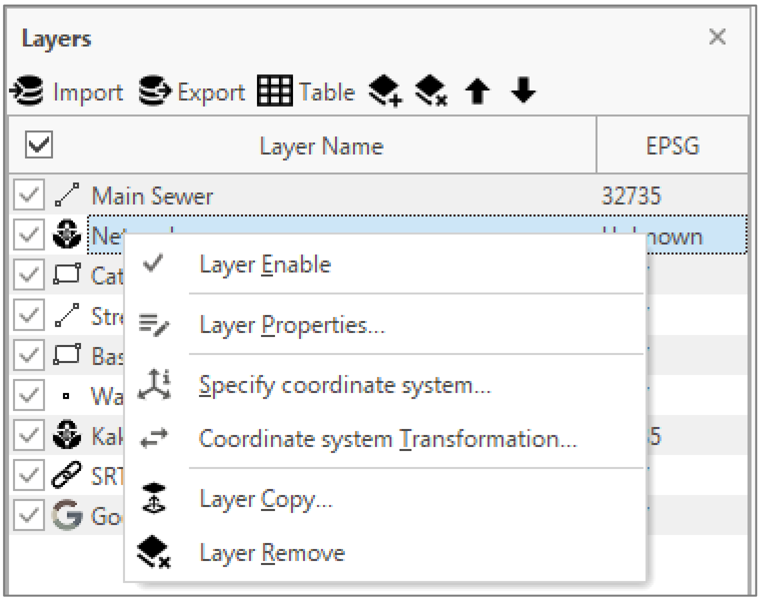


Figure 9-7: Layers context menu

This enables the user to select features as listed in **Table 9-3**.

Table 9-3: Layers context menu

Option	Description
Layer Enable	Turn layer visibility on/off.
Layer Properties	Open a dialog with detailed settings (see **Table 9-4**).
Specify Coordinate System	Define the layer's projection (see **Figure 9-15**).
Coordinate System Transformation	Reproject the layer to another EPSG (see **Figure 9-16**).
Layer Copy	Duplicate the selected layer.
Layer Remove	Delete the layer from the map.

Layer Properties Dialog

The Layer Properties dialog provides key controls such as listed in **Table 9-4** and the window shown in **Figure 9-8**.

In the Properties dialog window, the user can also open a saved style such as Thematic Tool Kit (*.ttkstyle), Initialization File (*.ini) or Styled Layer Descriptor (*.sld) which are different types of files used for storing styling and configuration information.

If a certain style has been set-up by a user this can also be saved (in *.ttkstyle format), for later use in other projects.

The parameters which can be changed will depend on the layer type, Raster or Vector or Other type off layer.

Table 9-4: Layer property settings

Option	Description
General Tab	
Parameters	
Caption	The display name or title of the layer as it appears in the map legend or layer list. See **Figure 9-8**.
Coordinate system	Specifies the spatial reference system (SRS) or projection used by the layer, such as WGS 84, UTM Zone 35S, or EPSG:4326. This ensures that the layer aligns properly with other spatial data and maps. If the coordinate system is incorrect or missing, features may appear in the wrong location or not at all.
Painting (see Figure 9-8)	
Base map	Specifies whether the layer serves as a base map (i.e., a background reference layer like satellite imagery or terrain). Base maps render beneath operational layers and are not interactive.
Cached paint	Enables or disables the use of a cached (pre-rendered) version of the layer for display. This improves rendering performance by avoiding re-drawing the layer each time the view changes.
Ignore shape parameters	When enabled, the software disregards shape-specific properties such as size, orientation, or border styling defined in the data. This is useful when a consistent or simplified rendering style is desired across all features, regardless of individual geometry attributes.
Multipass rendering (slower)	Allows the use of multiple rendering passes for complex symbology (e.g., layering shadows, outlines, gradients). It produces high-quality visual effects but may slow down performance due to repeated processing.
Prefer styling from config file	Prioritizes style settings from an external configuration file (e.g., .ttkstyle, .ini, or .sld) over default or manual settings. This ensures consistency across projects or environments by applying pre-defined styling rules.
Transparency	Sets the transparency level of the layer (from fully opaque to fully transparent) which allows layers beneath to be partially visible, enabling better visual integration or overlay comparison.
Interpretation	Defines how the layer's data should be interpreted or rendered, such as raster bands, categorical data, or elevation models. Options are Default, Pixel or Grid. This guides the software on how to handle and display the data meaningfully, based on its type and intended use.
Info	
File information	Displays metadata about the data source file associated with the layer, format (e.g., Shapefile, GeoTIFF), size, last modified date, and encoding.
User comments	A free-text field where users can add notes, descriptions, or observations related to the layer. This is to provide context, documentation, or reminders for future users or collaborators, such as data provenance, intended use, limitations, or special considerations.

Sections Tab	
Visible (see Figure 9-9)	
Sections	
Section	Represents logical groupings or thematic divisions within the layer, often used to manage complex datasets with sub-categories or to toggle visibility of specific parts of the data. This enables better organization and selective display of content within the layer, such as grouping features by type, function, or zone.
Minimum scale	Sets the smallest map scale (i.e., most zoomed out) at which the layer or section will be visible. This prevents clutter on the map by hiding detailed layers when the user is too zoomed out to see them meaningfully. For example, only show hydrants when zoomed into neighbourhood level. See **Figure 9-9**.
Maximum scale	Sets the largest map scale (i.e., most zoomed in) at which the layer or section will be visible. This hides layers when zoomed in too far, which can improve performance and relevance. For example, a regional water network might disappear when zoomed into street level. See **Figure 9-9**
3D (see Figure 9-10)	
Treat Layer as 2D	Renders the layer in a flat, two-dimensional view using only X (longitude) and Y (latitude) coordinates. This is suitable for most thematic and topographic maps where elevation or depth is not relevant. Simplifies visualization and improves performance.
Treat Layer as 3D Objects	Enables three-dimensional rendering of the layer by incorporating Z-values (elevation or depth). This is useful for visualizing terrain, infrastructure or subsurface networks in 3D scenes.
Normalized Z	Adjusts all Z-values relative to a base elevation (often ground level), converting absolute heights to relative ones. This is useful when data contains absolute elevation values but needs to be visualized relative to the surface.
Scale Z (%)	Multiplies all Z-values by a scaling factor (in percent) to enhance or compresses vertical exaggeration to improve visual interpretation of elevation differences, especially in flat terrain.
False Z	Assigns an artificial or fixed Z-value to features that lack true elevation data which allows non-elevated features (e.g., 2D lines or points) to be placed at a desired height in the 3D scene, avoiding overlap or hiding.
Pixel (see Figure 9-11)	
Colours	Defines how pixel values are mapped to display colours, either through colour Red, Green, Blue, Brightness, Contrast etc. This enhances interpretation by assigning meaningful or visually distinct colours to pixel values.
Bands	Specifies which raster bands (e.g., Red, Green, Blue, Alpha) are used for display or analysis. This allows control over image composition. Common configurations include:
Transparency	Adjusts the overall transparency of the raster layer or sets specific pixel values to be transparent (e.g., No Data areas). This improves map readability by allowing background layers to show through, or by hiding irrelevant or missing data areas.

Grid (see Figure 9-12)	
Grid Section	
Grid Band	Selects which raster band to display (Default or 1). Useful for multi-band datasets.
Shadow	Select to add shadow effect to grid rendering for better visual depth.
Antialiasing	Select to smooth out jagged edges in the raster rendering.
Minimum / Maximum Value	Defines the data value range to be rendered and visualized. Any values outside this range may be clipped
Height Threshold	
Minimum / Maximum Value	Similar to grid min/max, but may be used for visualizing a specific elevation band (e.g., for hillshading or analysis thresholds).
Ramp	
Min/Max/Legend	Displays and allows editing of value classes and their corresponding colour bands.
Smooth colours	Toggles between a smooth gradient and sharp colour changes between classes.
Grid Ramp Wizard Options (see **Figure 9-13**)	
Simple classification	Used for direct linear or continuous scaling between min and max values. The user chooses start, middle, and end values for colour gradation.
Unique values	Applies a colour to each unique raster value (used for categorical data).
Continuous values	Applies a gradient scale to a continuous range (e.g., elevation or temperature).
Minimum/Maximum value	Sets the value range for colour ramping.
Use middle value	Allows defining a midpoint value for 3-Colour gradient transitions.
Start Colour, Middle Colour, End Colour	Define the gradient colours used in the colour ramp.
Add new ramp to existing ramp	Combines the new ramp with previously defined ones.
Use HSL	Enables smoother transition by using Hue-Saturation-Lightness colour space.
Ramps	Choose from predefined palettes (e.g., Autumn, Terrain, Greyscale).
Reverse	Select to reverses the colour ramp.
Discrete	Select to display colour classes as distinct bands without gradient blending.
Show all	Select to display all ramp options for selection.
Level every	Defines spacing between data value contour levels.
Legend every	Defines spacing between legend labels displayed on the map.
Advanced classification	Allows more refined control, such as classification methods, number of classes, and band selection.
Method	Determines how the data range is divided. Natural Breaks which minimize variance within classes and maximizes variance between classes. Other methods may include Equal Interval, Quantile, Standard Deviation, etc.
Classes	Number of colour intervals or groupings to divide the raster data into

Interval	Automatically calculated based on data range and classification method; defines the step size
Band	For multi-band rasters, select which band to classify (e.g., Band 1 = elevation in DEM).
Start Colour, Middle Colour, End Colour	Define the gradient colours used in the colour ramp.
Add new ramp to existing ramp	Combines the new ramp with previously defined ones.
Use HSL	Enables smoother transition by using Hue-Saturation-Lightness colour space.
Ramps	Choose from predefined palettes (e.g., Autumn, Terrain, Greyscale).
Reverse	Select to reverses the colour ramp.
Discrete	Select to display colour classes as distinct bands without gradient blending.
Show all	Select to display all ramp options for selection.
Statistics Calculation – Fast Scan	Quickly estimates the raster stats (min/max, etc.). Faster, but less accurate.
Statistics Calculation – Full Scan	Scans the entire raster for precise statistics. Slower, but recommended for accurate classification.

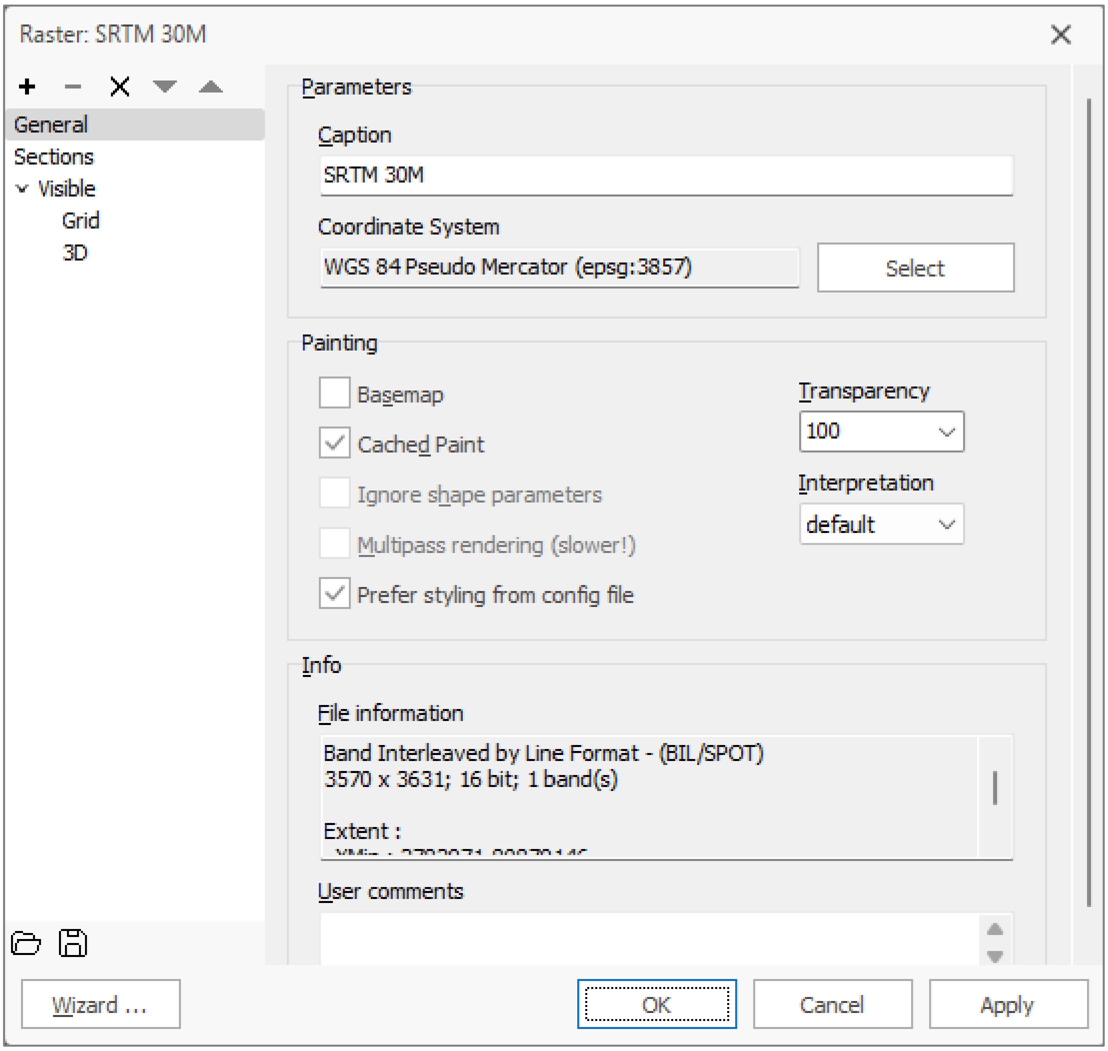


Figure 9-8: Layer properties

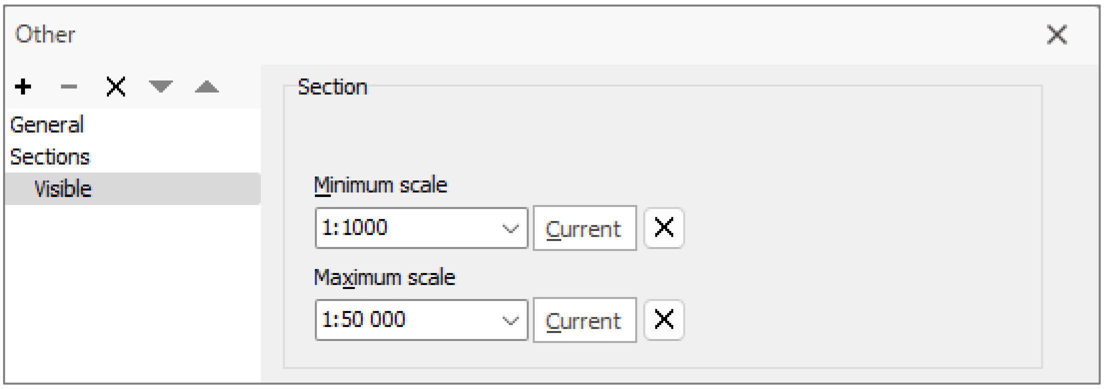


Figure 9-9: Layer properties - Visible settings

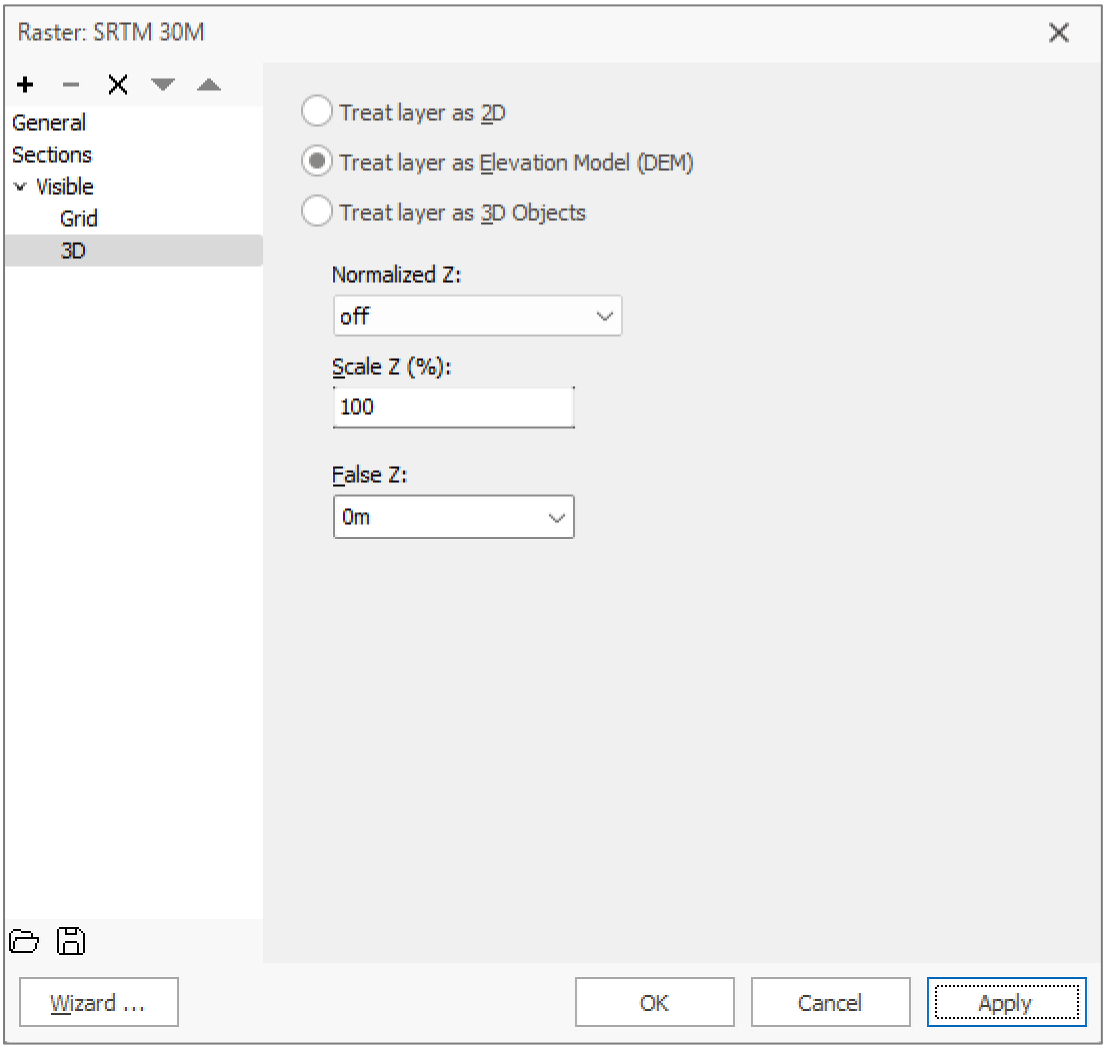


Figure 9-10: Layer property – Visible (3D) settings

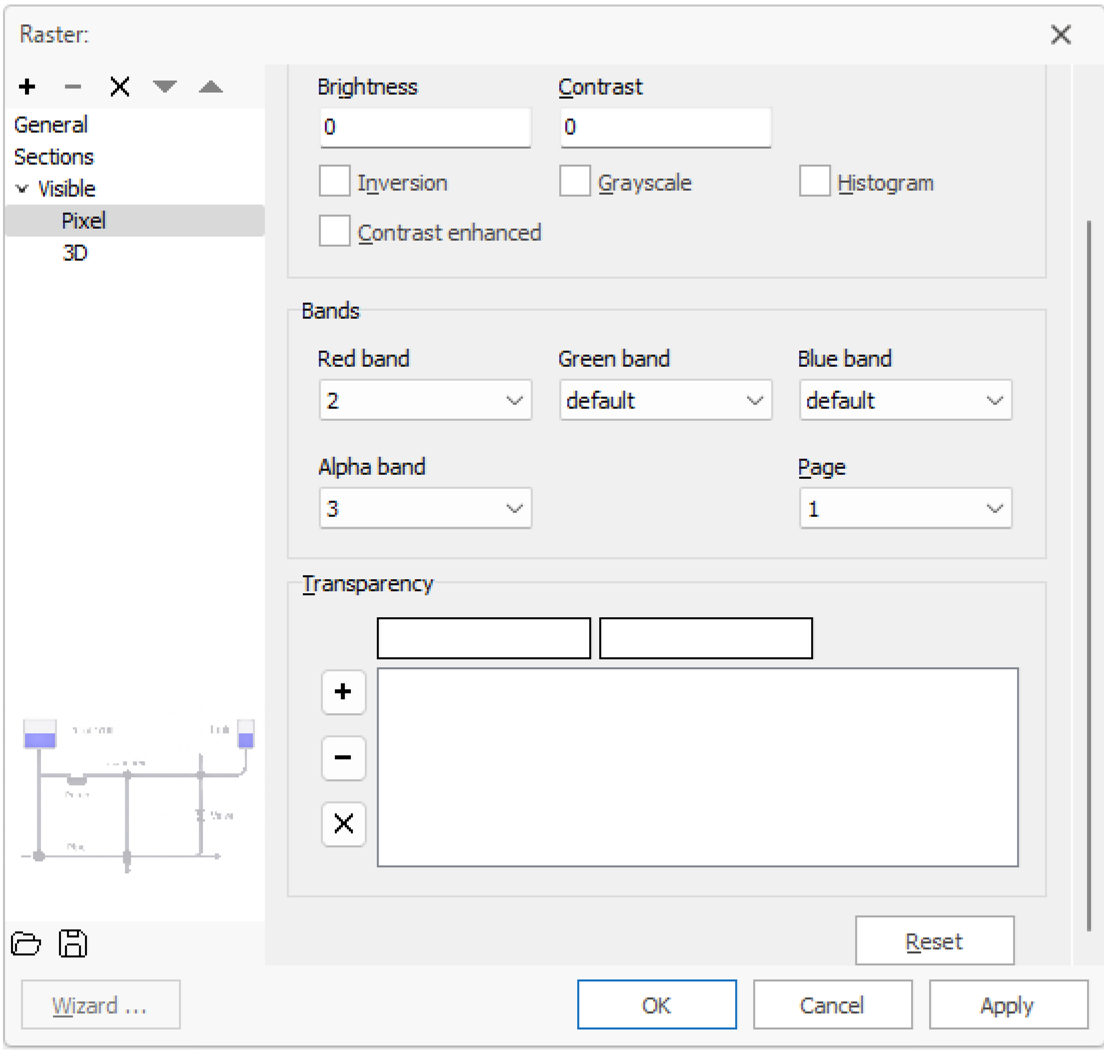


Figure 9-11: Layer property – Visible (Pixel) settings

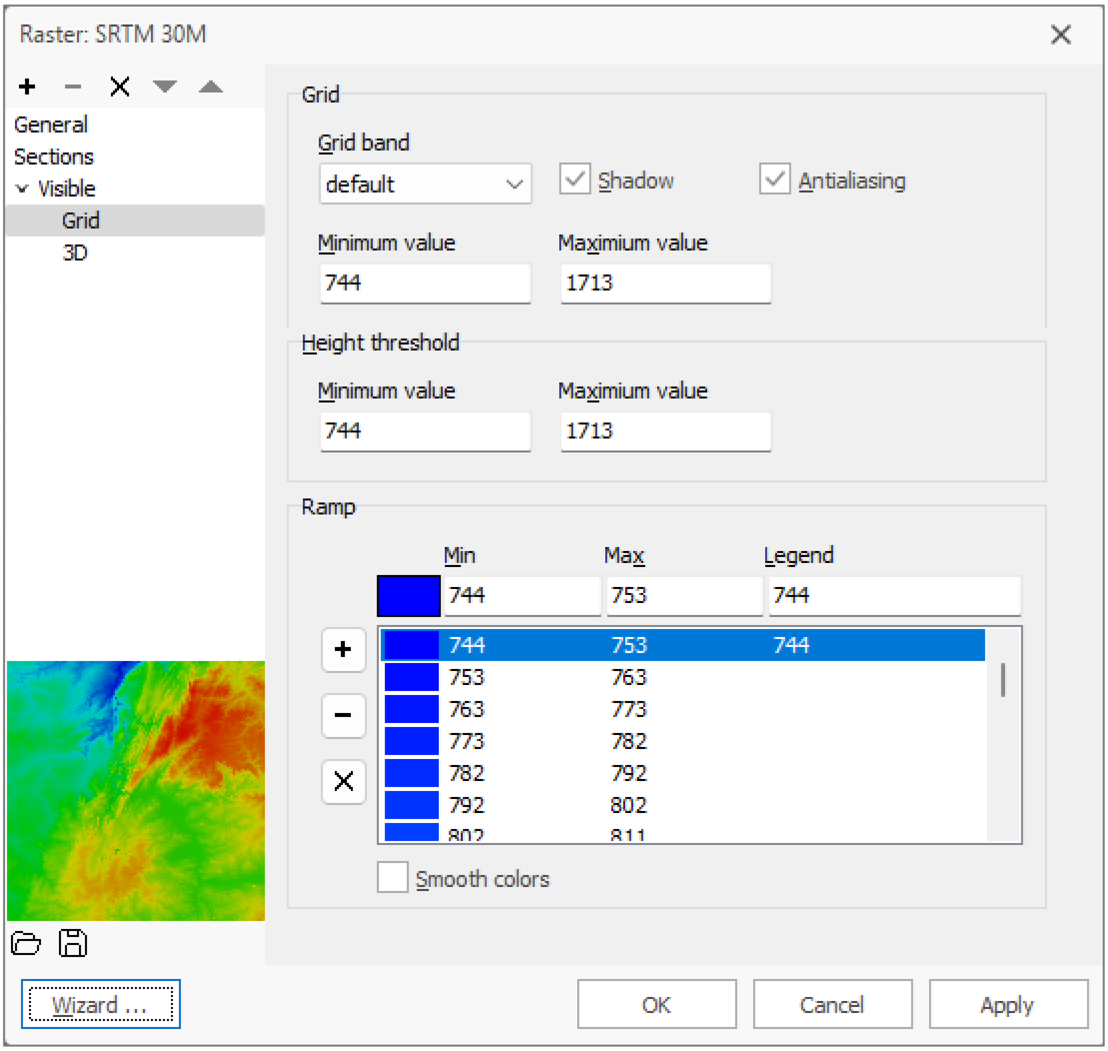


Figure 9-12: Layer property – Visible (Grid) settings

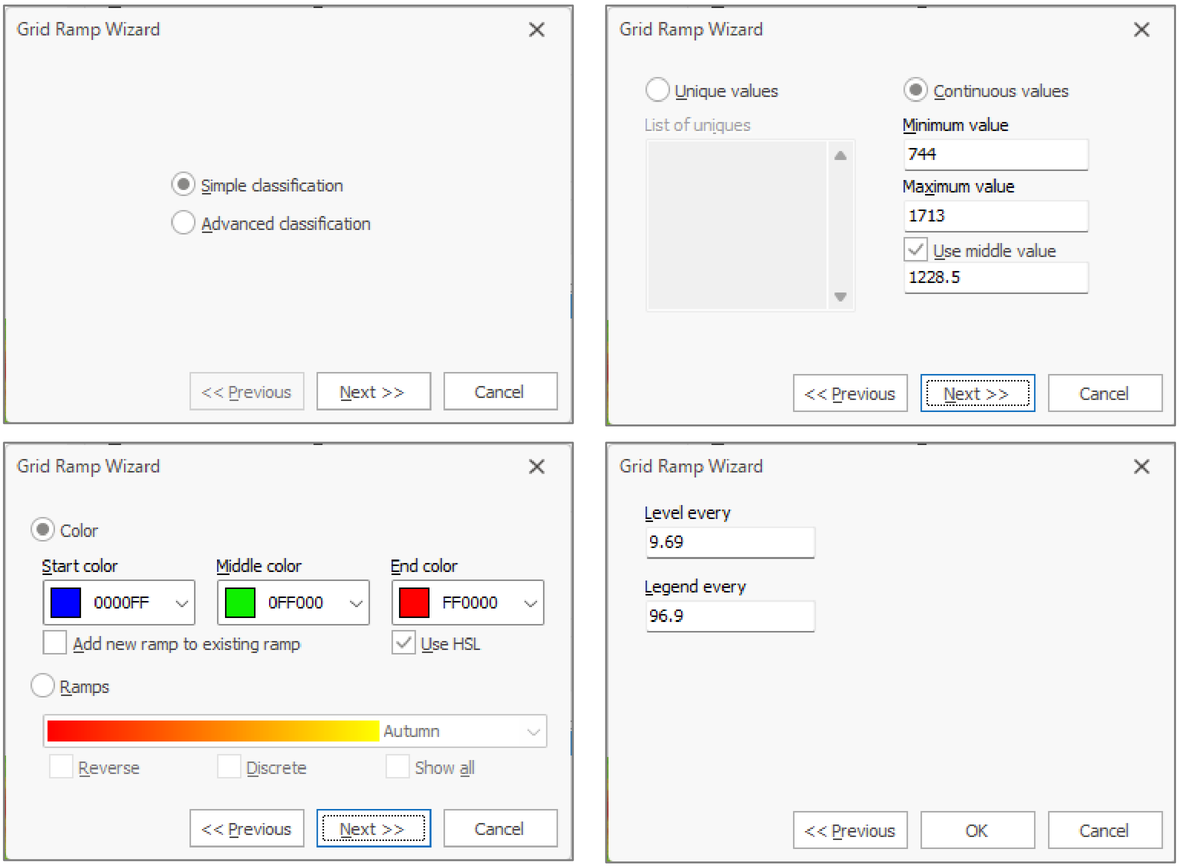


Figure 9-13: Layer property – Visible (Grid) settings – Wizard (Simple Classification)

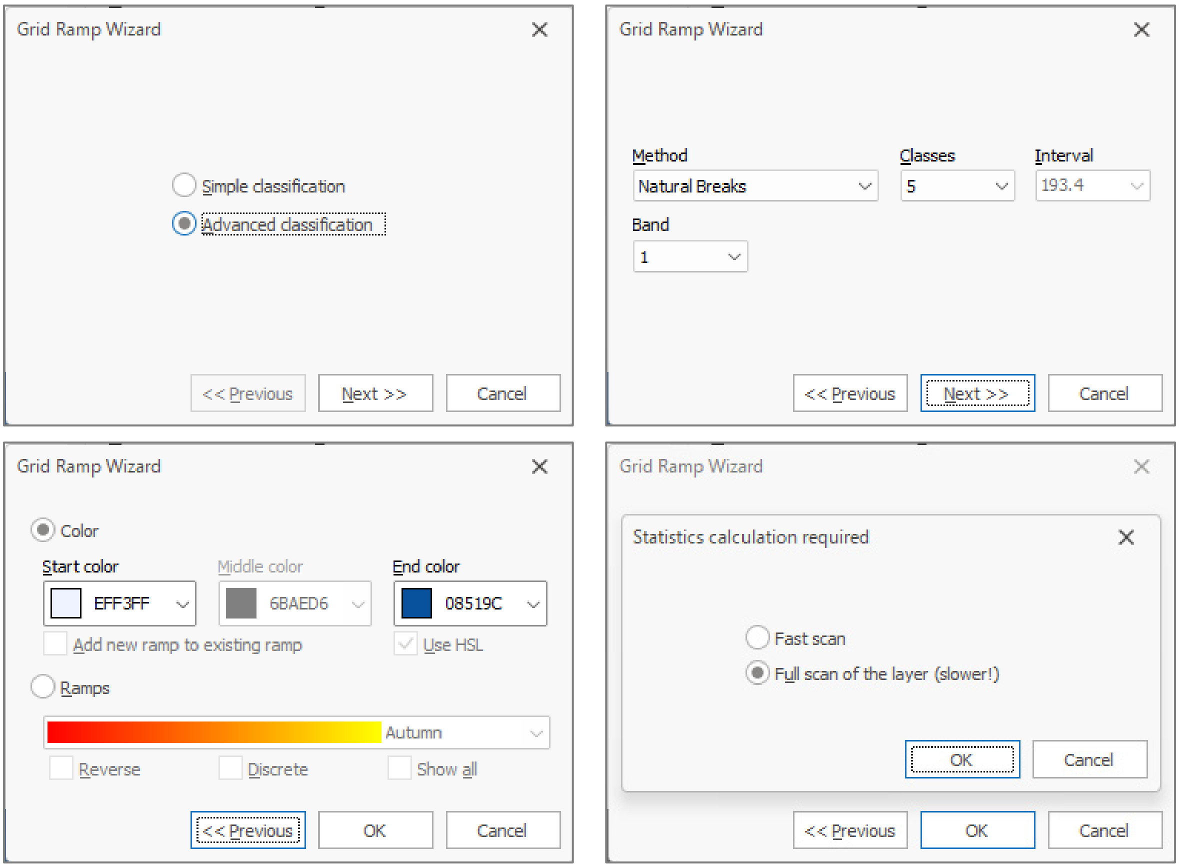


Figure 9-14: Layer property – Visible (Grid) settings – Wizard (Advanced Classification)

These tools help users visually classify and interpret raster data (e.g., elevation, runoff), and customize how information is displayed in GISpipe.

Coordinate System Settings

When working with spatial data in GISpipe, it is essential to assign the correct coordinate reference system (CRS) to ensure accurate alignment, measurements, and spatial analysis. The coordinate system defines how geographic coordinates are projected onto a flat surface.

To specify the coordinate system of a layer, right click on a layer to show the Layers context menu (**Figure 9-7**).

Left click on the *Specify Coordinate System* option to show the Select Coordinate System window (**Figure 9-15**) where the EPSG code for a newly imported layer (e.g., EPSG:3857 for Web Mercator, EPSG:32735 for WGS 84 UTM Zone 35S) can be set (**Table 9-5**).

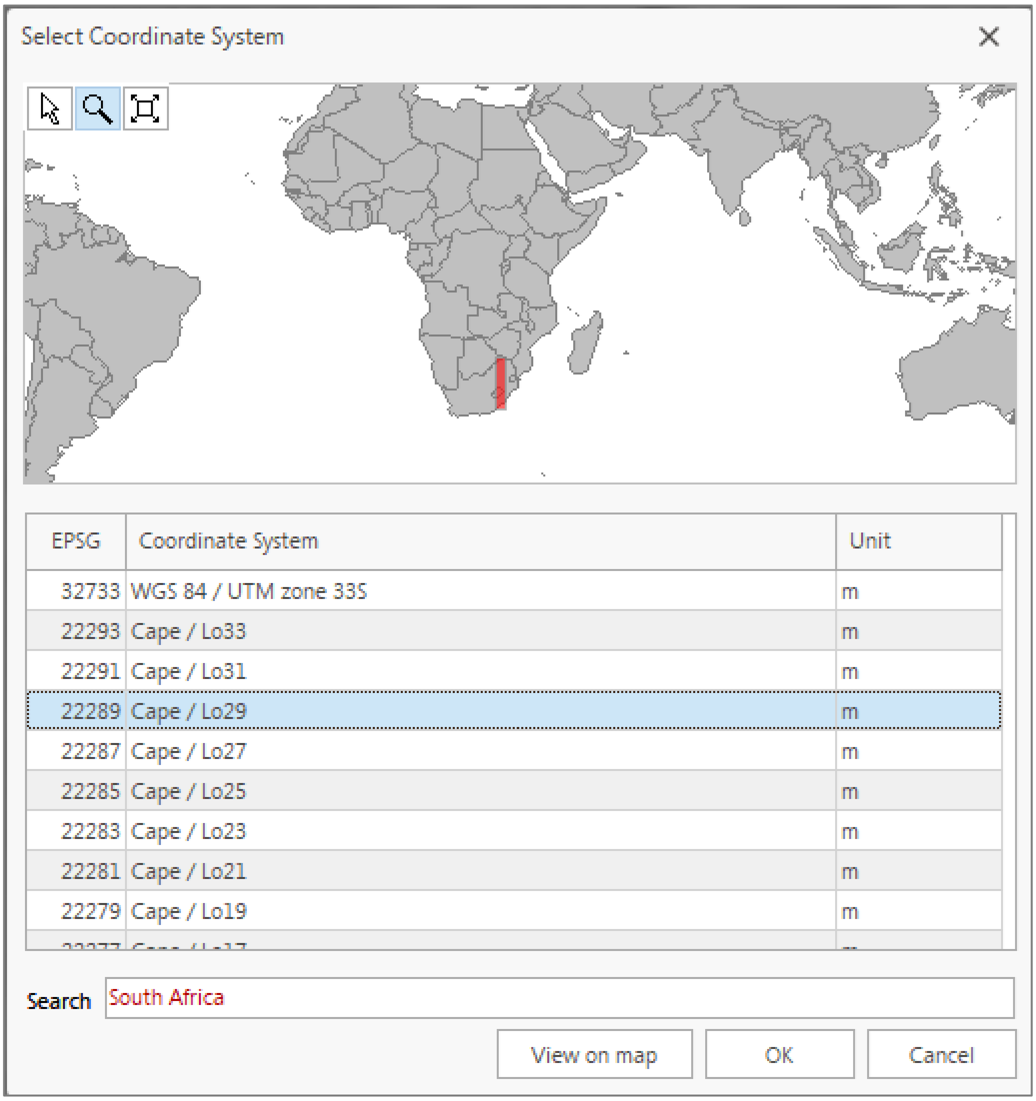


Figure 9-15: Coordinate System

Table 9-5: Coordinate System

Parameter	Description
EPSG	A unique identifier for the coordinate system.
Coordinate System	Typically includes the datum and the specific projection zone
Unit	Measurement unit (e.g., meters).

Steps to select the correct CRS

1. Identify the geographic region
   - In this case, a project located in South Africa is highlighted on the map.
   - A user can search for a country to narrow down the search for the most appropriate CRS system to select.

2. Choose the appropriate CRS

   - For South Africa, the recommended modern coordinate system is Hartebeesthoek94 (HB94), based on the WGS84 ellipsoid.
   - This system is split into Lo (Longitude) zones spaced every 2 degrees. Example zones:

     - Lo19 (EPSG:2051) for areas around 19°E
     - Lo29 (EPSG:2053) for areas around 29°E
     - Lo31 (EPSG:2054) for areas around 31°E, etc.

3. Match zone with project location

   - Find the longitude of your project site and select the corresponding HB94 / Lo zone. For example:
     - If your site is located at ~29°E longitude, use Hartebeesthoek94 / Lo29 (EPSG:2053).
     - If it is located at ~31°E, choose Lo31 (EPSG:2054).

4. Legacy systems

   - Older coordinate systems such as Cape / Lo (EPSG:22277 or EPSG:4222) may still appear. These are not recommended unless legacy data compatibility is required.

5. Final selection

   - Click the desired row (e.g., EPSG:2053 for HB94 / Lo29).
   - Click *OK button* to apply the coordinate system.

Tip: Click the *View on map* button to visually confirm the coverage of each CRS zone, aiding in the correct selection based on geographic location.

Coordinate System Transformation

Convert a layer from one CRS to another using a transformation dialog box (e.g., from 3857 to 32735).

To transform the coordinate system of a layer, right click on a layer to show the Layers context menu (**Figure 9-7**).

The Coordinate System Transformation window will be displayed as shown in **Figure 9-16**.

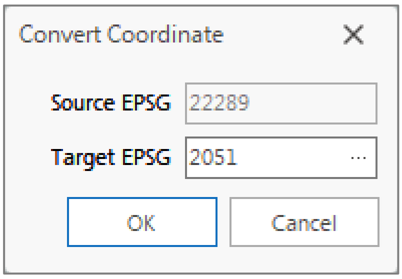


Figure 9-16: Coordinate System Transformation

The current Source EPSG is displayed and the Target ESPG can be entered in the input box or selected by clicking on the ellipse (…) to the right of the input box to show the Select Coordinate System window (**Figure 9-15**).

9.2.2 Physical components

GISpipe models a water distribution system as a collection of links connected to nodes. The links represent pipes, pumps, and control valves. The nodes represent junctions, tanks, and reservoirs. **Figure 9-17** below illustrates how these objects can be connected to one another to form a network.

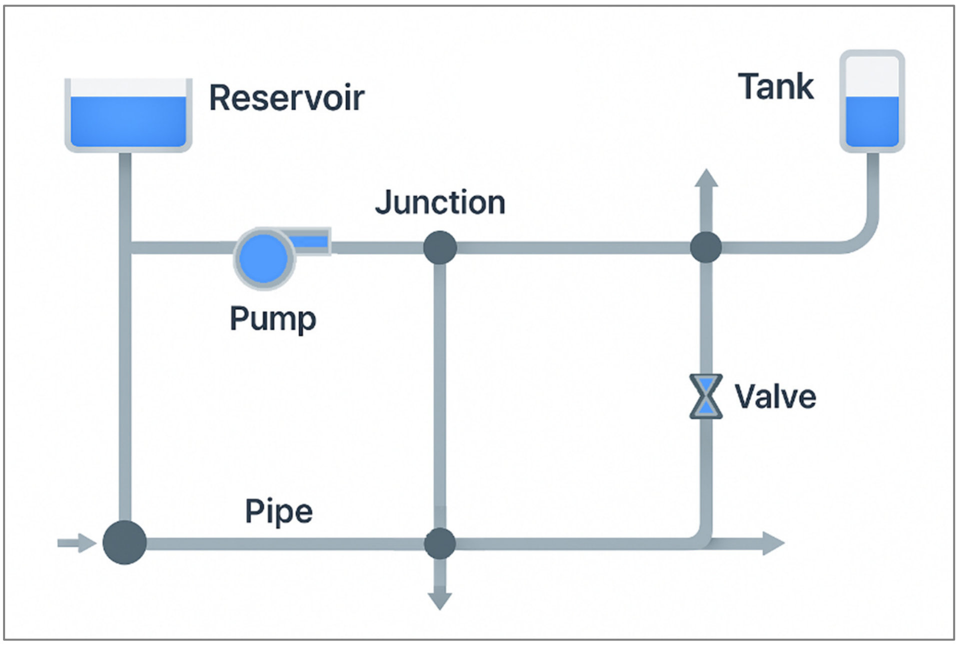


Figure 9-17: Components

Junctions

Junctions are points in the network where links join together and where water enters or leaves the network. The basic input data required for junctions are:

- Elevation above some reference (usually mean sea level)
- Water demand (rate of withdrawal from the network)
- Initial water quality

The output results computed for junctions at all time periods of a simulation are:

- Hydraulic head (internal energy per unit weight of fluid)
- Pressure
- Water quality

Junctions can also:

- Have their demand vary with time
- Have multiple categories of demands assigned to them
- Have negative demands indicating that water is entering the network
- Have pressure driven demand
- Be water quality sources where constituents enter the network
- Contain emitters (or sprinklers) which make the outflow rate depend on the pressure

Reservoirs

Reservoirs are nodes that represent an infinite external source or sink of water to the network. They are used to model such things as lakes, rivers, groundwater aquifers, and tie-ins to other systems. Reservoirs can also serve as water quality source points. The primary input properties for a reservoir are its hydraulic head (equal to the water surface elevation if the reservoir is not under pressure) and its initial quality for water quality analysis. Because a reservoir is a boundary point to a network, its head and water quality cannot be affected by what happens within the network. Therefore, it has no computed output properties. However, its head can be made to vary with time by assigning a time pattern to it (see Section 9.2.6).

Tanks

Tanks are nodes with storage capacity, where the volume of stored water can vary with time during a simulation. The primary input properties for tanks are:

- Bottom elevation (where water level is zero)
- Diameter (or shape if non-cylindrical)
- Initial, minimum and maximum water levels
- Initial water quality

The principal outputs computed over time are:

- Hydraulic head (water surface elevation)
- Water quality

Tanks are required to operate within their minimum and maximum levels. GISpipe stops outflow if a tank is at its minimum level and stops inflow if it is at its maximum level. Tanks can also serve as water quality source points.

Emitters

Emitters are devices associated with junctions that model the flow through a nozzle or orifice that discharges to the atmosphere. The flow rate through the emitter varies as a function of the pressure available at the node. Emitters are used to model flow through sprinkler systems and irrigation networks. They can also be used to simulate leakage in a pipe connected to the junction (if a discharge coefficient and pressure exponent for the leaking crack or joint can be estimated) or compute a fire flow at the junction (the flow available at some minimum residual pressure).

In the latter case one would use a very high value of the discharge coefficient (e.g., 100 times the maximum flow expected) and modify the junction’s elevation to include the equivalent head of the pressure target. GISpipe treats emitters as a property of a junction and not as a separate network component.

Note: The pressure-flow relation at a junction defined by an emitter should not be confused with the pressure demand relation when performing a pressure driven analysis (PDA).

Pipes

Pipes are links that convey water from one point in the network to another. GISpipe assumes that all pipes are full at all times. Flow direction is from the end at higher hydraulic head (internal energy per weight of water) to that at lower head. The principal hydraulic input parameters for pipes are:

- Start and end nodes
- Diameter
- Length
- Roughness coefficient (for determining headloss)
- Minor headloss coefficient
- Status (open, closed, or contains a check valve)

The status parameter allows pipes to implicitly contain shutoff (gate) valves and check (non-return) valves (which allow flow in only one direction).

The water quality inputs for pipes consist of:

- Bulk reaction coefficient
- Wall reaction coefficient

Computed outputs for pipes include:

- Flow rate
- Velocity
- Headloss
- Darcy-Weisbach friction factor
- Average reaction rate (over the pipe length)
- Average water quality (over the pipe length)

The hydraulic head lost by water flowing in a pipe due to friction with the pipe walls can be computed using one of three different formulas:

- Hazen-Williams formula
- Darcy-Weisbach formula
- Chezy-Manning formula

The Hazen-Williams formula is the most commonly used headloss formula in the United States. It cannot be used for liquids other than water and was originally developed for turbulent flow only. The Darcy-Weisbach formula is the most theoretically correct. It applies over all flow regimes and to all liquids. The Chezy-Manning formula is more commonly used for open channel flow. Each formula uses a different pipe roughness coefficient that must be determined empirically. With the Darcy-Weisbach formula GISpipe uses different methods to compute the friction factor depending on the flow regime:

- Hagen–Poiseuille formula is used for laminar flow (Re < 2 000).
- Swamee and Jain approximation to the Colebrook-White equation is used for fully turbulent flow (Re >4 000).
- Cubic interpolation from the Moody Diagram is used for transitional flow (2 000 < Re < 4 000).

Pipes can be set open or closed at pre-set times or when specific conditions exist, such as when tank levels fall below or above certain set points, or when nodal pressures fall below or above certain values. See the discussion of Controls in Section 9.2.6.

Minor head losses (also called local losses) are caused by the added turbulence that occurs at bends and fittings. The importance of including such losses depends on the layout of the network and the degree of accuracy required. They can be accounted for by assigning the pipe a minor loss coefficient.

Pumps

Pumps are links that impart energy to a fluid thereby raising its hydraulic head. The principal input parameters for a pump are its start and end nodes and its pump curve (the combination of heads and flows that the pump can produce). In lieu of a pump curve, the pump could be represented as a constant energy device, one that supplies a constant amount of energy to the fluid for all combinations of flow and head. The principal output parameters are flow and head gain. Flow through a pump is unidirectional and GISpipe will not allow a pump to operate outside the range of its pump curve.

Variable speed pumps can also be considered by specifying that their speed setting be changed under these same types of conditions. By definition, the original pump curve supplied to the program has a relative speed setting of 1. If the pump speed doubles, then the relative setting would be 2; if run at half speed, the relative setting is 0.5 and so on. Changing the pump speed shifts the position and shape of the pump curve.

As with pipes, pumps can be turned on and off at pre-set times or when certain conditions exist in the network. A pump's operation can also be described by assigning it a time pattern of relative speed settings. GISpipe can also compute the energy consumption and cost of a pump. Each pump can be assigned an efficiency curve and schedule of energy prices. If these are not supplied then a set of global energy options will be used.

Flow through a pump is unidirectional. If system conditions require more head than the pump can produce, GISpipe shuts the pump off. If more than the maximum flow is required, GISpipe extrapolates the pump curve to the required flow, even if this produces a negative head. In both cases a warning message will be issued.

Valves

Valves are links that limit the pressure or flow at a specific point in the network. Their principal input parameters include:

- Start and end nodes
- Diameter
- Setting
- Status

The computed outputs for a valve are flow rate and headloss. The different types of valves included in GISpipe are listed in **Table 9-6**.

Table 9-6: Types of valves

Valve type	Description	Image
Isolating Valve (IV)	IVs (ball, butterfly, gate) is simulated by a setting in the Initial Status property of a pipe. It can be either completely open or closed and are not considered as separate valve links.	
Check Valve (CV)	CVs (non-return, one-way) is simulated by a setting in the Initial Status property of a pipe. Select Check Valve if to ensure the fluid can only flow in one direction. CVs are not considered as separate valve links.	
Pressure Reducing Valve (PRV)	PRVs limit the pressure at a point in the pipe network. GISpipe computes in which of three different states a PRV can be in: • Partially opened (i.e., active) to achieve its pressure setting on its downstream side when the upstream pressure is above the setting • Fully open if the upstream pressure is below the setting • Closed if the pressure on the downstream side exceeds that on the upstream side (i.e., reverse flow is not allowed)	
Pressure Sustaining Valve (PSV)	PSVs maintain a set pressure at a specific point in the pipe network. GISpipe computes in which of three different states a PSV can be in: • Partially opened (i.e., active) to maintain its pressure setting on its upstream side when the downstream pressure is below this value • Fully open if the downstream pressure is above the setting • Closed if the pressure on the downstream side exceeds that on the upstream side (i.e., reverse flow is not allowed)	
Pressure Breaker Valve (PBV)	PBVs force a specified pressure loss to occur across the valve. Flow through the valve can be in either direction. PBV's are not true physical devices but can be used to model situations where a particular pressure drop is known to exist.	

Valve type	Description	Image
Flow Control Valve (FCV)	FCVs limit the flow to a specified amount. The program produces a warning message if this flow cannot be maintained without having to add additional head at the valve (i.e., the flow cannot be maintained even with the valve fully open).	
Throttle Control Valve (TCV)	TCVs simulate a partially closed valve by adjusting the minor head loss coefficient of the valve. A relationship between the degree to which a valve is closed and the resulting head loss coefficient is usually available from the valve manufacturer.	
General Purpose Valve (GPV)	GPVs are used to represent a link where the user supplies a special flow - head loss relationship instead of following one of the standard hydraulic formulas. They can be used to model turbines, well draw-down or reduced-flow backflow prevention valves.	

Each type of valve has a different type of setting parameter that describes its operating point (pressure for PRVs, PSVs, and PBVs; flow for FCVs; loss coefficient for TCVs, and head loss curve for GPVs). Valves can have their control status overridden by specifying they be either completely open or completely closed. A valve's status and its setting can be changed during the simulation by using control statements.

Because of the ways in which valves are modelled the following rules apply when adding valves to a network:

- A PRV, PSV or FCV cannot be directly connected to a reservoir or tank (use a length of pipe to separate the two)
- PRVs cannot share the same downstream node or be linked in series.
- Two PSVs cannot share the same upstream node or be linked in series.
- A PSV cannot be connected to the downstream node of a PRV.

9.2.3 Types of objects

GISpipe contains both physical objects that can appear on the Map Viewer, and non-physical objects that encompass design and operational information. These objects can be classified as followed:

1. Nodes (Junctions, Reservoirs & Tanks)
2. Links (Pipes, Pumps, Valves)
3. Map labels
4. Time Patterns
5. Curves
6. Controls (Simple and/or Rule-Based)

9.2.4 Adding objects

GISpipe allows manual digitization of new components directly on the map.

To Add Network elements:
Go to the *Edit* tab (**Figure 9-18**)

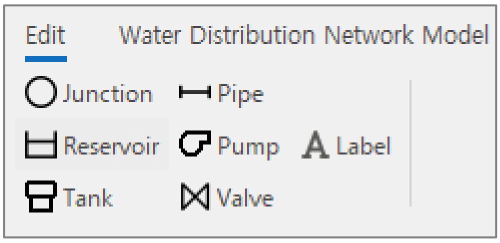


Figure 9-18: Edit Tab (adding network elements)

Select the object type:

- Junction
- Pipe
- Reservoir
- Tank
- Pump
- Valve

Or

- Label

Adding a Node

Select the type of node (junction, reservoir, or tank) and move the mouse cursor to the desired location on the Viewer Map and click to place the object on the specified location (**Figure 9-19**).

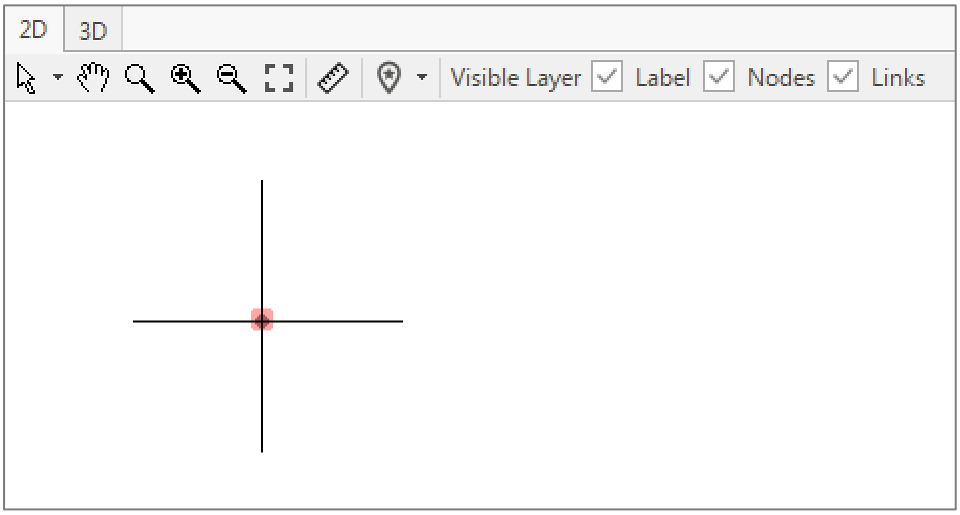


Figure 9-19: Added a junction node

Once the object was added it would have been included in the list of objects and its properties can be set, as shown in **Figure 9-20**.

Attributes will be auto-filled with default values (editable via the Property Editor, example shown in **Figure 9-20**).

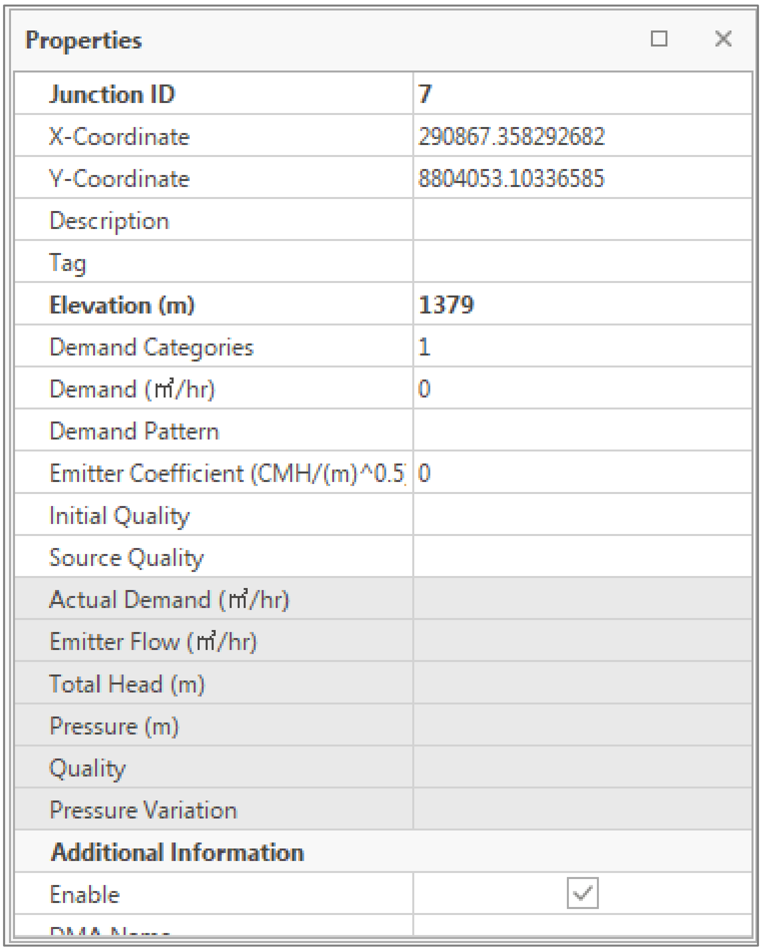


Figure 9-20: Object properties

The user has the option to move the object to a different place by either selecting in in the Map Viewer and dragging it to a new location or the object's map coordinates (X and Y) could be set in the Properties Editor. To be able to drag the object to a new position would however require the user to first change the cursor selection function by clicking on the button and changing the cursor to the *Select* option as shown in **Figure 9-21**.

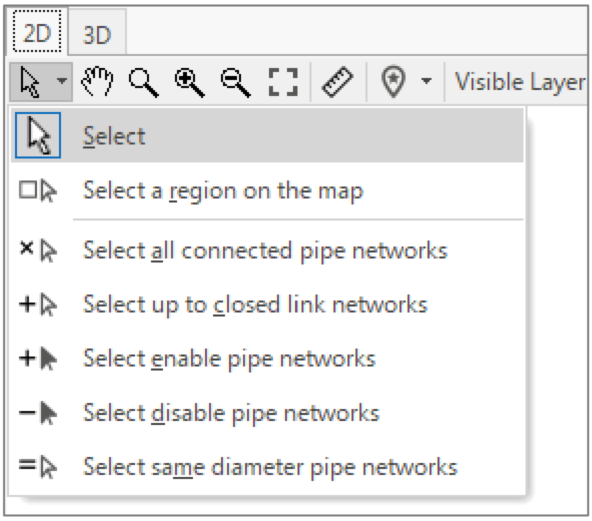


Figure 9-21: Changing cursor function

The cursor pointer would change to a cross with a square block when moving over the Map Viewer area and change to a cross with a green circle (depending on setting, see **Figure 7-12**) when hovering over the object as shown in **Figure 9-22** and **Figure 9-23**.

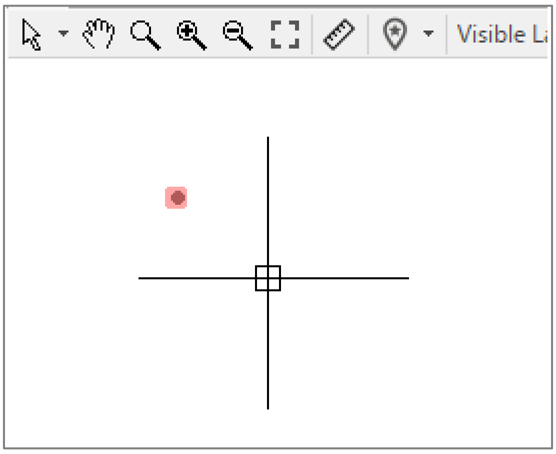


Figure 9-22: Mouse cursor over Map Viewer Area

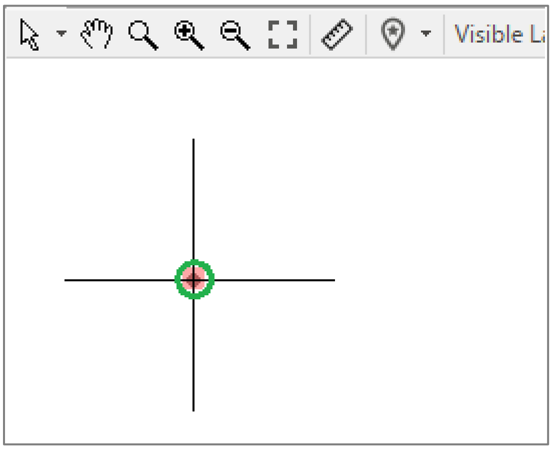


Figure 9-23: Mouse cursor over object

Adding a Link

Select the type of link (Pipes, Pumps or Valves) (**Figure 9-18**) and move the mouse cursor to the desired location on the Viewer Map.

Click the mouse when over the link's start node.

Move mouse to the link's end node and *click* on that object (**Figure 9-24**).

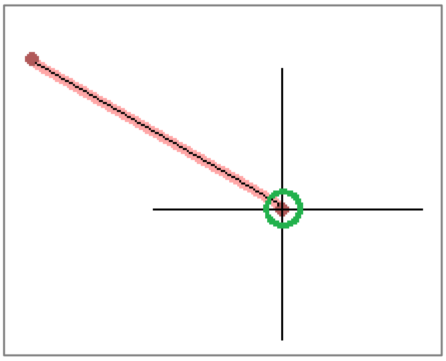


Figure 9-24: Add a pipe link

Tip: Use the *Pan* and *Zoom* tools for precise placement (**Figure 9-25**). Pipes must snap to valid start and end nodes.

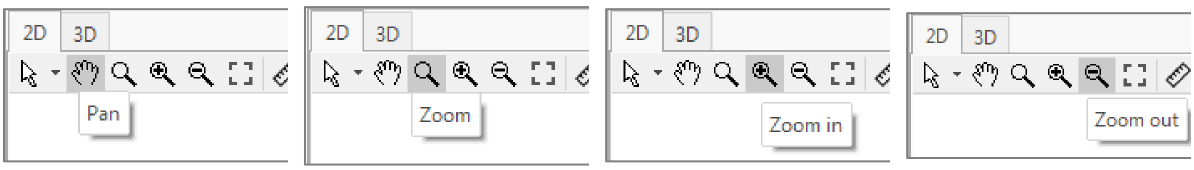


Figure 9-25: Pan and Zoom tools of Map Viewer Area

If the link is a pipe and the user requires it to have a curvature or change direction move the mouse in the direction of the link's end node, *clicking* it at those intermediate points where it is necessary to change the link's direction. Then *click* the mouse a final time over the link's end node as shown in **Figure 9-26**.

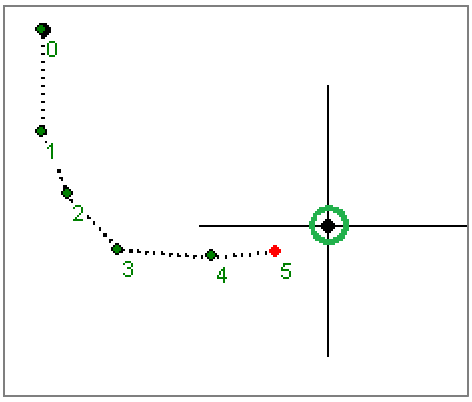


Figure 9-26: Creating a pipe with a curvature

Pressing the right mouse button or the Escape key while drawing a link will cancel the operation.

Use the *Edit Vertices* tool, available from the Edit Facilities toolbar (**Figure 9-27**) to adjust the geometry of pipes. As described intermediate points could be added to model curved or offset alignments and can be moved after initially drawing these on the map.

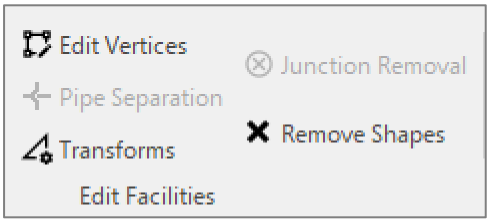


Figure 9-27: Edit Facilities toolbar

Select the *Edit Vertices* tool and then select a pipe with vertices. *Clicking* on an intermediate point will change the colour of the selected point to a red dot (**Figure 9-28**). *Click* and *hold* in the left mouse button allows the user to drag it to a new position. Release mouse button when in the correct position. Right *click* to accept the change.

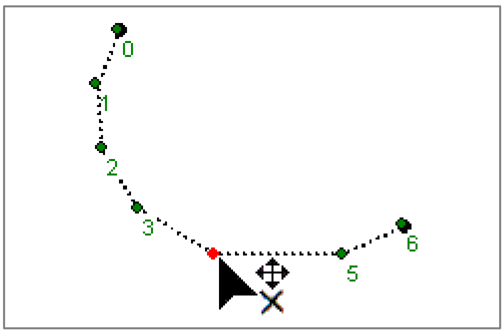


Figure 9-28: Editing vertices

When hovering over a selected link the cursor would change to a green cross as shown in **Figure 9-29**.

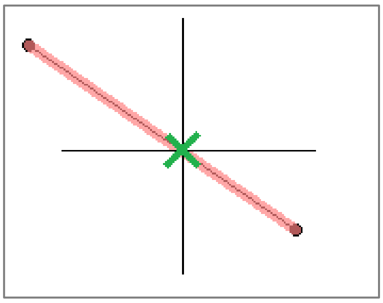


Figure 9-29: Selecting a link

Valves and pumps are added in a similar way (**Figure 9-30**). These links would require specific information as described in Section 9.2.5 as well as **Table 9-12** and **Table 9-13**.

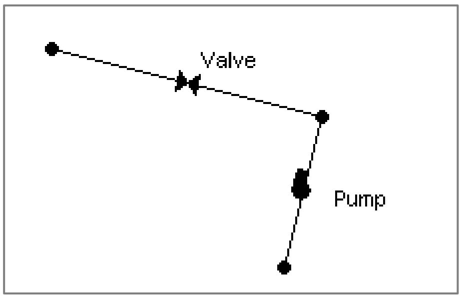


Figure 9-30: Links - Pumps and valves

Additional editing functions can be found by selecting a link or node and right clicking the mouse as shown in **Figure 9-31** with functions described in **Table 9-7**.

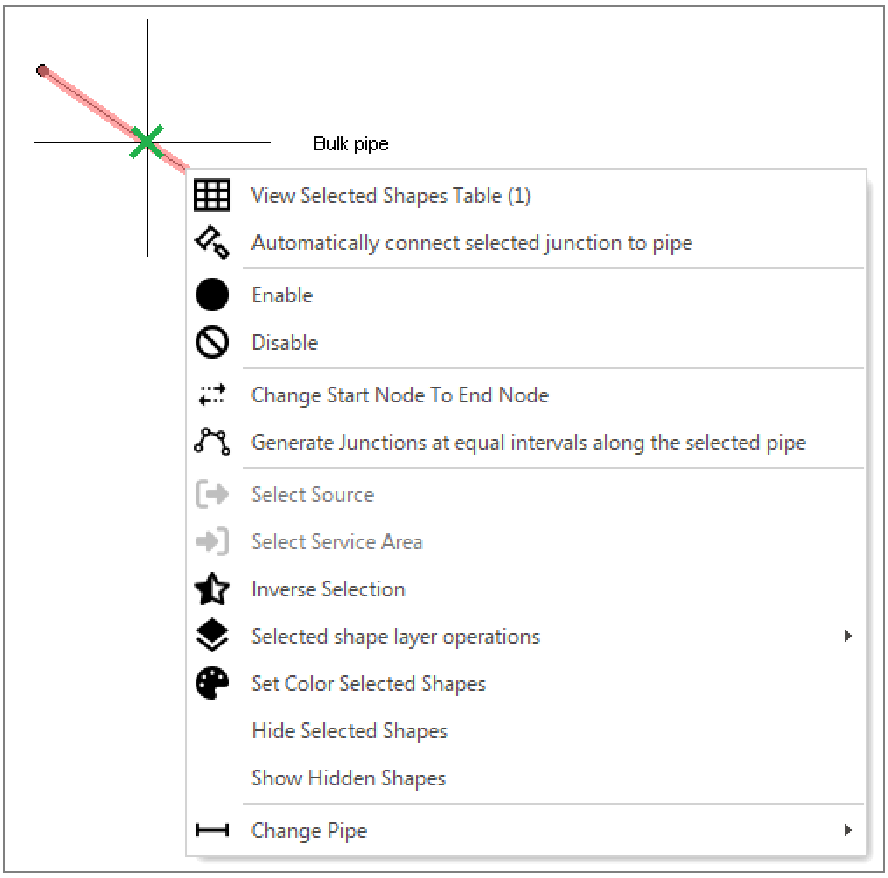


Figure 9-31: Additional editing functions

Table 9-7: Additional editing functions

Option	Description
View Selected Shapes Table (1)	View all the selected objects in a table format. Value in brackets indicates number of objects selected). A mixture of objects can be selected simultaneously and the similar objects will be grouped together in tables.
Automatically connect selected junction to pipe	Connects a selected junction (node) to the nearest or chosen pipe in the model. This is useful when placing a new junction as it automatically snaps and links into the network.
Enable	Activates or "turns on" the selected elements (pipes, pumps, valves, etc.) so that they are included in the hydraulic simulation.
Disable	Deactivates or "turns off" the selected elements. Disabled objects remain visible in the network but are ignored in the simulation (no flow or headloss is calculated through them).
Change Start Node To End Node	Changes the links begin and end point. Important for pumps and valves.
Generate Junctions at equal intervals along the selected pipe	Junctions will be added along the selected pipe based on a distance entered by the user.
Select Source	Highlights and selects all upstream elements in the network that contribute flow to the currently selected node, junction, or pipe. Useful for tracing the origin of water supply.
Select Service Area	Selects all downstream elements supplied from the selected source (junction, reservoir, or tank). This allows defining quickly the service area fed by a given node.
Inverse Selection	Reverses the current selection. All unselected objects become selected, and all selected objects become unselected.
Selected shape layer operations	Allows operations (copy, move, delete, or edit attributes) to be performed on the layer of the selected shapes, treating them as a group.
Set Colour Selected Shapes	Changes the colour of the selected objects
Hide Selected Shapes	Hides a selected object
Show Hidden Shapes	Shows any hidden objects
Change Pipe	A link can be changed to a Pipe, Pump or Valve. A node can be changed to a Junction, Reservoir or Tank.

Adding a Map Label

To add a label to the map, *click* the *Label* button on the Edit toolbar (**Figure 9-18**).

Click the mouse on the map where the label should be placed.

Enter the caption for the label and press the *Enter* key.

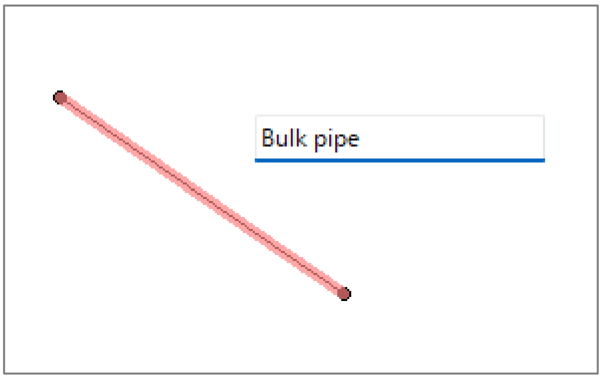


Figure 9-32: Adding a label to the map

The properties of the label can be changed in the Properties Editor, if the label was selected (**Figure 9-33**).

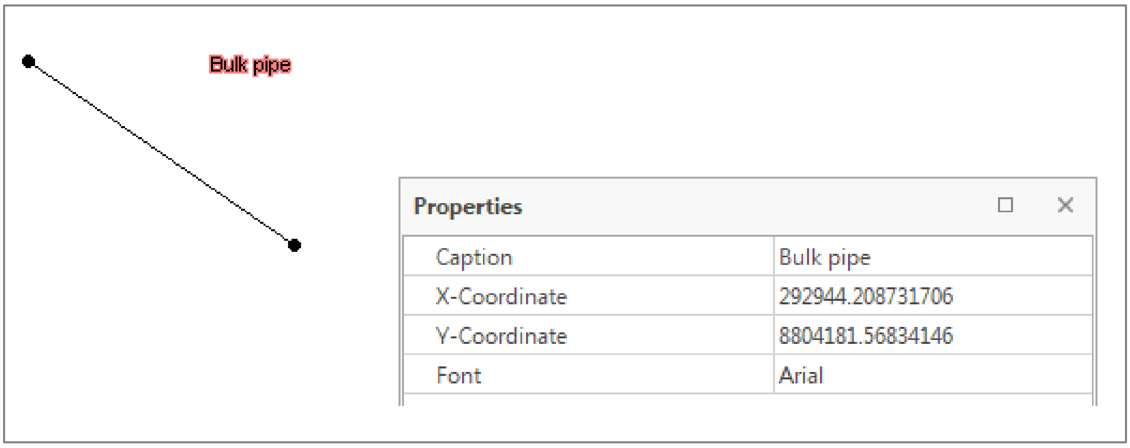


Figure 9-33: Properties of the label

To add a non-visual object (Curve, Time Pattern, Controls) to the network see Section 9.2.6.

9.2.5 Editing Visual Objects

Each component in GISpipe has editable attributes accessible via the Property Editor example shown in **Figure 9-34**, or the Attribute Table, **Figure 9-35**. It is used to edit the properties of objects that can appear on the Viewer Map (Junctions, Reservoirs, Tanks, Pipes, Pumps, Valves, or Labels).

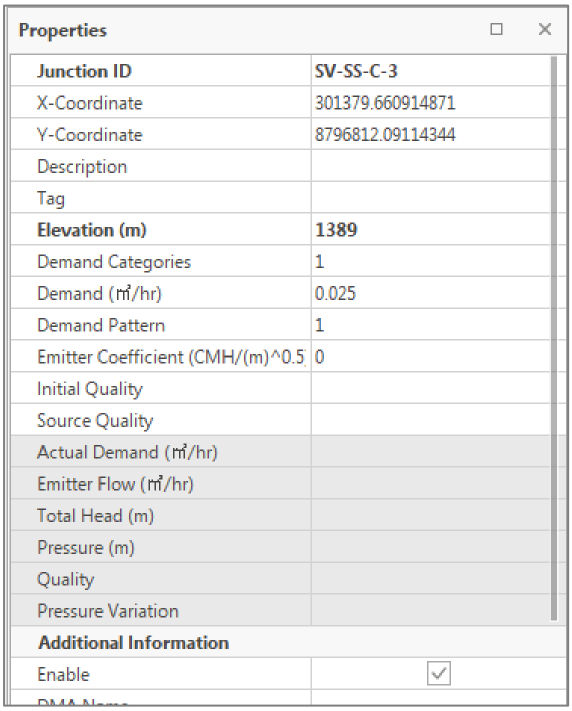


Figure 9-34: Properties editor (example)

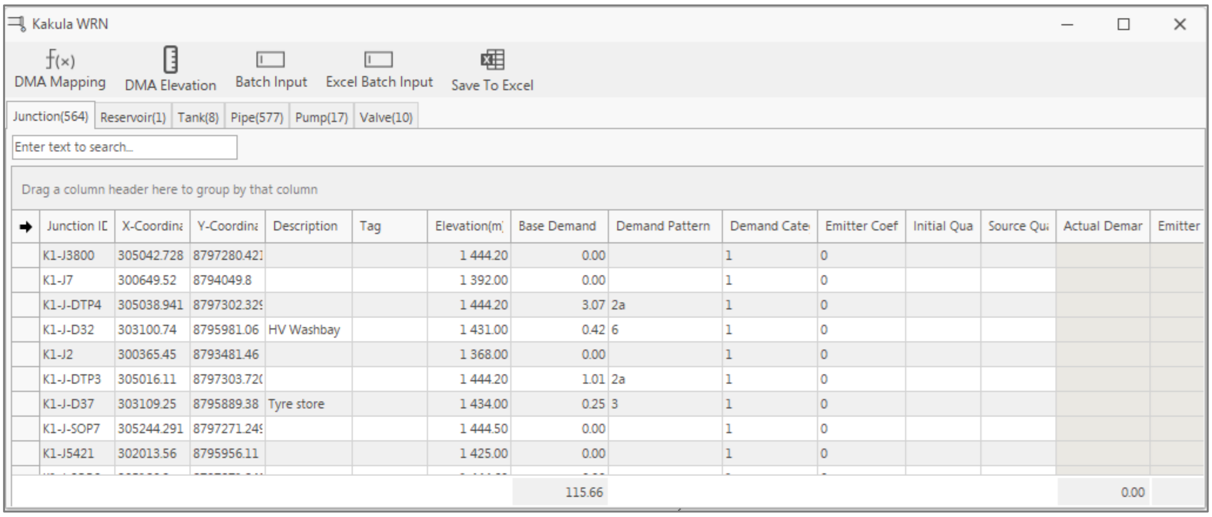


Figure 9-35: Attribute table (example)

To edit one of these objects, select the object on the Viewer Map or double-click a cell in the Table view.

Entered values/changes are saved automatically.

The associated properties of each of these types of objects are described in **Table 9-8** to **Table 9-14**.

Tip: Use the *Search/Filter* option in the table to isolate and bulk-edit similar elements.

Note: The unit system in which object properties are expressed depends on the choice of units for flow rate. Using a flow rate expressed in litres or cubic meters means that SI metric units will be used. Using a flow rate expressed in cubic feet, gallons or acre-feet means that US units will be used for all quantities. Flow units are selected by clicking the *Hydraulics* toolbar button (see **Figure 9-36**) or *Unit* on the Status Bar (see **Figure 9-37**). The units used for all properties are summarized in **Appendix B**.

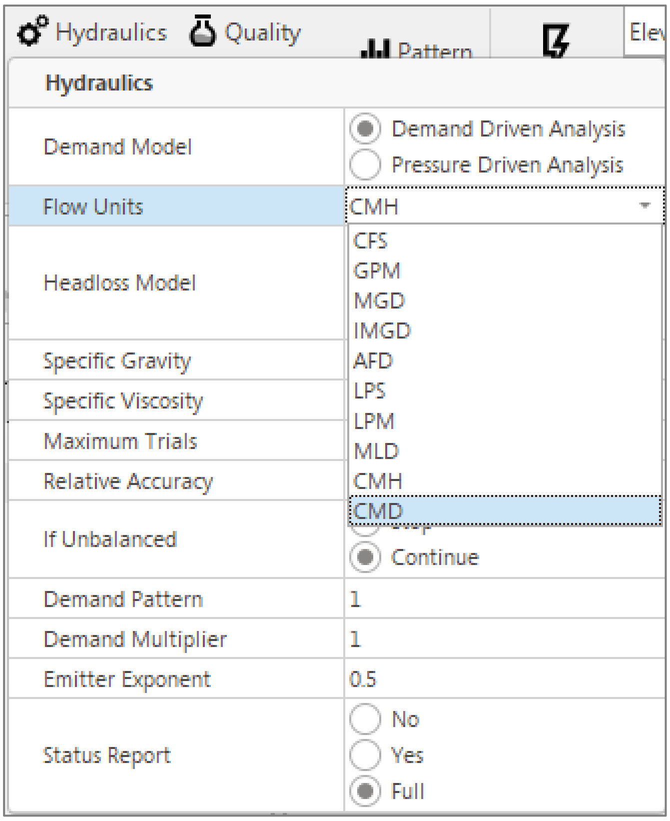


Figure 9-36: Hydraulics options

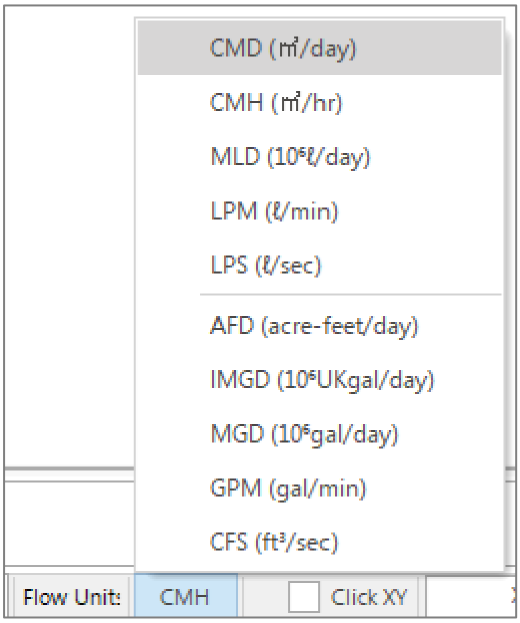


Figure 9-37: Flow Unit options

The junction properties are provided in **Table 9-8**.

Table 9-8: Junction properties

Property	Description
Junction ID	A unique label used to identify the junction. It can consist of a combination of up to 15 numerals or characters. It cannot be the same as the ID for any other node. This is a required property.
X-Coordinate	The horizontal location of the junction on the map, measured in the map's distance units. If left blank the junction will not appear on the network map.
Y-Coordinate	The vertical location of the junction on the map, measured in the map's distance units. If left blank the junction will not appear on the network map.
Description	An optional text string that describes other significant information about the junction.
Tag	An optional text string (with no spaces) used to assign the junction to a category, such as a pressure zone.
Elevation	The elevation in meters (feet) above some common reference of the junction. This is a required property. Elevation is used only to compute pressure at the junction. It does not affect any other computed quantity.
Base Demand	The average or nominal demand for water by the main category of consumer at the junction, as measured in the current flow units. A negative value is used to indicate an external source of flow into the junction. If left blank then demand is assumed to be zero.
Demand Pattern	The ID label of the time pattern used to characterize time variation in demand for the main category of consumer at the junction. The pattern provides multipliers that are applied to the Base Demand to determine actual demand in a given time period. If left blank then the Default Time Pattern assigned in the Hydraulic Options (see Section 9.4.1) will be used.
Demand Categories	Number of different categories of water users defined for the junction. Click the ellipsis button (or hit the Enter key) to bring up a special Demands Editor which will let the user assign base demands and time patterns to multiple categories of users at the junction. Ignore if only a single demand category will suffice.
Emitter Coefficient	Discharge coefficient for emitter (sprinkler or nozzle) placed at junction. The coefficient represents the flow (in current flow units) that occurs at a pressure drop of 1 psi (or meter). Leave blank if no emitter is present. See the Emitters topic in Section 9.2.2 for more details.
Initial Quality	Water quality level at the junction at the start of the simulation period. Can be left blank if no water quality analysis is being made or if the level is zero.
Source Quality	Quality of any water entering the network at this location. Click the ellipsis button (or hit the Enter key) to bring up the Source Quality Editor (see Section 9.2.6 below).

The reservoir properties are provided in **Table 9-9**.

Table 9-9: Reservoir properties

Property	Description
Reservoir ID	A unique label used to identify the reservoir. It can consist of a combination of up to 15 numerals or characters. It cannot be the same as the ID for any other node. This is a required property.
X-Coordinate	The horizontal location of the reservoir on the map, measured in the map's distance units. If left blank the reservoir will not appear on the network map.
Y-Coordinate	The vertical location of the reservoir on the map, measured in the map's distance units. If left blank the reservoir will not appear on the network map.
Description	An optional text string that describes other significant information about the reservoir.
Tag	An optional text string (with no spaces) used to assign the reservoir to a category, such as a pressure zone
Total Head	The hydraulic head (elevation + pressure head) of water in the reservoir in feet (meters). This is a required property.
Head Pattern	The ID label of a time pattern used to model time variation in the reservoir's head. Leave blank if none applies. This property is useful if the reservoir represents a tie-in to another system whose pressure varies with time.
Initial Quality	Water quality level at the reservoir. Can be left blank if no water quality analysis is being made or if the level is zero.
Source Quality	Quality of any water entering the network at this location. Click the ellipsis button (or hit the Enter key) to bring up the Source Quality Editor (**Figure 9-40**).

The tank properties are provided in **Table 9-10**.

Table 9-10: Tank properties

Property	Description
Tank ID	A unique label used to identify the tank. It can consist of a combination of up to 15 numerals or characters. It cannot be the same as the ID for any other node. This is a required property.
X-Coordinate	The horizontal location of the tank on the map, measured in the map's scaling units. If left blank the tank will not appear on the network map.
Y-Coordinate	The vertical location of the tank on the map, measured in the map's scaling units. If left blank the tank will not appear on the network map.
Description	Optional text string that describes other significant information about the tank.
Tag	Optional text string (with no spaces) used to assign the tank to a category, such as a pressure zone
Elevation	Elevation above a common datum in feet (meters) of the bottom shell of the tank. This is a required property.
Initial Level	Height in feet (meters) of the water surface above the bottom elevation of the tank at the start of the simulation. This is a required property.
Minimum Level	Minimum height in feet (meters) of the water surface above the bottom elevation that will be maintained. The tank will not be allowed to drop below this level. This is a required property.

Property	Description
Maximum Level	Maximum height in feet (meters) of the water surface above the bottom elevation that will be maintained. The tank will not be allowed to rise above this level. This is a required property.
Diameter	The diameter of the tank in feet (meters). For cylindrical tanks this is the actual diameter. For square or rectangular tanks, it can be an equivalent diameter equal to 1.128 times the square root of the cross-sectional area. For tanks whose geometry will be described by a curve (see below) it can be set to any value. This is a required property.
Minimum Volume	The volume of water in the tank when it is at its minimum level, in cubic feet (cubic meters). This is an optional property, useful mainly for describing the bottom geometry of non-cylindrical tanks where a full volume versus depth curve will not be supplied (see below).
Volume Curve	The ID label of a curve used to describe the relation between tank volume and water level. If no value is supplied then the tank is assumed to be cylindrical.
Mixing Model	The type of water quality mixing that occurs within the tank. The choices include - MIXED (fully mixed) - 2COMP (two-compartment mixing) - FIFO (first-in first-out plug flow) - LIFO (last-in first-out plug flow) See the Mixing Models Topic in Section 9.1.2 for more information.
Mixing Fraction	The fraction of the tank's total volume that comprises the inlet-outlet compartment of the two-compartment (2COMP) mixing model. Can be left blank if another type of mixing model is employed.
Reaction Coefficient	The bulk reaction coefficient for chemical reactions in the tank. Time units are 1/days. Use a positive value for growth reactions and a negative value for decay. Leave blank if the Global Bulk reaction coefficient specified in the project's Reactions Options will apply. See Water Quality Reactions in **Table 9-24** for more information.
Initial Quality	Water quality level in the tank at the start of the simulation. Can be left blank if no water quality analysis is being made or if the level is zero.
Source Quality	Quality of any water entering the network at this location. Click the ellipsis button (or hit the Enter key) to bring up the Source Quality Editor (see **Figure 9-40**).

The pipe properties are provided in **Table 9-11**.

Table 9-11: Pipe properties

Property	Description
Pipe ID	A unique label used to identify the pipe. It can consist of a combination of up to 15 numerals or characters. It cannot be the same as the ID for any other link. This is a required property.
Start Node	The ID of the node where the pipe begins. This is a required property.
End Node	The ID of the node where the pipe ends. This is a required property.
Description	An optional text string that describes other significant information about the pipe.
Tag	An optional text string (with no spaces) used to assign the pipe to a

Property	Description
	category, perhaps one based on age or material.
Length	The actual length of the pipe in feet (meters). This is a required property.
Diameter	The pipe diameter in inches (mm). This is a required property.
Roughness	The roughness coefficient of the pipe. It is unitless for Hazen-Williams or Chezy-Manning roughness and has units of millifeet (mm) for Darcy-Weisbach roughness. This is a required property.
Loss Coefficient	Unitless minor loss coefficient associated with bends, fittings, etc. Assumed 0 if left blank.
Initial Status	Determines whether the pipe is initially open, closed, or contains a check valve. If a check valve is specified then the flow direction in the pipe will always be from the Start node to the End node.
Bulk Coefficient	The bulk reaction coefficient for the pipe. Time units are 1/days. Use a positive value for growth and a negative value for decay. Leave blank if the Global Bulk reaction coefficient from the project's Reaction Options will apply. See Water Quality Reactions in **Table 9-24** for more information.
Wall Coefficient	The wall reaction coefficient for the pipe. Time units are 1/days. Use a positive value for growth and a negative value for decay. Leave blank if the Global Wall reaction coefficient from the project's Reactions Options will apply. See Water Quality Reactions in **Table 9-24** for more information.

Note:
Pipe lengths can be automatically computed as pipes are added or repositioned on the Viewer Map. If the Auto-Length setting is turned on. Toggle this setting On/Off, by clicking the *Options* toolbar button and *Checking*/*Unchecking* the Auto-length (see **Figure 7-12**). Be sure to provide meaningful dimensions for the Viewer Map before using the Auto-Length feature.

The pump properties are provided in **Table 9-12**.

Table 9-12: Pump properties

Property	Description
Pump ID	A unique label used to identify the pump. It can consist of a combination of up to 15 numerals or characters. It cannot be the same as the ID for any other link. This is a required property.
Start Node	The ID of the node on the suction side of the pump. This is a required property.
End Node	The ID of the node on the discharge side of the pump. This is a required property.
Description	An optional text string that describes other significant information about the pump.
Tag	An optional text string (with no spaces) used to assign the pump to a category, perhaps based on age, size or location.
Pump Curve	The ID label of the pump curve used to describe the relationship between the head delivered by the pump and the flow through the pump. Leave blank if the pump will be a constant energy pump (see below).

Property	Description
Power	The power supplied by the pump in horsepower (kw). Assumes that the pump supplies the same amount of energy no matter what the flow is. Leave blank if a pump curve will be used instead. Use when pump curve information is not available.
Speed	The relative speed setting of the pump (unitless). For example, a speed setting of 1.2 implies that the rotational speed of the pump is 20% higher than the normal setting.
Pattern	The ID label of a time pattern used to control the pump's operation. The multipliers of the pattern are equivalent to speed settings. A multiplier of zero implies that the pump will be shut off during the corresponding time period. Leave blank if not applicable.
Initial Status	State of the pump (open or closed) at the start of the simulation period.
Efficiency Curve	The ID label of the curve that represents the pump's wire-to-water efficiency (in percent) as a function of flow rate. This information is used only to compute energy usage. Leave blank if not applicable or if the global pump efficiency supplied with the project's Energy Options (see **Table 9-25**) will be used.
Energy Price	The average or nominal price of energy in monetary units per kW-hr. Used only for computing the cost of energy usage. Leave blank if not applicable or if the global value supplied with the project's Energy Options (**Table 9-25**) will be used.
Price Pattern	The ID label of the time pattern used to describe the variation in energy price throughout the day. Each multiplier in the pattern is applied to the pump's Energy Price to determine a time-of-day pricing for the corresponding period. Leave blank if not applicable or if the global pricing pattern specified in the project's Energy Options (**Table 9-25**) will be used.

The valve properties are provided in **Table 9-13**.

Table 9-13: Valve properties

Property	Description
ID Label	A unique label used to identify the valve. It can consist of a combination of up to 15 numerals or characters. It cannot be the same as the ID for any other link. This is a required property.
Start Node	The ID of the node on the nominal upstream or inflow side of the valve. (PRVs and PSVs maintain flow in only a single direction.) This is a required property.
End Node	The ID of the node on the nominal downstream or discharge side of the valve. This is a required property.
Description	An optional text string that describes other significant information about the valve.
Tag	An optional text string (with no spaces) used to assign the valve to a category, perhaps based on type or location.
Diameter	The valve diameter in inches (mm). This is a required property.
Type	The valve type (PRV, PSV, PBV, FCV, TCV, or GPV). See **Table 9-6** for descriptions of the various types of valves. This is a required property.

Property	Description
Setting	A required parameter for each valve type that describes its operational setting: PRV - Pressure (psi or m) PSV - Pressure (psi or m) PBV - Pressure (psi or m) FCV - Flow (flow units) TCV - Loss Coeff (unitless) GPV - ID of head loss curve
Loss Coefficient	Unitless minor loss coefficient that applies when the valve is completely opened. Assumed 0 if left blank.
Fixed Status	Valve status at the start of the simulation. If set to OPEN or CLOSED then the control setting of the valve is ignored and the valve behaves as an open or closed link, respectively. If set to NONE, then the valve will behave as intended. A valve's fixed status and its setting can be made to vary throughout a simulation by the use of control statements. If a valve's status was fixed to OPEN/CLOSED, then it can be made active again using a control that assigns a new numerical setting to it.

The label properties are provided in **Table 9-14**.

Table 9-14: Label properties

Property	Description
Caption	The label's text.
X-Coordinate	The horizontal location of the upper left corner of the label on the map, measured in the map's scaling units. This is a required property.
Y-Coordinate	The vertical location of the upper left corner of the label on the map, measured in the map's scaling units. This is a required property.
Font	Launches a Font dialog that allows selection of the label's font, size, and style.

9.2.6 Editing Non-Visual Objects

Curves, Time Patterns, Controls and Calibration have special editors that are used to define their properties. To edit one of these objects, from the Settings Toolbar click on the object (see **Figure 9-38**) to edit.

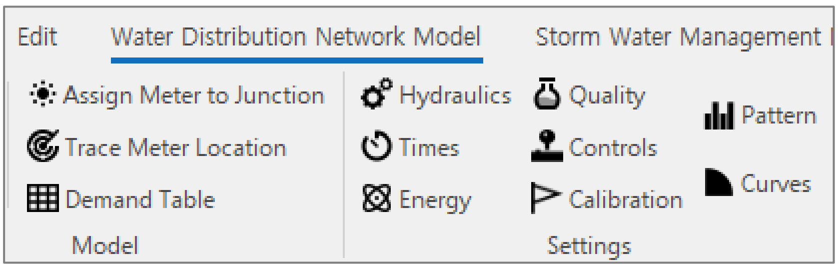


Figure 9-38: Non-visual objects settings

In addition, the Property Editor for Junctions contains an ellipsis button in the field for Demand Categories that brings up a special Demand Editor when clicked (**Figure 9-39**).

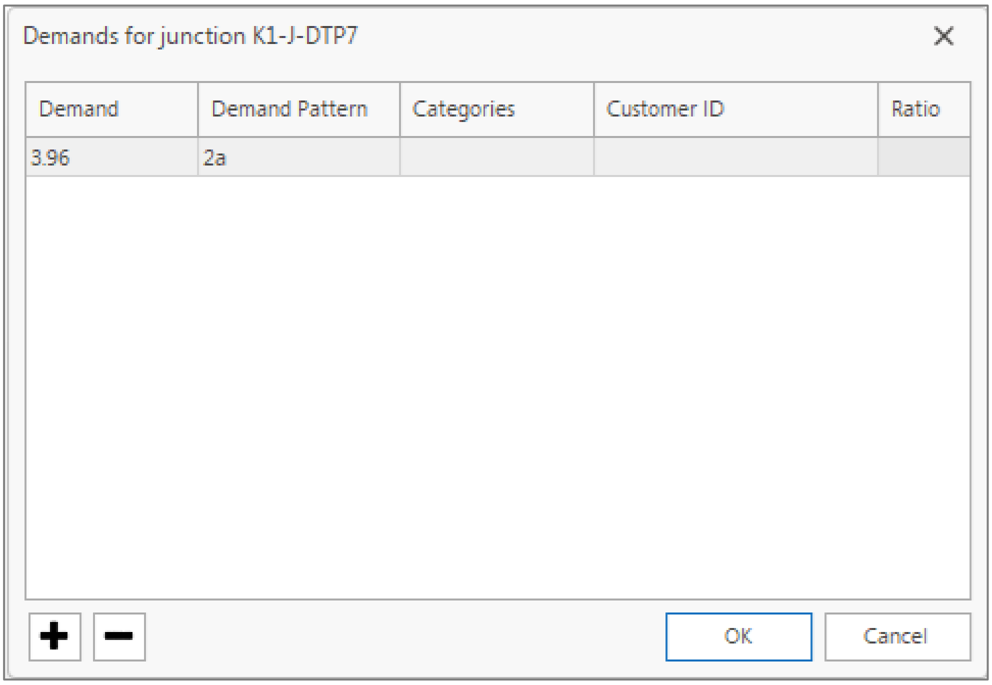


Figure 9-39: Demand Categories Editor

Similarly, the Source Quality field in the Property Editor for Junctions, Reservoirs, and Tanks has a button that launches a special Source Quality Editor (**Figure 9-40**). Each of these specialized editors is described next.

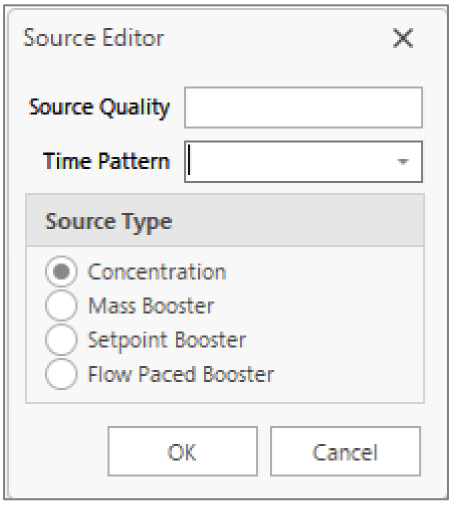
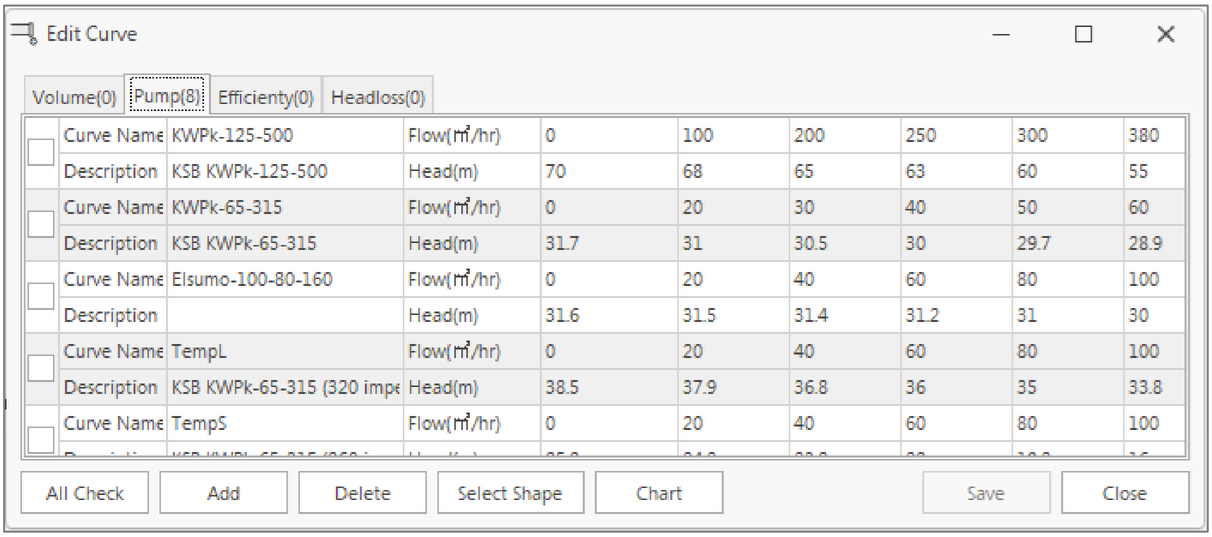


Figure 9-40: Source Quality Editor

Curves Editor

The Curves Editor is a dialog form as shown in **Figure 9-41.** To use the Curve Editor, enter values for the items as listed in **Table 9-15**. The Curve Editor can be used to enter a Volume, Pump, Efficiency or Headloss curve by simply selecting the tab at the top of the dialog form.



Curve Name	KWPk-125-500	Flow(m³/hr)	0	100	200	250	300	380
Description	KSB KWPk-125-500	Head(m)	70	68	65	63	60	55
Curve Name	KWPk-65-315	Flow(m³/hr)	0	20	30	40	50	60
Description	KSB KWPk-65-315	Head(m)	31.7	31	30.5	30	29.7	28.9
Curve Name	Elsumo-100-80-160	Flow(m³/hr)	0	20	40	60	80	100
Description		Head(m)	31.6	31.5	31.4	31.2	31	30
Curve Name	TempL	Flow(m³/hr)	0	20	40	60	80	100
Description	KSB KWPk-65-315 (320 impe	Head(m)	38.5	37.9	36.8	36	35	33.8
Curve Name	TempS	Flow(m³/hr)	0	20	40	60	80	100

Figure 9-41: Curves Editor

A Volume Curve determines how storage tank volume (Y) varies as a function of water level (X). It is used when it is necessary to accurately represent tanks whose cross-sectional area varies with height. The lower and upper water levels supplied for the curve must contain the lower and upper levels between which the tank operates.

A Pump Curve represents the relationship between the head and flow rate that a pump can deliver at its nominal speed setting. Head is the head gain imparted to the water by the pump and is plotted on the vertical (Y) axis of the curve in head units. Flow rate is plotted on the horizontal (X) axis in flow units. A valid pump curve must have decreasing head with increasing flow.

An Efficiency Curve, only used for energy calculations, determines pump efficiency (Y in percent) as a function of pump flow rate (X in flow units). Efficiency should represent wire-to-water efficiency that takes into account mechanical losses in the pump itself and the electrical losses in the pump's motor. If not supplied for a specific pump, then a fixed global pump efficiency will be used.
A Headloss Curve is used to described the headloss (Y) through a General-Purpose Valve (GPV) as a function of flow rate (X). It provides the capability to model devices and situations with unique headloss-flow relationships, such as reduced flow - backflow prevention valves, turbines, and well draw-down behaviour.

Select the curve that needs to be entered (by selecting the tab at the top of the dialogue form) and click on the *Add* button to generate a blank curve that can be populated. Similarly, a specific curve can be selected (checking the box next to the curve) and click on the *Delete* button. Selecting a curve or multiple curves (checking the box) and clicking on the *Chart* button will plot the selected curve(s) as depicted in **Figure 9-42**, an example of a pump curve. Click on the *Save* button to save the entered curves and click on the *Close* button to close the Edit Curve Editor.

Table 9-15: Curves Editor

Item	Description
	Volume
Curve name	A unique name that can be used to refer to this specific curve
Description	Optional description of what the curve represents
Height	The height (units is linked to the selected flow units)
Volume	The volume (units is linked to the selected flow units)
	Pump
Curve name	ID label of the curve (maximum of 15 numerals or characters)
Description	Optional description of what the curve represents
Curve Type	Type of curve
Head	The pumping head, corresponding to a specific flow, i.e., X-Y point on the curve (units is linked to the selected flow units)
Flow	The flow, corresponding to a specific pumping head, i.e., X-Y point on the curve (units is linked to the selected flow units)
	Efficiency
Curve name	A unique name that can be used to refer to this specific curve
Description	Optional description of what the curve represents
Flow	The flow rate, corresponding to a specific efficiency (units is linked to the selected flow units)
Efficiency	The efficiency, corresponding to a specific flow rate (units is linked to the selected flow units)
	Headloss
Curve name	A unique name that can be used to refer to this specific curve
Description	Optional description of what the curve represents
Flow	The flow rate, corresponding to a specific headloss (units is linked to the selected flow units)
Headloss	The headloss, corresponding to a specific flow rate (units is linked to the selected flow units)

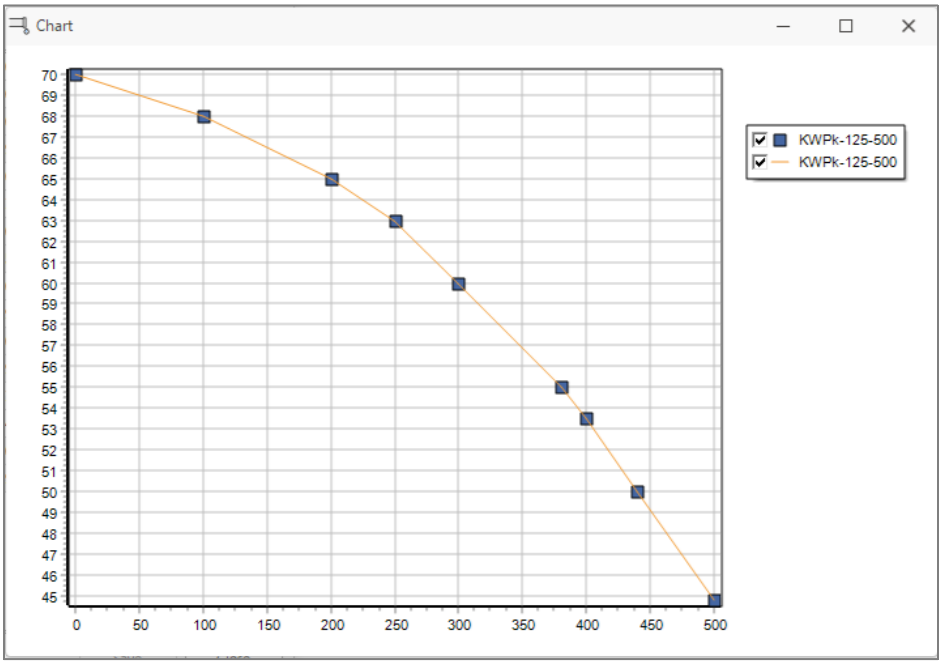


Figure 9-42: Chart of pump curve

Pattern Editor

A Time Pattern is a collection of multipliers that can be applied to a quantity to allow it to vary over time. Nodal demands, reservoir heads, pump schedules, and water quality source inputs can all have time patterns associated with them. The time interval used in all patterns is a fixed value, set with the project's Time Options. Within this interval a quantity remains at a constant level, equal to the product of its nominal value and the pattern's multiplier for that time period. Although all time patterns must utilize the same time interval, each can have a different number of periods. When the simulation clock exceeds the number of periods in a pattern, the pattern wraps around to its first period again.

The Pattern Editor, displayed in **Figure 9-43**, edits the properties of a time pattern object. To use the Pattern Editor, enter values for the items as listed in **Table 9-16**. A small chart is drawn above the column for each entered pattern, see **Figure 9-43**.

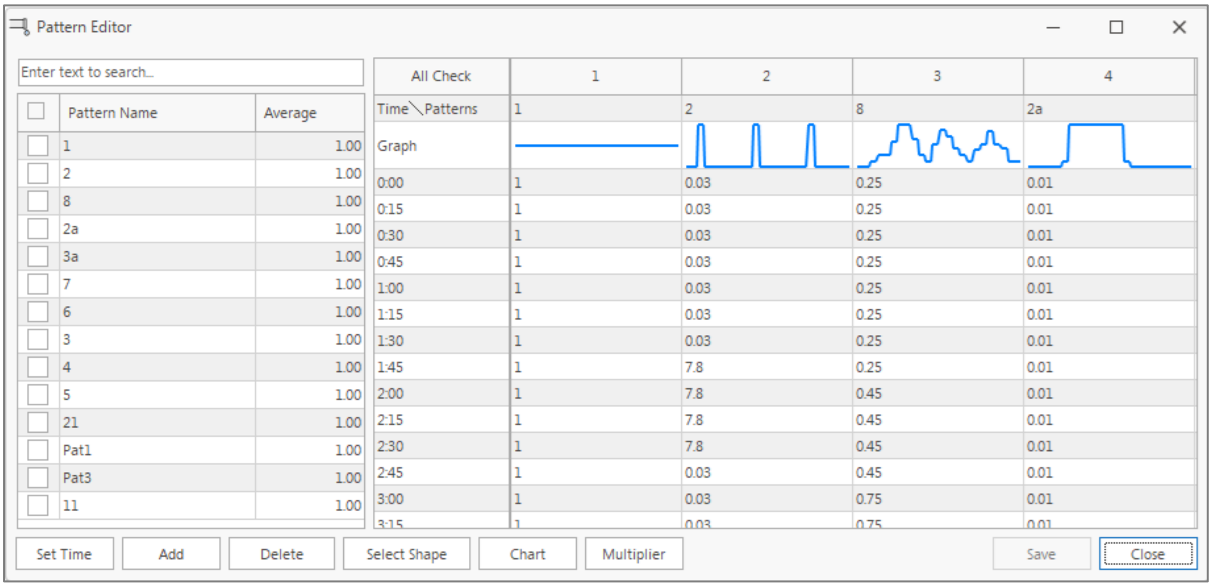


Figure 9-43: Pattern Editor

Table 9-16: Pattern Editor

Item	Description
Pattern Name	Unique name of the time pattern.
Description	Optional description of what the pattern represents.
Multipliers	Multiplier value for each time period of the pattern.

Click on the *Set Time* button (**Figure 9-43**) to set the *Duration*, *Start Time* and *Time Step* as shown in **Figure 9-44**. Click on the *Add* button to generate a blank pattern (column) that can be populated. Similarly, a specific pattern can be selected (checking the box next to the Pattern Name) and click on the *Delete* button.

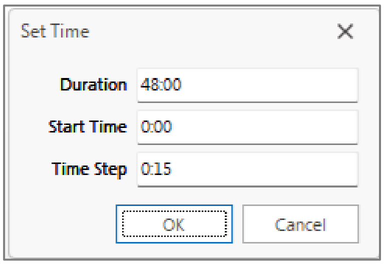


Figure 9-44: Set time dialogue form

Selecting a pattern or multiple patterns (checking the box next to the Pattern Name) and clicking on the *Chart* button will plot the selected pattern(s) as depicted in **Figure 9-45**, an example of a pump curve. Click on the *Save* button to save the entered curves and click on the *Close* button to close the Time Pattern Editor.

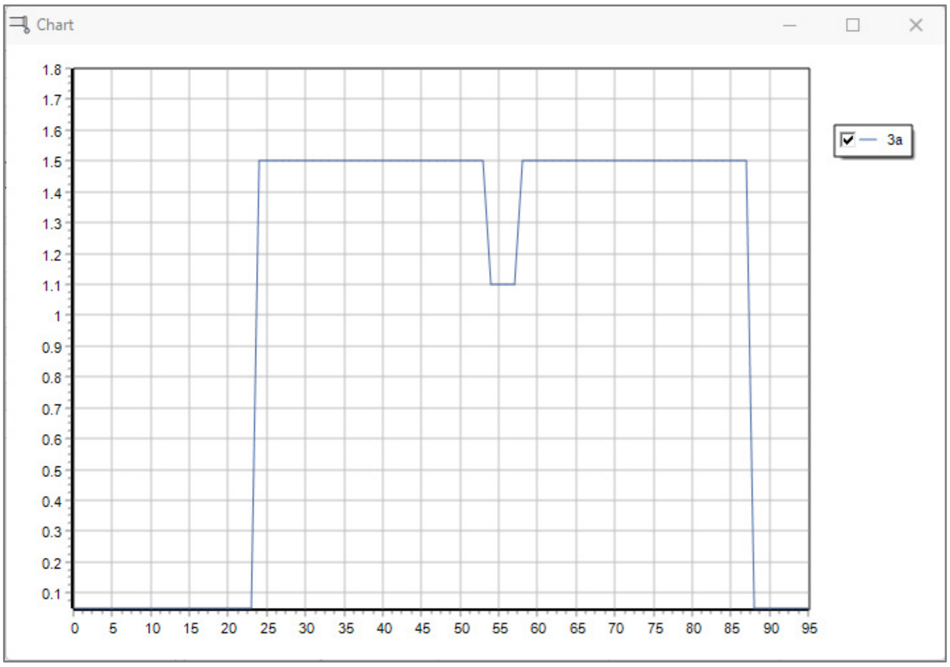


Figure 9-45: Time pattern chart

Right click anywhere on the Chart (**Figure 9-45**) will show the various options to modify the General look (colours, views and title), Horizontal Axis (minimum, maximum, increments, grid lines & title), Vertical Axis (minimum, maximum, increments, grid lines & title), Legend (position, colour, size, framed & visibility) and Series (series, title, symbology of lines, markers and labels) as depicted in **Figure 9-46**.

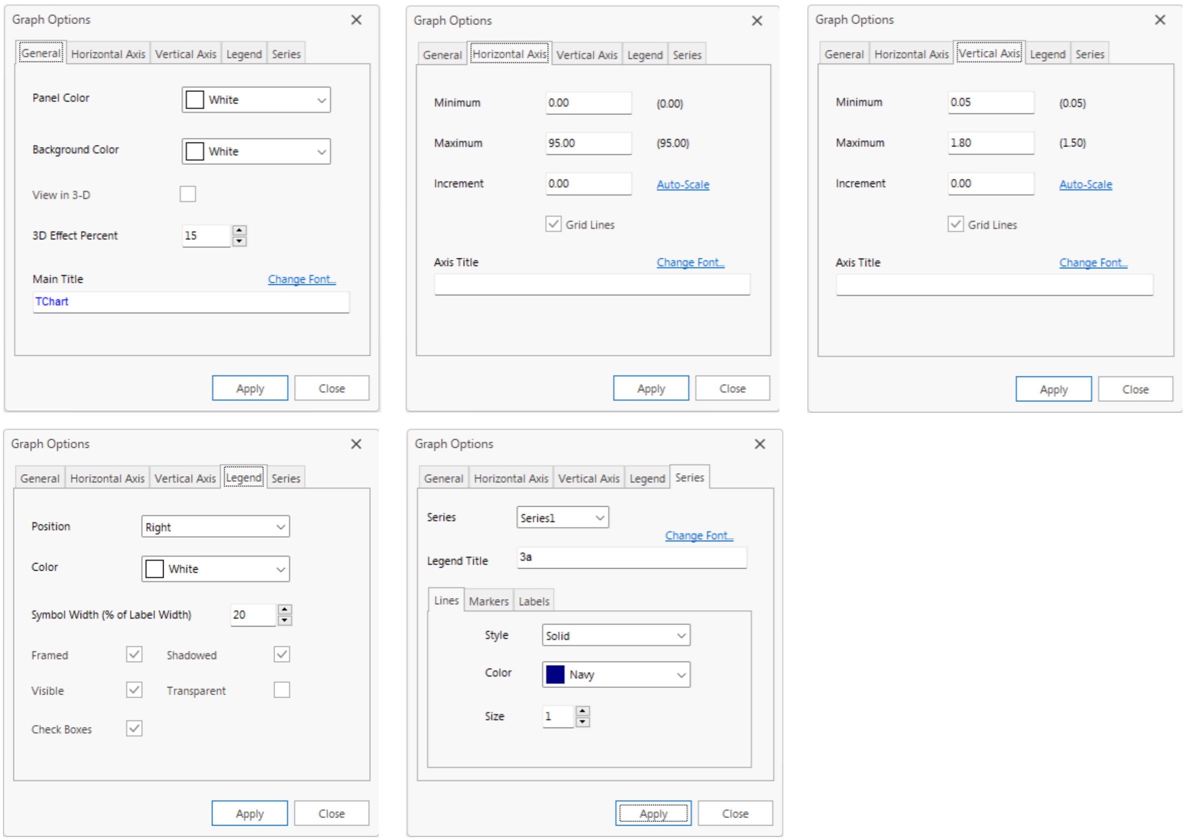


Figure 9-46: Graph options

Click the *Save* button to save the current pattern's data to a file.

Controls Editor

The Controls Editor, shown in **Figure 9-47**, is a text editor window used to edit both simple and rule-based controls.

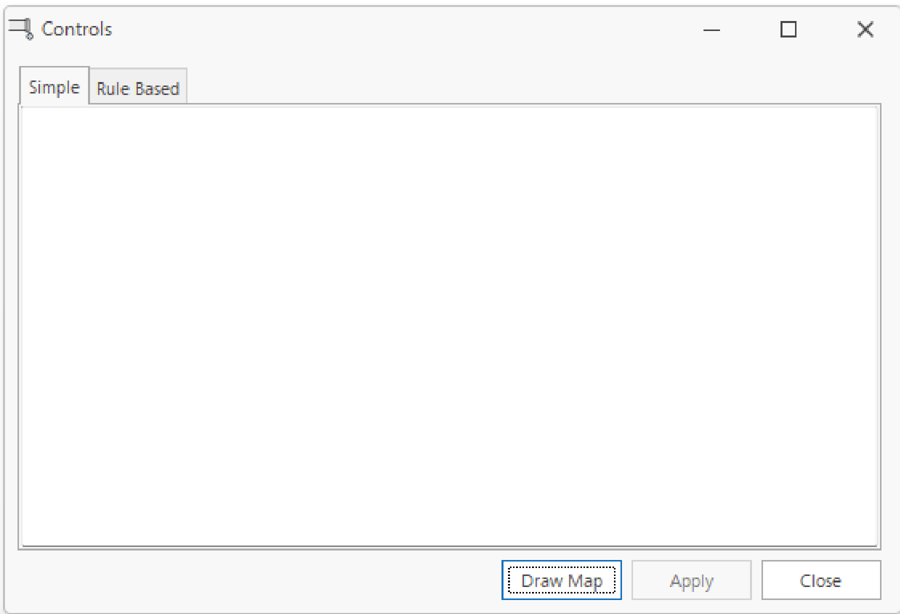


Figure 9-47: Control Editor

Controls are statements that determine how the network is operated over time. They specify the status of selected links as a function of time, tank water levels, and pressures at select points within the network. There are two categories of controls that can be used:

- Simple Controls
- Rule-Based Controls Simple Controls

Simple Controls

Simple controls change the status or setting of a link based on:
• The water level in a tank
• The pressure at a junction
• The time into the simulation
• The time of day
They are statements expressed in one of the following three formats:

LINK	x	status	IF	NODE	y	ABOVE/BELOW	z
LINK	x	status	AT	TIME	t		
LINK	x	status	AT	CLOCKTIME	c	AM/PM	

where:
x = a link ID label
status = OPEN or CLOSED, a pump speed setting, or a control valve setting
y = a node ID label
z = a pressure for a junction or a water level for a tank
t = a time since the start of the simulation (decimal hours or hours:minutes)
c = a 24-hour clock time

There is no limit on the number of simple control statements that can be used.
Note: Level controls are stated in terms of the height of water above the tank bottom, not the elevation (total head) of the water surface.

Note: Using a pair of pressure controls to open and close a link can cause the system to become unstable if the pressure settings are too close to one another. In this case using a pair of Rule-Based controls might provide more stability.

Rule-Based Controls

Rule-Based Controls allow link status and settings to be based on a combination of conditions that might exist in the network after an initial hydraulic state of the system is computed. Here is an example of a Rule-Based Control. This set of rules shuts down a pump and opens a by-pass pipe when the level in a tank exceeds a certain value and does the opposite when the level is below another value.

```
RULE 1
    IF TANK 2 LEVEL ABOVE 6.0
    THEN PUMP 10 STATUS IS CLOSED
    AND PIPE 55 STATUS IS OPEN
RULE 2
    IF TANK 2 LEVEL BELOW 3.9
    THEN PUMP 10 STATUS IS OPEN
    AND PIPE 55 STATUS IS CLOSED
```

Source Quality Editor

The Source Quality Editor is a pop-up dialog used to describe the quality of source flow entering the network at a specific node. This source might represent the main treatment works, a well head or satellite treatment facility, or an unwanted contaminant intrusion. The dialog form, shown in **Figure 9-48**, contains the following fields (**Table 9-17**).

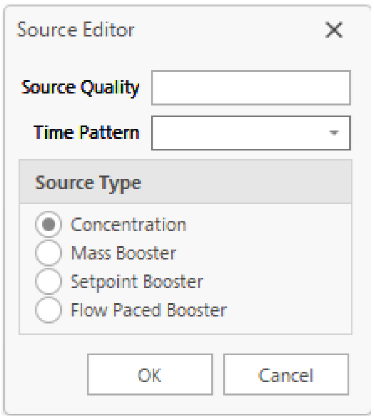


Figure 9-48: Source Editor

Table 9-17: Source quality parameters

Field	Description
Source Quality	Baseline or average concentration (or mass flow rate per minute) of source – leave blank to remove the source.
Quality Pattern	ID label of time pattern used to make source quality vary with time, leave blank if not applicable.
Source Type	Select either: - Concentration - Mass Booster - Setpoint Booster - Flow Paced Booster

A water quality source can be designated as a concentration or booster source.

- A concentration source fixes the concentration of any external inflow entering the network, such as flow from a reservoir or from a negative demand placed at a junction.
- A mass booster source adds a fixed mass flow to that entering the node from other points in the network.
- A flow paced booster source adds a fixed concentration to that resulting from the mixing of all inflow to the node from other points in the network.
- A setpoint booster source fixes the concentration of any flow leaving the node (as long as the concentration resulting from all inflow to the node is below the setpoint).

The concentration-type source is best used for nodes that represent source water supplies or treatment works (e.g., reservoirs or nodes assigned a negative demand). The booster-type source is best used to model direct injection of a tracer or additional disinfectant into the network or to model a contaminant intrusion.

9.2.7 Other modelling parameters

GISpipe provides additional useful functionalities such as assigning a meter to a junction, tracing meter location and viewing/editing of the demand table. These functionalities can be accessed from the Water Distribution Network Model Toolbar as depicted in **Figure 9-49**.

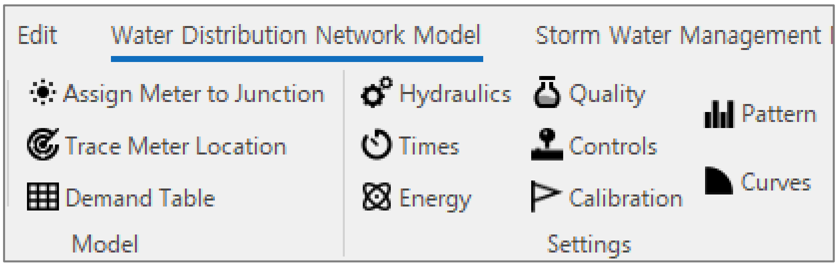


Figure 9-49: Water Distribution Network Model

Assign meter to junction

This dialog box, **Figure 9-50**, is for assigning water meter points (from the GIS water meter layer) to junctions in the water distribution model. This step links customer demand (from meters) to the hydraulic model, see **Table 9-18**.

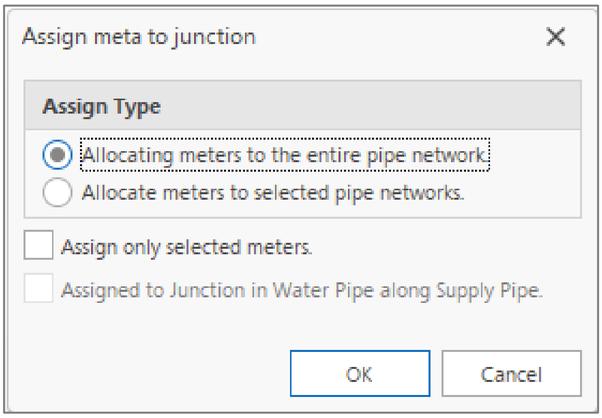


Figure 9-50: Assign meter to a junction

Table 9-18: Assign meters to junctions

Item	Description
Assign type	
Allocating meters to the entire pipe network	All meters in the meter layer are assigned automatically to the nearest junctions in the full network. Use this when you want a complete allocation across the model.
Allocate meters to selected pipe networks	Only the meters within the user-selected pipe network(s) are allocated to the corresponding junctions. Use this when working on a sub-network or when testing a smaller area.
Additional options	
Assign only selected meters	Instead of allocating all meters in the layer, only the currently selected subset of meters will be assigned to junctions. This is useful when manually controlling which meters are linked.
Assigned to Junction in Water Pipe along Supply Pipe	Ensures that meters are linked to a junction that lies directly along the supplying pipe (rather than just the nearest junction by distance). This is a more hydraulically correct allocation when working with complex networks.

Trace meter location

The Trace meter location dialog window is shown **Figure 9-51**. This function allows the user to see which meters and demands are connected to which junction and allows the user to drag and drop the meter to the desired junction which changes the assignment to the new junction, see **Figure 9-52**.

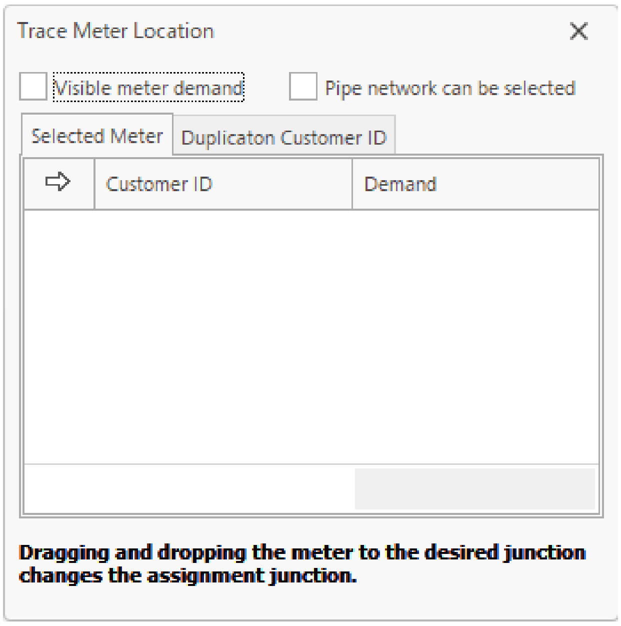


Figure 9-51: Trace meter location dialog window

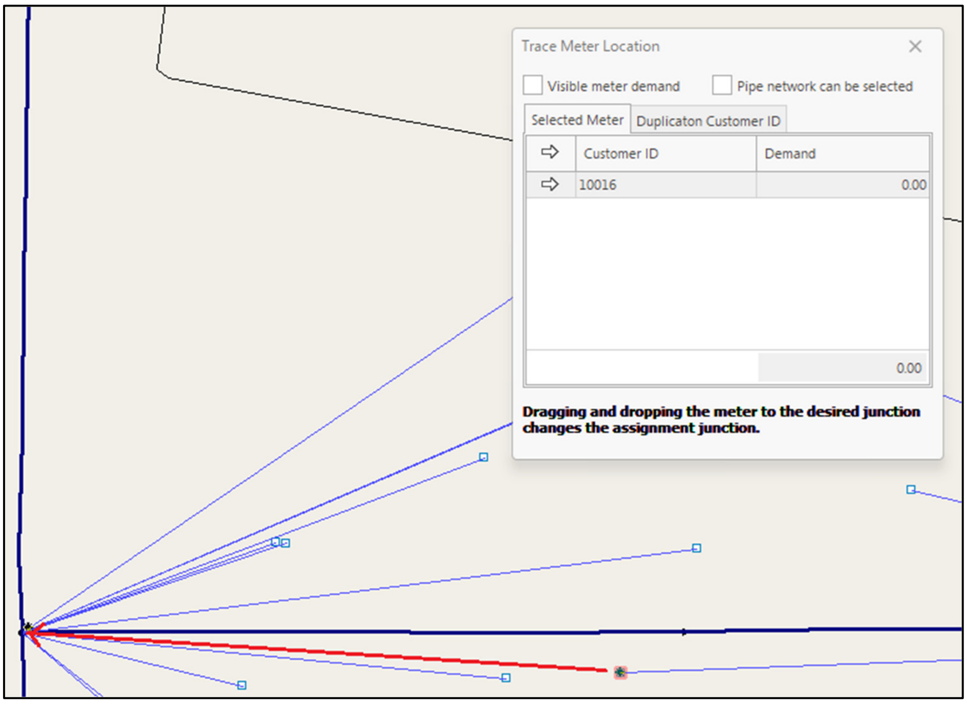


Figure 9-52: Dragging and dropping the meter onto new junction

Demand Editor

The Demand Editor is shown in **Figure 9-53**. It is used to assign base demands and time patterns when there is more than one category of water user at a junction. The editor is invoked from the Property Editor by clicking the ellipsis button (or hitting the *Enter* key) when the Demand Categories field has the focus.

Figure 9-53: Demand Table Editor

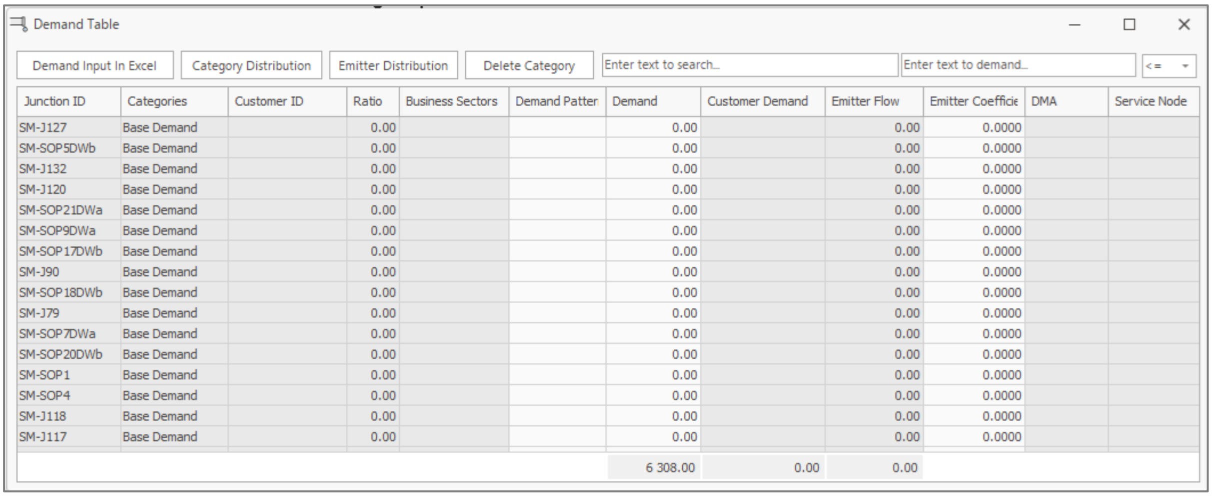
The editor is a table containing five columns. Each category of demand is entered as a new row in the table. Rows are added or deleted by clicking on the  or  buttons. The columns contain the following information:

- Demand: baseline or average demand for the category (required)
- Demand Pattern: Pattern name (can be s3eletced from drop down list, as show in Figure 9-53). This is used to allow demand to vary with time (optional).
- Categories: text label used to identify the demand category (optional)
- Customer ID: Used to define the category/user.
- Ratio: The ratio of the demand of each customer of the total demand at the specific junction.

Note: By convention, the demand placed in the first row of the editor will be considered the main category for the junction and will appear in the Base Demand field of the Property Editor.

A summary of all the Demands and Categories, Demand patterns linked to every junction is shown by clicking on the *Demand Table* toolbar button (**Figure 9-38**). This will provide an overview of all the demands, patterns, categories, emitter flows, emitter coefficients, District Metered Areas (DMA) as depicted in **Figure 9-54**.

This Demand Table provides advanced functionality allowing demand values to be imported from Microsoft Excel (**Figure 9-55**), click on the *Demand Input in Excel* button, change the viewed values by clicking on the *Category Distribution* button and setting the preference (**Figure 9-56**) to filter the table.



Junction ID	Categories	Customer ID	Ratio	Business Sectors	Demand Patter	Demand	Customer Demand	Emitter Flow	Emitter Coefficie	DMA	Service Node
SM-J127	Base Demand		0.00			0.00		0.00	0.0000		
SM-SOP5DWb	Base Demand		0.00			0.00		0.00	0.0000		
SM-J132	Base Demand		0.00			0.00		0.00	0.0000		
SM-J120	Base Demand		0.00			0.00		0.00	0.0000		
SM-SOP21DWa	Base Demand		0.00			0.00		0.00	0.0000		
SM-SOP9DWa	Base Demand		0.00			0.00		0.00	0.0000		
SM-SOP17DWb	Base Demand		0.00			0.00		0.00	0.0000		
SM-J90	Base Demand		0.00			0.00		0.00	0.0000		
SM-SOP18DWb	Base Demand		0.00			0.00		0.00	0.0000		
SM-J79	Base Demand		0.00			0.00		0.00	0.0000		
SM-SOP7DWa	Base Demand		0.00			0.00		0.00	0.0000		
SM-SOP20DWb	Base Demand		0.00			0.00		0.00	0.0000		
SM-SOP1	Base Demand		0.00			0.00		0.00	0.0000		
SM-SOP4	Base Demand		0.00			0.00		0.00	0.0000		
SM-J118	Base Demand		0.00			0.00		0.00	0.0000		
SM-J117	Base Demand		0.00			0.00		0.00	0.0000		
						6 308.00	0.00	0.00			

Figure 9-54: Demand Table

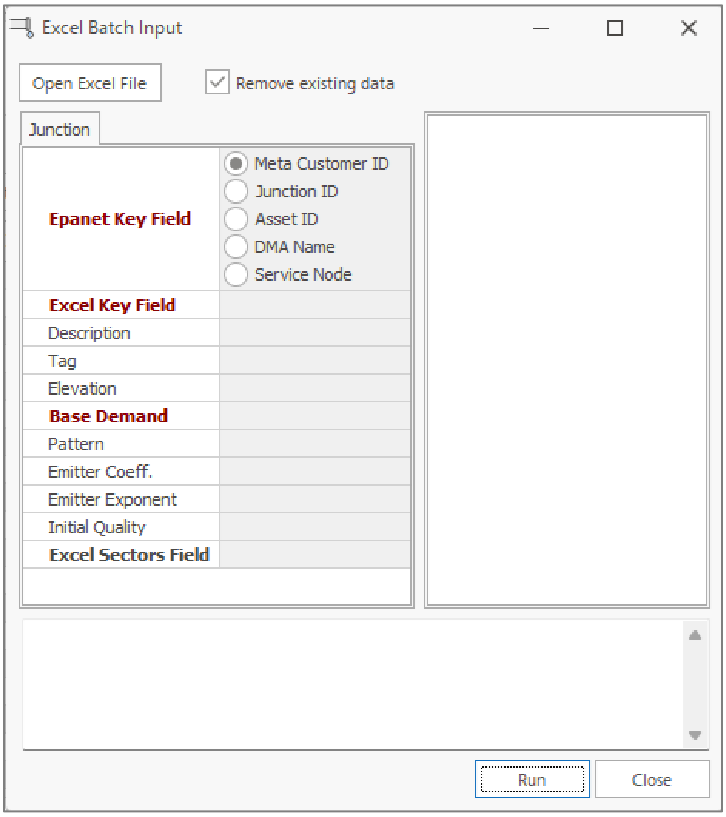


Figure 9-55: Excel Batch Input

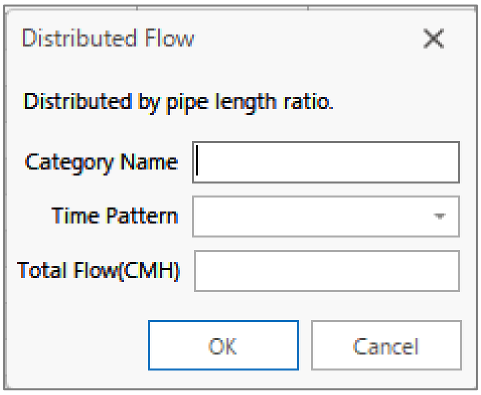


Figure 9-56: Category Distribution

9.3 Editing Tools

9.3.1 Map Tools and Navigation

Undo and Redo

This is one of the most useful functions in GISpipe to assist the user to edit the network. Simply click on the buttons above the menu bar to undo or redo the last step.

Undo 

Redo 

Options

GISpipe allows the setting of some of the general preferences of the program by clicking on the *Options* button on the Toolbar, which will display **Figure 9-57** with descriptions of the meaning of theses settings provided in **Table 9-19**.

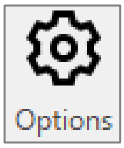


Table 9-19: Description of Option settings

Option	Description
Use snap feature	When drawing a pipe, snap may help the cursor lock to an existing node or pipe, ensuring alignment of elements in the network model. It makes network building more accurate and efficient.
Show delete objects dialog	When enabled, the software will display a pop-up dialog asking the user to confirm before deleting selected objects. This dialog indicates the objects to be deleted and offer the options "Yes" and "No". When disabled, objects are deleted immediately without confirmation, which may speed up workflows but increases the risk of accidental loss.
Node elevation closest to node elevation	Automatically assigns a node's elevation based on the elevation of the nearest existing node. Useful when importing or creating nodes without specified elevations. It automatically fills in missing elevation data and thus maintains topographic continuity in a model when elevation data is incomplete.
Auto length	If the Auto length setting is turned (Check box is ticked) then drawing a pipe link will automatically determine the length of that pipe based on the scale of the map. Be sure to provide meaningful dimensions for the Viewer Map before using the Auto-Length feature.

Option	Description
Scroll Mode	The user has 3 options: - Classic: Uses standard mouse-based scrolling. Scrollbars behave traditionally; users click and drag the scrollbar or use the scroll wheel. - Touch: Optimized for touch input. Scrolls like a mobile/tablet interface, swiping moves the view, often with inertia (momentum). Scrollbars may be hidden or overlaid. - Hybrid: Combines both Classic and Touch modes. It detects the input method (mouse or touch) and adapts accordingly.
Cursor	The Selection Precision, Cursor Size and Selection Colour can be set
Map Background Colour	One of two options can be selected *White* or *Black* by selecting the radio button

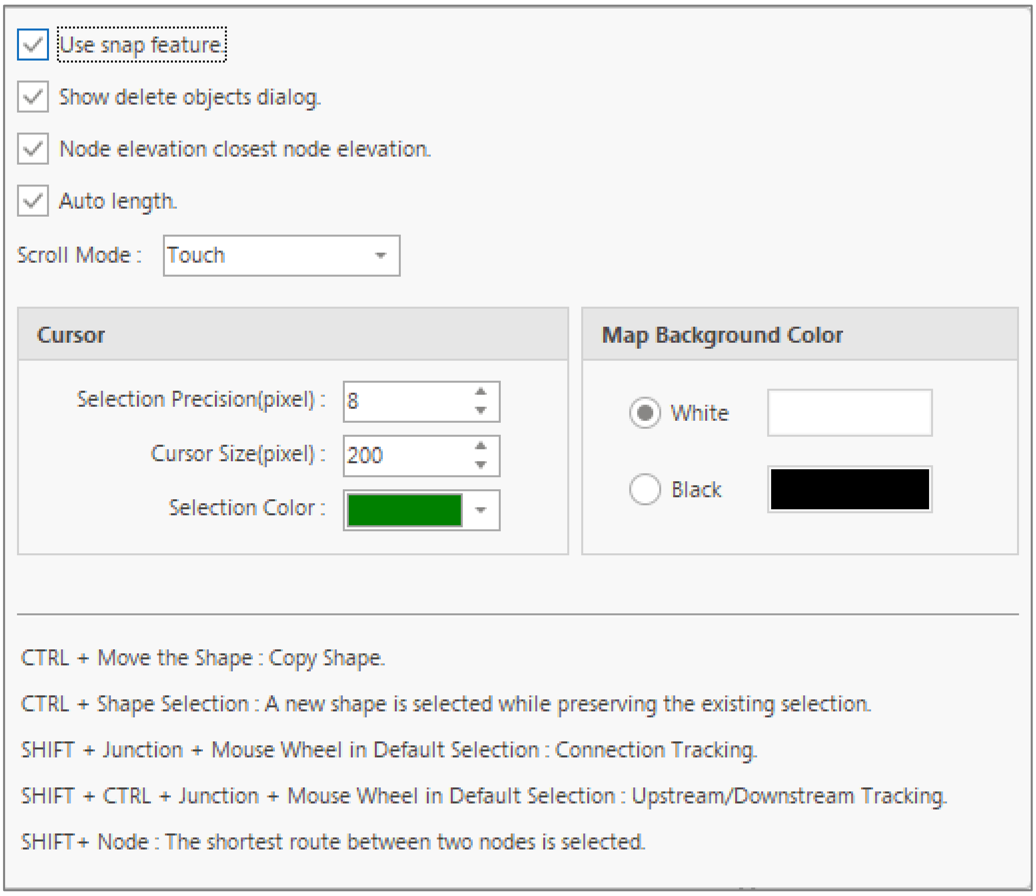
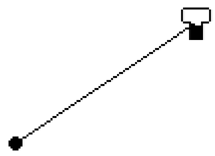
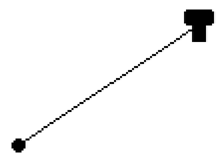
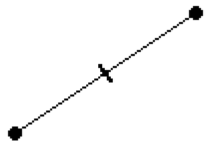
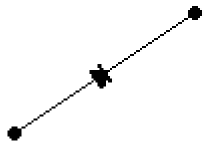
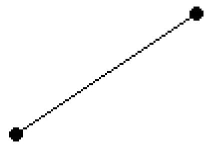
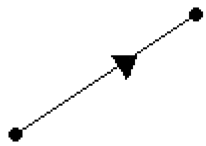
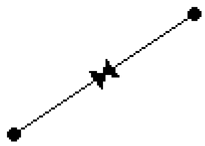
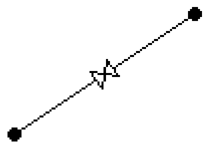
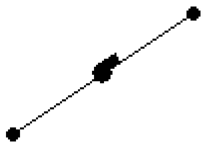
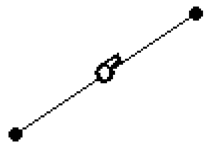


Figure 9-57: Options

Table 9-20: Object's statuses on the map

A tank which shows its status (50% full)			A tank which shows its status (100% full)	
A pipe which is closed on the map screen			A pipe which also functions as a check valve	
A pipe which is open on the map screen			A pipe with flow arrow indicating the direction of flow.	
A valve which is in the open position. A control or its function could change this status during an extended period analysis.			A valve which is in the closed position. A control or its function could change this status during an extended period analysis.	
A pump which is in the open position. A control or its function could change this status during an extended period analysis.			A pump which is in the closed position. A control or its function could change this status during an extended period analysis.	

Junction removal

Select the junction J2 as shown in **Figure 9-58** and drag it towards junction J3 (by holding in left-mouse button). When releasing the mouse button once on the point, junction J2 would be deleted, pipe P2 will be deleted and Junction J1 will now be directly joined to junction J3 with pipe P1.

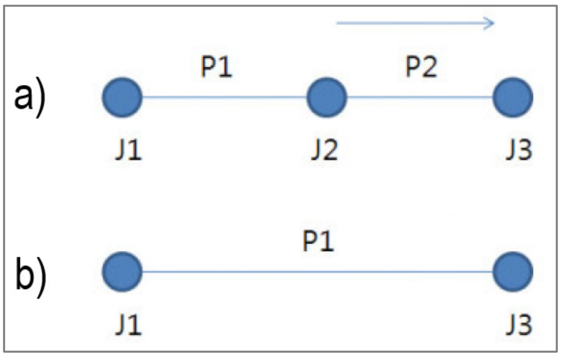


Figure 9-58: Removing a junction

Junction merging

Select junction J3 and drag it towards junction J5 (by holding in left-mouse button), see **Figure 9-59**. When releasing the mouse button once on the point, junction J3 would be deleted and pipe P2 will now be connected to junction J5.

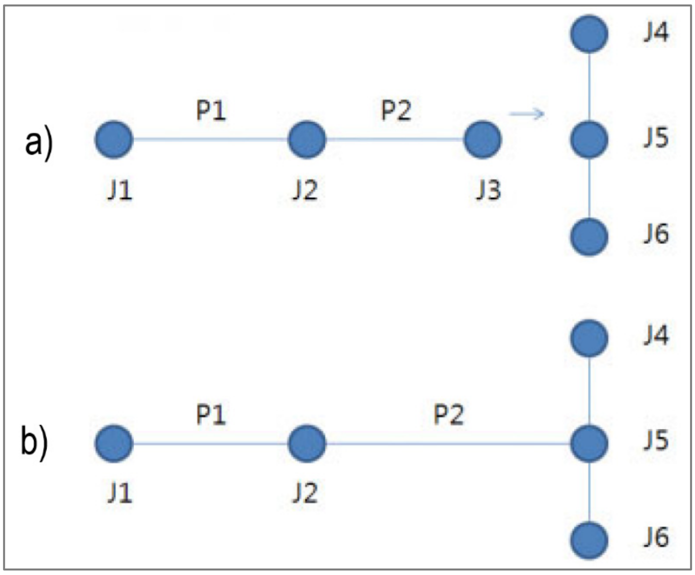


Figure 9-59: Merging junctions

Clipboard function

The Copy, Cut, Paste functionalities can be selected form the Clipboard toolbar, **Figure 9-60**.

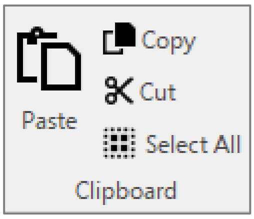


Figure 9-60: Clipboard functions

Alternatively selecting any object or group of objects and pressing *Ctrl + C* on the keyboard copies these objects to the clipboard.

Pressing *Ctrl + V* pastes these objects.

These functions can be used between different layers and the objects will be copied onto the actively selected layer.

9.4 Water Distribution Network Model Settings

The Water Distribution Network Model Settings provide the foundational parameters required to simulate the hydraulic and water quality behaviour of a water supply system. These settings govern aspects such as simulation time steps, flow routing, pressure constraints, and quality analysis options, ensuring that the model accurately reflects real-world conditions and operates efficiently under various scenarios. Proper configuration of these settings is essential for reliable results and meaningful analysis. These Settings can be accessed from the Water Distribution Network Model toolbar, see **Figure 9-61**.

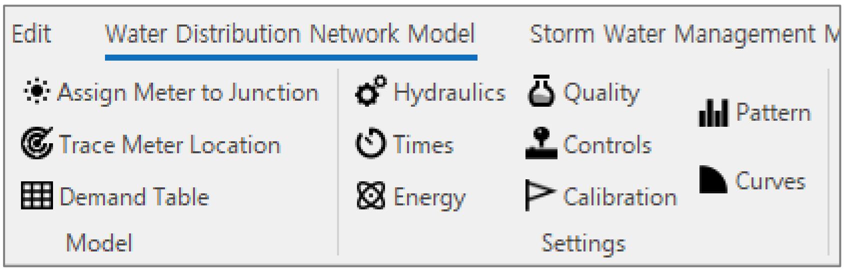


Figure 9-61: WDNM Settings

The hydraulic, water quality, energy and time settings can be set to analyse the numerical model in a specific manner and are described in the following sections.

9.4.1 Hydraulic Options

The Hydraulic options windows, **Figure 9-62**, can be accessed by clicking on the *Hydraulics* button on the Water Distribution Network Model Toolbar.

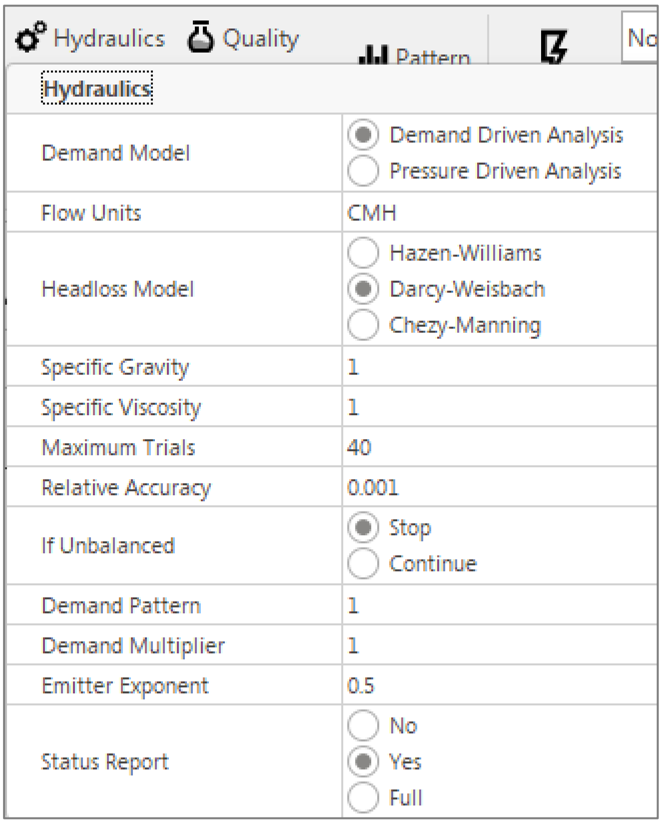


Figure 9-62: Hydraulic options

Table 9-21: Hydraulic options for hydraulic analysis

Option	Description
Demand Model	Selects between demand or pressure driven analysis – DDA or PDA, respectively. DDA assumes demands are fixed at a given point in time, while PDA assumes demands are a function of pressure. The PDA option can be used to find a solution when negative pressures are present in a DDA.
Flow Units	Units in which nodal demands and link flow rates are expressed. Choosing units in gallons, cubic feet, or acre-feet implies that the units for all other network quantities are Customary US. Selecting litres or cubic meters causes all other units to be SI metric. Use caution when changing flow units as it might affect all other data supplied to the project. (See **Appendix B**, Units of Measurement)
Headloss Model	Formula used to compute headloss as a function of flow rate in a pipe. Choices are: - Hazen-Williams - Darcy-Weisbach - Chezy-Manning Because each formula measures pipe roughness differently, switching formulas might require that all pipe roughness coefficients be updated.
Specific Gravity	Ratio of the density of the fluid being modelled to that of water at 4 deg. C (unitless).
Relative Viscosity	Ratio of the kinematic viscosity of the fluid to that of water at 20 deg. C (1.0 centistokes or 0.94 sq ft/day) (unitless).
Maximum Trials	Maximum number of trials used to solve the nonlinear equations that govern network hydraulics at a given point in time. Suggested value is 40.
Relative Accuracy	Convergence criterion used to signal that a solution has been found to the nonlinear equations that govern network hydraulics. Trials end when the sum of all flow changes divided by the sum of all link flows is less than this number. Suggested value is 0.001.
If Unbalanced	Action to take if a hydraulic solution is not found within the maximum number of trials. Choices are STOP to stop the simulation at this point or CONTINUE to use another 10 trials, with no link status changes allowed, in an attempt to achieve convergence.
Default Pattern	ID label of a time pattern to be applied to demands at those junctions where no time pattern is specified. If no such pattern exists then demands will not vary at these locations.
Demand Multiplier	Global multiplier applied to all demands to make total system consumption vary up or down by a fixed amount (e.g., 2.0 doubles all demands, 0.5 halves them, and 1.0 leaves them as is).
Emitter Exponent	Power to which pressure is raised when computing the flow through an emitter device. The textbook value for nozzles and sprinklers is 0.5. This may not apply to pipe leakage.
Status Report	Amount of status information to report after an analysis is made. Choices are: - NO (no reporting) - YES (link status changes) - FULL (normal plus convergence) Full status reporting is only useful for debugging purposes.
Minimum Pressure	In a PDA, the pressure below which demand is assumed to be zero.

Option	Description
Required Pressure	In a PDA, the pressure required to deliver the full demand.
Pressure Exponent	PDA assumes a pressure demand relation raised to an exponent. Standard value is 0.5.
Max. Head Error	Augments the ACCURACY option. Specifies the maximum head loss error any network link can have for hydraulic convergence to occur. The default value of 0 indicates that no head error limit applies. Units of this parameter are feet (US) or meters (SI).
Max. Flow Change	Augments the ACCURACY option. Specifies the largest change in flow that any network element (link, emitter, or pressure- dependent demand) can have for hydraulic convergence to occur. The default value of 0 indicates that no flow change limit applies. It is specified based on the current project flow unit setting.
CHECKFREQ	This sets the number of solution trials that pass during hydraulic balancing before the status of pumps, check valves, flow control valves and pipes connected to tanks are once again updated. The default value is 2, meaning that status checks are made every other trial. A value equal to the maximum number of trials would mean that status checks are made only after a system has converged. (Whenever a status change occurs the trials must continue since the current solution may not be balanced.) The frequency of status checks on pressure reducing and pressure sustaining valves (PRVs and PSVs) is determined by the DAMPLIMIT option (see **Table 9-22**).
MAXCHECK	This is the number of solution trials after which periodic status checks on pumps, check valves, flow control valves and pipes connected to tanks are discontinued. Instead, a status check is made only after convergence is achieved. The default value is 10, meaning that after 10 trials, instead of checking status every CHECK- FREQ trials, status is checked only at convergence.
DAMPLIMIT	This is the accuracy value at which solution damping and status checks on PRVs and PSVs should begin. Damping limits all flow changes to 60% of what they would otherwise be as future trials unfold. The default is 0 which indicates that no damping should be used and that status checks on control valves are made at every iteration. Damping might be needed on networks that have trouble converging, in which case a limit of 0.01 is suggested.

Table 9-22: Typical values for status checking parameters

CHECKFREQ	MAXCHECK	DAMPLIMIT	Remarks
2	10	0	Frequent status checking; tends to produce solutions in the least number of iterations.
10	100	0.01	Less frequent status checking; might be needed for networks that have difficult in converging.
Max. Trials	Max. Trials	Convergence Accuracy	Status checks made only after convergence is achieved; might produce convergence when other settings fail.

9.4.2 Water Quality Options

Water quality options control how the water quality analysis is carried out. On the *Settings* toolbar click on the *Quality* button as shown in **Figure 9-63** to view the water quality options that can be set.

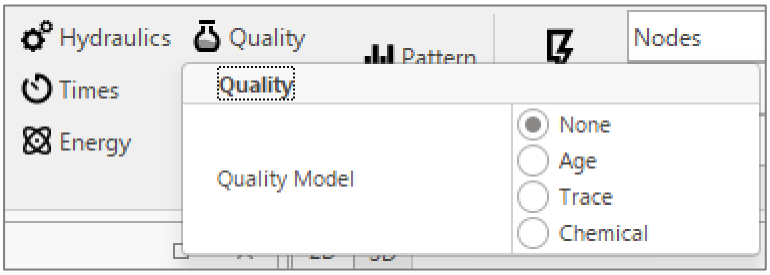


Figure 9-63: Quality options

Select the applicable *Quality Model* by selecting the radio button.

If Age was selected the *Tolerance* value (described in **Table 9-23**) must be entered as show in **Figure 9-64**.

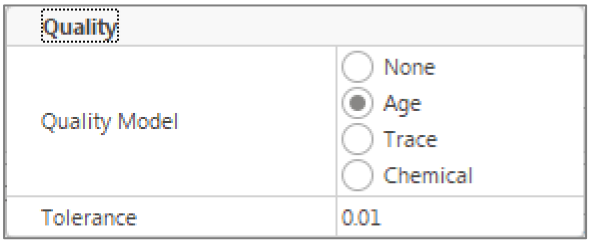


Figure 9-64: Quality model - Age

If Trace was selected the *Trace Node* (described in **Table 9-23**) must be entered as show in **Figure 9-65.**

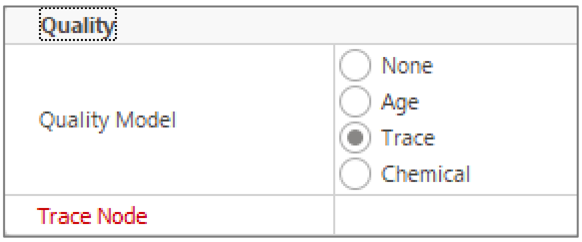


Figure 9-65: Quality model - Trace

If Chemical was selected, as shown in **Figure 9-66**, the following additional input parameters were required: Units, Diffusivity, Tolerance, Quality name (described in **Table 9-23**) and the reaction parameters (see **Table 9-24**).

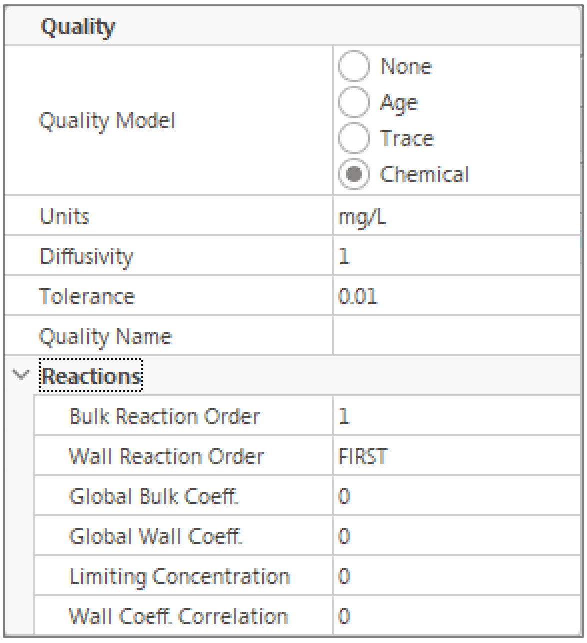


Figure 9-66: Quality model - Chemical

Table 9-23: Water quality options for water quality analysis

Option	Description
Parameter	Type of water quality parameter being modelled. Choices include: - None (no quality analysis) - Age (estimate water age) - Trace (percent flow from node) - Chemical (compute concentration) In lieu of Chemical, the user can enter the actual name of the chemical being modelled (e.g., Chlorine).
Units	Mass units used to express concentration. Choices are mg/L ug/L. Units for Age and Trace analyses are fixed at hours and percent, respectively.
Diffusivity	Ratio of the molecular diffusivity of the chemical being modelled to that of chlorine at 20 deg. C (0.00112 sq ft/day). Use 2 if the chemical diffuses twice as fast as chlorine, 0.5 if half as fast, etc. Applies only when modelling mass transfer for pipe wall reactions. Set to zero to ignore mass transfer effects.
Tolerance	Smallest change in quality that will cause a new parcel of water to be created in a pipe. A typical setting might be 0.01 for chemicals measured in mg/L as well as water age and source tracing.
Trace Node	ID label of the node whose flow is being traced. Applies only to flow tracing analyses.

Note: The Quality Tolerance determines when the quality of one parcel of water is essentially the same as another parcel. For chemical analysis this might be the detection limit of the procedure used to measure the chemical, adjusted by a suitable factor of safety. Using too large a value for this tolerance might affect simulation accuracy. Using too small a value will affect computational efficiency. Some experimentation with this setting might be called for.

Reaction options set the types of reactions that apply to a water quality analysis and are listed in **Table 9-24**.

Table 9-24: Reaction options for water quality analysis

Option	Description
Bulk Reaction Order	Power to which concentration is raised when computing a bulk flow reaction rate. Use 1 for first-order reactions, 2 for second-order reactions, etc. Use any negative number for Michaelis-Menton kinetics. If no global or pipe-specific bulk reaction coefficients are assigned then this option is ignored.
Wall Reaction Order	Power to which concentration is raised when computing a bulk flow reaction rate. Choices are FIRST (1) for first-order reactions or ZERO (0) for constant rate reactions. If no global or pipe-specific wall reaction coefficients are assigned then this option is ignored.
Global Bulk Coefficient	Default bulk reaction rate coefficient K_b assigned to all pipes. This global coefficient can be overridden by editing this property for specific pipes. Use a positive number for growth, a negative number for decay, or 0 if no bulk reaction occurs. Units are concentration raised to the (1-n) power divided by days, where n is the bulk reaction order.
Global Wall Coefficient	Wall reaction rate coefficient K_w assigned to all pipes. Can be overridden by editing this property for specific pipes. Use a positive number for growth, a negative number for decay, or 0 if no wall reaction occurs. Units are ft/day (US) or m/day (SI) for first-order reactions and mass/sq ft/day (US) or mass/sq m/day (SI) for zero-order reactions.
Limiting Concentration	Maximum concentration that a substance can grow to or minimum value it can decay to. Bulk reaction rates will be proportional to the difference between the current concentration and this value. Set to zero if not applicable.
Wall Coefficient Correlation	Factor correlating wall reaction coefficient to pipe roughness. Set to zero if not applicable.

9.4.3 Energy Options

Energy Analysis Options provide default values used to compute pumping energy and cost when no specific energy parameters are assigned to a given pump. The Energy Analysis Options can be set by clicking the *Energy* button on the *Settings* toolbar, see **Figure 9-67**, and the parameters that can be set are listed and described in **Table 9-25.**

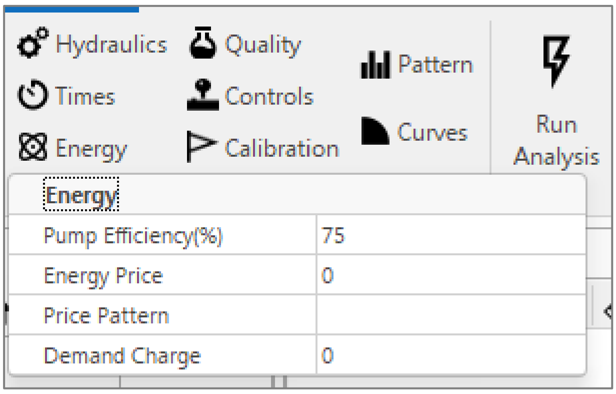


Figure 9-67: Energy options

Table 9-25: Energy options for hydraulic analysis

Option	Description
Pump Efficiency (%)	Default pump efficiency.
Energy Price	Price of energy per kilowatt-hour. Monetary units are not explicitly represented.
Price Pattern	ID label of a time pattern used to represent variations in energy price with time. Leave blank if not applicable.
Demand Charge	Additional energy charge per maximum kilowatt usage.

9.4.4 Times

The Times options windows, **Figure 9-68**, can be accessed by clicking on the *Times* button on the Water Distribution Network Model Toolbar. Times option set values for the various time steps used in an extended period simulation. These are listed below in **Table 9-26** (times can be entered as decimal hours or in hours:minutes notation).

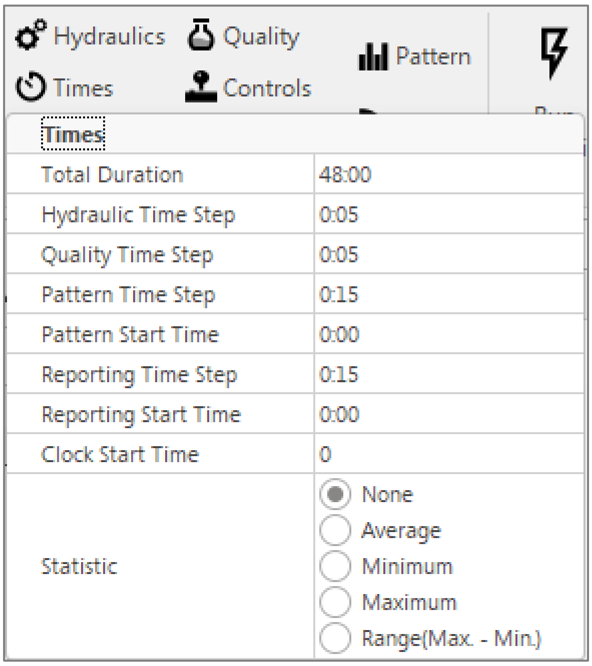


Figure 9-68: Times options

Table 9-26: Times Options for Hydraulic and Water Quality Analysis

Option	Description
Total Duration	Total length of a simulation in hours. Use 0 to run a single period (snapshot) hydraulic analysis.
Hydraulic Time Step	Time interval between re-computation of system hydraulics. Normal default is 1 hour.
Quality Time Step	Time interval between routing of water quality constituent. Normal default is 5 minutes (0:05 hours).
Pattern Time Step	Time interval used with all-time patterns. Normal default is 1 hour.
Pattern Start Time	Hours into all-time patterns at which the simulation begins (e.g., a value of 2 means that the simulation begins with all-time patterns starting at their second hour). Normal default is 0.
Reporting Time Step	Time interval between times at which computed results are reported. Normal default is 1 hour.

Option	Description
Report Start Time	Hours into simulation at which computed results begin to be reported. Normal default is 0.
Clock Start Time	Defines the starting time of day for a simulation run, typically expressed in 24-hour format (e.g., 0:00, 6:00, 18:00).
Starting Time of Day	Clock time (e.g., 7:30 am, 10:00 pm) at which simulation begins. Default is 12:00 am (midnight).
Statistic	Type of statistical processing used to summarize the results of an extended period simulation. Choices are: None (current time step results) Average (time-averaged results) Minimum (minimum value results) Maximum (maximum value results) Range (Max-Min) Statistical processing is applied to all node and link results obtained between the Report Start Time and the Total Duration.

Note: To run a single-period hydraulic analyses (also called a snapshot analysis) enter 0 for Total Duration. In this case entries for all of the other time options, with the exception of Starting Time of Day, are not used. Water quality analyses always require that a non-zero Total Duration be specified.

9.4.5 Calibration Tools

GISpipe allows the user to compare results of a simulation against measured field data. This can be done via Time Series plots for selected locations in the network or by special Calibration Reports that consider multiple locations. Before GISpipe can use such calibration data it has to be entered into a file and imported into the project.

A Calibration File is a text file containing measured data for a particular quantity taken over a particular period of time within a distribution system. The file provides observed data that can be compared to the results of a network simulation. Separate files should be created for different parameters (e.g., pressure, fluoride, chlorine, flow, etc.) and different sampling studies. Each line of the file contains the following items:

- Location ID - ID label (as used in the network model) of the location where the measurement was made.
- Time - Time (in hours) when the measurement was made
- Value - Result of the measurement

See example of data set of Pressure head data shown at two nodes for a number of specific different times (**Figure 9-70**).

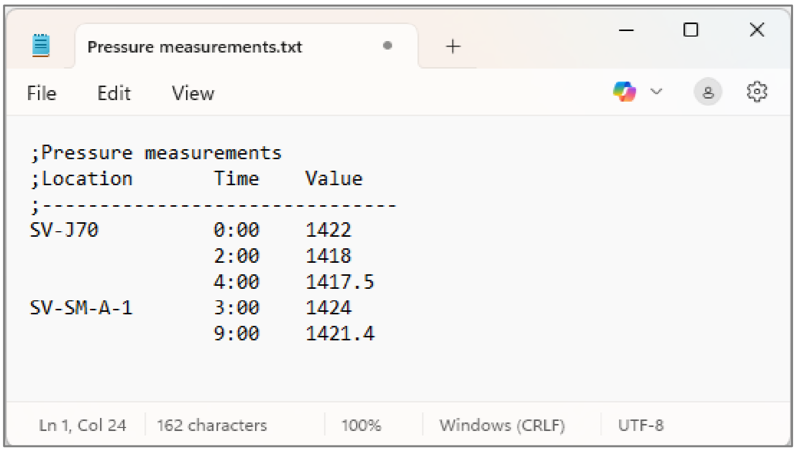


Figure 9-69: Example Calibration File

The measurement time is with respect to time zero of the simulation to which the Calibration File will be applied. It can be entered as either a decimal number (e.g., 27.5) or in hours:minutes format (e.g., 27:30). For data to be used in a single period analysis all time values can be 0. Comments can be added to the file by placing a semicolon (;) before them. For a series of measurements made at the same location the Location ID does not have to be repeated.

The Calibration File can be imported by clicking on *Calibration* button on the Settings toolbar, and will display the observed values and compare these with the computed values as shown in **Figure 9-70**. Nodes can be added or deleted by clicking on the *Add* or *Delete* buttons. The Calibration File is added to the project by clicking on the *Import* button. Checking the *Show values on map* check box shows the results and comparison of the observed and computed values on the Viewer Map for the specific component.

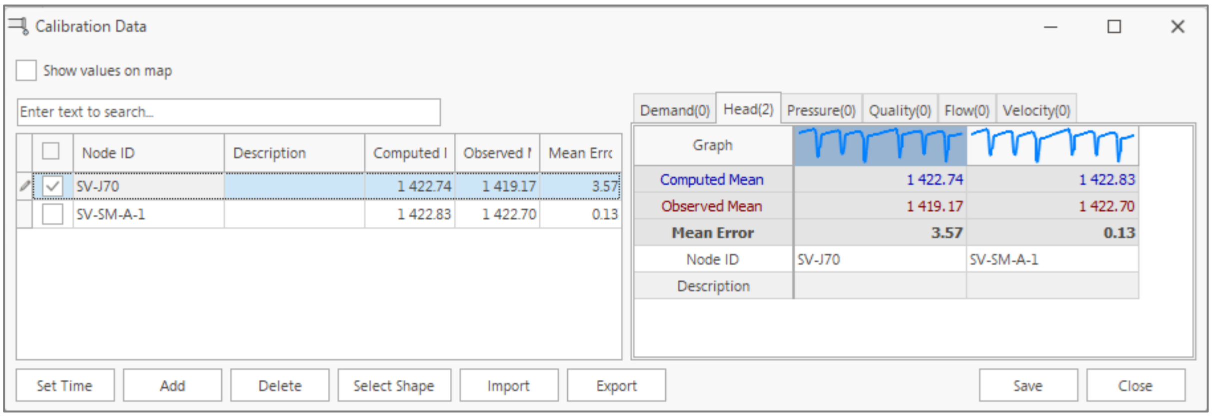


Figure 9-70: Calibration Data

9.4.6 Adding Controls and Rules

The Controls options windows, **Figure 9-71** and **Figure 9-72**, can be accessed by clicking on the *Controls* button on the Water Distribution Network Model Toolbar. Controls are statements that determine how the network is operated over time. They specify the status of selected links as a function of time, tank water levels, and pressures at select points within the network.

There are two categories of controls that can be used:

- Simple Controls
- Rule-Based Controls Simple Controls

A description of the setting of control statements is described in Section 9.2.6.

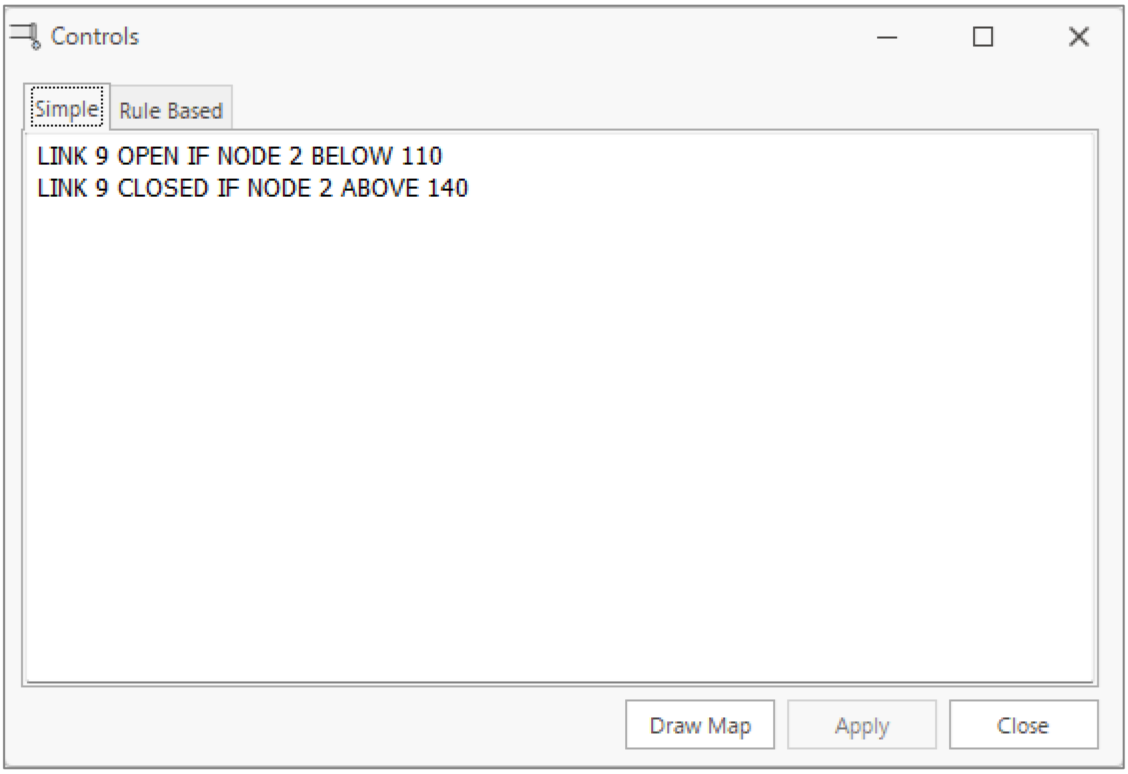


Figure 9-71: Setting Simple Controls

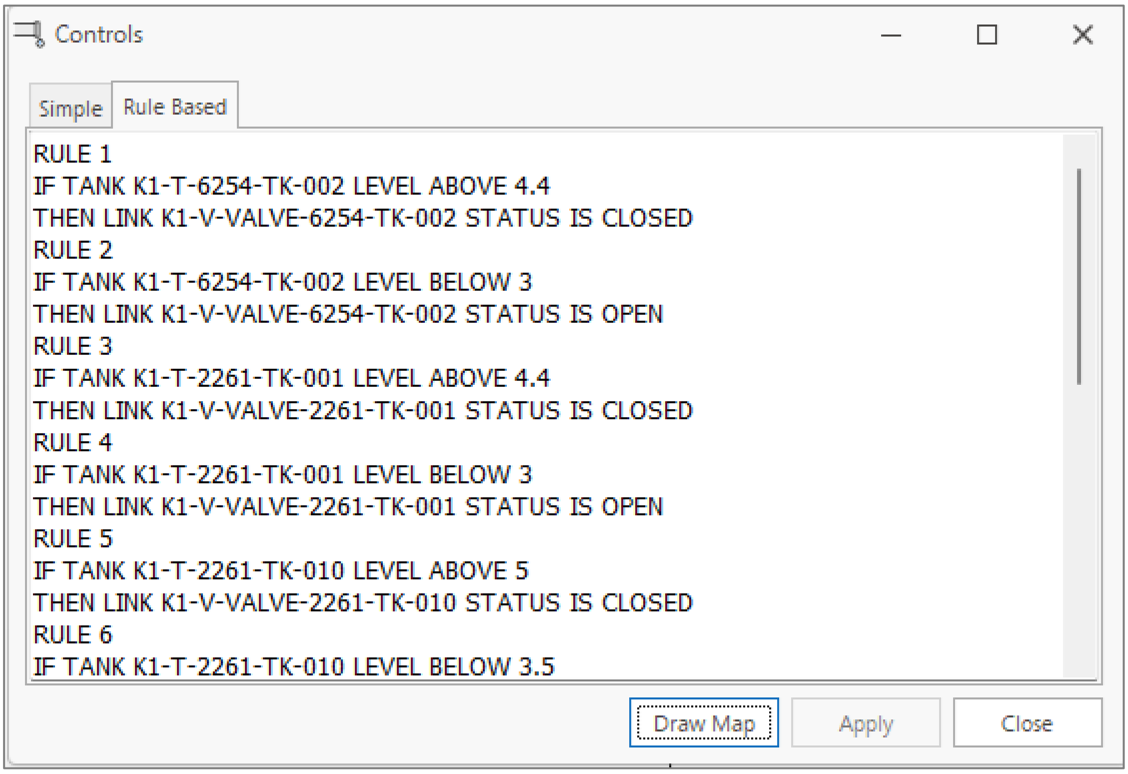


Figure 9-72: Setting Rule Based Controls

9.5 Running Simulations

9.5.1 Setting Up a Simulation

There are four categories of options that control how GISpipe analyses a network: Hydraulics, Quality, Times and Energy. To set any of these options click on the buttons on the Water Distribution Network Model toolbar (**Figure 9-73**).

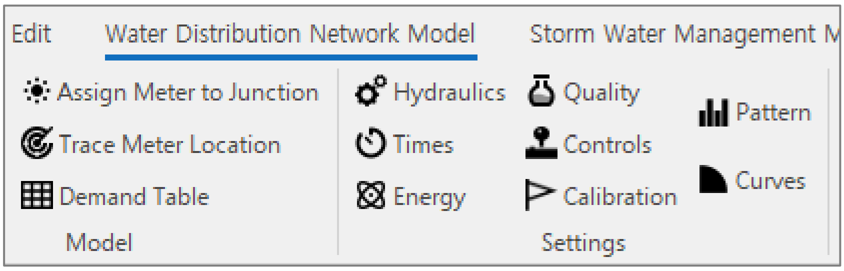


Figure 9-73: WDNM Settings

Table 9-27: Simulation options

Simulation option	Description
Hydraulics	Hydraulics settings define how flow and pressure are calculated throughout the network, including solution accuracy, flow units, and convergence criteria, ensuring a reliable simulation of system behaviour, see **Figure 9-62**.
Times	Times settings control the duration, frequency, and scheduling of the simulation, aligning model results with real-world operating patterns and enabling time-based analysis, see **Figure 9-68**.
Energy	Energy settings evaluate the energy consumption and cost associated with pump operations, supporting assessments of operational efficiency and optimization, see **Figure 9-67**.
Quality	Quality settings determine how water quality constituents (e.g., chlorine, age, source tracing) are tracked and analysed as they move and react within the distribution system, see **Figure 9-63**.

9.5.2 Run an Analysis

To run a hydraulic/water quality analysis click on the *Run Analysis* button on the Analysis & Simulation toolbar (**Figure 9-74**).

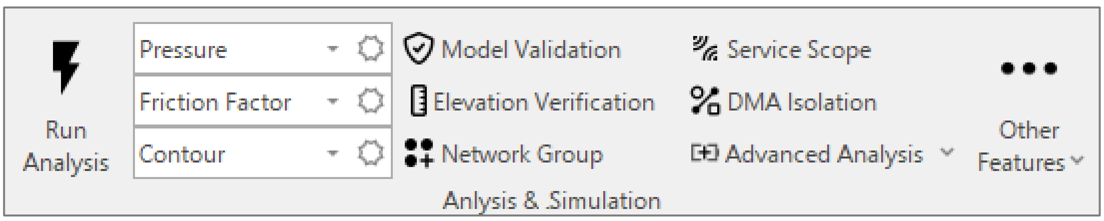


Figure 9-74: Analysis and Simulation

The progress of the analysis will be displayed in a Run Status window (**Figure 9-75**).

Click on the *Close* button when the analysis ends.

Any error or warning messages will appear in the Run Status window. When double clicking on any message in the Run Status window would take the user to that component in the model.

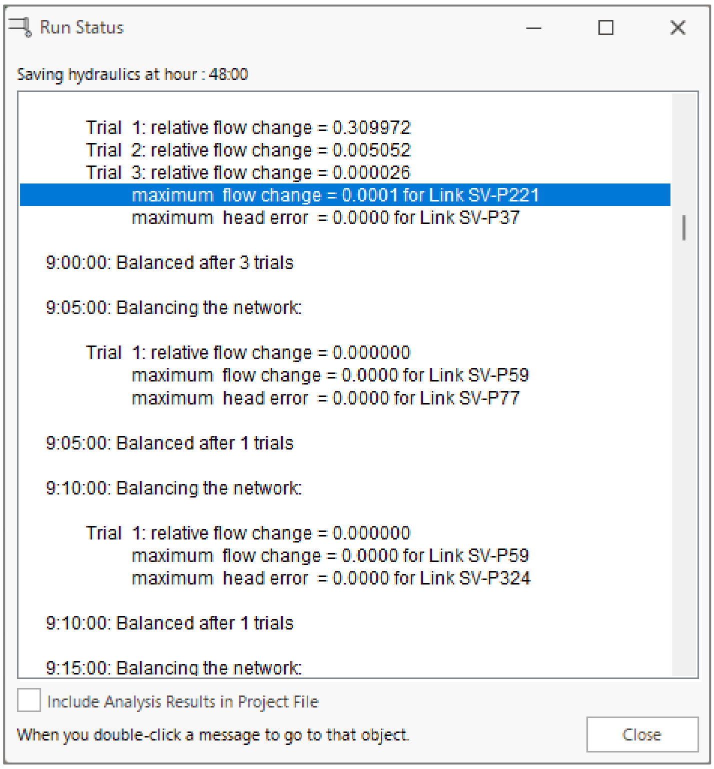


Figure 9-75: Run status

Model Validation

Before running an analysis, users may consider running a Model Validation. This will evaluate the selected model and determine if there are any potential problems that could impact the successful analysis of the model.

Click on the 

button.

If the network is correct the user will receive a message: 

Any potential problems will be listed, **Figure 9-76**, which the user can then address.

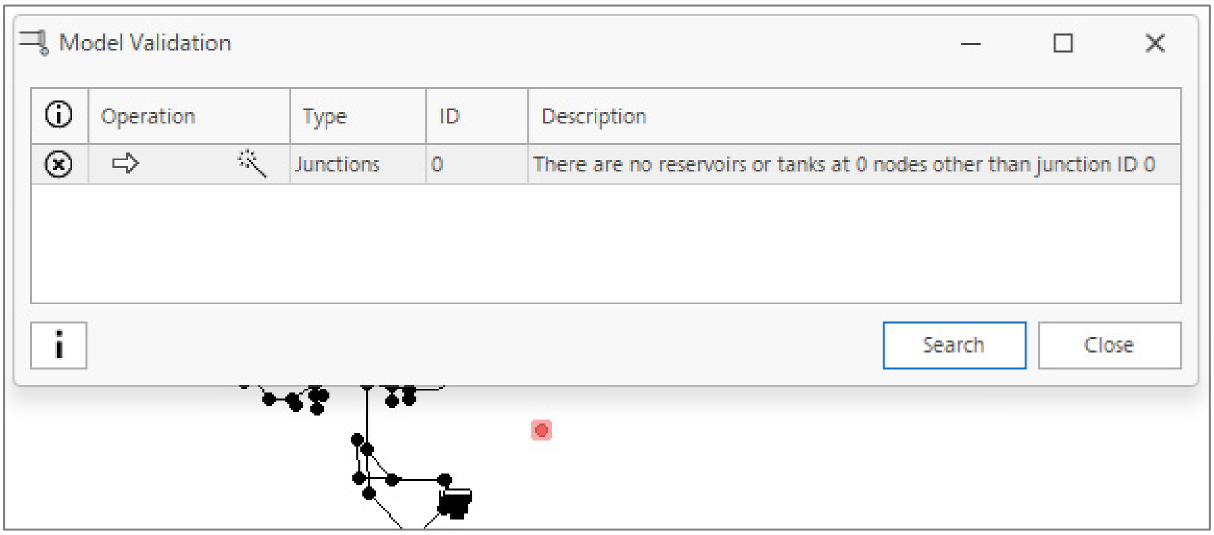


Figure 9-76: Validation Evaluation results

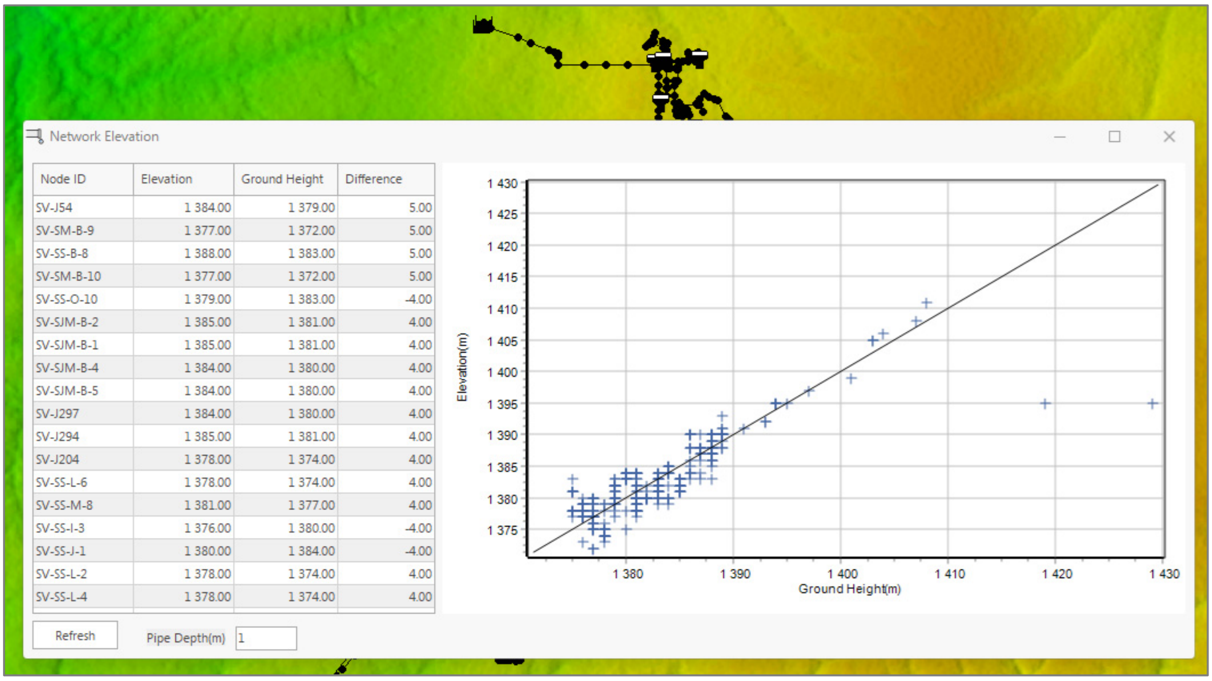
Elevation Verification

The verification of the elevations is a tool where GISpipe compares the entered elevation values of the objects with the Digital Elevation Model (DEM) layer values. To use this function a DEM layer needs to be added.

In a case where no DEM is available the SRTM30M DEM from the Server could be loaded.

From the Tools menu select Importing SRTM30M DEM From Server

The comparison of the DEM and the entered elevations is provided in table format as well as a graphical presentation of a direct comparison, see **Figure 9-77**. The *Pipe depth* can be entered to correct for the depth below the surface of the node as it compares it with the ground elevation from the DEM.



Node ID	Elevation	Ground Height	Difference
SV-J54	1 384.00	1 379.00	5.00
SV-SM-B-9	1 377.00	1 372.00	5.00
SV-SS-B-8	1 388.00	1 383.00	5.00
SV-SM-B-10	1 377.00	1 372.00	5.00
SV-SS-O-10	1 379.00	1 383.00	-4.00
SV-SJM-B-2	1 385.00	1 381.00	4.00
SV-SJM-B-1	1 385.00	1 381.00	4.00
SV-SJM-B-4	1 384.00	1 380.00	4.00
SV-SJM-B-5	1 384.00	1 380.00	4.00
SV-J297	1 384.00	1 380.00	4.00
SV-J294	1 385.00	1 381.00	4.00
SV-J204	1 378.00	1 374.00	4.00
SV-SS-L-6	1 378.00	1 374.00	4.00
SV-SS-M-8	1 381.00	1 377.00	4.00
SV-SS-I-3	1 376.00	1 380.00	-4.00
SV-SS-J-1	1 380.00	1 384.00	-4.00
SV-SS-L-2	1 378.00	1 374.00	4.00
SV-SS-L-4	1 378.00	1 374.00	4.00

Figure 9-77: Elevation Verification results table

Network Group

The Network Group feature determines the different network groups in the network model.

From the Tools menu select Network Group

The feature identifies all the distinct closed link groups and can organise these by colour as well as summarises the number of nodes, in this network and the total pipe length as shown in **Figure 9-78**.

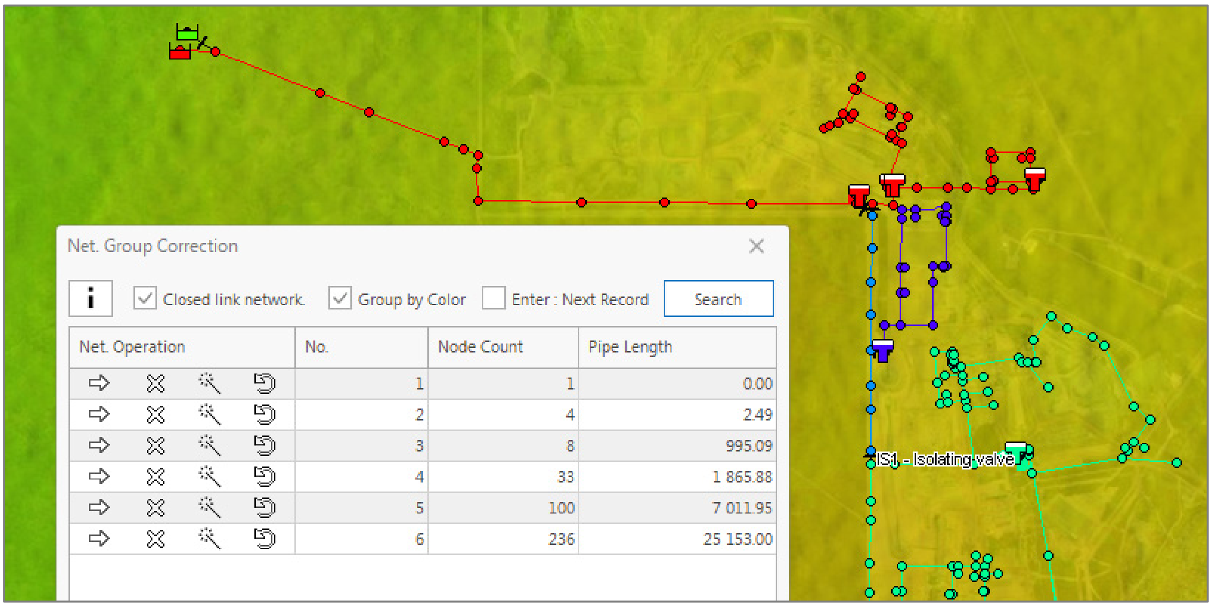


Figure 9-78: Network Group results table

Service Scope

The Service Scope analysis determines the areas which are serviced by a specific Tank or Reservoir.

From the Tools menu select Service Scope

The feature identifies all the distinct network sections which are supplied from a particular source as shown in **Figure 9-79**.

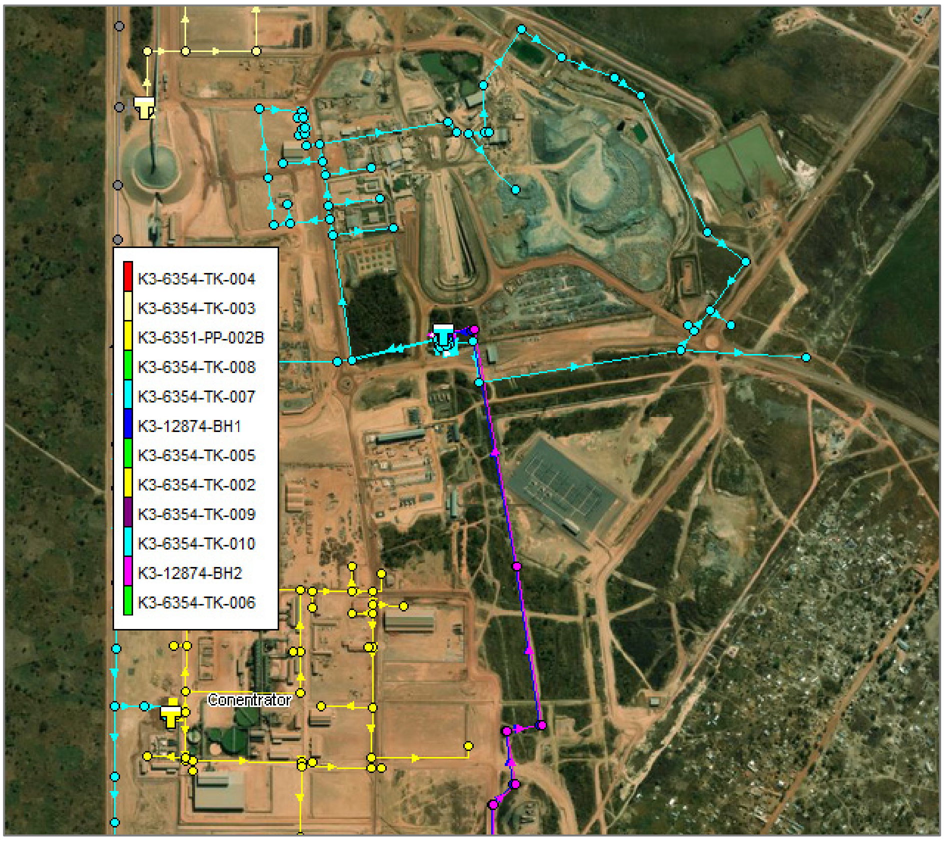
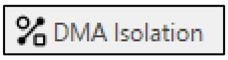


Figure 9-79: Service scope analysis

DMA Isolation

The DMA Isolation function is used to verify, manage, and simulate the isolation of district metered areas, ensuring they are properly bounded for operational control, leakage detection, and demand management. A new layer is created (DMA) and the DMA zones/shapes are drawn.

The DMA Isolation function is then used to identify and simulate the hydraulic isolation of a District Metered Area (DMA) within the distribution network. A DMA is a defined zone in the water supply system, typically bounded by closed valves and monitored through bulk meters, used to measure inflows/outflows and manage demand and losses.

Click on the 

button.

This will evaluate the network based on the defined DMA zones and indicate on the Viewer Map the different zones. If there is a zone which is linked with another DMA zone it will highlight where these are linked by means of a larger junction, indicating that it is supplied from the adjacent DMA zone, see **Figure 9-80**.

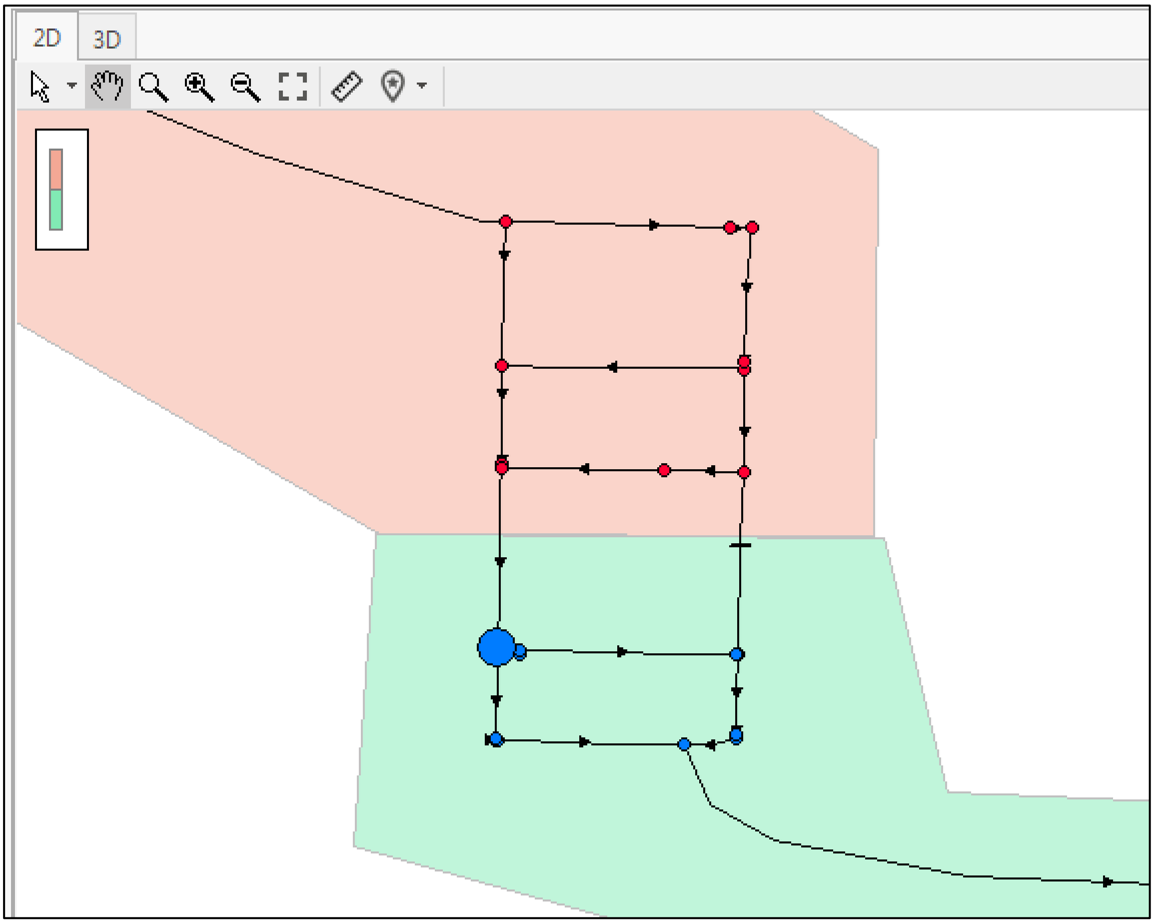


Figure 9-80: DMA zones with water distribution network

Advanced Analysis

GISpipe provides some additional analysis option. These options can be selected from the Analysis & Simulation toolbar, by clicking on *Advanced Analysis* button, as shown in **Figure 9-81**.

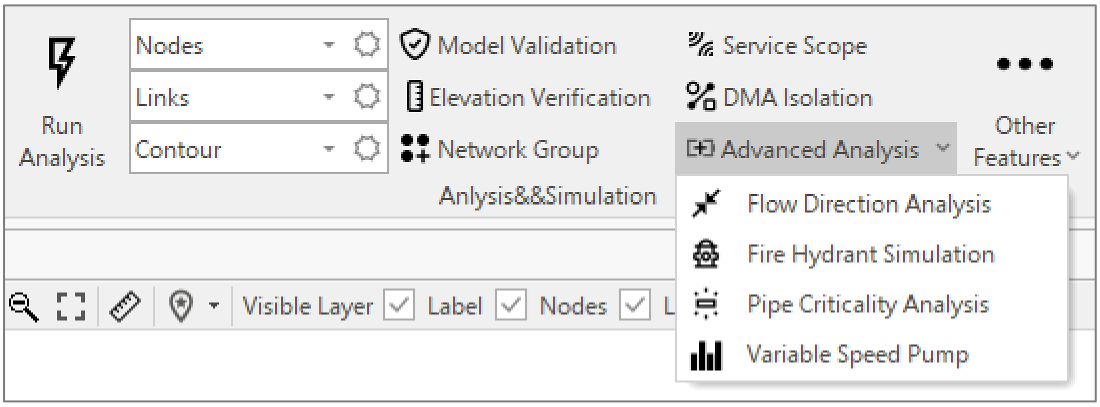


Figure 9-81: Advanced Analysis options

Flow Direction Analysis

On the Analysis & Simulation Toolbar, click on *Advance Analysis* (**Figure 9-81**) and select *Flow Direction Analysis* from the drop-down menu to show the Flow Direction Analysis window (**Figure 9-82**).

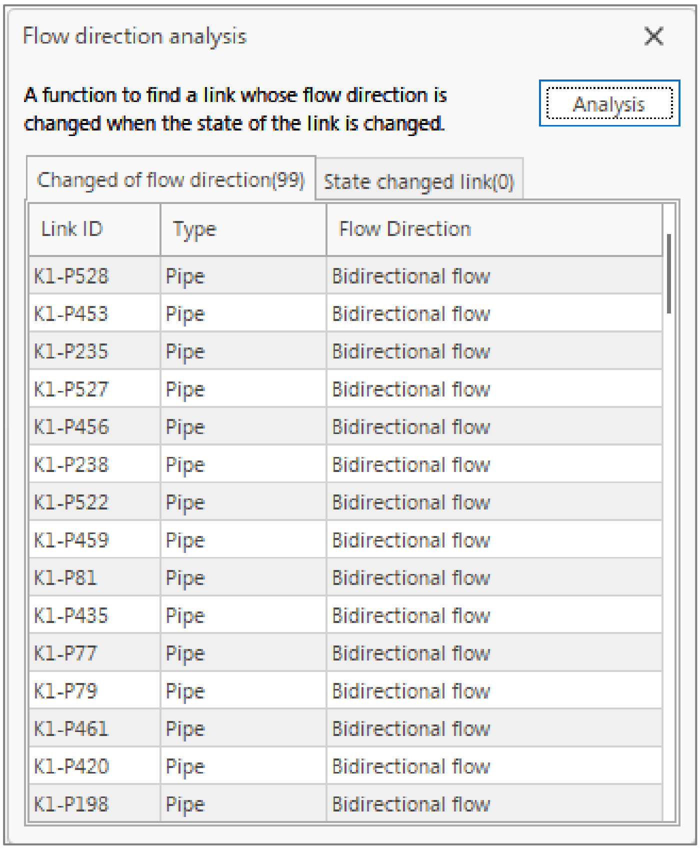


Figure 9-82: Flow Direction Analysis

The tool identifies which links will experience a change in flow direction when the state of a particular link (open/closed, enabled/disabled) is modified. This helps in evaluating operational decisions, detecting possible backflows, and understanding the hydraulic impact of link changes on the system, **Figure 9-83**.

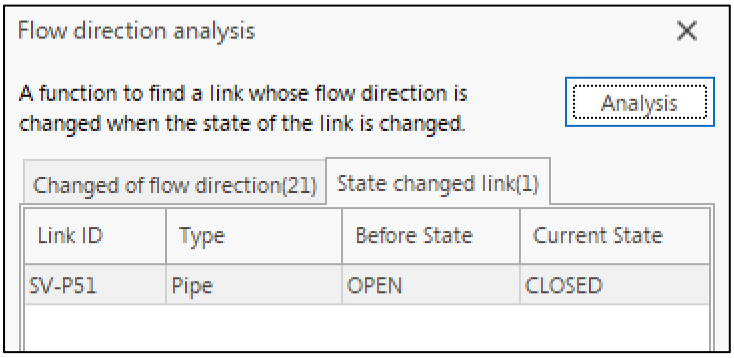


Figure 9-83: State changed link - Flow Direction Analysis

Fire Hydrant Analysis

On the Analysis & Simulation Toolbar, click on *Advance Analysis* (**Figure 9-81**) and select *Fire Hydrant Analysis* from the drop-down menu to show the Fire Hydrant Analysis window (**Figure 9-84**).

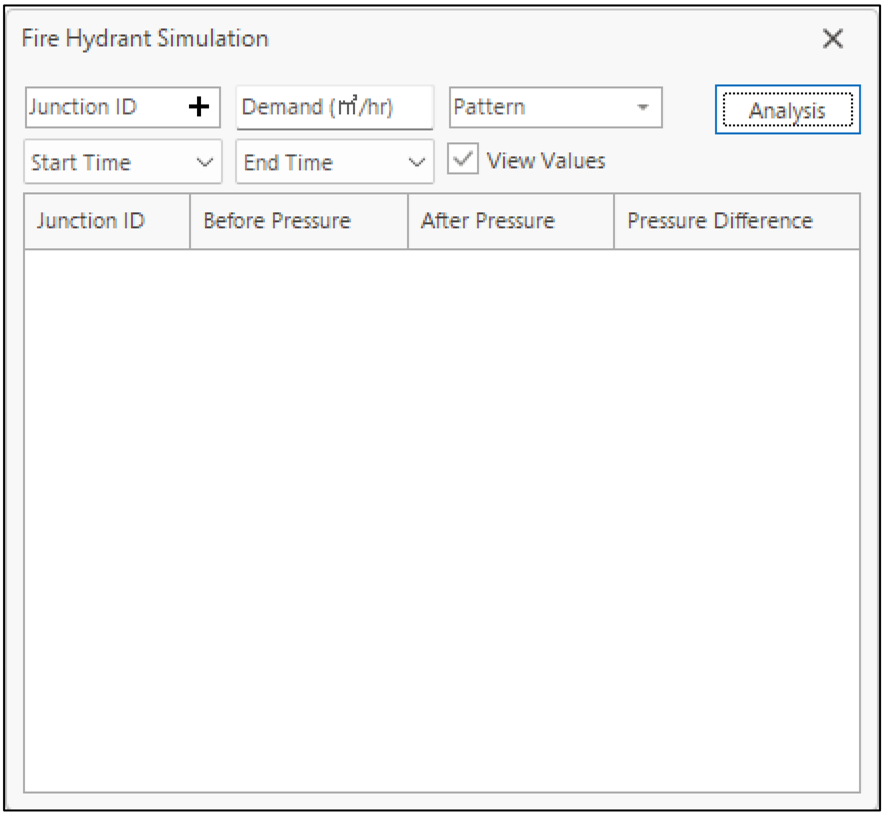


Figure 9-84: Fire Hydrant Analysis

This function is to simulate the impact of a fire hydrant on the pressures in the network. It allows users to test the effect of opening a fire hydrant (or simulating a sudden withdrawal of flow) on system pressures. The inputs and outputs are described in **Table 9-28**. The results are show on the Fire Hydrant Analysis dialogue box, **Figure 9-85**, and can also be seen on the map viewer window.

Table 9-28: Input and output of Fire Hydrant Simulation

Parameter	Description
Input	
Junction ID	The specific junction (hydrant location) where the fire flow will be applied.
Demand (m³/day)	The simulated fire flow demand, expressed as an outflow rate. This represents the volume of water withdrawn by the fire hydrant.
Pattern	An optional demand pattern to apply, if the fire flow is time-dependent (e.g., varying throughout the simulation).
Start Time / End Time	The simulation period during which the fire flow demand is applied.
View Values	If checked, the tool displays the calculated pressures before and after applying the fire hydrant demand on the map viewer window.
Results	
Junction ID	Reference to the results of the junction.
Before Pressure	The pressure at the junction before the fire flow demand is applied.
After Pressure	The pressure at the junction once the fire flow demand is applied.
Pressure Difference	The drop in pressure due to the fire hydrant withdrawal. This is the critical result for evaluating system resilience and firefighting capacity.

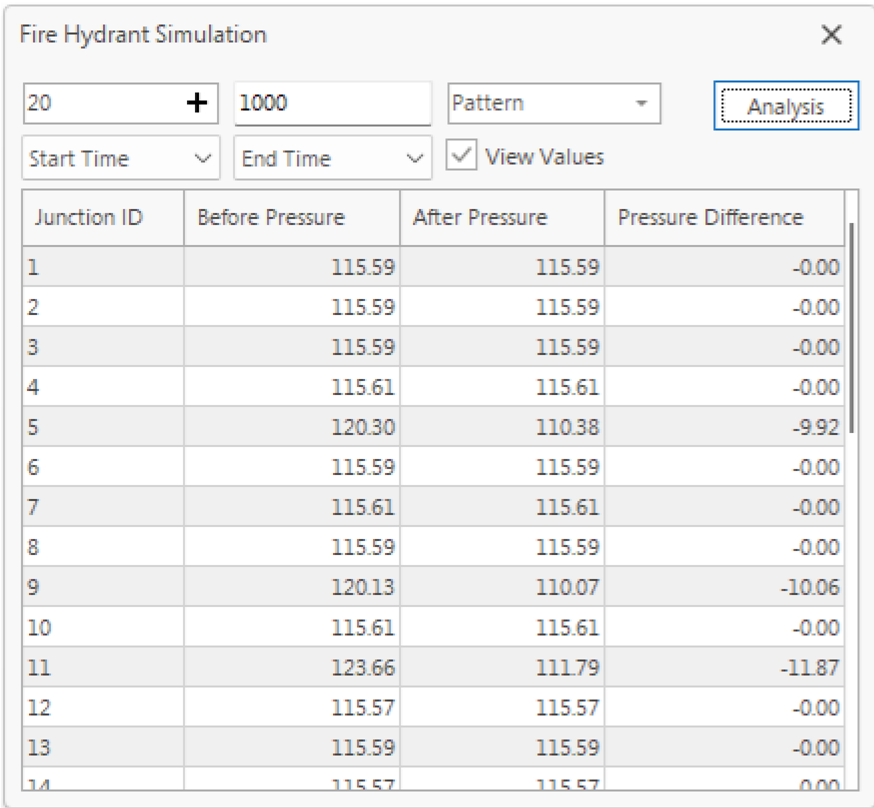


Figure 9-85: Fire Hydrant Analysis - Results

Pipe Criticality Analysis

On the Analysis & Simulation Toolbar, click on *Advance Analysis* (**Figure 9-81**) and select *Pipe Criticality Analysis* from the drop-down menu to show the Pipe Criticality Analysis window (**Figure 9-86**). The input parameters and results are listed in **Table 9-29**.

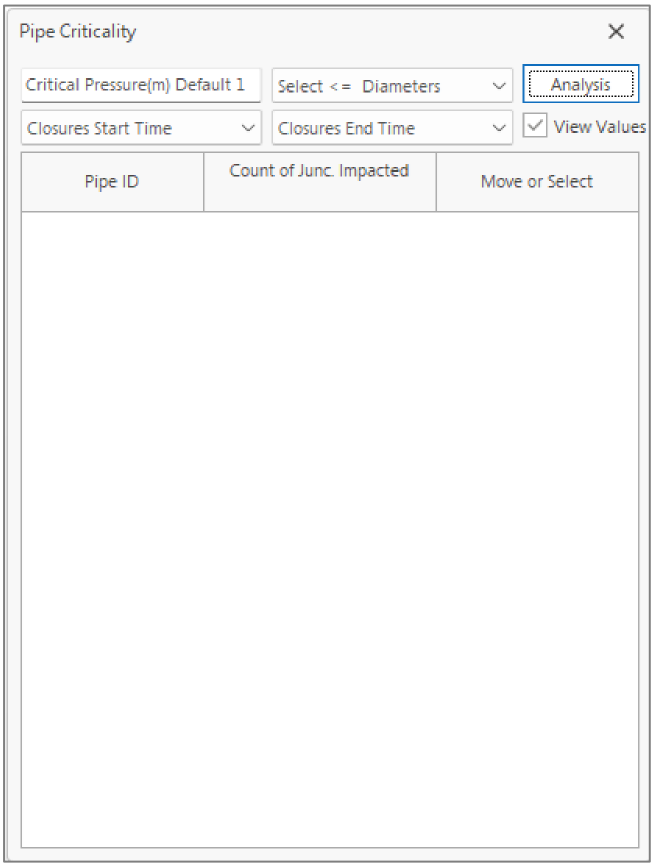


Figure 9-86: Pipe Criticality Analysis

Table 9-29: Input and output of Pipe Criticality Analysis

Parameter	Description
Input	
Critical Pressure (m)	The minimum allowable pressure (in meters of head) at junctions. If pressure falls below this threshold during a pipe closure, those junctions are considered impacted. Default 1 m
Select <= Diameters	A filter to include only pipes with diameters smaller than or equal to the specified size. This allows analysis to focus on critical smaller mains or laterals.
Closures Start Time / Closures End Time	Defines the period over which the pipe closure is simulated. Useful for extended period simulations where the timing of closures can influence results.
View Values	If checked, the results (pressures, number of impacted junctions, etc.) are displayed in map viewer window.
Analysis	Runs the pipe-by-pipe closure simulation based on the criteria above.
Results	
Pipe ID	Identifier of the pipe that was analysed for closure.
Count of Junc. Impacted	The number of junctions where pressure dropped below the specified critical threshold due to closure of the pipe.
Move or Select	Allows the user to either zoom to (move) or highlight (select) the impacted pipe in the network for further inspection.

The results of a Pipe Criticality Analysis are depicted in **Figure 9-87**.

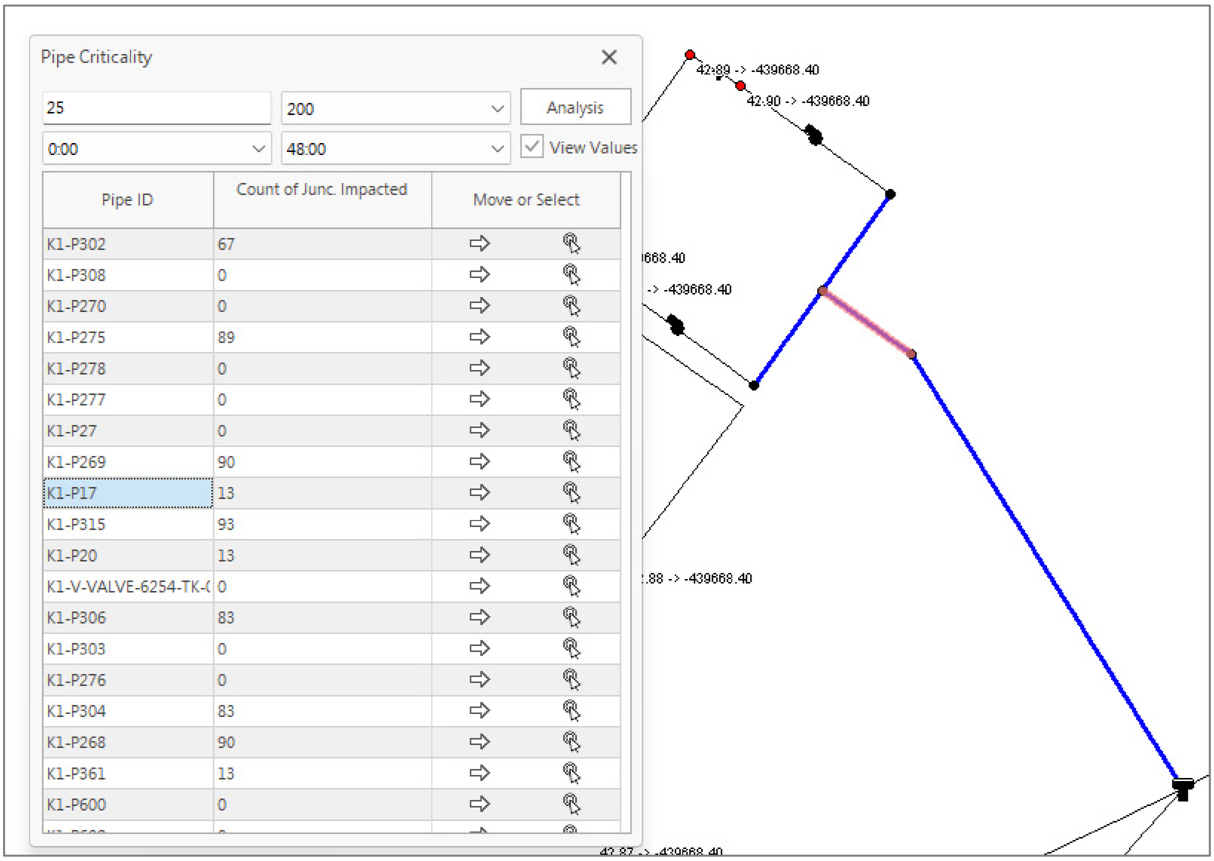


Figure 9-87: Pipe Criticality results

Variable Speed Pump

On the Analysis & Simulation Toolbar, click on *Advance Analysis* (**Figure 9-81**) and select *Variable speed Pump* from the drop-down menu to show the Variable Speed Pump Analysis window (**Figure 9-88**).

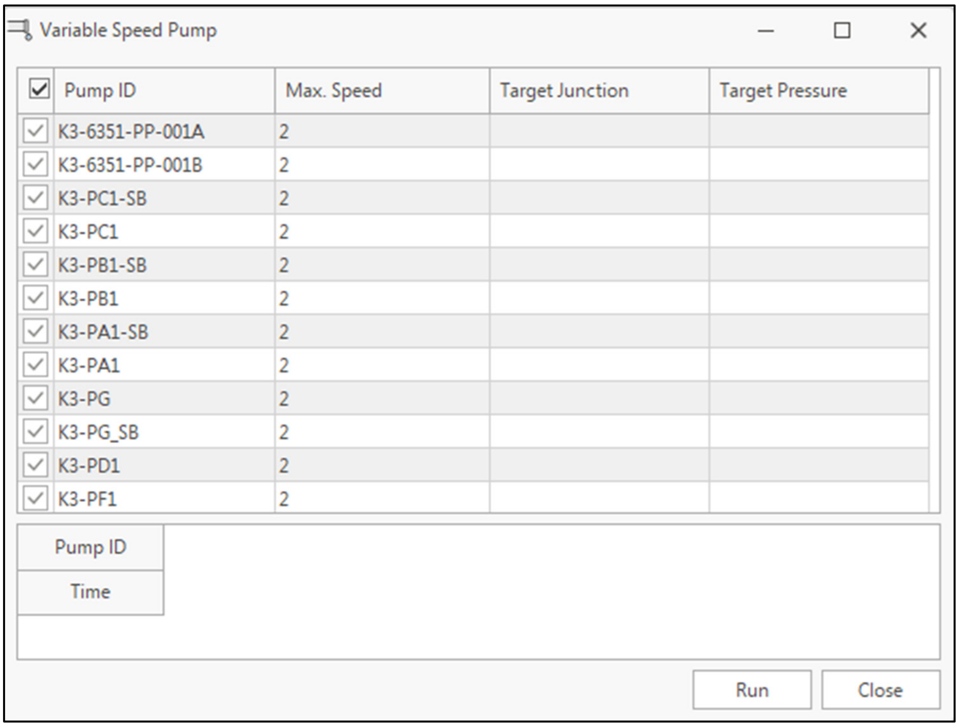


Figure 9-88: Variable Speed Pump Analysis

Table 9-30: Input and output of Variable Speed Pump Analysis

Parameter	Description
Input	
Pump ID	The list of pumps in the model. Select the pump(s) to include in the variable speed analysis. Checkboxes used to select.
Max. Speed	The maximum allowable relative pump speed (typically expressed as a multiplier, e.g., “2” means up to twice the nominal speed).
Target Junction	The junction/node in the network where the pressure requirement must be satisfied.
Target Pressure	The required pressure (in meters of head) at the target junction. The program will adjust pump speed to maintain this pressure.
Run	Run button executes the analysis and generates time-step results for the selected pump(s).
Results	
Pump ID	Identifies the pump being analyzed (example: K3-PB1).
Time	The simulation time step (e.g., 0:00, 0:15, 0:30, etc.).
Speed Pattern	The relative pump speed multiplier required at each time step to maintain the target pressure at the junction (1.0 = nominal speed, >1.0 = faster, <1.0 = slower).
Pressure	The resulting pressure at the target junction when operating at the calculated pump speed.

The results of a Pipe Criticality Analysis are depicted in **Figure 9-89**.

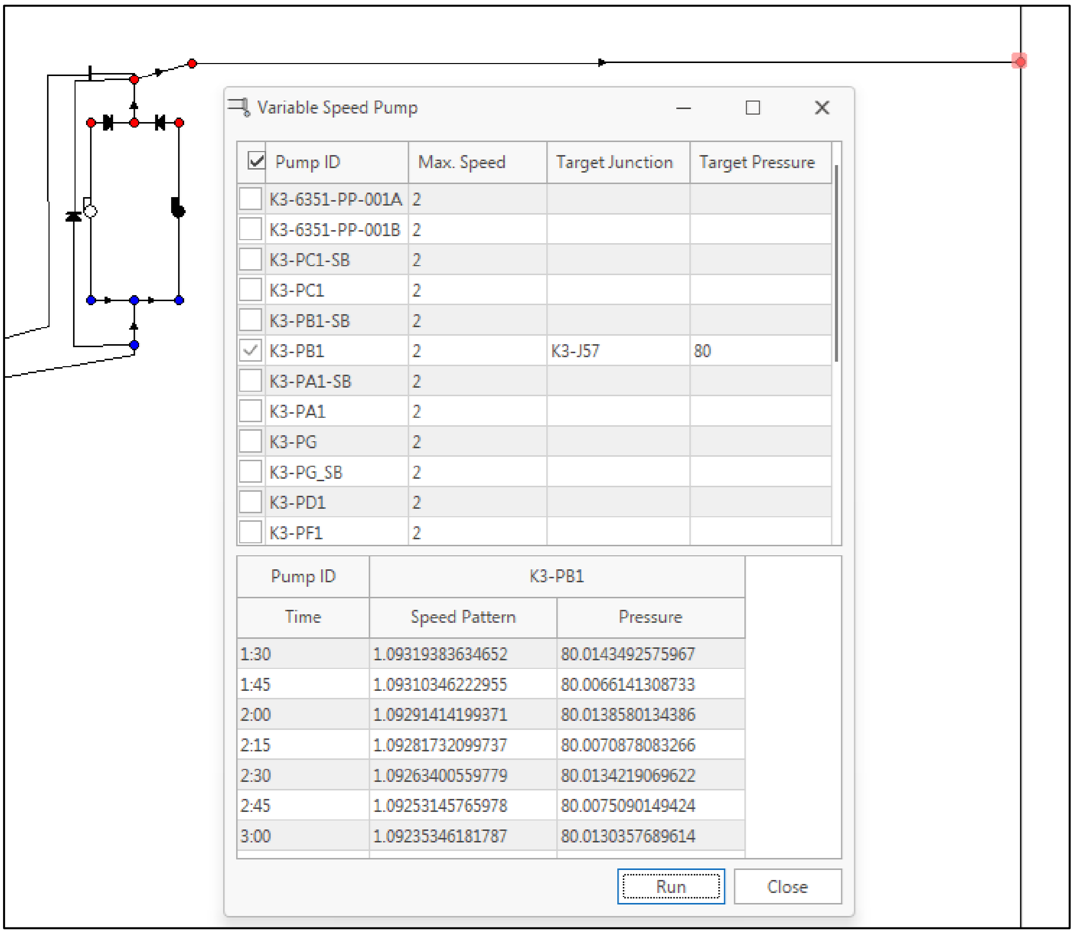


Figure 9-89: Variable Speed Pump Analysis results

9.5.3 Viewing Status and Errors

GISpipe will issue specific Error and Warning messages when problems are encountered in running a hydraulic/water quality analysis (see **Appendix D** for a complete listing). The most common problems are discussed below.

Pumps cannot deliver flow or head
GISpipe will issue a warning message when a pump is asked to operate outside the range of its pump curve. If the pump is required to deliver more head than its shutoff head, GISpipe will close the pump down. This might lead to portions of the network becoming disconnected from any source of water.

Network is disconnected
GISpipe classifies a network as being disconnected if there is no way to provide water to all nodes that have demands. This can occur if there is no path of open links between a junction with demand and either a reservoir, a tank, or a junction with a negative demand. If the problem is caused by a closed link GISpipe will still compute a hydraulic solution (probably with extremely large negative pressures) and attempt to identify the problem link in its Status Report. If no connecting link(s) exist GISpipe will be unable to solve the hydraulic equations for flows and pressures and will return an Error 110 message when an analysis is made. Under an extended period simulation, it is possible for nodes to become disconnected as links change status over time.

Negative pressures exist
When performing a demand driven analysis (DDA), GISpipe will issue a warning message when it encounters negative pressures at junctions that have positive demands. This usually indicates that there is some problem with the way the network has been designed or operated. Negative pressures can occur when portions of the network can only receive water through links that have been closed off. In such cases an additional warning message about the network being disconnected is also issued. Alternatively, a pressure driven analysis (PDA) can be performed to determine a hydraulic solution assuming a pressure-demand relationship at junctions. The hydraulic solution found will have reduced or zero demands and negative pressures will be largely eliminated. This is considered a more "realistic" solution since large negative pressures in a network are not physically realistic.

System unbalanced
A system unbalanced condition can occur when GISpipe cannot converge to a hydraulic solution in some time period within its allowed maximum number of trials. This situation can occur when valves, pumps, or pipelines keep switching their status from one trial to the next as the search for a hydraulic solution proceeds. For example, the pressure limits that control the status of a pump may be set too close together. Or a pump's head curve might be too flat causing it to keep shutting on and off. To eliminate the unbalanced condition one can, try to increase the allowed maximum number of trials or loosen the convergence accuracy requirement. Both of these parameters are set with the project's Hydraulic Options. If the unbalanced condition persists, then another hydraulic option, labelled "If Unbalanced", offers two ways to handle it.

1. One is to terminate the entire analysis once the condition is encountered.
2. The other is to continue seeking a hydraulic solution for another 10 trials with the status of all links frozen to their current values. If convergence is achieved then a warning message is issued about the system possibly being unstable. If convergence is not achieved then a "System unbalanced" warning message is issued. In either case, the analysis will proceed to the next time period.

If an analysis in a given time period ends with the system unbalanced then the user should recognize that the hydraulic results produced for this time period are inaccurate. Depending on circumstances, such as errors in flows into or out of storage tanks, this might affect the accuracy of results in all future periods as well.

Hydraulic equations unsolvable

Error 110 is issued if at some point in an analysis the set of equations that model flow and energy balance in the network cannot be solved. This can occur when some portion of a system demands water but has no links physically connecting it to any source of water. In such a case GISpipe will also issue warning messages about nodes being disconnected. The equations might also be unsolvable if unrealistic numbers were used for certain network properties.

9.6 Viewing and Interpreting Results

There are several ways in which database values and results of a simulation can be viewed directly on the Viewer Map. On the Analysis & Simulation toolbar drop down lists allows the selection of object, parameter and elements to visualise (see **Figure 9-90**).

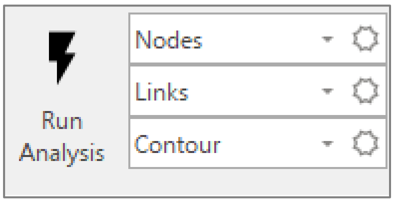


Figure 9-90: View results option toolbar

The Viewing options that can be selected and set are shown in **Figure 9-91**.

There are three types of map legends that can be displayed. The Node and Link Legends associate a colour with a range of values for the current parameter being viewed on the map (see **Figure 9-92**). The Time Legend displays the clock time of the simulation time period being viewed. To display the legend, select the parameter and to hide any of these legends double click with the left mouse button on the specific legend.

To move a legend to another location:

- Press the left mouse button over the legend.
- With the button held down, drag the legend to its new location and release the button.

To edit the Node Legend:

1. Right-click on the legend if it is visible.
2. Use the Legend Editor dialog form that appears (see **Figure 9-93**) to modify the legend's colours and intervals.

A similar method is used to edit the Link Legend.

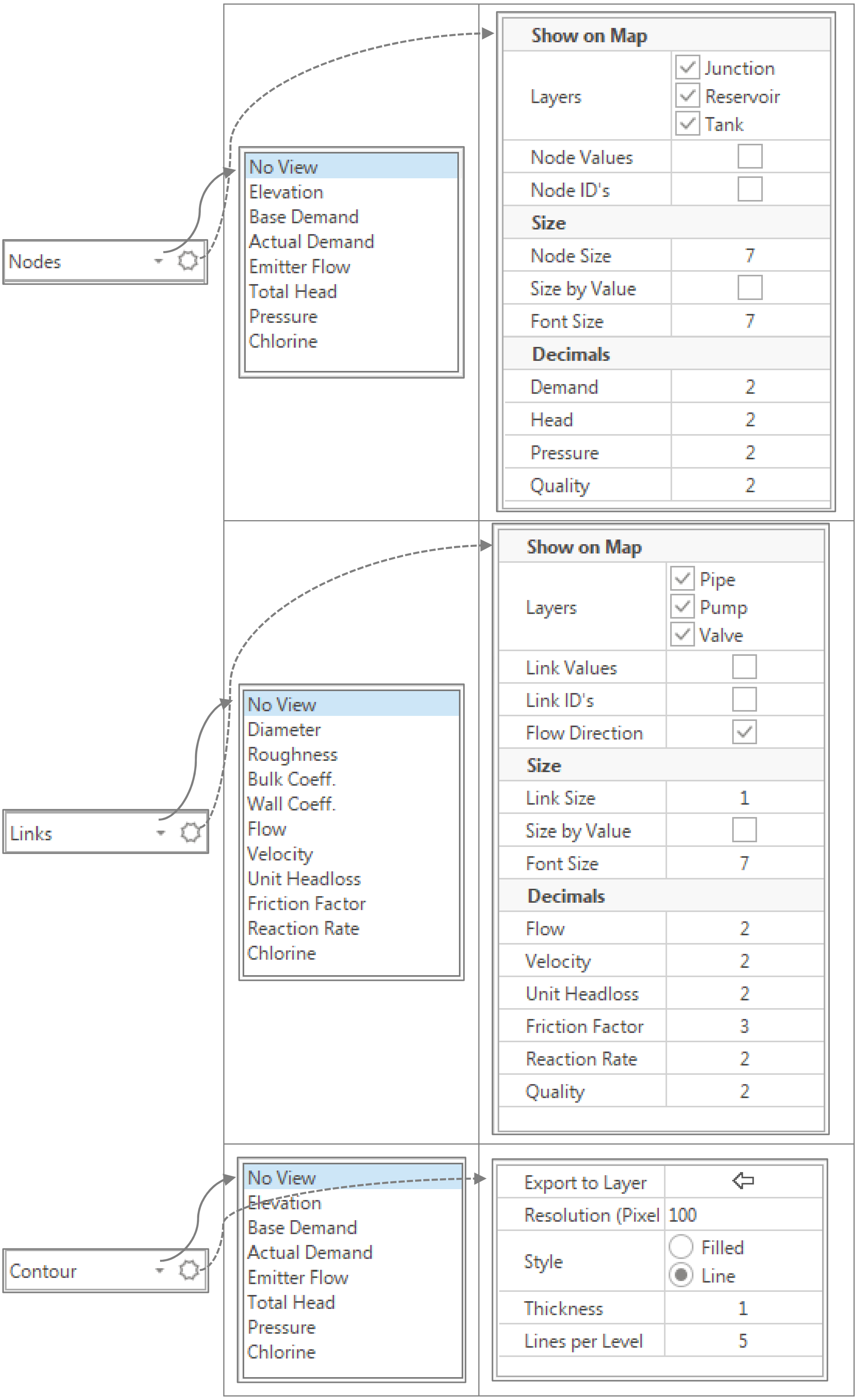
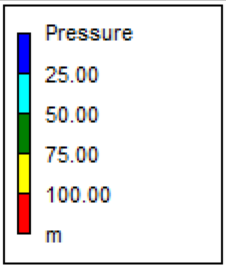
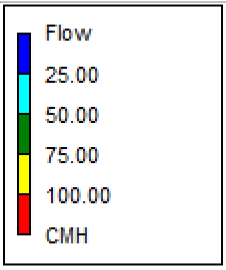


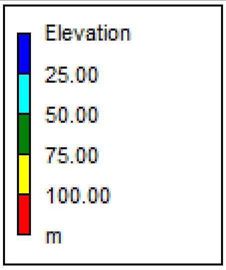
Figure 9-91: Viewing options



Setting Nodes view to Pressure



Setting Links view to Flow



Setting Contour view to Elevation

Figure 9-92: Legend options

The Legend Editor (**Figure 9-93**) is used to set numerical ranges to which different colours are assigned for viewing a particular parameter on the network map. It works as follows:

- Numerical values, in increasing order, are entered in the edit boxes to define the ranges. Not all four boxes need to have values.
- To change a colour, click on its colour band in the Editor and then select a new colour from the Colour Dialog box that will appear.
- Click the Equal Intervals button to assign ranges based on dividing the range of the parameter at the current time period into equal intervals.
- Click the Equal Quantiles button to assign ranges so that there are equal numbers of objects within each range, based on values that exist at the current time period.
- The Reverse Colours button reverses the ordering of the current set of colours (the colour in the lowest range becomes that of the highest range and so on).

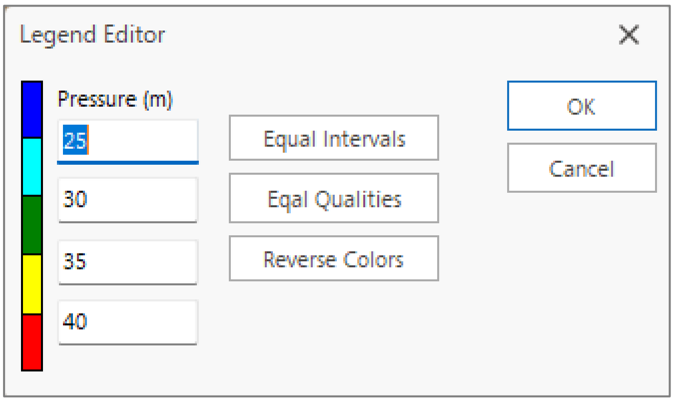


Figure 9-93: Legend Editor

A description of the options that could be displayed as shown in **Figure 9-91** is listed in **Table 9-31** and shown in an example in **Figure 9-94**.

Table 9-31: Display options on Viewer Map

Option	Description
Nodes	
Layer - Junction	Toggles visibility of junctions on the map
Layer - Reservoir	Toggles visibility of reservoirs on the map
Layer - Tank	Toggles visibility of tanks on the map
Node Values	Displays computed results (e.g., pressure, head) at each node
Node ID's	Displays identification labels for nodes
Size - Node Size	Controls the display size of nodes

Option	Description
Size - Size by Value	Adjusts node size dynamically based on hydraulic or quality values
Size - Font Size	Sets font size for node-related labels
Decimals - Demand	Sets decimal places for demand values at nodes
Decimals - Head	Controls decimal precision for hydraulic head values
Decimals - Pressure	Controls decimal places for pressure values
Decimals - Quality	Sets decimal precision for water quality at nodes
Links	
Layer - Pipe	Toggles visibility of pipes on the map
Layer - Pump	Toggles visibility of pumps on the map
Layer - Valve	Toggles visibility of valves on the map
Link Values	Displays computed values (e.g., flow, velocity) on each link
Link ID's	Displays the identification labels of links
Flow Direction	Shows arrows indicating direction of flow in pipes
Size - Link Size	Sets the display thickness of links on the map
Size - Size by Value	Adjusts link thickness dynamically based on hydraulic values
Size - Font Size	Sets font size for link-related labels
Decimals - Flow	Sets the number of decimal places shown for flow
Decimals - Velocity	Sets decimal precision for velocity values
Decimals - Unit Headloss	Controls decimal places for headloss per unit length
Decimals - Friction Factor	Controls display precision of the pipe friction factor
Decimals - Reaction Rate	Sets decimals for chemical reaction rates in links
Decimals - Quality	Sets decimal precision for water quality in links
Contours	
Export to Layer	Exports contour map to a GIS layer for visualization or further processing
Resolution (Pixel)	Sets the pixel resolution for the exported contour map
Style	Chooses between Filled or Line contour styles for the map
Thickness	Defines the line thickness for contour lines
Lines per Level	Specifies how many lines are drawn per contour level

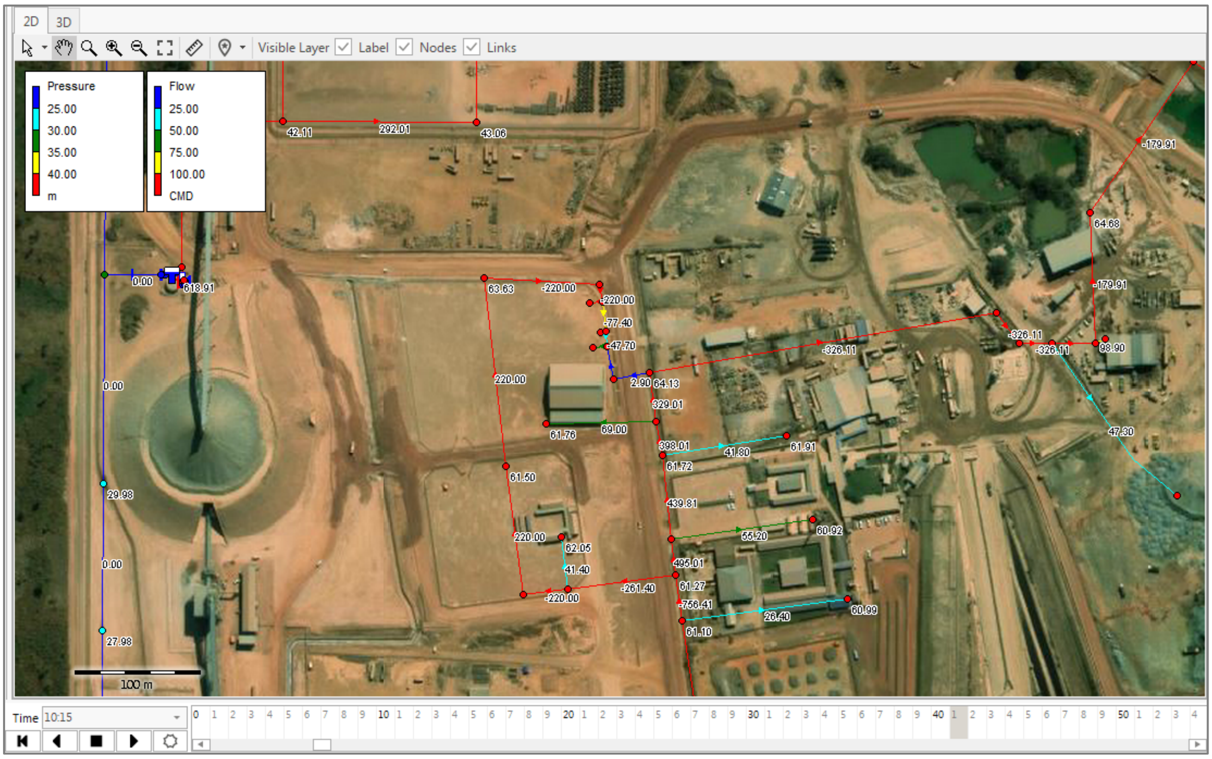


Figure 9-94: Map view hydraulic analysis results

The results as displayed in the map view is based on the simulation for that specific analysis period. The specific time period that the user wants to display can be selected from the timeline shown at the bottom of the screen, simply click on the time period block.

Alternatively, the time period can be selected from the drop-down list in the Time Controller (**Figure 9-95**).

The user also has the option to view an animation of the changes which occurs in the distribution network. The buttons in the Timer control can animate the map through time. Click the **Forward** button to start the animation and the **Stop** button to stop it. Click the **Option** button to view the Simulated Speed which can be set on the slider bar (+ increase or – decrease).

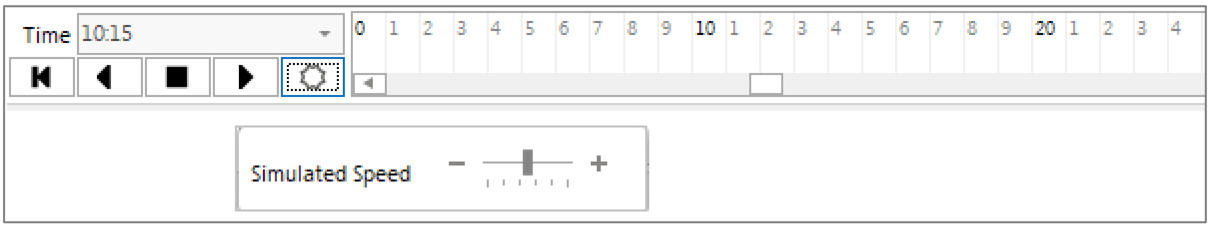


Figure 9-95: Time controller

The timeline at the bottom of the Map Viewer area will show each of the time periods analysed. If a block is light blue it indicates that there were some of the nodes that have experienced negative pressures during that period. Double-click the left mouse button on the block to reveal all the locations where negative pressures were experienced in the network as shown in **Figure 9-96**.

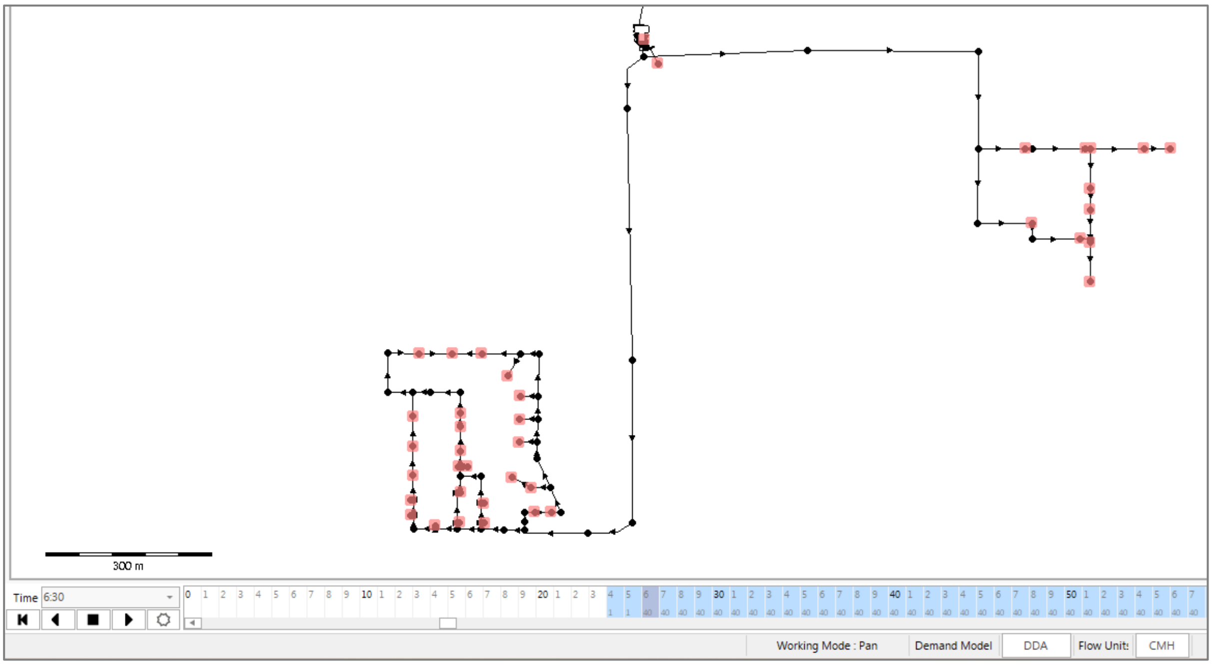


Figure 9-96: Extended Period Simulation - Results

9.6.1 Viewing Results with a Graph

Analysis results, as well as some design parameters, can be viewed using several different types of graphs. Graphs can be printed, copied to the Windows clipboard, or saved as a data file or Windows metafile. **Table 9-32** lists the types of graphs that can be used to view values for a selected parameter.

Table 9-32: Types of Graphs Available to View Results

Type of plot	Description	Applies to
Time Series Graph	Plots value versus time (Examples shown in **Figure 9-97** and **Figure 9-99**)	Specific nodes or links over all time periods
Profile Graph	Plots value versus distance (Example shown in **Figure 9-100**)	A list of nodes at a specific time
Frequency Graph	Plots value versus fraction of objects at or below the value (Example shown in **Figure 9-101** and **Figure 9-102**)	All nodes or links at a specific time or for all analysed time periods
Contour Plot	Shows regions of the map where values fall within specific intervals	All nodes at a specific time

Time Series graphs and Profile graphs require one or more objects be selected for plotting. To select items into the Time Series or Profile Graphs for plotting:

- Select the object (node or link) on the Viewer Map. (Time Series Graph Window will remain visible during this process).
- Click the *Add* button on the Time Series Graph Window to add the selected item to the list.

Time Series Graph

The parameters which can be selected to plot in a Time Series graph are listed in **Table 9-33**.

Table 9-33: Time series graph parameters

Object	Parameter
Nodes (**Figure 9-97**)	Actual Demand Total Head Pressure Quality Demand Deficit (PDA) Pressure Deficit (PDA)
Links (**Figure 9-99**)	Flow Velocity Unit headloss Friction factor Reaction rate Quality Status

An example of a Time Series Graph of nodes and the Pressure vs Time of 3 selected junctions is depicted in **Figure 9-97**.

The added nodes can also be selected and deleted by clicking on the *Delete* button and new nodes can be added by clicking on the *Add* button.

Clicking in the check boxes in the legend above the graph will show or hide the selected node.

The order of the selected nodes can also be changed by clicking on the *Move Up* or *Move Down* buttons.

Clicking the *Table* Tab (see **Figure 9-97**) switches to the same selected nodes but shows the results in table format as show in **Figure 9-98**.

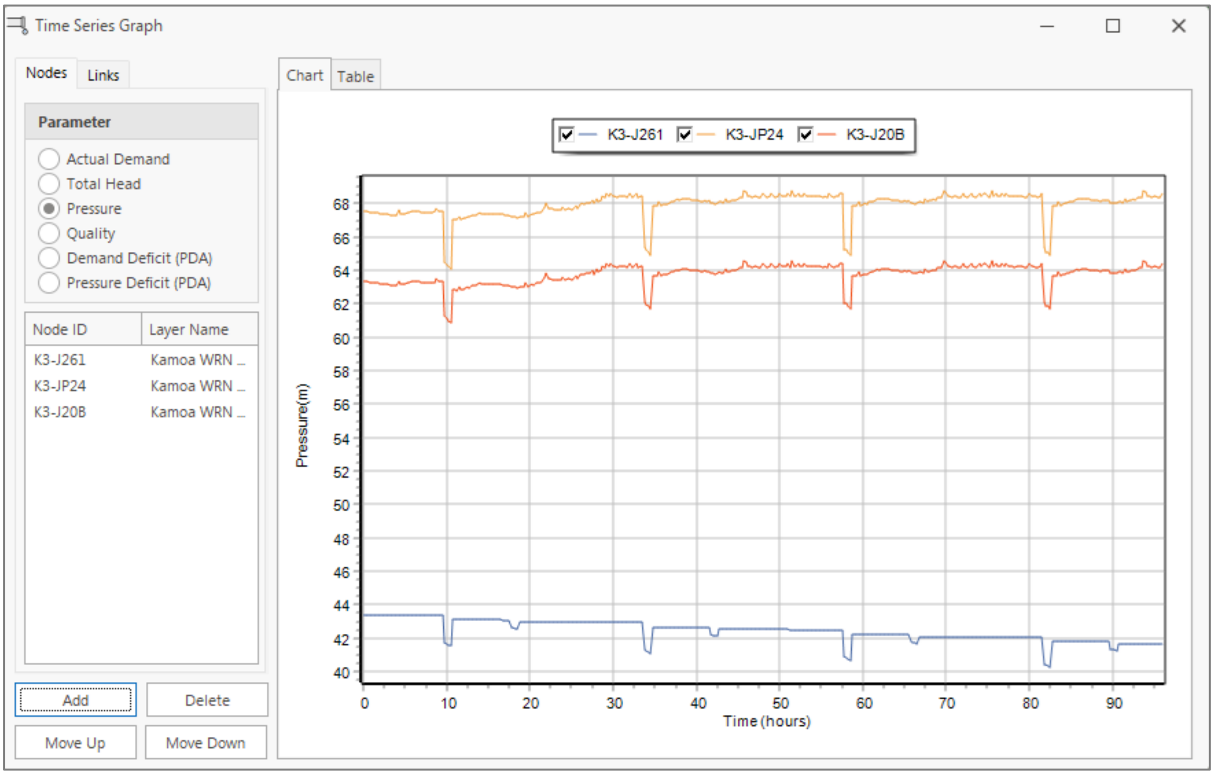
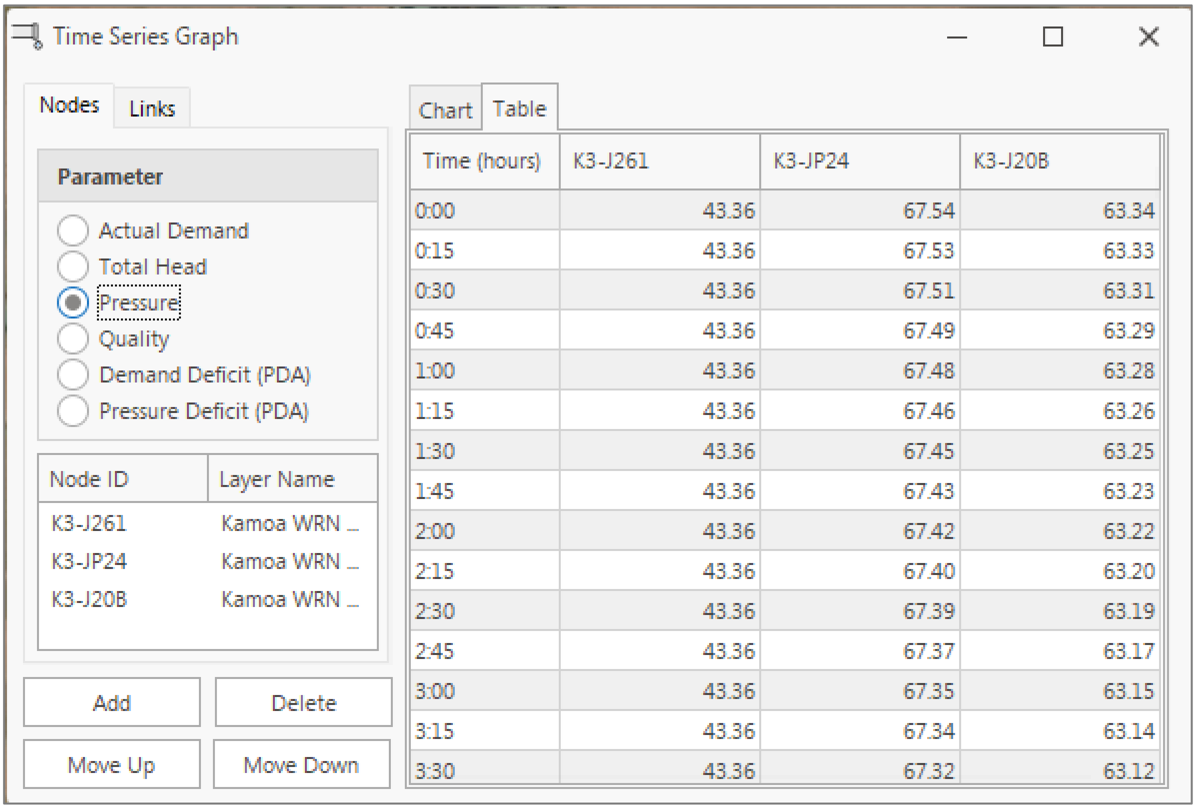


Figure 9-97: Time Series graph - Nodes



Time (hours)	K3-J261	K3-JP24	K3-J20B
0:00	43.36	67.54	63.34
0:15	43.36	67.53	63.33
0:30	43.36	67.51	63.31
0:45	43.36	67.49	63.29
1:00	43.36	67.48	63.28
1:15	43.36	67.46	63.26
1:30	43.36	67.45	63.25
1:45	43.36	67.43	63.23
2:00	43.36	67.42	63.22
2:15	43.36	67.40	63.20
2:30	43.36	67.39	63.19
2:45	43.36	67.37	63.17
3:00	43.36	67.35	63.15
3:15	43.36	67.34	63.14
3:30	43.36	67.32	63.12

Figure 9-98: Time Series graph - Nodes - Table format

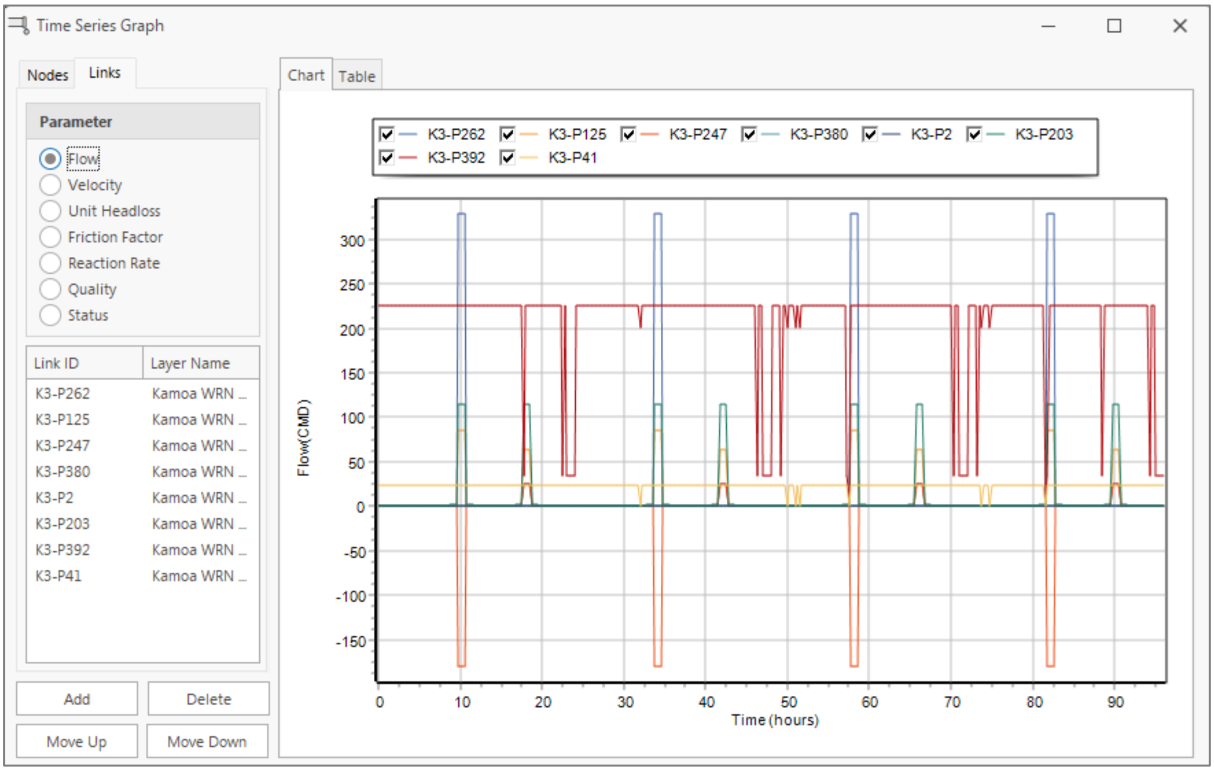


Figure 9-99: Time Series graph - Links

Profile Graph

To generate a Profile Graph the start and end node should be selected. Click on the Start node and then the *Add* button, select the end node and click on the *Add* button. All the nodes in between these nodes are automatically added and the graph is automatically drawn as shown in **Figure 9-100**.

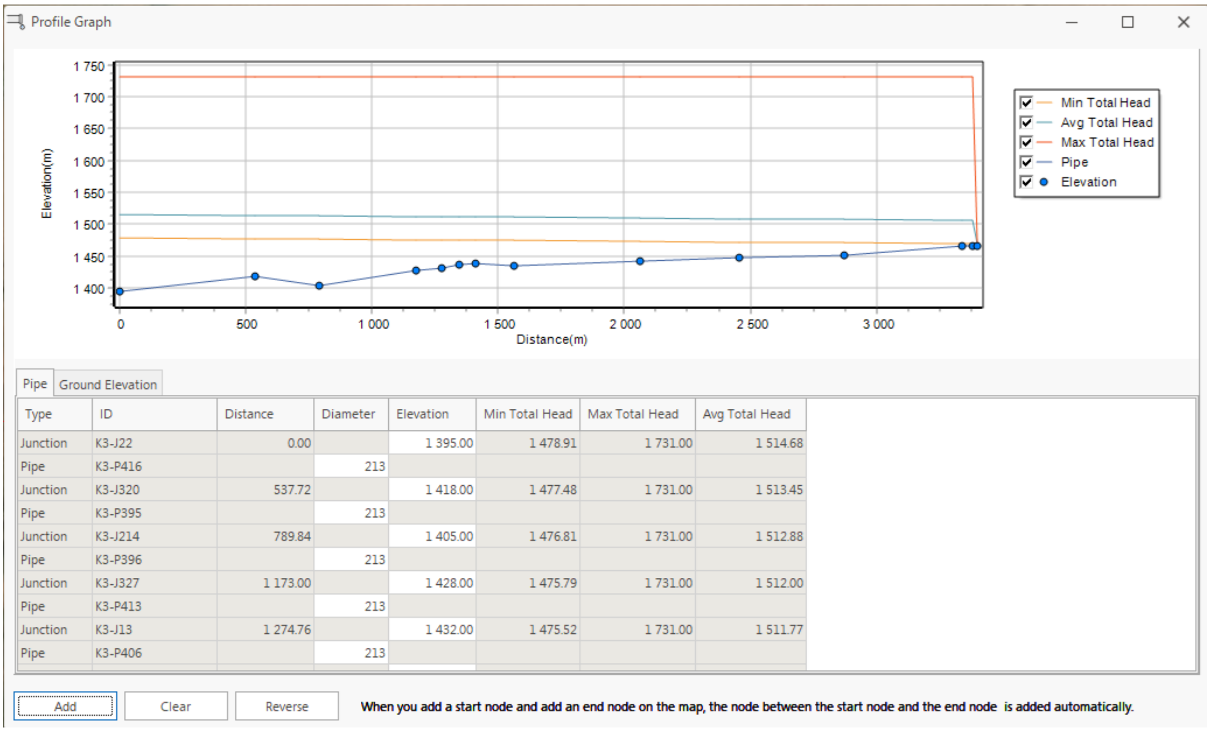


Figure 9-100: Profile graph

The graph displays the Min. Total Head, Avg. Total Head, Max. Total Head, Pipe and Elevation with option in the legend to display or hide through the toggle buttons.

The user also has the option to reverse the direction of the profile plot by clicking on the *Reverse* button. Clicking on the *Clear* button will delete all the displayed nodes and show a clear graph.

Frequency Graph

The Frequency Graph displays the value versus fraction of objects at or below the value as shown in **Figure 9-101** and **Figure 9-102**.

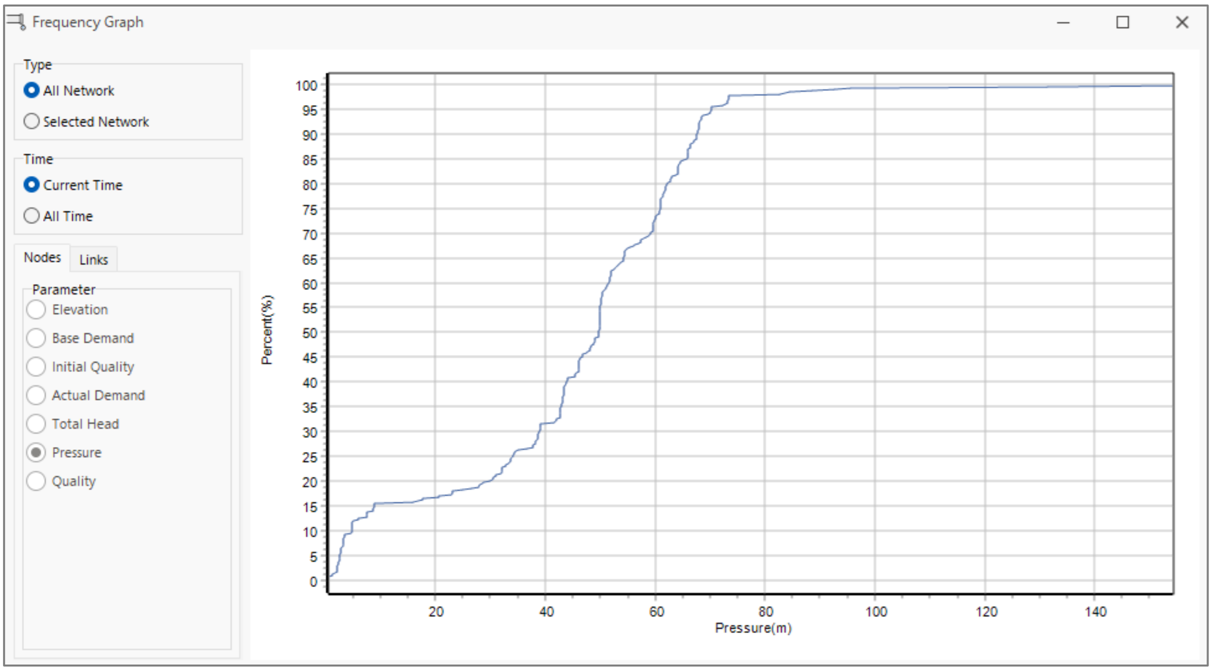


Figure 9-101: Frequency graph – Nodes - Pressure (Current time)

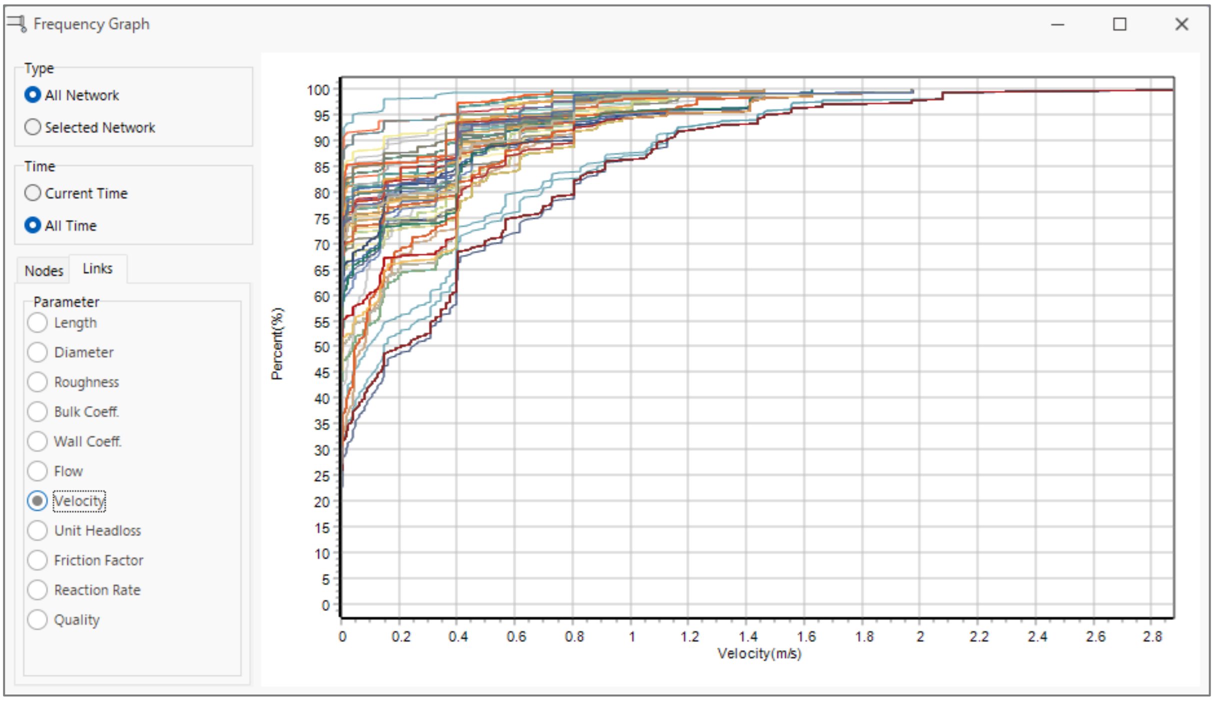


Figure 9-102: Frequency graph – Links - Velocity (All time)

When drawing the frequency plot the user has the option to set the Type:

- All network
- Selected network

This will allow user to select only a section of the network for which a frequency diagram is required

When drawing the frequency plot the user has the option to set the Time:

- Current time (see **Figure 9-101**)
- All time (see **Figure 9-102**)

This will allow user to select only a specific time period or draw a frequency diagram with all the time periods analysed.

Table 9-34: Frequency graph parameters

Object	Parameter
Nodes (**Figure 9-101**)	Elevation Base Demand Initial Quality Actual Demand Total Head Pressure Quality
Links (**Figure 9-102**)	Length Diameter Roughness Bulk coefficient Wall coefficient Flow Velocity Unit headloss Friction factor Reaction rate Quality

Note: A Time Series, Profile, or Frequency Graph can be zoomed by holding down the Shift key while drawing a zoom rectangle with the mouse's left button held down. Drawing the rectangle from left to right zooms in, drawing from right to left zooms out. The graph can also be panned in any direction by holding down the Ctrl key and moving the mouse across the plot with the right button held down.

Right click anywhere on the Graph (**Figure 9-97**, **Figure 9-99** and **Figure 9-101**) will show the various options to modify the General look (colours, views and title), Horizontal Axis (minimum, maximum, increments, grid lines & title), Vertical Axis (minimum, maximum, increments, grid lines & title), Legend (position, colour, size, framed & visibility) and Series (series, title, symbology of lines, markers and labels) as depicted in **Figure 9-103**.

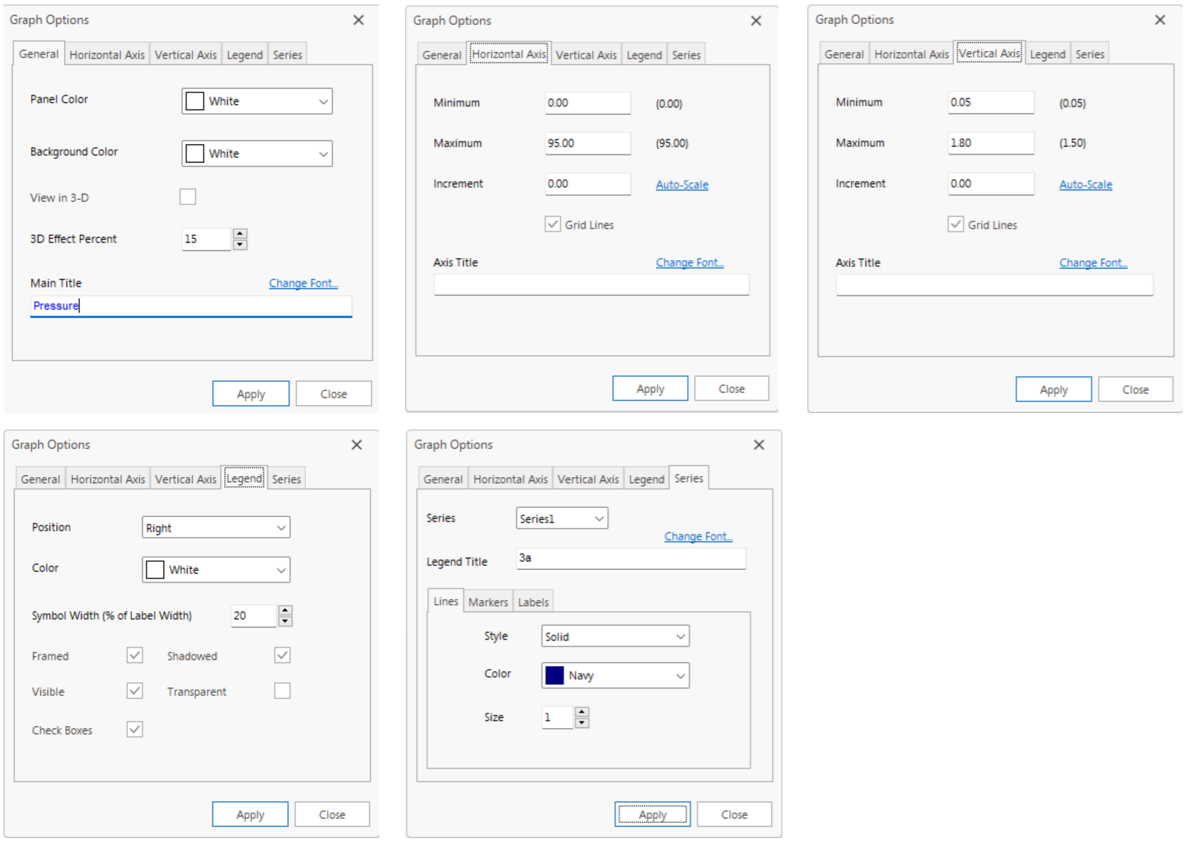


Figure 9-103: Graph options

The options contained on each tab of the Graph Options dialog window are described below in **Table 9-35**.

Table 9-35: Graph Options General, Horizontal and Vertical Axis and Legend Tabs

Option	Description
	General
Panel Colour	Colour of the panel which surrounds the graph's plotting area
Background Colour	Colour of graph's plotting area
View in 3D	Check if graph should be drawn in 3D
3D Effect Percent	Degree to which 3D effect is drawn
Main Title	Text of graph's main title
Font	Changes the font used for the main title
	Horizontal and Vertical Axis
Minimum	Sets minimum axis value (minimum data value is shown in parentheses). Can be left blank.
Maximum	Sets maximum axis value (maximum data value is shown in parentheses). Can be left blank.
Increment	Sets increment between axis labels. Can be left blank.
Auto Scale	If checked then Minimum, Maximum, and Increment settings are ignored.
Gridlines	Selects type of gridline to draw.
Axis Title	Text of axis title
Font	Click to select a font for the axis title.
	Legend
Position	Selects where to place the legend.

Option	Description
Colour	Selects colour to use for legend background.
Symbol Width	Selects width to use (in pixels) to draw symbol portion of the legend.
Framed	Places a frame around the legend.
Visible	Makes the legend visible.
Position	Selects where to place the legend.

Click the *Font* button to change the font used for the legend. (Other legend properties are selected on the Legend page of the dialog.)

Select a property of the data series you would like to modify. The choices are: Lines, Markers, Patterns and Labels. (Not all properties are available for some types of graphs.)

The data series properties that can be modified on the Series tab are listed in **Table 9-36**.

Table 9-36: Graph Options Series Tab

Category	Option	Description
Lines	Style	Selects line style.
	Colour	Selects line colour.
	Size	Selects line thickness (only for solid line style).
	Visible	Determines if line is visible.
Markers	Style	Selects marker style.
	Colour	Selects marker colour.
	Size	Selects marker size.
	Visible	Determines if marker is visible.
Patterns	Style	Selects pattern style.
	Colour	Selects pattern colour.
	Stacking	Not used with GISpipe.
Labels	Style	Selects what type of information is displayed in the label.
	Colour	Selects the colour of the label's background.
	Transparent	Determines if graph shows through label or not.
	Show Arrows	Determines if arrows are displayed on pie charts.
	Visible	Determines if labels are visible or not.

Contour plot

Using the View results option toolbar, **Figure 9-90**, the user can select to show the contours. This is a for example a very useful plot to identify specific areas of high or low pressures as shown in **Figure 9-104**.

The created contour map can also be exported to a separate layer by clicking on the *Export to Layer* option see **Figure 9-91**.

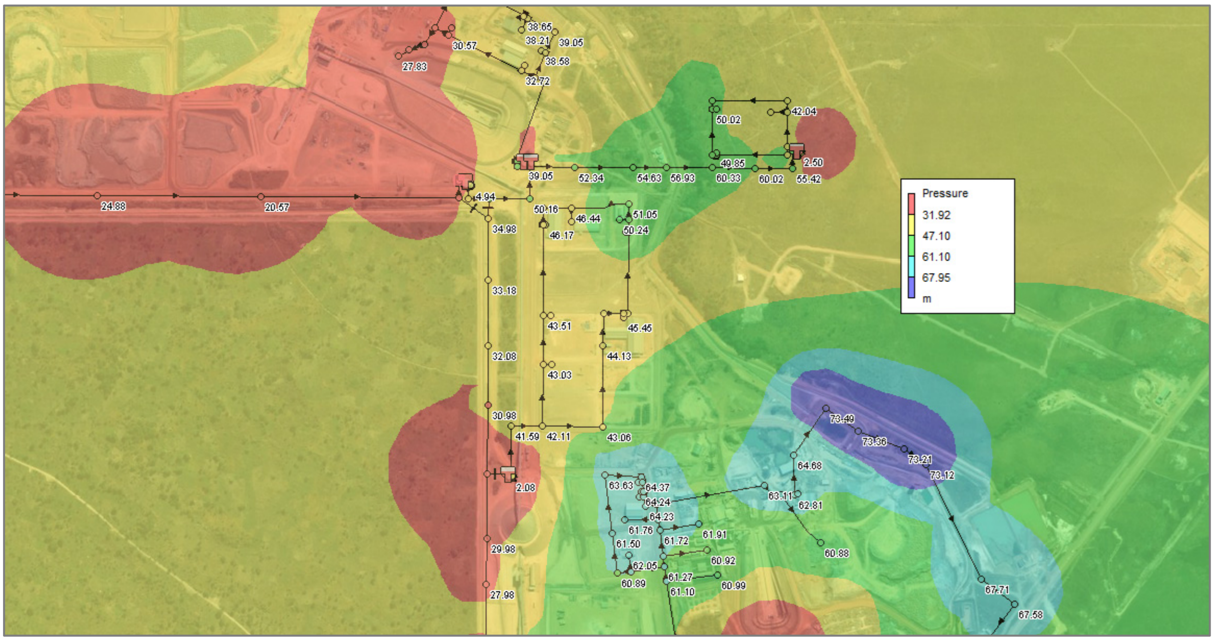


Figure 9-104: Contour plot

9.6.2 Viewing Results in Tables

There are numerous ways to generate and view the results in table format. When viewing a graph (**Figure 9-97**), clicking the Table Tab switches to the same selected nodes but shows the results in table format as show in **Figure 9-98**.

Alternatively selecting the junctions and/or links on the Map viewer window, right clicking on the window and selecting View Selected Shapes Table from the pop-up menu, see **Figure 9-105**, will show the results in a table format, as shown in **Figure 9-106**.

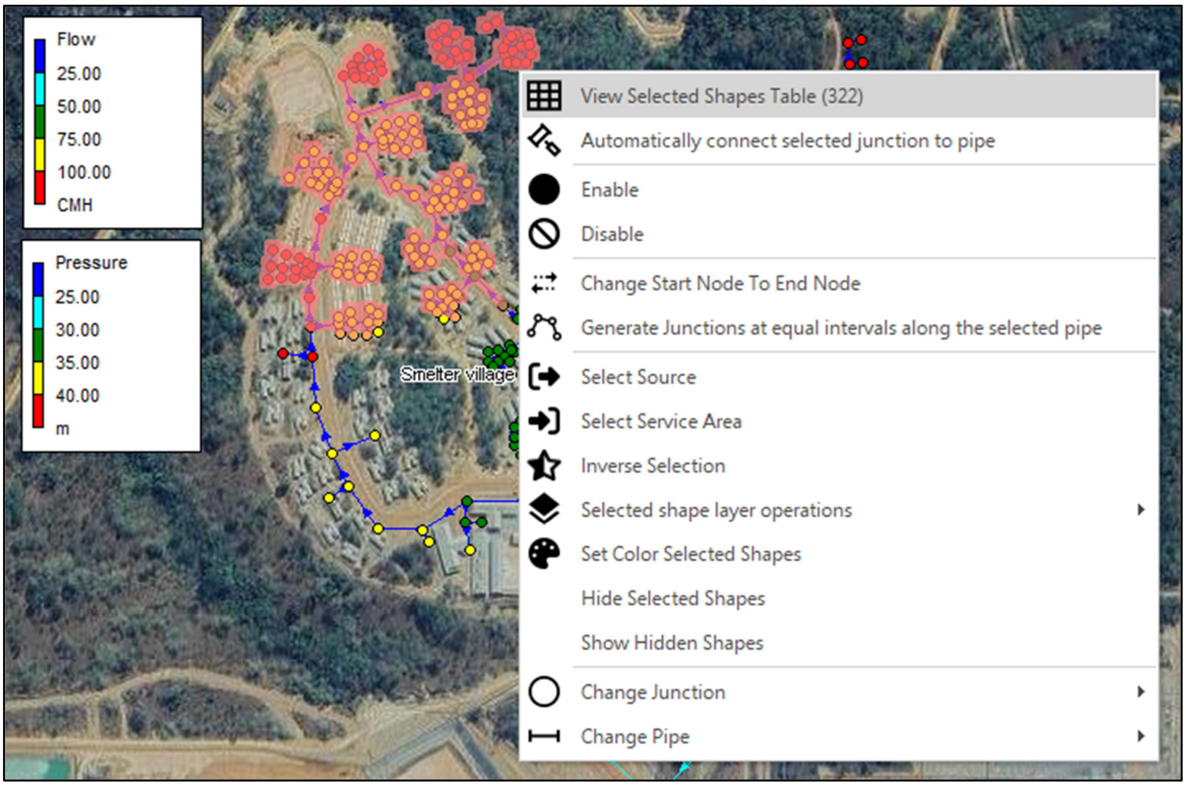
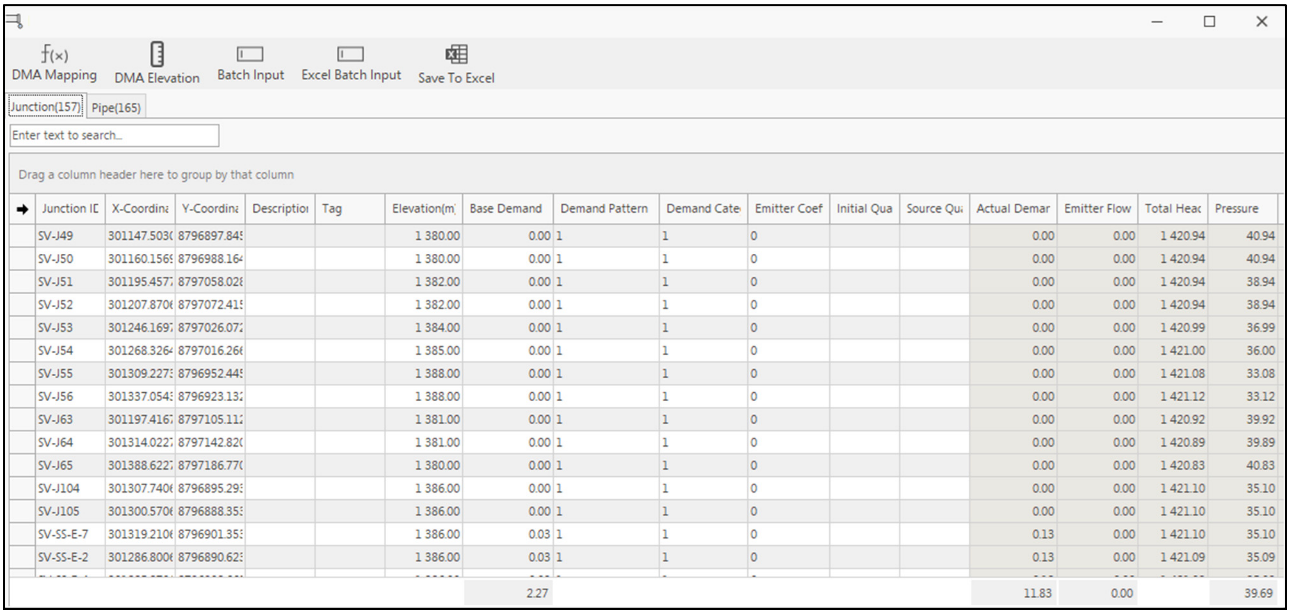


Figure 9-105: Selecting junctions and pipes



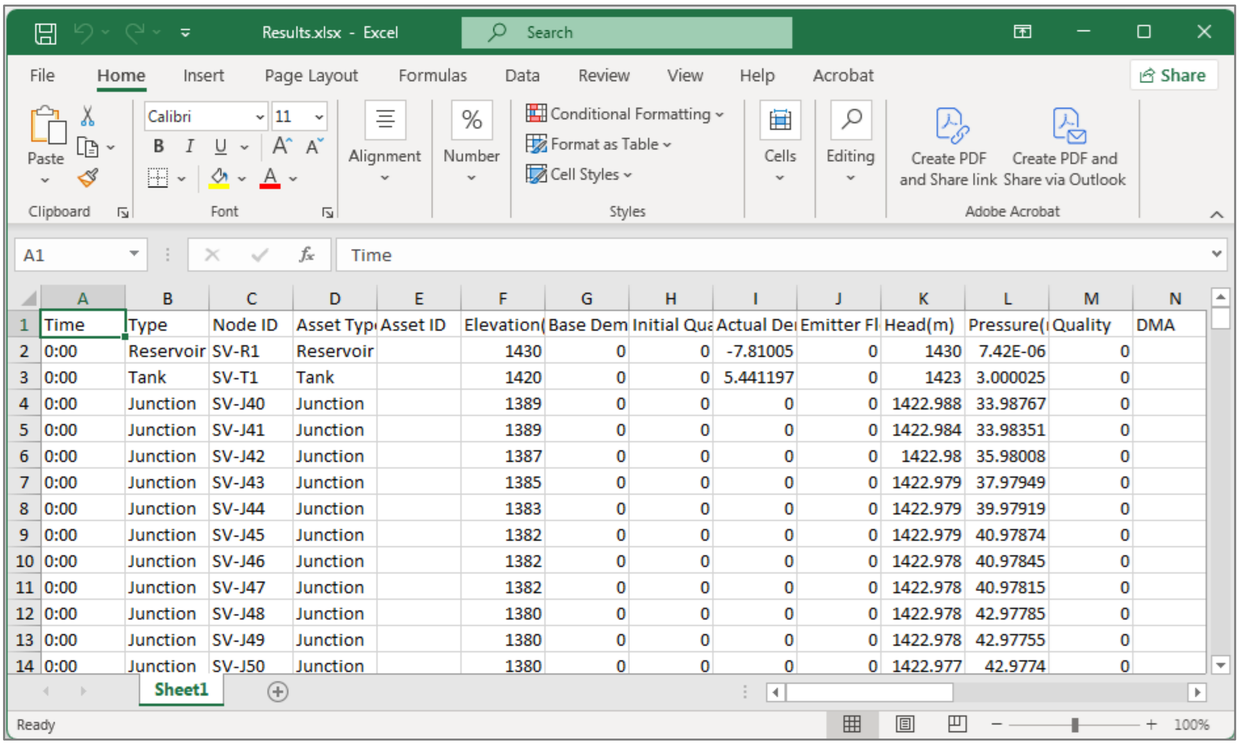
Junction ID	X-Coordina	Y-Coordina	Descriptio	Tag	Elevation(m)	Base Demand	Demand Pattern	Demand Cate	Emitter Coef	Initial Qua	Source Qua	Actual Demar	Emitter Flow	Total Heac	Pressure
SV-J49	301147.503(	8796897.84!			1 380.00	0.00	1	1	0			0.00	0.00	1 420.94	40.94
SV-J50	301160.156!	8796988.16			1 380.00	0.00	1	1	0			0.00	0.00	1 420.94	40.94
SV-J51	301195.457	8797058.02			1 382.00	0.00	1	1	0			0.00	0.00	1 420.94	38.94
SV-J52	301207.870(	8797072.41			1 382.00	0.00	1	1	0			0.00	0.00	1 420.94	38.94
SV-J53	301246.169	8797026.07			1 384.00	0.00	1	1	0			0.00	0.00	1 420.99	36.99
SV-J54	301268.326	8797016.26(			1 385.00	0.00	1	1	0			0.00	0.00	1 421.00	36.00
SV-J55	301309.227	8796952.44!			1 388.00	0.00	1	1	0			0.00	0.00	1 421.08	33.08
SV-J56	301337.054	8796923.13			1 388.00	0.00	1	1	0			0.00	0.00	1 421.12	33.12
SV-J63	301197.416	8797105.11			1 381.00	0.00	1	1	0			0.00	0.00	1 420.92	39.92
SV-J64	301314.022	8797142.82(			1 381.00	0.00	1	1	0			0.00	0.00	1 420.89	39.89
SV-J65	301388.622	8797186.77(			1 380.00	0.00	1	1	0			0.00	0.00	1 420.83	40.83
SV-J104	301307.740(	8796895.29			1 386.00	0.00	1	1	0			0.00	0.00	1 421.10	35.10
SV-J105	301300.570(	8796888.35			1 386.00	0.00	1	1	0			0.00	0.00	1 421.10	35.10
SV-SS-E-7	301319.210(	8796901.35			1 386.00	0.03	1	1	0			0.13	0.00	1 421.10	35.10
SV-SS-E-2	301286.800(	8796890.62			1 386.00	0.03	1	1	0			0.13	0.00	1 421.09	35.09
						2.27						11.83	0.00		39.69

Figure 9-106: Viewing results in table format

It must be remembered that the results shown would only be for the specific time step in an extended period simulation.

9.6.3 Exporting Simulation Results

On the *Report* Toolbar click on the *Export* button to save all the simulated results to a Microsoft Excel file, **Figure 9-107**.

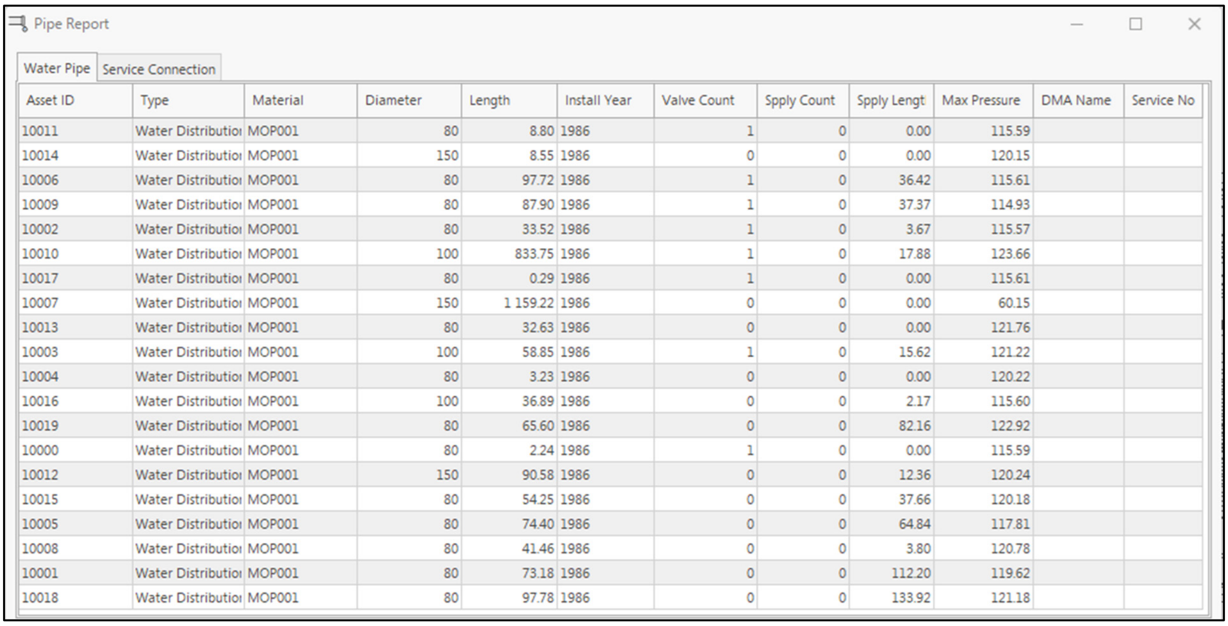


	A	B	C	D	E	F	G	H	I	J	K	L	M	N
1	Time	Type	Node ID	Asset Typ	Asset ID	Elevation(	Base Dem	Initial Qua	Actual De	Emitter Fl	Head(m)	Pressure(	Quality	DMA
2	0:00	Reservoir	SV-R1	Reservoir		1430	0	0	-7.81005	0	1430	7.42E-06	0	
3	0:00	Tank	SV-T1	Tank		1420	0	0	5.441197	0	1423	3.000025	0	
4	0:00	Junction	SV-J40	Junction		1389	0	0	0	0	1422.988	33.98767	0	
5	0:00	Junction	SV-J41	Junction		1389	0	0	0	0	1422.984	33.98351	0	
6	0:00	Junction	SV-J42	Junction		1387	0	0	0	0	1422.98	35.98008	0	
7	0:00	Junction	SV-J43	Junction		1385	0	0	0	0	1422.979	37.97949	0	
8	0:00	Junction	SV-J44	Junction		1383	0	0	0	0	1422.979	39.97919	0	
9	0:00	Junction	SV-J45	Junction		1382	0	0	0	0	1422.979	40.97874	0	
10	0:00	Junction	SV-J46	Junction		1382	0	0	0	0	1422.978	40.97845	0	
11	0:00	Junction	SV-J47	Junction		1382	0	0	0	0	1422.978	40.97815	0	
12	0:00	Junction	SV-J48	Junction		1380	0	0	0	0	1422.978	42.97785	0	
13	0:00	Junction	SV-J49	Junction		1380	0	0	0	0	1422.978	42.97755	0	
14	0:00	Junction	SV-J50	Junction		1380	0	0	0	0	1422.977	42.9774	0	

Figure 9-107: Export to Microsoft Excel - Results

9.6.4 Pipes report

On the *Report* Toolbar click on the *Pipes* button to view the water pipes results, **Figure 9-108**. The results in each of the columns are listed in **Table 9-37**.



Asset ID	Type	Material	Diameter	Length	Install Year	Valve Count	Spply Count	Spply Lengt	Max Pressure	DMA Name	Service No
10011	Water Distributio	MOP001	80	8.80	1986	1	0	0.00	115.59		
10014	Water Distributio	MOP001	150	8.55	1986	0	0	0.00	120.15		
10006	Water Distributio	MOP001	80	97.72	1986	1	0	36.42	115.61		
10009	Water Distributio	MOP001	80	87.90	1986	1	0	37.37	114.93		
10002	Water Distributio	MOP001	80	33.52	1986	1	0	3.67	115.57		
10010	Water Distributio	MOP001	100	833.75	1986	1	0	17.88	123.66		
10017	Water Distributio	MOP001	80	0.29	1986	1	0	0.00	115.61		
10007	Water Distributio	MOP001	150	1 159.22	1986	0	0	0.00	60.15		
10013	Water Distributio	MOP001	80	32.63	1986	0	0	0.00	121.76		
10003	Water Distributio	MOP001	100	58.85	1986	1	0	15.62	121.22		
10004	Water Distributio	MOP001	80	3.23	1986	0	0	0.00	120.22		
10016	Water Distributio	MOP001	100	36.89	1986	0	0	2.17	115.60		
10019	Water Distributio	MOP001	80	65.60	1986	0	0	82.16	122.92		
10000	Water Distributio	MOP001	80	2.24	1986	1	0	0.00	115.59		
10012	Water Distributio	MOP001	150	90.58	1986	0	0	12.36	120.24		
10015	Water Distributio	MOP001	80	54.25	1986	0	0	37.66	120.18		
10005	Water Distributio	MOP001	80	74.40	1986	0	0	64.84	117.81		
10008	Water Distributio	MOP001	80	41.46	1986	0	0	3.80	120.78		
10001	Water Distributio	MOP001	80	73.18	1986	0	0	112.20	119.62		
10018	Water Distributio	MOP001	80	97.78	1986	0	0	133.92	121.18		

Figure 9-108: Pipes report

Table 9-37: Pipes report results

Column	Description
Asset ID	Unique identifier for the pipe, usually linked to the GIS asset register
Type	Classification of the pipe (e.g., Water Distribution)
Material	Pipe construction material
Diameter	Internal diameter of the pipe
Length	Length of the pipe segment
Install Year	Year the pipe was installed, used for age tracking and renewal planning
Valve Count	Number of valves located on or associated with the pipe
Supply Count	Number of direct customer connections served by the pipe
Supply Length	Total length of service connections attached to the pipe
Max Pressure	Maximum pressure experienced in the pipe during simulation
DMA Name	District Metered Area (DMA) that the pipe belongs to
Service No	Number of individual service connections (customer meters) linked to the pipe

9.6.5 Energy report

On the *Report* Toolbar click on the *Energy* button to view the Energy Report results, **Figure 9-109**. This report summarizes the energy performance of pumps in the network, helping identify high energy users, inefficient pumps, and opportunities for cost savings.

The results in each of the columns are listed in **Table 9-38**.

Selecting the Graph Tab (**Figure 9-109**), provides a visual comparison of energy and efficiency performance across all pumps in the system, making it easy to identify the most energy-intensive or least efficient pumps, see **Figure 9-110**. This is done by providing 5 different graph options similar to that in **Table 9-38**.

Energy Report

Table | Graph

Pump	Percent Utiliztion	Average Efficiency	Kw-hr/m³	Average Kwatts	Peak Kwatts	Cost/day
K3-6351-PP-001A	100.00	75.00	1.11	32.27	35.49	0.00
K3-6351-PP-001B	100.00	75.00	1.12	32.10	35.30	0.00
K3-PC1-SB	0.00	0.00	0.00	0.00	0.00	0.00
K3-PC1	100.00	75.00	0.21	12.06	40.82	0.00
K3-PB1-SB	0.00	0.00	0.00	0.00	0.00	0.00
K3-PB1	100.00	75.00	0.23	3.74	45.96	0.00
K3-PA1-SB	0.00	0.00	0.00	0.00	0.00	0.00
K3-PA1	100.00	75.00	0.22	6.30	49.01	0.00
K3-PG	100.00	75.00	0.15	0.07	0.68	0.00
K3-PG_SB	0.00	0.00	0.00	0.00	0.00	0.00

Figure 9-109: Energy Report - Table output

Table 9-38: Energy reports - Output

Field	Description
Pump	Identifier of the pump being analysed
Percent Utilization	Percentage of the total simulation time that the pump was operating
Average Efficiency	Average hydraulic-to-mechanical efficiency (%) of the pump during operation.
kW-hr/m³	Energy consumed per unit volume of water pumped (kilowatt-hours per cubic metre)
Average kWatts	Average power consumption (kW) of the pump over the simulation period
Peak kWatts	Maximum (peak) power demand (kW) recorded during the simulation
Cost/day	Daily cost of operating the pump, based on energy consumption and tariff data

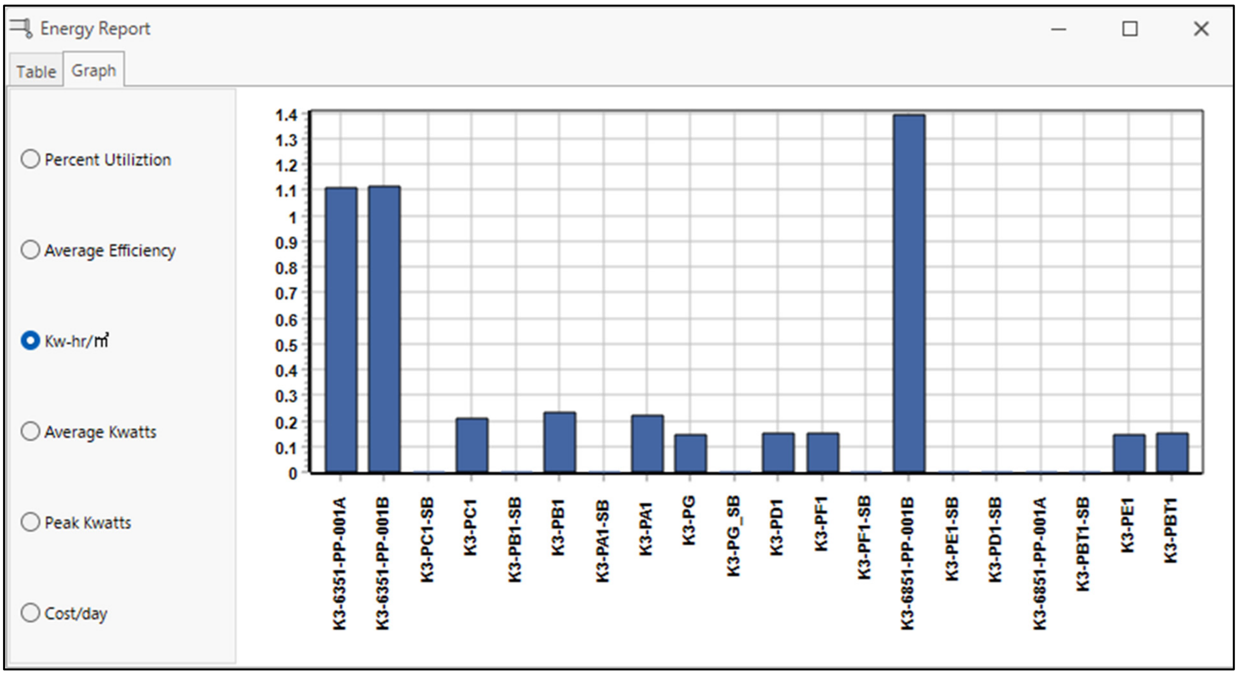


Figure 9-110: Energy Report - Graph output

9.7 Other Features

On the Results Toolbar click on the *Export* button to save all the simulated results to a Microsoft Excel file, **Figure 9-107**.

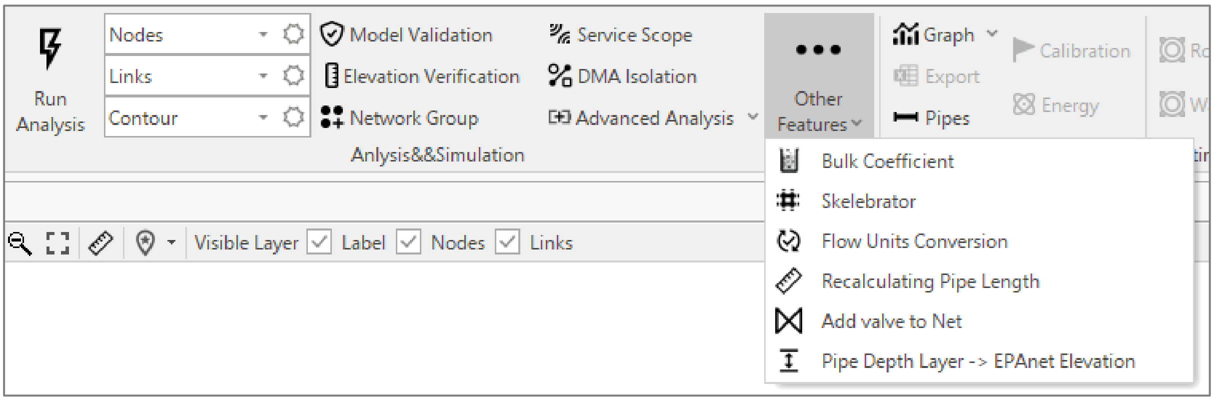


Figure 9-111: Other Features

9.7.1 Bulk Coefficient

The Bulk Coefficient window, as shown in **Figure 9-112**, helps users determine how the bulk reaction coefficient (Kb), used in water quality modelling, changes with temperature. This is especially relevant for chlorine decay and other disinfectant reactions.

This tool is selected by clicking on the *Other Features* button on the *Analysis & Simulation* toolbar and then click on the *Bulk Coefficient* button.

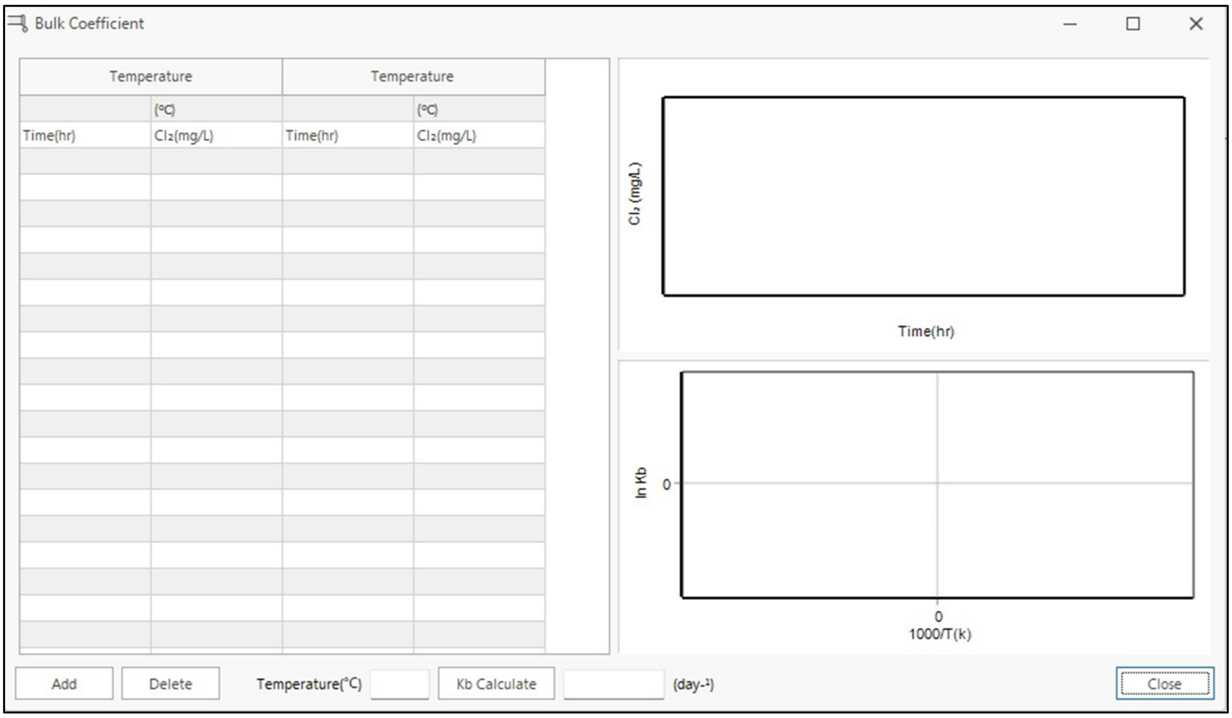


Figure 9-112: Bulk Coefficient

A detailed breakdown of the inputs and outputs are listed in **Table 9-39**.

Table 9-39: Input and output of Bulk Coefficient

Parameter	**Description**
Input	
Time (hr)	The elapsed time of the test, in hours, for chlorine decay measurements.
Cl_2 (mg/L)	The measured chlorine concentration at that time step. These values are typically obtained from laboratory or field tests where a water sample is monitored over time.
Temperature (°C)	The temperature at which the chlorine decay test is performed. Multiple sets of decay tests can be run at different temperatures to establish temperature dependence.
Add	This button adds a new row of input data (time, chlorine concentration, and temperature).
Delete	Removes a selected row of input data.
Temperature (°C)	Sets or edits the test temperature associated with the dataset.
Kb Calculate	Runs the regression analysis to calculate the bulk reaction coefficient (Kb) for each dataset.
Results	
Graph (Cl_2 vs Time)	Displays chlorine concentration decay over time for the entered dataset. The slope of the decay curve is used to estimate the bulk coefficient.
Graph (ln Kb vs 1000/T)	Plots the natural logarithm of the bulk coefficient against inverse temperature (Arrhenius plot). This shows how reaction rates depend on temperature and allows extrapolation to other temperatures.
Calculated Kb (day^{-1})	The computed bulk reaction coefficient, shown numerically after analysis. This coefficient is then used in water quality simulations (e.g., chlorine decay modelling).

9.7.2 Skelebrator

The Skelebrator tool, as shown in **Figure 9-113**, is used to simplify a complex water distribution network by selectively removing less significant parts of the system. This process is commonly referred to as network skeletonization, and it helps reduce computational complexity while retaining the essential hydraulic characteristics of the network.

This tool is selected by clicking on the *Other Features* button on the *Analysis & Simulation* toolbar and then click on the *Skelebrator* button.

The Skelebrator simplifies the network by:

- Removing pipes with diameters smaller than a user-defined threshold.
- Merging junctions where there are minimal elevation differences.
- Focusing either on the entire network or a selected portion.

This is especially useful for large, detailed models where a more abstracted version is needed for planning or scenario analysis. The information required to apply the skeletonization is listed in **Table 9-40**.

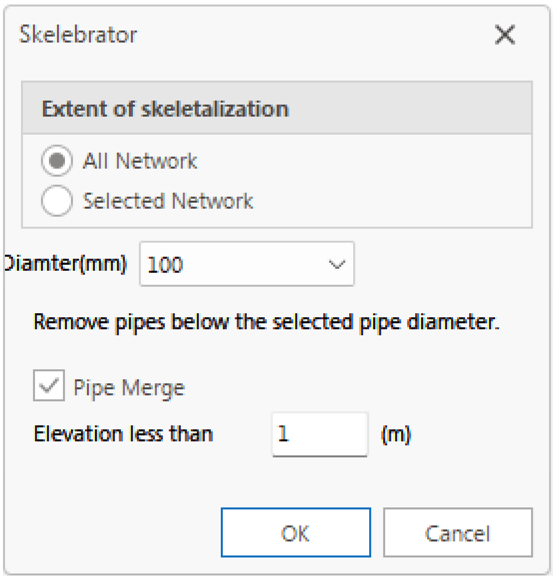


Figure 9-113: Skelebrator option

Table 9-40: Skelebrator input

Option	Description
Extent of Skeletalization	Choose between applying simplification to the entire network or just a selected portion.
Diameter	Select, from a dropdown list, the pipes with diameters smaller than this value that should be removed from the model.
Pipe Merge	When enabled, the software merges connected pipes at junctions where the elevation change is minor.
Elevation less than	Specifies the elevation difference threshold below which junctions can be merged.

9.7.3 Flow Units Conversion

This tool is selected by clicking on the *Other Features* button on the *Analysis & Simulation* toolbar and then click on the *Flow Units Conversion* button. The project is set to a specific flow unit (**see Appendix B**). This tool allows the user to change the units and, in this process, change all the demand values to the new units. The Flow Units Conversion window, **Figure 9-114**, will show the current flow units (Source Units) and provide a drop-down list (Target Units) of flow units that the user can select from. Click on the *Ok* button to confirm the conversion.

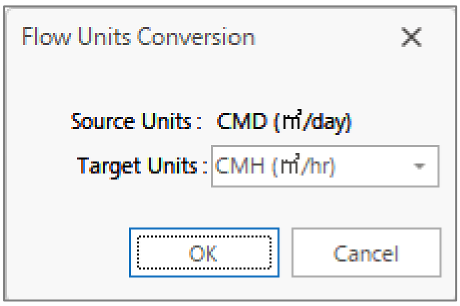


Figure 9-114: Flow unit conversion

9.7.4 Pipe Length Recalculation

This tool is selected by clicking on the *Other Features* button on the *Analysis & Simulation* toolbar and then click on the *Recalculating Pipe Length* button. Based on the scale all pipe length will be recalculated. Once complete a confirmation pop-up window will be shown, **Figure 9-115**, to confirm the procedure.

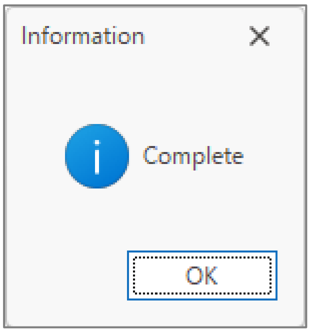


Figure 9-115: Completed Pipe Recalculating Length procedure

9.7.5 Add Valve to Net

Valves can be added to the network model by importing a valve layer with the positions of the valves. The valve can be selected in the Viewer map see **Figure 9-116**.

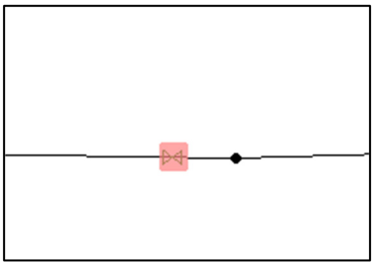


Figure 9-116: Selecting a valve on the Valve layer

This Add Valve to Net tool is selected by clicking on the *Other Features* button on the *Analysis & Simulation* toolbar and then click on the *Add Valve to Net* button. The valve will be inserted into the network model as shown in **Figure 9-117** .

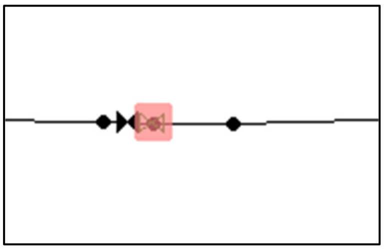


Figure 9-117: Valve added to network

9.7.6 Pipe Depth to EPANET Elevation

This tool is selected by clicking on the *Other Features* button on the *Analysis & Simulation* toolbar and then click on the *Pipe Depth Layer -> EPAnet Elevation* button. This function automatically converts GIS pipe depth and surface elevation data into EPANET junction elevations, ensuring the hydraulic model reflects real-world ground levels.

From GIS elevation data (DEM, contour layers, or pipe survey data), the surface elevation at each pipe/junction location is determined. A Pipe Depth layer is required and when the network model is selected click on the *Pipe Depth Layer -> EPAnet Elevation* button to show the dialogue box shown in **Figure 9-115**. Select if the information should be extracted from the DEM or the pipe depth data layer. Select if the entire network (*All Network* radio button) or only a *Selected Network* should be modified.

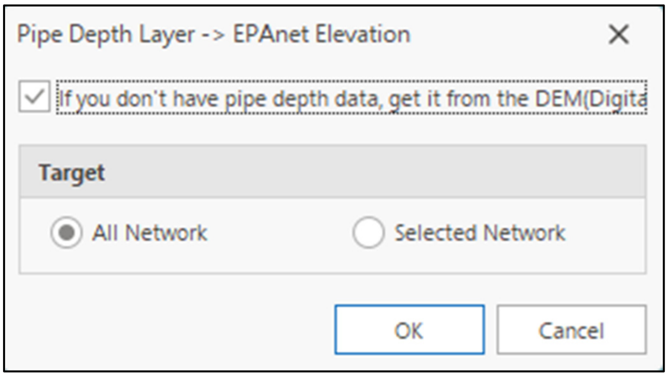


Figure 9-118: Pipe Depth Layer -> EPAnet Elevation tool

9.8 Optimization

On the *Optimization* toolbar there are two options to select from *Roughness* and *Wall Coeff*.

9.8.1 Roughness automatic calibration

Selecting *Roughness* shows the Roughness automatic calibration window as depicted in **Figure 9-119**. The various input and results parameters are listed in **Table 9-41**.

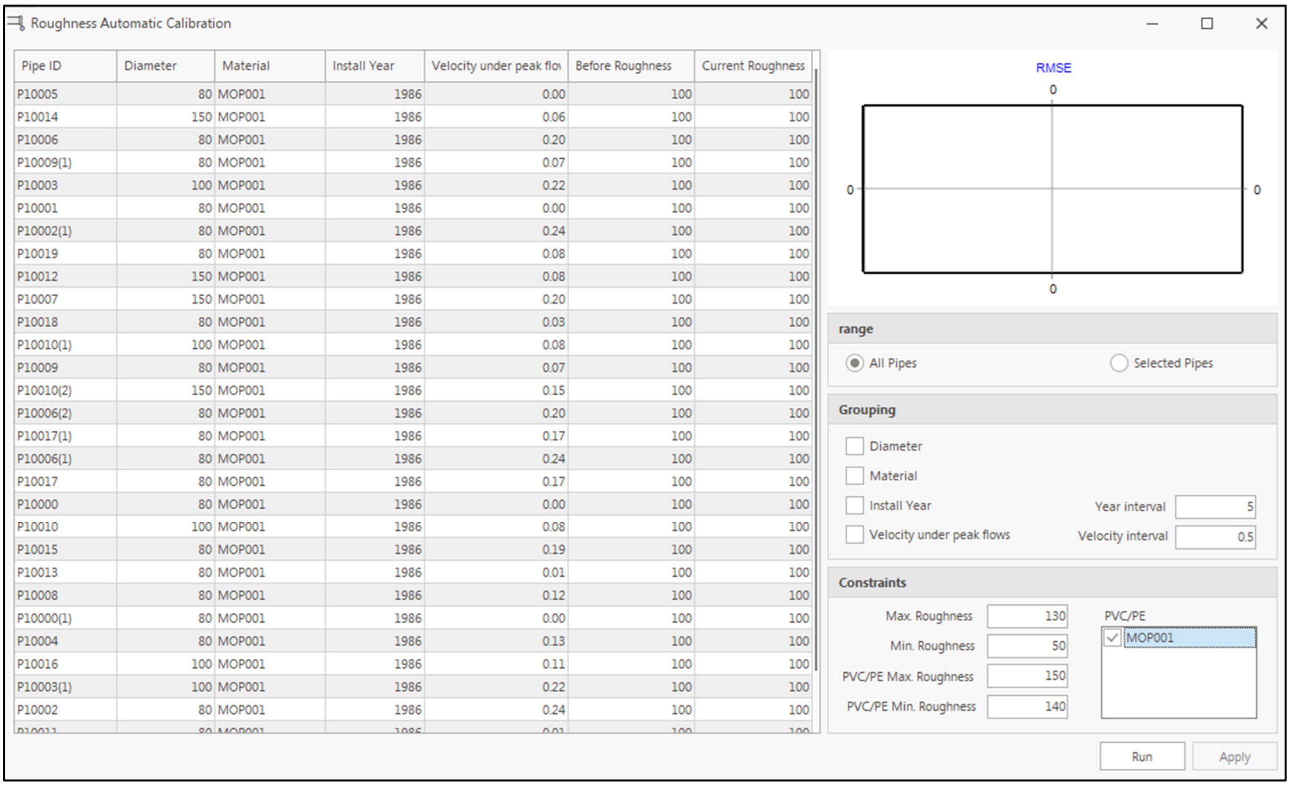


Figure 9-119: Roughness Automatic Calibration window

Table 9-41: Roughness automatic calibration

Field / Control	Description
Pipe Data	
Pipe ID	Unique identifier of each pipe in the network
Diameter	Pipe internal diameter
Material	Pipe material type (e.g., MOP001, PVC/PE)
Install Year	Year the pipe was installed, used for age-based grouping
Velocity under peak flow	Simulated velocity in the pipe at peak demand
Before Roughness	Initial/default roughness value before calibration
Current Roughness	Updated/calibrated roughness value after running the tool
Range Options	
All Pipes	Apply calibration to the entire network
Selected Pipes	Apply calibration only to highlighted/selected pipes
Grouping Options	
Diameter	Groups pipes by diameter for calibration
Material	Groups pipes by material type
Install Year	Groups pipes by age; interval controlled by **Year Interval** field.
Year Interval	Interval in years for grouping (e.g., 5 years)
Velocity under peak flows	Groups pipes by velocity; interval controlled by **Velocity Interval** field.
Velocity Interval	Interval in m/s for velocity grouping (e.g., 0.5 m/s)
Constraints	
Max. Roughness	Maximum allowable calibrated roughness value
Min. Roughness	Minimum allowable calibrated roughness value
PVC/PE Max. Roughness	Maximum roughness allowed for PVC/PE pipes
PVC/PE Min. Roughness	Minimum roughness allowed for PVC/PE pipes
Pipe Material Selector	Enables setting constraints for specific pipe materials (e.g., MOP001)
Analyse	
Run	Executes the automatic calibration process
Apply	Applies the updated/calibrated roughness values to the model
Results	
RMSE Graph	Displays Root Mean Square Error between observed and simulated data (fit check).
Current Roughness (updated)	Shows the final calibrated roughness values in the table.

9.8.2 Wall coefficient automatic calibration

Selecting *Wall Coeff.* shows the Wall Coefficient Automatic Calibration window as depicted in **Figure 9-120**. The various input and results parameters are listed in **Table 9-42**.

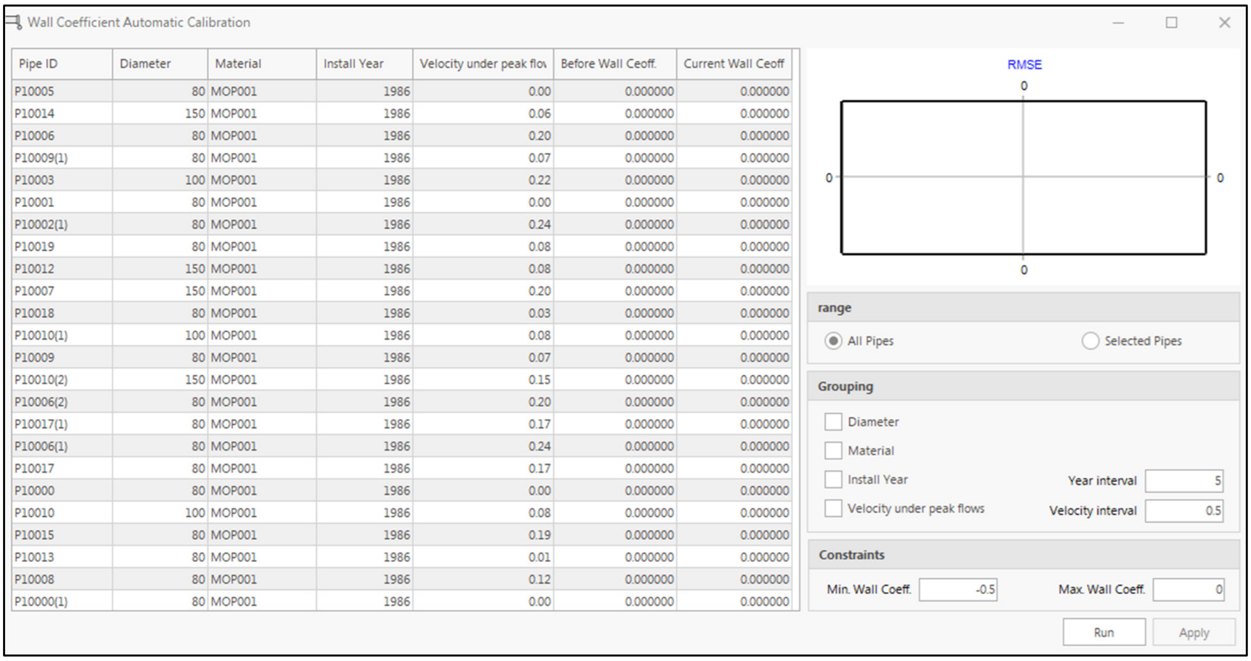


Figure 9-120: Wall Coefficient Automatic Calibration window

Table 9-42: Wall coefficient automatic calibration

Field / Control	Description
Pipe Data	
Pipe ID	Unique identifier of each pipe in the network
Diameter	Pipe internal diameter
Material	Pipe material type (e.g., MOP001, PVC/PE)
Install Year	Year the pipe was installed, used for age-based grouping
Velocity under peak flow	Simulated velocity in the pipe at peak demand
Before Wall Coeff.	Initial/default wall reaction coefficient before calibration
Current Wall Coeff.	Updated/calibrated wall reaction coefficient after running the tool
Range Options	
All Pipes	Apply calibration to the entire network
Selected Pipes	Apply calibration only to highlighted/selected pipes
Grouping Options	
Diameter	Groups pipes by diameter for calibration
Material	Groups pipes by material type
Install Year	Groups pipes by age; interval controlled by **Year Interval** field
Year Interval	Interval in years for grouping (e.g., 5 years)
Velocity under peak flows	Groups pipes by velocity; interval controlled by **Velocity Interval** field
Velocity Interval	Interval in m/s for velocity grouping (e.g., 0.5 m/s)
Constraints	
Min. Wall Coeff.	Minimum allowable wall reaction coefficient (e.g., –0.5)
Max. Wall Coeff.	Maximum allowable wall reaction coefficient
Analyse	
Run	Executes the automatic calibration process
Apply	Applies the updated/calibrated wall coefficients to the model
Results	
RMSE Graph	Displays Root Mean Square Error between observed and simulated data (fit check)
Current Wall Coeff. (updated)	Shows the final calibrated wall reaction coefficient values in the table

10 STORM WATER MANAGEMENT MODEL (To be completed)

GISpipe integrates the EPA Storm Water Management Model (SWMM) into a Geographical Information System environment. This creates a dynamic rainfall-runoff simulation model used for single event or long-term (continuous) simulation of runoff quantity and quality from primarily urban areas running on a GIS platform. The runoff component of SWMM operates on a collection of subcatchment areas that receive precipitation and generate runoff and pollutant loads (**Figure 10-1**). The routing portion of SWMM transports this runoff through a system of pipes, channels, storage/treatment devices, pumps, and regulators. SWMM tracks the quantity and quality of runoff generated within each subcatchment, and the flow rate, flow depth, and quality of water in each pipe and channel during a simulation period comprised of multiple time steps. Running under Windows, SWMM 5 provides an integrated environment for editing study area input data, running hydrologic, hydraulic and water quality simulations, and viewing the results in a variety of formats. These include colour-coded drainage area and conveyance system maps, time series graphs and tables, profile plots, and statistical frequency analyses. This user's manual describes in detail how to run GISpipe built storm water management models. It includes instructions on how to build a drainage system model, how to set various simulation options, and how to view results in a variety of formats. It also describes the different types of files used by SWMM and provides useful tables of parameter values. Detailed descriptions of the theory behind SWMM 5 (the engine used in GISpipe) and the numerical methods it employs can be found in a separate set of reference manuals (www.epa.gov/water-research/storm-water-management-model-swmm).

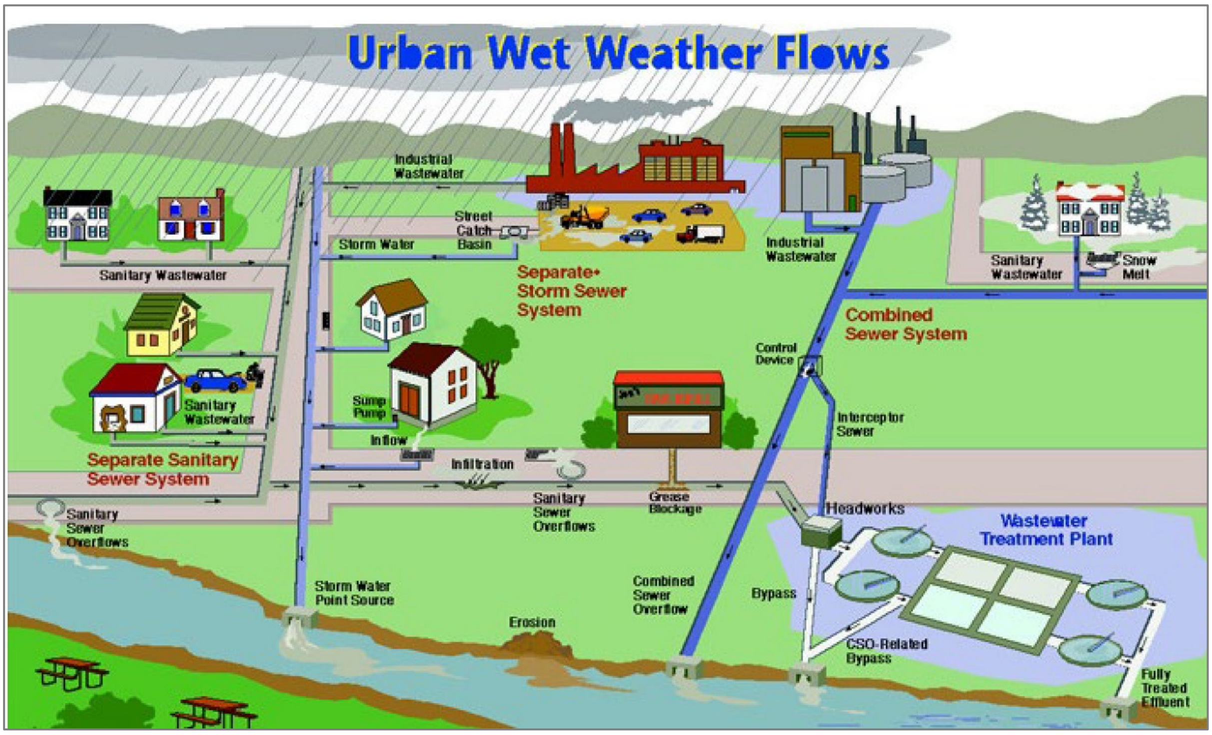


Figure 10-1: SWMM (USEPA, 2022)

SWMM is used throughout the world for planning, analysis, and design related to stormwater runoff, combined and sanitary sewers, and other drainage systems. It can be used to evaluate gray infrastructure stormwater control strategies, such as pipes and storm drains, and is a useful tool for creating cost-effective green/gray hybrid stormwater control solutions. SWMM was developed to help support local, state, and national stormwater management objectives to reduce runoff through infiltration and retention, and help to reduce discharges that cause impairment of waterbodies.

GISpipe

11 SURFACE RUNOFF MODEL (To be completed)

When working on Surface Runoff Models selecting *Surface Runoff Model* Menu will show the analysis and viewing functions associated with the surface runoff model as shown in **Figure 11-1**.

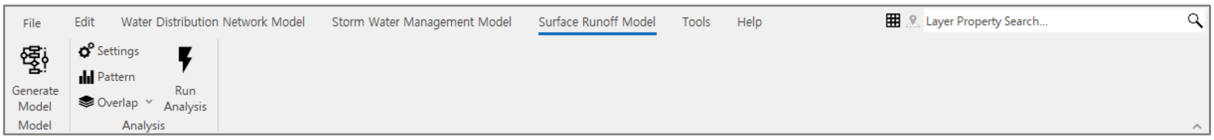


Figure 11-1: Tool bar options when selecting the Surface Runoff Model Menu

12 TOOLS

Selecting the *Tools* menu will provide the user with specific tools related to the DEM data, Shapefiles and generating of basins or streams as shown in **Figure 12-1** with descriptions listed in **Table 12-1**.

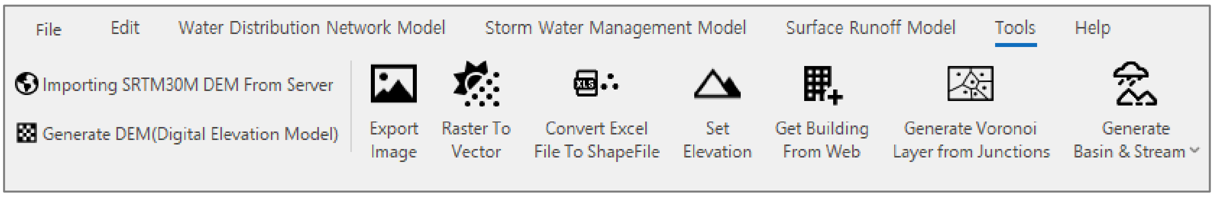


Figure 12-1: Tool bar options when selecting the Tools Menu

Table 12-1: Layer tools

Icon	Description
Importing SRTM30M DEM From Server	Downloads 30-meter resolution Digital Elevation Model (DEM) data from an online SRTM (Shuttle Radar Topography Mission) server. This provides elevation data for the selected geographic area and is typically used for terrain analysis or hydraulic modelling.
Generate DEM (Digital Elevation Model)	Creates a DEM surface from available elevation data (e.g., points, contours). This is useful for analysing topography, drainage paths, and basin delineation in water distribution planning.
Export Image	Creates a DEM surface from available elevation data (e.g., points, contours). This is useful for analysing topography, drainage paths, and basin delineation in water distribution planning.
Raster To Vector	Converts raster data (pixel-based images such as scanned maps or DEMs) into vector features (points, lines, polygons). Typically used to digitize features like pipelines, roads, or contours from a background image.
Convert Excel File To ShapeFile	Transforms coordinate data stored in an Excel file (e.g., node coordinates, elevations) into a shapefile format. This enables integration with GIS systems and spatial analysis workflows.
Set Elevation	Assigns elevation values to nodes or junctions in the water distribution model, either manually or based on DEM data. Accurate elevation data is crucial for hydraulic modelling.
Get Building From Web	Imports building footprints or structural data from web-based GIS services (e.g., OpenStreetMap). This helps visualize infrastructure and assess impacts in urban water modelling.
Generate Voronoi Layer from Junctions	Creates a Voronoi (Thiessen) polygon layer based on junctions or nodes. Each polygon defines the area closest to a particular node, useful for estimating demand zones or service areas.
Generate Basin & Stream	Identifies watershed boundaries and stream paths based on the terrain surface (DEM). Useful for integrating surface runoff, catchment analysis, and planning stormwater management alongside water distribution systems.

Importing SRTM30M DEM from server

This tool connects to an online data source (typically NASA or USGS servers) to automatically download SRTM (Shuttle Radar Topography Mission) Digital Elevation Model data with a 30-meter spatial resolution for a selected area. The DEM provides essential topographic information used to understand the terrain of a region, which directly affects water flow, pressure zones, drainage patterns, and catchment behaviour. Integrating this elevation data into a GIS-based model enhances the accuracy of hydraulic simulations and is particularly valuable for regions without local topographic surveys.

Clicking on the *Importing SRTM30M DEM From Server* button will import the data and create a new layer (called SRTM 30M, the name of which can be changed). Selecting the created layer and setting its Layer Properties to show a colour ramp of the elevations **Figure 12-2** (and in this case also set the transparency level to see the Google Earth image behind it.

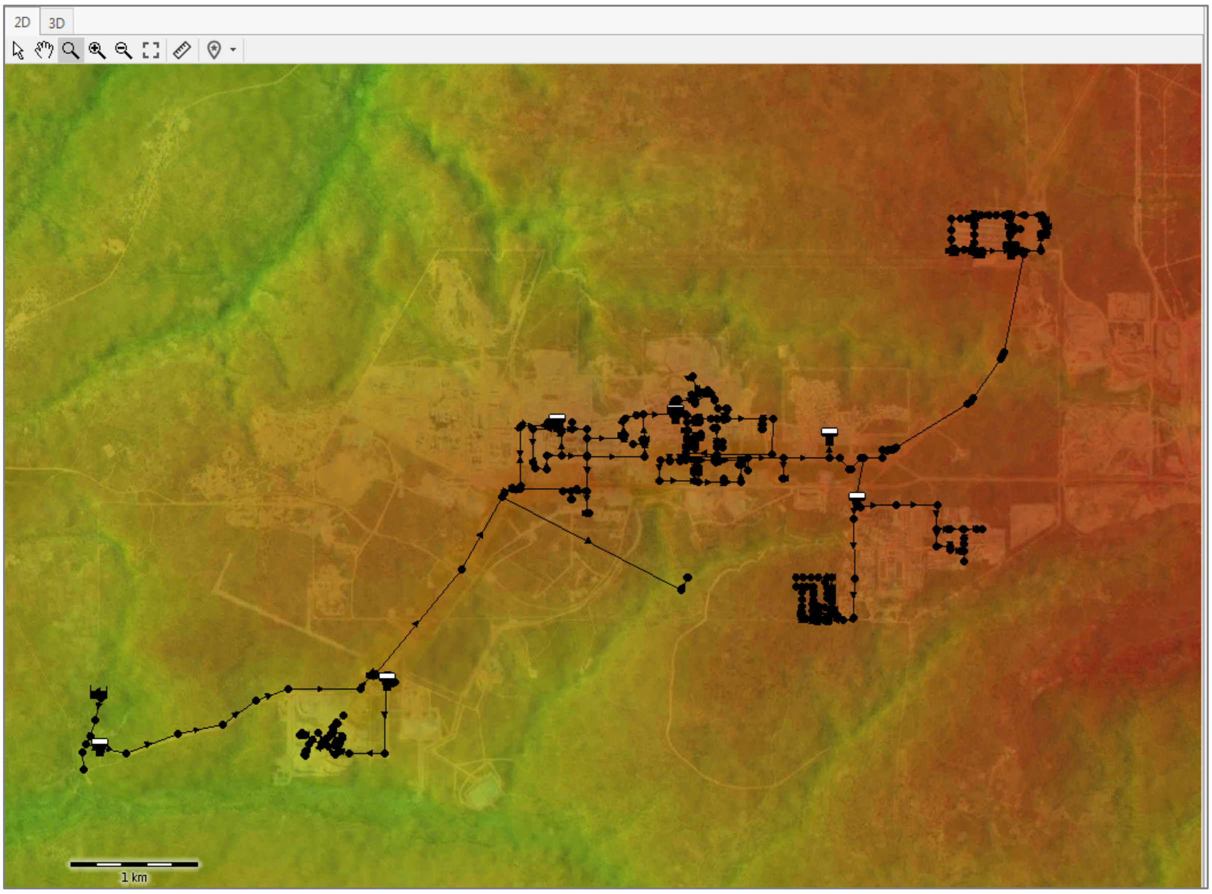


Figure 12-2: SRTM 30M DEM imported from server

Generate DEM (Digital Elevation Model)

This function allows the user to create a custom Digital Elevation Model from local data sources such as surveyed elevation points, GPS data, or contour lines. It uses interpolation algorithms to generate a continuous elevation surface that reflects the terrain's shape. The generated DEM can then be used for hydrologic and hydraulic modelling, including flow direction analysis, slope calculations, and defining natural drainage patterns. This tool is essential when more detailed or localized elevation data is available and preferred over global datasets like SRTM.

Select a DEM Clicking on the *Generate DEM (Digital Elevation Model)* button the user will receive a prompt to enter the Cell size as shown in **Figure 12-3**. A DEM is a grid of cells, each representing the average elevation over a square area. The cell size determines how much ground area each cell covers. For example, a cell size of 10 meters means each pixel in the DEM covers a 10 × 10 meter area on the ground.

- Smaller cell sizes provide more detailed elevation data, capturing finer terrain features.
- Larger cell sizes result in coarser DEMs with smoother, more generalized terrain.

Set the Cell Size and click on the *OK* button.

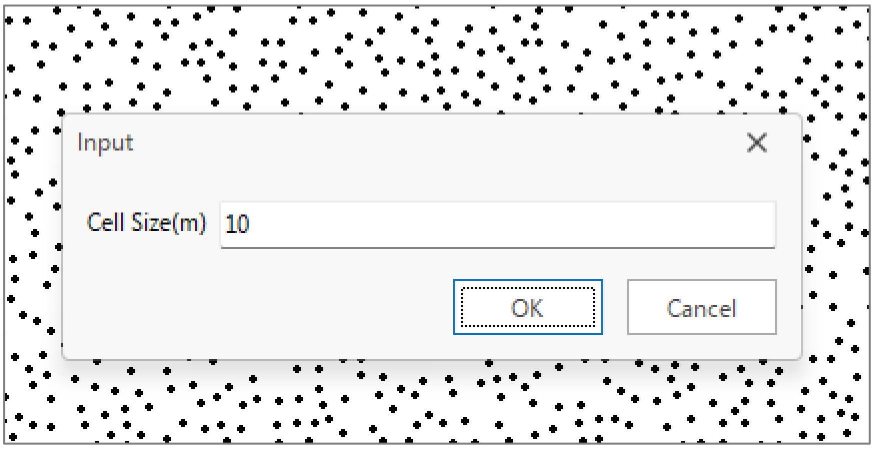


Figure 12-3: Generate DEM - Set Cell Size

The DEM will be created, **Figure 12-4**, and listed as a new Layer in the project.

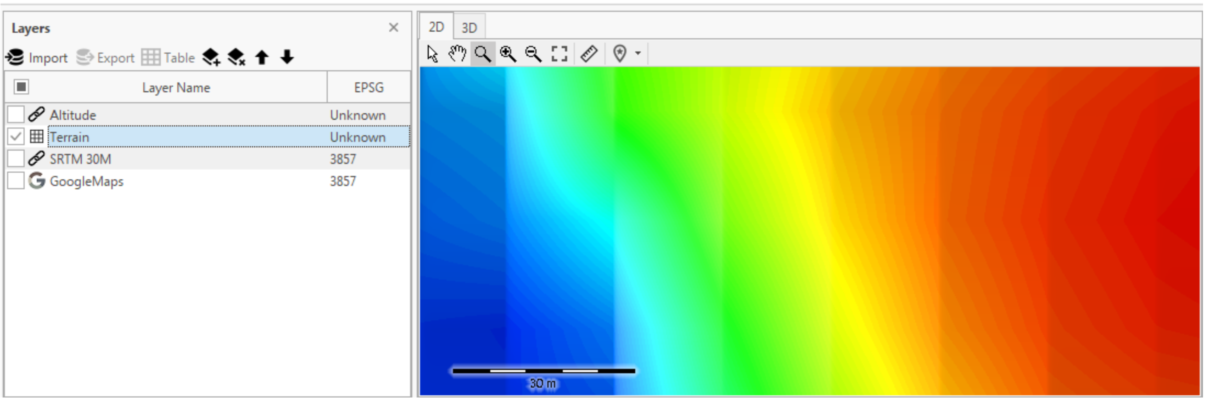


Figure 12-4: Generated terrain

Export Image

The Export Image tool allows users to capture the current view of the GIS project and save it as a raster image file, such as PNG or JPEG. This is especially useful for preparing visuals for reports, presentations, or documentation where a snapshot of the model layout, pipe network, or terrain features is required. Users can typically select the resolution and extent of the exported image, making it a valuable tool for communicating the spatial layout and design features of a water distribution or drainage system.

Clicking on the *Export Image* button will show the dialog window as shown in **Figure 12-5**. Set the following parameters as listed in **Table 12-2**, and click on the OK button. The user can then select the directory and file name where the image should be stored.

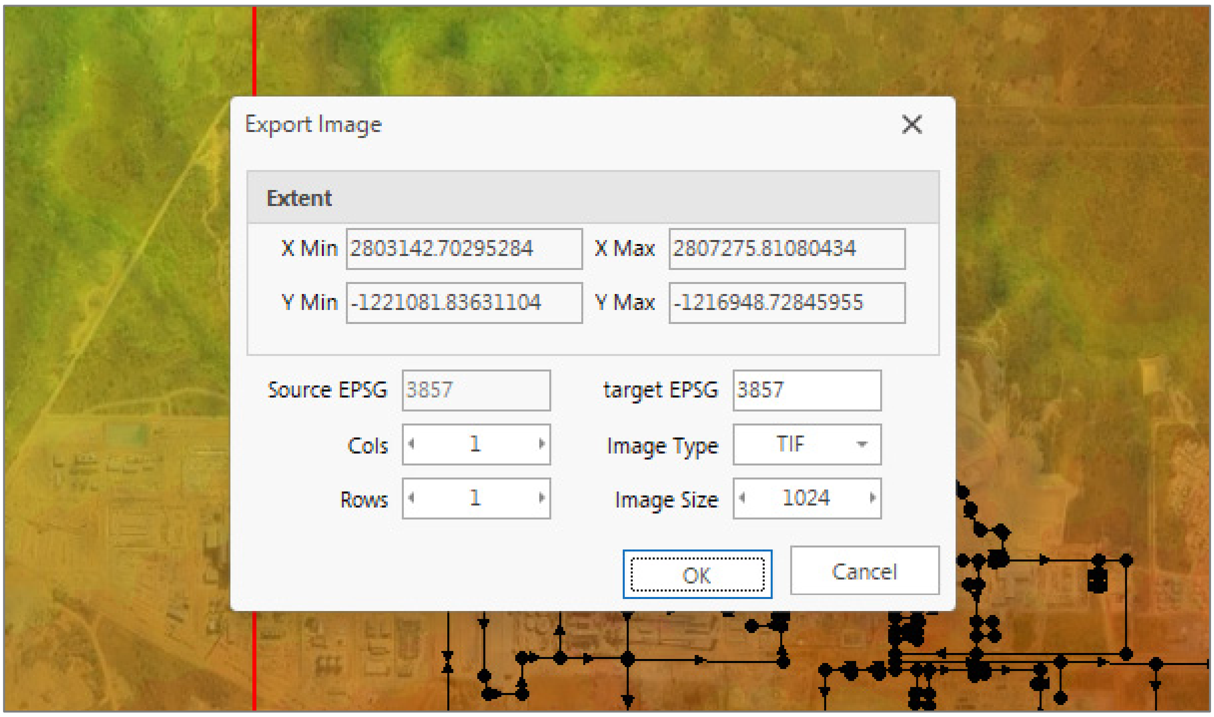


Figure 12-5: Export Image dialog window

Table 12-2: Export image parameters

Parameter	Description
X Min	The minimum X (longitude or easting) coordinate of the bounding box (lower-left corner) of the selected spatial extent.
X Max	The maximum X (longitude or easting) coordinate of the bounding box (upper-right corner) of the selected spatial extent.
Y Min	The minimum Y (latitude or northing) coordinate of the bounding box (lower-left corner) of the selected spatial extent.
Y Max	The maximum Y (latitude or northing) coordinate of the bounding box (upper-right corner) of the selected spatial extent.
Source EPSG	The EPSG code of the coordinate reference system used for the input spatial data. EPSG:3857 is Web Mercator projection commonly used in web maps. This would be the current source of the layer selected
Target EPSG	The EPSG code of the coordinate reference system to which the data will be transformed or reprojected.
Cols	The number of columns (grid cells horizontally) in the raster grid to be generated. Maximum number is 15.
Rows	The number of rows (grid cells vertically) in the raster grid to be generated. Maximum number is 15.
Image Type	The output format of the image or raster, e.g., TIF (Tagged Image File Format), which supports georeferencing or JPG, PNG & BMP.
Image Size	The resolution or size of the output image in pixels. In this case, it defines a square image of 1024 × 1024 pixels.

Raster to Vector

Raster to Vector conversion transforms pixel-based data (e.g., scanned maps, aerial photos, or DEMs) into vector features (points, lines, and polygons) that can be edited and analysed in GIS. For example, contours from a scanned topographic map can be traced and converted into elevation lines, or roads and infrastructure visible on a satellite image can be digitized into vector paths. This tool automates part of the digitizing process, allowing for more efficient creation of geospatial features that are necessary for modelling and analysis in GISpipe.

When the user selects a raster layer and click on the *Raster to Vector* button the Tolerance can be set as shown in **Figure 12-6**. The tolerance refers to a numerical threshold that determines how precisely the vector shapes follow the boundaries or features in the raster image. For example, if a scanned map of a pipeline is vectorized:

- A low tolerance (e.g., 0.5) will trace every minor bend or pixel-level fluctuation.
- A high tolerance (e.g., 5.0) will produce straighter, smoother lines, potentially ignoring minor deviations.

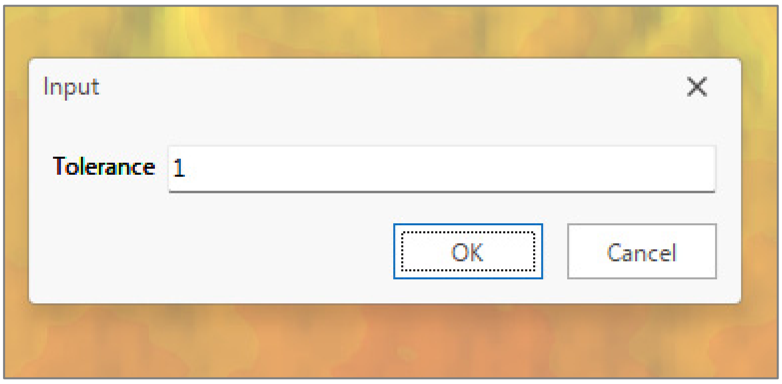


Figure 12-6: Setting Tolerance for raster conversion

Once the process has been completed a new Vector layer is created as shown in **Figure 12-7**.

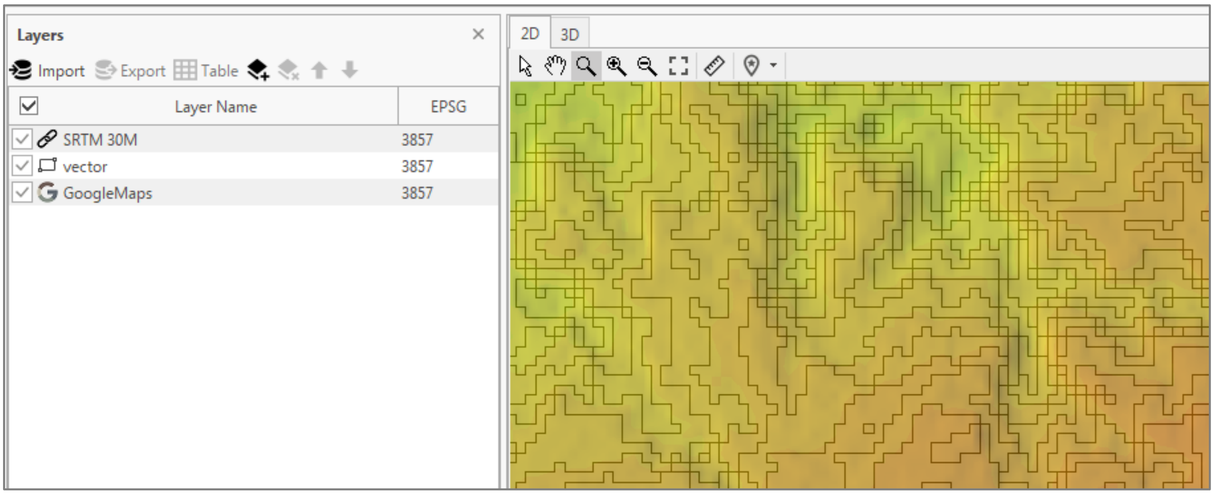


Figure 12-7: Vector layer created

Convert Excel File to Shapefile

This tool enables the conversion of tabular data from Excel spreadsheets into spatial vector data in the form of shapefiles. If an Excel file contains columns for coordinates (e.g., latitude and longitude or X and Y), along with attribute data like elevation, node IDs, or demand values, this tool creates corresponding point features in the GIS environment. It is particularly useful for importing survey data, pump station locations, or customer demand nodes into the GISpipe system, allowing for easy integration of non-spatial data into a spatially-aware model.

Clicking on the *Convert Excel File to Shapefile* button will show the dialog window as shown in **Figure 12-8**. Click on the Open Excel File button and after the file was loaded select which column is the X- and which is the Y-coordinate. Click on the *OK* button to create the Shapfile. The user can then select the directory and file name where the shapefile should be stored. It is not automatically added as a layer. The user has the option to add it to the project using the procedure as described in Section 9.2.1.

Figure 12-8: Convert Excel File to Shapefile

Set Elevation

The Set Elevation function assigns elevation values to specific nodes, such as junctions, tanks, and reservoirs, within the water distribution network. Elevations can be set manually or derived automatically from an underlying DEM or contour data. Accurate elevation information is critical for hydraulic modelling because it affects pressure calculations, head loss, and flow dynamics throughout the network. This tool ensures that every element in the model reflects realistic terrain influences, improving the fidelity of simulations and design evaluations.

The user can select the layer with the nodes that needs to be obtained from the DEM. Click on the *Set Elevation* button after which the user will be prompted to enter the Pipe Depth as well as select if this function should only be applied to selected nodes (toggle option) or if it will be applied to all the nodes in the layer as shown in **Figure 12-9**.

Set the Pipe depth and click on the *OK* button. The nodes on the selected layer will all be changed based on the DEM information.

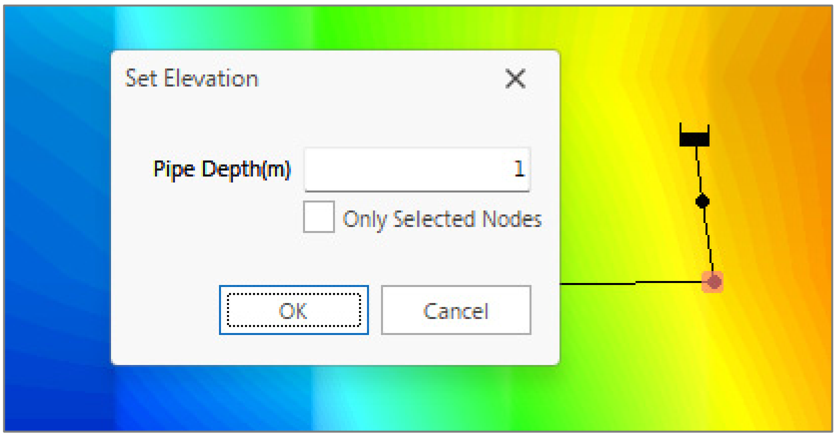


Figure 12-9: Set elevation

Get Building from Web

This feature connects to online geospatial databases—such as OpenStreetMap or other public GIS services—to import building footprints and structural data into the project area. This allows users to visualize residential, commercial, or industrial buildings, aiding in estimating water demand, planning service coverage, or evaluating the impact of infrastructure placement. The buildings are downloaded as vector polygons and can be used for further spatial analysis, such as proximity to pipelines, population density mapping, or identifying high-demand zones.

Click on the *Get Building from Web* button to start the process to download this information (this process can take a few minutes depending on the size of the area). A 3D view of the terrain will show the variations in elevations and structures which are shown based on their heights, see **Figure 12-10**.

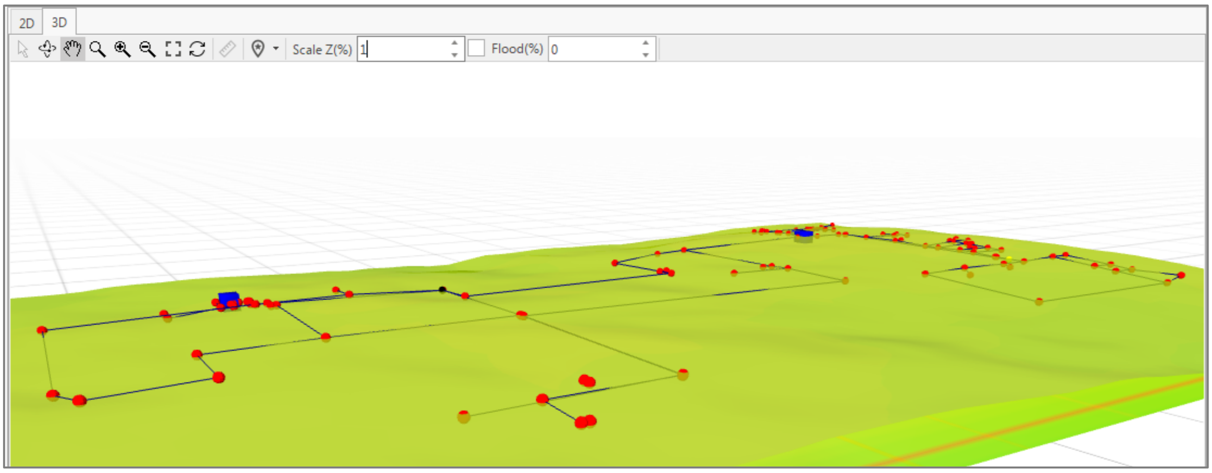


Figure 12-10: 3D view

Generate Voronoi Layer from Junctions

This tool generates a Voronoi (or Thiessen) diagram from selected network junctions or nodes, creating polygons around each node such that any point within a polygon is closer to its central node than to any other. These polygons represent service areas or demand zones and are helpful in water distribution modelling to allocate population, demand, or land use data to the nearest supply point. This spatial partitioning supports network optimization, balancing demand, and locating new infrastructure more effectively.

Select the layer with nodes. Click on the *Generate Voronoi* Layer from Junctions button to generate the Voronoi (or Thiessen) diagram from the junctions as shown in **Figure 12-11**.

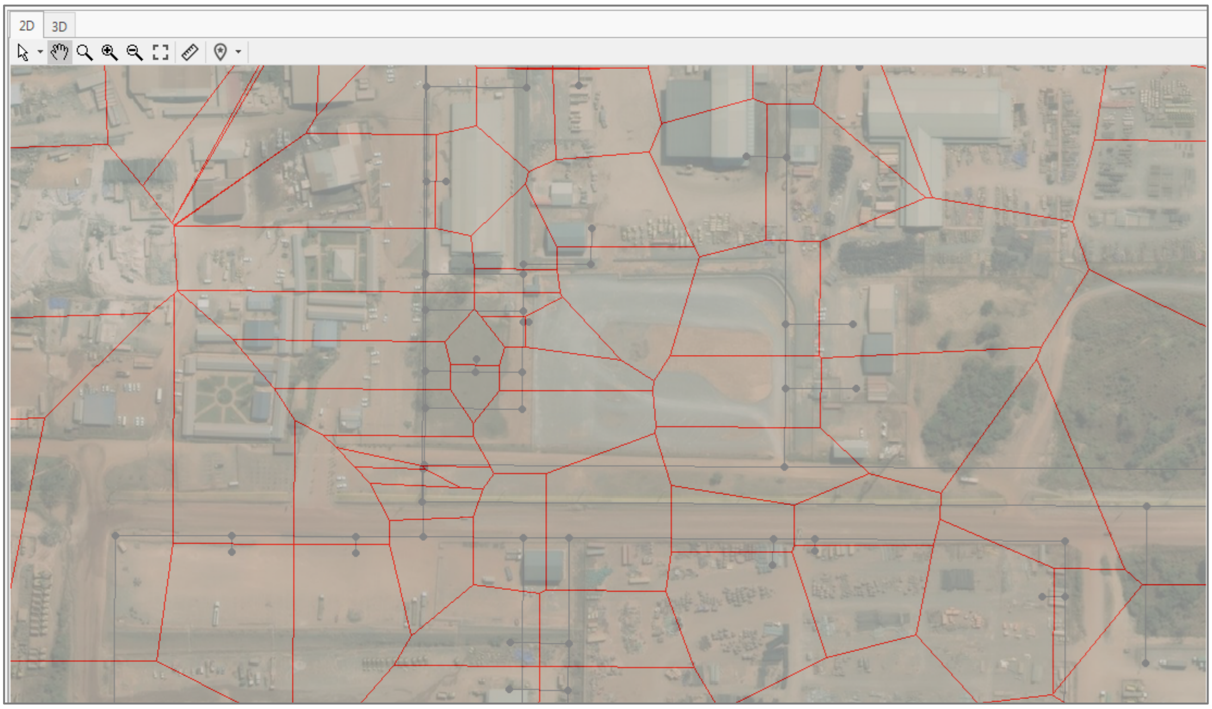


Figure 12-11: Generated Voronoi Layer from Junctions

Once the process has been completed a new Voronoi layer is created and its properties can be set as previously described in Section 9.2.1.

Generate Basin & Stream

The Generate Basin & Stream tool performs hydrologic analysis using elevation data (DEM) to delineate watershed boundaries and identify natural stream paths. It calculates flow direction and accumulation based on terrain slope, then defines catchment areas and the likely routes of surface water runoff. This tool is useful for integrating surface hydrology with piped infrastructure, assessing flood risk, designing stormwater systems, or planning sustainable drainage. The output helps visualize how natural terrain influences water movement, supporting comprehensive water resource management.

Select the layer that contains the DEM data. Click on the *Generate Basin & Stream* button after which the user will be prompted about the Precision Generation and the Accuracy can be selected from the drop-down list (1 to 9) as shown in **Figure 12-12**.

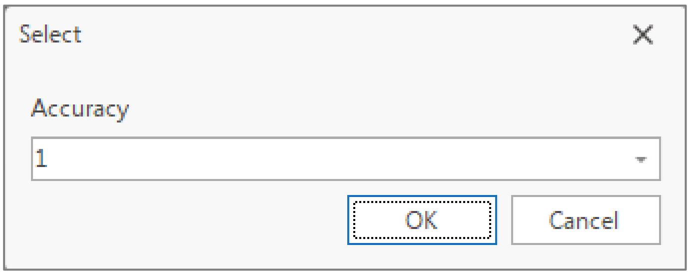


Figure 12-12: Precision Generation for Basins and Streams

The accuracy level sets the internal grid resolution or refinement level used during processing:

- 1 = Lowest precision (faster, less detailed)
- 9 = Highest precision (slower, more detailed)

The selection of the Accuracy will impact the following:

- Flow Direction and Accumulation Grids - Higher accuracy results in finer computation of how water moves across the terrain.
- Stream network delineation - More accurate identification of stream paths, avoiding false channels or missing small tributaries.
- Basin boundaries - More precise and often more complex watershed outlines that better follow natural terrain features.
- Smoothing and thresholds - At low precision, small topographic variations may be ignored. At high precision, subtle elevation changes are preserved and reflected in the output.

High accuracy (7–9) would provide a more realistic and detailed stream and basin outputs but would mean a slower processing time and higher memory use and is recommended for small areas or when accuracy is critical (e.g., urban drainage). The Low accuracy (1–3) will execute faster and is suitable for quick assessments or large regional models where detail is less important.

The resulting streams and basins are determined and three new layers (Catchments, Basins and Streams) are created. See example of Generated Basins and Streams in **Figure 12-13** and of Generated Catchments and Streams **Figure 12-14**.

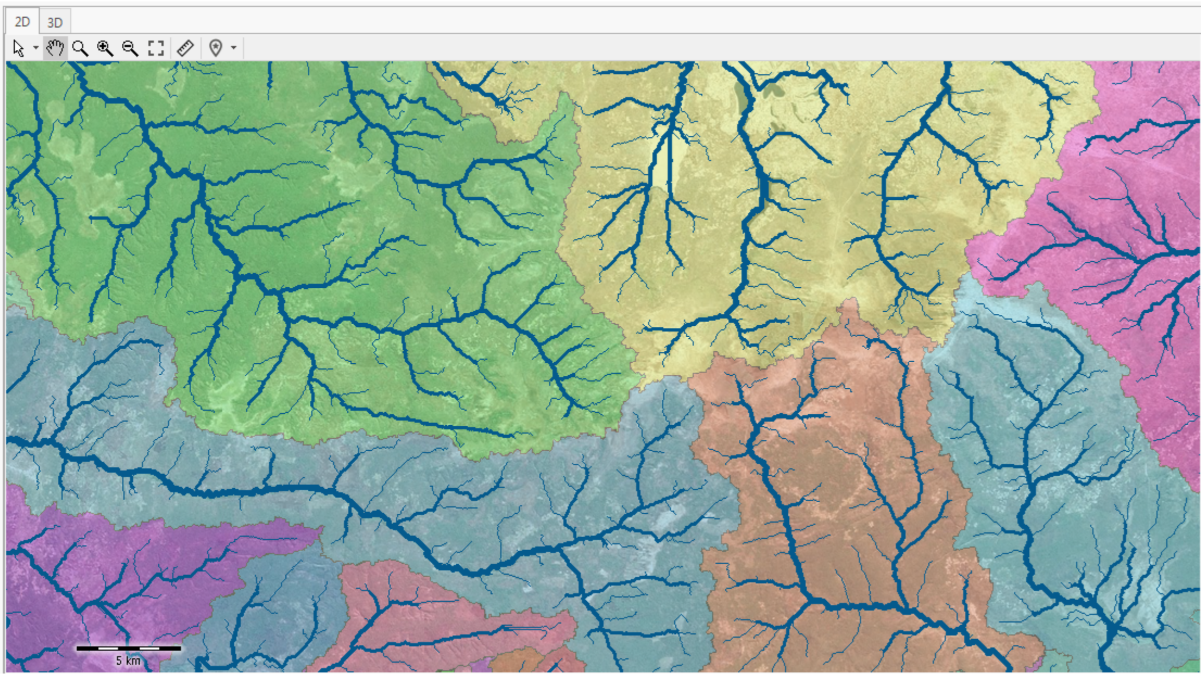


Figure 12-13: Generated Basins and Streams

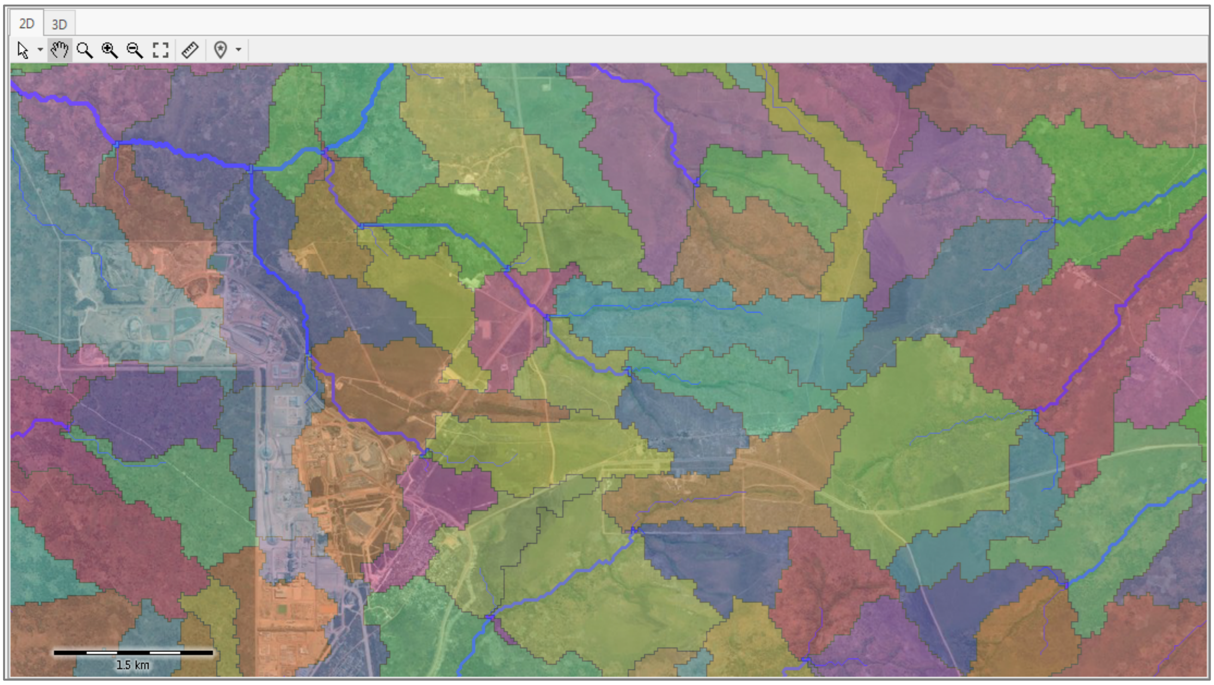


Figure 12-14: Generated Catchments and Streams

13 HELP

Selecting the *Help* menu will provide the user with toolbar options related to the use of the software as well as the licensing thereof as shown in **Figure 13-1.** A detailed description of each of these functionalities are provided in **Table 13-1**.

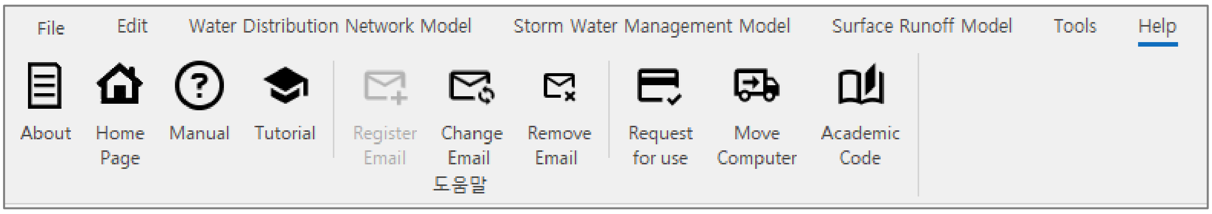


Figure 13-1: Help menu tool bar options

Table 13-1: Help toolbar options

Toolbar icon button	Description
About	Displays general information about the software, including version, authors, and acknowledgements.
Home Page	Opens the official website or homepage of the software developer (Jinbosoft) in your default web browser.
Manual	Opens the user manual, in PDF format, for detailed software guidance.
Tutorial	Provides access to a tutorial or guided walkthrough of the software features and modelling process. Link to the online webpage where tutorials can be found.
Register Email	Allows the user to register an email address for activation, notifications, and licensing purposes. (Grayed out if already registered)
Change Email	Enables the user to change the registered email address associated with the software license.
Remove Email	Deletes the registered email address from the system, deactivating the license.
Request for use	Opens a form to request a new license or usage rights for the software.
Move Computer	Provides tools or instructions to transfer the license or software installation to a different computer.
Academic Code	Grants access to an academic version of the license code, with some usage limitations, for educational purposes.

14 CONTACT US

Our highly dedicated team is available at all times to assist with any problems.

E-mail: **support@gispipe.co.za**
Internet: **www.gispipe.co.za**

15 REFERENCES

Rossman, L.A. and U.S. Environmental Protection Agency (2020) *EPANET 2.2 User Manual*. EPA/600/R-20/133. Available at: https://www.epa.gov/water-research/epanet (Accessed: 1 July 2025).

U.S. Environmental Protection Agency (USEPA) (2022) *Storm Water Management Model (SWMM) User's Manual Version 5.2*. EPA/600/R-22/195. Available at: https://www.epa.gov/water-research/storm-water-management-model-swmm (Accessed: 1 July 2025).

Appendices

A. Supported File Formats

B. Units of Measurements

C. Coordinate System Reference (EPSG)

D. Errors and Warnings

E. Glossary of Terms

Appendix A
Supported file formats

Category	Format	Extension(s)
GISpipe Project and Simulation	GISpipe Project File	*.pgp
	EPANET Input File	*.inp
	SWMM Input File	*.inp
	EPANET Network File	*.net
Vector formats	ESRI Shapefile	*.shp
	MapInfo Interchange Format	*.mif
	Arc/Info Export Format	*.e00
	MicroStation Format	*.dgn
	Geographic Markup Language	*.gml
	Google Earth Format	*.kml
	GPS Exchange Format	*.gpx
	OpenStreetMap Format	*.osm
	MapInfo Native Format	*.tab
	Digital Line Graph	*.dlg
	Census 2000 TIGER/Line	*.rt1
	Vector Product Format (VPF)	*.lfz, *.pft, *.tff
Raster and Grid formats	Arc/Info ASCII Grid	*.asc, *.agr
	Golden Software Grid	*.grd
	IHO S-57 ENC	*.000
	Arc/Info Binary Grid	*.adf
	Binary Terrain Grid	*.bt
	Digital Terrain Elevation Data	*.dt0, *.dt1, *.dt2
	Arc/Info Float Grid	*.flt
	SDTS Vector Profile	*.ddf
	Digital Elevation Model	*.dem
	FME Feature Store	*.ffs
CAD Formats	AutoCAD Drawing	*.dxf
	AutoCAD DWG R2000	*.dwg
Raster compression formats	Enhanced Compressed Wavelet	*.ecw
	ERDAS Imagine Format	*.img
Image formats	Windows Bitmap	*.bmp
	Portable Network Graphics	*.png
	Pictometry Oblique Image	*.jpg, *.jpeg
	Tagged Image File Format	*.tif, *.tiff
	JPEG 2000	*.jp2, *.j2k, *.jpf, *.jpx, *.jpc, *.j2c
	ERDAS Imagine Raw	*.raw
3D and Terrain formats	SDTS DEM	*.ddf
	Digital Terrain Elevation Data (DTED)	*.dt0, *.dt1, *.dt2
Miscellaneous	Standard Tessellation Language (3D CAD)	*.stl

Appendix B
Units of measurements

Note:
US Customary units apply when CFS, GPM, AFD, or MGD is chosen as flow units.
SI Metric units apply when flow units are expressed using either litres or cubic meters.

Parameter	US customary	SI metric
Concentration	mg/L or ug/L	mg/L or ug/L
Demand	(see Flow units)	(see Flow units)
Diameter (Pipes)	inch	millimetre
Diameter (Tanks)	foot	meter
Efficiency	percent	percent
Elevation	foot	meter
Emitter Coefficient	flow unit/ $(psi)^{1/2}$	flow unit/ $(meter)^{1/2}$
Energy	kilowatt - hour	kilowatt - hour
Flow	CFS (cu foot/sec) GPM (gal/min) MGD (M gal/day) IMGD (Imperial MGD) AFD (ac-foot/day)	LPS (litre/sec) LPM (litre/min) MLD (M litre/day) CMH (cu meter/hr) CMD (cu meter/day)
Friction Factor	unitless	unitless
Hydraulic Head	foot	meter
Length	foot	meter
Minor Loss Coeff.	unitless	unitless
Power	horsepower	kilowatt
Pressure	pounds per square inch	meter
Reaction Coeff. (Bulk)	1^{st}-order 1/day	1^{st}-order 1/day
Reaction Coeff. (Wall)	0-order mass/L/day 1st-order ft/day	0-order mass/L/day 1st-order meter/day
Roughness Coefficient	Darcy-Weisbach 10^{-3}foot Otherwise unitless	Darcy-Weisbach millimetre Otherwise unitless
Source Mass Injection	mass/minute	mass/minute
Velocity	foot/second	meter/second
Volume	cubic foot	cubic meter
Water Age	hour	hour

GISpipe

Appendix C
Coordinate system reference (EPSG)

Spatial data in GISpipe must be referenced to a consistent and accurate Coordinate Reference System (CRS) to enable integration, analysis, and map alignment. The EPSG code is a unique identifier for each CRS, maintained by the International Association of Oil & Gas Producers (IOGP).

This appendix lists commonly used European Petroleum Survey Group (EPSG) codes. These can be used when assigning or transforming coordinate systems in GISpipe. Useful websites for a full list of EPSG definitions and transformation parameters which can be searched by EPSG code or location:

- MapTiler Team's Coordinate Systems Worldwide
  https://epsg.io
- SpatialReference.org is a comprehensive online resource dedicated to providing detailed information and reference materials on spatial coordinate reference systems
  https://spatialreference.org/
- The IOGP GeoRepository site contains the authoritative EPSG Dataset
  https://epsg.org/home.html

C.1 South African Coordinate Systems

The Hartebeesthoek94 (HB94) datum, based on WGS84, is the national standard for geospatial data in South Africa. It uses Gauss Conformal Transverse Mercator projections with 2-degree longitudinal zones.

Hartebeesthoek94 – Lo-Series

Zone Name	Central Meridian	EPSG Code	Description
HB94 / Lo15	15°E	EPSG:2046	Northern Cape, Namibia border region
HB94 / Lo17	17°E	EPSG:2047	Western parts of Northern and Western Cape
HB94 / Lo19	19°E	EPSG:2048	Western Cape / Northern Cape (central)
HB94 / Lo21	21°E	EPSG:2049	Western/Northern Cape and central Karoo
HB94 / Lo23	23°E	EPSG:2050	Eastern Northern Cape, parts of Free State
HB94 / Lo25	25°E	EPSG:2051	Free State and western Gauteng
HB94 / Lo27	27°E	EPSG:2052	Gauteng, North West, northern Free State
HB94 / Lo29	29°E	EPSG:2053	Eastern Free State, KZN (common in hydrology)
HB94 / Lo31	31°E	EPSG:2054	KwaZulu-Natal and eastern Mpumalanga
HB94 / Lo33	33°E	EPSG:2055	Eastern Mpumalanga, parts of Mozambique border

Tip: When working across multiple provinces or integrating national datasets, reproject all data layers to the most dominant EPSG zone, or use a regional projection like UTM (see below).

UTM is a global projected CRS divided into 6-degree longitudinal zones.

Universal Transverse Mercator (UTM)

Zone	Hemisphere	EPSG Code	Area covered
33S	Southern	EPSG:32733	Eastern parts of South Africa, eSwatini
35S	Southern	EPSG:32735	Central to eastern South Africa (common UTM)
36S	Southern	EPSG:32736	Parts of Mozambique and Limpopo River Basin

Notes on EPSG usage for South Africa:

- Always verify the datum and projection before assigning an EPSG code.
- Mixing data in different coordinate systems without proper transformation can lead to spatial misalignment.
- Use projected systems (e.g., EPSG:2053 or EPSG:32735) for tasks involving distance, area, elevation, or hydraulic calculations.

C.2 South Korean Coordinate Systems

South Korea uses both geographic and projected coordinate systems for spatial data. The official national coordinate system is based on Korean 2000 (Korea 2000 / Central Belt), but legacy systems based on the Tokyo datum are still occasionally used in historical datasets.

Korean 2000 / Transverse Mercator Projections

Zone Name	Central Meridian	EPSG Code	Description
Korea 2000 / Central Belt	127°E	EPSG:5179	National standard; used for cadastral, engineering, and GIS
Korea 2000 / West Belt	125°E	EPSG:5180	Used in western regions (e.g., Incheon, parts of Seoul)
Korea 2000 / East Belt	129°E	EPSG:5181	Used in eastern provinces (e.g., Gangwon, Gyeongsangbuk-do)
Korea 2000 / East Sea Belt	131°E	EPSG:5182	Rarely used; covers far eastern regions or ocean zones

Note: EPSG:5179 (Central Belt) is the most widely adopted in South Korean government and infrastructure projects.

UTM Zones for South Korea

Zone	Hemisphere	EPSG Code	Area covered
52N	Northern	EPSG:32652	Covers most of South Korea (western/central)
51N	Northern	EPSG:32651	Used near the western edge (e.g., Yellow Sea coast)
53N	Northern	EPSG:32653	Eastern coastal regions and marine zones

Tip: UTM zones are used primarily for scientific applications or integration with international datasets. Korean 2000 should be preferred for most local engineering work.

Notes on EPSG usage for South Korea:

- Korean 2000 is based on the GRS80 ellipsoid and is compatible with modern GNSS.
- Be cautious when working with older datasets using the Tokyo datum (EPSG:4301), coordinate transformation will be required.
- GISpipe supports CRS transformation to align datasets from different EPSG codes within Korea or between Korea and other regions (e.g., EPSG:5179 to EPSG:4326).

C.3 Geographic Coordinate Systems (Lat/Lon)

Used for global data and GPS-based datasets. Not ideal for distance or area calculations due to lack of linear units.

Geographic Coordinate Systems

Name	EPSG Code	Description
WGS 84	EPSG:4326	Global default; used in GPS, satellite datasets
Hartebeesthoek94	EPSG:4148	South African geographic CRS (HB94, unprojected)
Korea 2000	EPSG:4162	Geographic CRS based on GRS80 ellipsoid
Tokyo	EPSG:4301	Older datum used before 2000; check legacy data

Appendix D
Errors and warnings

ID	Explanation (EPANET Error Codes)
101	An analysis was terminated due to insufficient memory available.
110	An analysis was terminated because the network hydraulic equations could not be solved. Check for portions of the network not having any physical links back to a tank or reservoir or for unreasonable values for network input data.
200	One or more errors were detected in the input data. The nature of the error will be de- scribed by the 200-series error messages listed below.
201	There is a syntax error in a line of the input file created from your network data. This is most likely to have occurred in .INP text created by a user outside of EPANET.
202	An illegal numeric value was assigned to a property.
203	An object refers to undefined node.
204	An object refers to an undefined link.
205	An object refers to an undefined time pattern.
206	An object refers to an undefined curve.
207	An attempt is made to control a check valve. Once a pipe is assigned a Check Valve status with the Property Editor, its status cannot be changed by either simple or rule-based controls.
208	Reference was made to an undefined node. This could occur in a control statement for example.
209	An illegal value was assigned to a node property.
210	Reference was made to an undefined link. This could occur in a control statement for example.
211	An illegal value was assigned to a link property.
212	A source tracing analysis refers to an undefined trace node.
213	An analysis option has an illegal value (an example would be a negative time step value).
214	There are too many characters in a line read from an input file. The lines in the *.INP file are limited to 255 characters.
215	Two or more nodes or links share the same ID label.
216	Energy data were supplied for an undefined pump.
217	Invalid energy data were supplied for a pump.
219	A valve is illegally connected to a reservoir or tank. A PRV, PSV or FCV cannot be directly connected to a reservoir or tank. Use a length of pipe to separate the two.
220	A valve is illegally connected to another valve. PRVs cannot share the same downstream node or be linked in series, PSVs cannot share the same upstream node or be linked in series, and a PSV cannot be directly connected to the downstream node of a PRV.
221	A rule-based control contains a misplaced clause.
223	There are not enough nodes in the network to analyse. A valid network must contain at least one tank/reservoir and one junction node.
224	There is not at least one tank or reservoir in the network.

ID	Explanation (EPANET Error Codes)
225	Invalid lower/upper levels were specified for a tank (e.g., the lower lever is higher than the upper level).
226	No pump curve or power rating was supplied for a pump. A pump must either be assigned a curve ID in its Pump Curve property or a power rating in its Power property. If both properties are assigned then the Pump Curve is used.
227	A pump has an invalid pump curve. A valid pump curve must have decreasing head with increasing flow.
230	A curve has non-increasing X-values.
233	A node is not connected to any links.
302	The system cannot open the temporary input file. Make sure that the EPANET Temporary Folder selected has write privileges assigned to it.
303	The system cannot open the status report file. See Error 302.
304	The system cannot open the binary output file. See Error 302.
308	Could not save results to file. This can occur if the disk becomes full.
309	Could not write results to report file. This can occur if the disk becomes full.

Appendix E
Glossary of Terms

Term	Definition
Analysis	The process of computing hydraulic or water quality behaviour across the network using EPANET as the simulation engine.
Bulk coefficient	A reaction rate coefficient that models the growth or decay of a chemical constituent within the bulk flow of a pipe.
Calibration	The process of adjusting model parameters to match observed system behaviour such as pressures, flows, or chlorine residuals.
Contour map	A graphical display in GISpipe showing Colour-coded parameter values (e.g., elevation, pressure) across the network.
Coordinate system (ESPG)	A spatial reference system used to georeference the model in map space. GISpipe supports custom EPSG codes (e.g., 32735 for UTM Zone 35S).
Demand	The rate of water withdrawal from the system at a node, typically expressed in m^3/hr or CMH.
Demand pattern	A time-varying multiplier applied to the base demand at a junction to simulate fluctuations over time.
DMA (district metered area)	A defined sub-network or zone within the larger network used for monitoring and managing water distribution.
Emitter coefficient	Represents flow through an emitter (e.g., sprinkler or leak), as a function of pressure
EPANET	The simulation engine that computes pressure, flow, and water quality in the network. GISpipe uses EPANET to execute hydraulic and water quality models.
Flow units	Units used to define flow throughout the model (e.g., CMH, LPS, GPM). GISpipe allows selection and conversion of flow units.
Graph tool	A utility in GISpipe that allows users to generate time-series or parameter-specific plots for nodes or links.
Hydraulics	The study of water movement (flow, pressure, headloss) through the pipe network under given conditions.
Initial status	The operational state of a link (e.g., pipe, pump, valve) at the start of a simulation, open, closed, or active.
Junction	A network node where pipes connect and where demand can be placed.
Layer	A GIS element containing network data such as pipes, junctions, valves, or tanks. GISpipe layers are EPSG-registered and may be toggled on/off.
Link	A connection between two nodes in the network, typically a pipe, pump, or valve.
Minor loss coefficient	A coefficient representing additional head loss due to bends, fittings, or valves in a pipe.
Node	A point feature in the network model, such as a junction, reservoir, or tank, that may have elevation, demand, or pressure.
Pattern	A time-based array of multipliers used to model temporal changes in demand, head, or energy price.
Pipe	A conduit in the network model that carries pressurized water flow from one node to another.
Pressure	The force per unit area exerted by water at a node, typically measured in meters of water column (mWC) or psi.
Property editor	The interface section in GISpipe that allows users to view or modify attribute data for selected network components.
Quality analysis	Simulation of chemical transport, age of water, or source tracing in the

Term	Definition
	network.
Recalculation (pipe length)	A GISpipe function to automatically recalculate pipe lengths based on spatial geometry.
Reservoir	A boundary node representing an infinite water source with fixed hydraulic head.
Roughness coefficient	A value used in headloss equations (e.g., Hazen-Williams C-factor) to represent friction in a pipe.
Rule-based controls	Logic-based instructions that control pump or valve operations in response to system conditions.
Simulation	Execution of the model using user-defined parameters, controls, and patterns to compute system behaviour.
Skelabrator	A tool to simplify the network layout by removing redundant nodes or links for faster simulation.
Tank	A node representing storage volume that can fill or empty during the simulation.
Valve	A network component that controls flow or pressure, including PRV, PSV, FCV, and isolation valves.
Velocity	The speed at which water moves through a pipe, typically expressed in meters per second (m/s).
Working mode	The selected mode of operation in GISpipe (e.g., Select, Add Node, Edit Vertex) that controls mouse behaviour in the map view.