TdhNet

23.08.07

Tim Hirrel (timhirrel@tdhnet.com)

Overview

The purpose of TdhNet is to provide the essential functions of hydraulic network analysis and the power of a relational database without the need for purchasing additional external software. TdhNet accomplishes this objective with a surprising degree of functionality, including

- graphical data input
- change data management
- rule based controls
- fire flow analysis
- water quality modeling
- pumping cost analysis
- automatic status determination for control valves
- pressure adjusted demands
- automatic pump speed contols

A relational database is used to store and retrieve input data without traditional file IO, to store and query model results and to manipulate graphics based on results. Data can be extracted from a network to create a new network and multiple networks can be merged into a new network.

TdhNet runs under the Microsoft Windows and Linux operating systems. The maximum size network which can be modeled will depend on the amount of RAM available. An approximation for the amount of RAM required is 1 KB for every 4 pipes.

Limited Copyright Transfer

TdhNet is copyrighted software. Permission to use TdhNet is granted by the author and copyright owner, Tim Hirrel (timhirrel@tdhnet.com), under that terms defined within the tdhnet.com website.

Much effort has been devoted to the development of this software, however, it is impossible to guarantee it will perform as expected under all conditions. The author does not assume or acknowledge liability for damages due to the use of this software and neither expresses or implies any guarantees regarding its performance.

Model Components and Theory

For the purposes of this model, a water distribution system is composed of pipes which have endpoints, referred to as nodes, and various other attributes. Based on the input data, the model calculates the flow in each pipe and the hydraulic grade or the inflow/outflow at each node. The calculation of flow is based on the principals of conservation of mass at all nodes and the conservation of energy around all loops and between points of fixed energy. These calculations involve the repeated solution of a system of simultaneous linear equations until the total change in flows between consecutive iterations divided by the total flow is less than or equal to a predetermined value, 0.005 by default. The solution accuracy will be improved automatically, if necessary, to determine the proper status of hydraulic control valves.

Nodes

Inflow and outflow from the distribution system can only occur at nodes. Nodes are the only points where the hydraulic grade will be known or calculated. A node may be the endpoint for any number of pipes. A node is either a junction node or a fixed grade node.

For junction nodes, inflow or outflow (demand) is specified within the input data and the hydraulic grade is calculated by the model. If a junction node is not connected to a fixed grade node with an open path then no hydraulic grade is calculated and the inflow/outflow has no effect on the system.

For a fixed grade node, the hydraulic grade is specified within the input data and the inflow or outflow is calculated by the model. At least one fixed grade node must be connected to the distribution system for the model to produce any results.

Demand Pts

In addition to a single demand value included with a junction node record, any number of Demand Pt records may be associated with a junction node. A Demand Pt has a demand value and is associated with a Junction Group. The demand assigned to a junction during a model solution is the summation of the demand value in the junction record and the demand values for all demand pts belonging to both the junction and an active junction group. Demand factors may be applied to demand pts based on their group association. Demand factors are explained in more detail in Change Data.

A possible use of demand pts is to create demand pts associated with groups that represent different demand scenarios, such as current demands and demands for each of several future years. In this case it would probably be best to not have demand values in the junction data and to turn off all junctions groups containing demand pts except for the one containing the desired demand scenario. While it is possible to associate a single junction group with both junction data and demand pt data, this should probably be avoided. Demand Pt data can be imported from comma separated files as explained in Tdhnet Data Files.

Junction Groups

A junction group is used to establish a relationship between junction nodes or between Demand Pts. A junction node may belong to junction group; a demand pt must belong to a junction group. Demand factors may be applied to all demands belonging to a junction group.

Junction Groups may be turned on or off. If a Junction Group is turned off, any demand data associated with that group will not be incorporated in model solutions or included in demand summaries.

A junction group may be owned by another junction group, in which case any demand multiplier applied to the owner group will be applied to the owned group. To remove an owner relationship, the group owner can be set to itself. Otherwise, it must be ensured that there are no circular owner relationships (e.g. if A is owned by B and B is owned by C, C must not be owned by A).

A junction group may also be useful for conducting queries based on group association.

Pressure Adjusted Demands

When pressure is below a specified value or above a specified value demands can be automatically adjusted by creating a Pressure Dependency record. A Pressure Dependency record can apply to all junction nodes or just to junctions nodes belonging to a specified group. To create or edit a Pressure Dependency record, select System **|** Pressure/Demand Data from the Main menu. A Pressure Dependency record has the following fields:

- Dependency ID: identifies the record.
- Active: determines whether the dependency will be used.
- Pressure FullQ: the pressure below which demands will be adjusted downward. If zero, no demand adjustment for low pressure will be applied.
- Pressure NoQ: the pressure at and below which there will be no demand. Linear interpolation is used to determine the demand between Pressure FullQ and Pressure NoQ.
- Pressure Tolerance: the required agreement, in pressure units, between the calculated pressure and the pressured used to determine the demand.
- Pressure High: the pressure above which demand will be adjusted upward. If zero, no demand adjustment for high pressure will be applied.
- Slope High: the factor used to increase demand for every unit of pressure above Pressure High.
- Junction Group: the junction group to which the dependency will apply. If blank, the dependency will apply to all junctions.

Junctions with an elevation of zero and junctions without a demand greater than zero will not be affected by pressure adjustments.

The demand for a node can be adjusted by only one Pressure Dependency during a solution. After being adjusted by a Dependency, all other Dependencies will be ignored, for that node. Pressure Dependencies are evaluated in order of their Dependency ID (i.e. the same order they appear in the edit dialog).

When using pressure adjusted demands, the required iterations for obtaining an accurate solution may increase by a factor of 3 or more. The Max Trials parameter in the System Data record should be adjusted accordingly.

Pipes

The input data for a pipe determines the assumed direction of flow and the relationship of flow vs. change in hydraulic grade between the nodes connected by the pipe. The assumed direction of flow is from the first node listed to the second node. Flow and losses greater than zero indicate that the assumed direction of flow was correct; values less than zero indicate that the direction of flow was opposite the assumed direction.

The three components of the change in hydraulic grade for a pipe are:

1. Hydraulic losses due to friction between the water and the pipe wall.
2. Hydraulic losses due to minor loss factors.
3. Added energy due to a pump.

Friction loss and minor loss cause a decrease in hydraulic grade in the direction of flow while a pump causes an increase in hydraulic grade in the direction of flow. The friction loss is calculated for every pipe while the minor loss and pump grade are calculated only if appropriate.

Friction Loss

Friction loss can be calculated by either the Hazen-Williams equation or by the Darcy-Weisbach equation. The Hazen-Williams equation is expressed as follows:

```
Loss = 10.51*length*(flow**1.85)/(cval**1.85)/(diam**4.87)
```

where:

1. Length is in feet.
2. Flow is in gallons per minute.
3. C-value is a unitless measure of the internal condition of the pipe as determined either through field measurements or estimates based on a pipe's age, diameter and material.
4. Diameter is in inches.
5. Hydraulic loss is in feet.

The Darcy-Weisbach equation is expressed as follows:

```
Loss = 0.02517*length*ffac*(flow**2)/(diam**5)
```

where:

1. length is in feet.
2. flow is in cubic feet per second.
3. diameter is in feet.
4. Hydraulic loss is in feet.

The friction factor (ffac) is calculated as follows:

```
if Rey > 2000 then
  ffac = 0.25/sqr(ln(roughness/3700*diam)+5.74/rey**0.9)

if 2000 => Rey > 8 then
  ffac = 64/rey

if  Rey <= 8 then
  ffac = 8
```

where:

rey is the Reynolds number

rey = 1.274*abs(flow)/diam/kinematic viscosity

roughness is in milli-feet.

TdhNet stores the roughness parameter for both the Hazen-Williams formula and the Darcy-Weisbach formula for each pipe. When editing or solving, the appropriate roughness parameter is accessed based on the friction formula being used. When importing data that contains only Hazen-Williams roughness data, a Darcy-Weisbach parameter is calculated for each pipe (Travis and Mays’s (2007) equation).

Minor Loss

The change in hydraulic grade due to minor loss is calculated using the Minor Loss equation:

Loss = minor*sqr(velocity)/(2*g)

where:

1. Minor is a unitless parameter and represents the sum of the minor loss coefficients for all obstructions within a pipe.
2. Velocity is in feet per second.
3. g is the gravitational acceleration.
4. Loss is in feet.

Minor losses are often negligible when compared to friction losses. However, a pipe that contain numerous valves and fittings, as might be the case within a pumping station or when connecting to a storage facility, might best be simulated using minor losses.

Pumps

The change in hydraulic grade due to a pump is calculated by one of 3 methods. The first method is based on constant power:

Pump grade = power*550/(flow*62.4*specific gravity)

where:

1. Power is in horsepower.
2. Flow is in cubic feet per second.
3. Specific gravity has no units
4. Pump grade is in feet.

Since the power supplied by a pump varies with flow care should be taken that the actual power for the calculated flow is within an acceptable tolerance of the assumed power.

The second method is based on a set of three input operating points for the pump:

Pump grade = a*(flow**b) + c

where:

1. a, b, and c are calculated based on the three operating points, one of which is at zero flow.
2. Flow is in the same units as the operating points.
3. Pump grade is in the same units as the operating points.

If the calculated flow exceeds the highest flow specified within the operating points the constant power equation is used with the power calculated based on the operating points.

The third method is based on a set of more than 3 operating points. The head vs. flow relationship is determined by interpolating between 2 operating points closest to the values compatible with the hydraulic conditions.

When pump operating data is provided, it is possible to simulate variable speed pumps. A pump speed factor can be provided so that new operating points will be calculated automatically. For each of the original operating points, a new operating point is calculated according to the following equations:

new TDH = original TDH * pump speed factor**2

new Q = original Q * pump speed factor

Valves

TdhNet provides several types of valves used to control pressure, flow and head loss.

Check Valves

A check valve will allow flow in only one direction. Incorporating a check valve in the data for a pipe does not change the head loss characteristics of the pipe. Pipes with pumps or active pressure regulating valves and tanks with an empty or full water elevation will automatically include a check valve. TDHNET ensures that closed check valves will not inappropriately isolate any portions of the network.

Pressure Reducing Valves

A pressure reducing valve adjusts flow to maintain a specified downstream hydraulic grade. The program simulates a pressure reducing valve in a specified pipe by automatically replacing the upstream junction node (i.e. the first node) with a fixed grade node. The flow into what had been the upstream node (which must still be connected to other pipes and a fixed grade node) is increased by the magnitude of the flow from the new fixed grade node, maintaining the net system flow. (It may help to visualize the new fixed grade node as representing the valve's pressure sensor and the upstream node as connected to the supply pipe).

If the downstream side of a reducing valve is supplied from an alternate source which provides a grade exceeding the PRV grade but less than the upstream grade, the reducing valve will automatically close. (The incorporation of a check valve within the pipe containing the reducing valve is not necessary for this occurrence.)

If the upstream grade is less than the PRV grade and greater than the downstream grade, the reducing valve is automatically removed. Flow through the pipe will be calculated as though no PRV existed.

If the downstream side of a reducing valve is supplied from an alternate source which provides a grade exceeding the upstream grade, the reducing valve is removed. Flow through the pipe will be calculated as though no PRV existed. Since the flow will necessarily be less than zero the absence or presence of a check valve within the pipe record will determine whether the pipe is open or closed, respectively.

Pressure Relief Valves

A pressure relief valve adjusts flow to maintain a maximum upstream hydraulic grade. The program simulates a relief valve in a specified pipe by automatically replacing the downstream junction node (i.e. the second node) with a fixed grade node. The flow from what had been the downstream node (which must still be connected to other pipes and a fixed grade node) is increased by the magnitude of the flow into the new fixed grade node, maintaining the net system flow. (It may help to visualize the new fixed grade node as representing the valve's pressure sensor and the downstream node as connected to the discharge pipe).

If the grade upstream of the relief valve is less than the PRV grade but greater than the downstream grade then the relief valve will automatically close. (The incorporation of a check valve within the pipe containing the reducing valve is not necessary for this occurrence.)

If the downstream grade is greater than the PRV grade and less than the upstream grade, the relief valve is automatically removed. Flow through the pipe will be calculated as though no PRV existed.

If the upstream grade is less than the downstream grade the relief valve is removed. Flow through the pipe will be calculated as though no PRV existed. Since the flow will necessarily be less than zero the absence or presence of a check valve will determine whether the pipe is open or closed, respectively.

Flow Control Valves

A Flow Control Valve restricts flow to a specified value in the assumed direction of flow. Both the upstream and downstream nodes must have an open path to a Fixed Grade Node. If the calculated hydraulic grade for the upstream node drops below the calculated hydraulic grade for the downstream node, the Flow Control Valve will be removed and the flow will drop below the specified value.

General Purpose Valves

A general purpose valve allows a flow vs. head loss relationship to be specified for the valve. A flow vs. head loss relationship is specified within a Valve Data record, which can then be assigned to any number of valves.

A Valve Data record contains a Valve Data ID and any number of Valve Pts (flow, loss).

Throttle Valves

A throttle valve allows the user to specify values for Cv, Opening and Exponent in the following equation for pressure loss vs flow:

pressure loss = sqr (flow/(Cv*Opening^Exponent)) * spgrav

The status of a pipe containing a Throttle valve is controlled by the valve. An Opening of zero closes the pipe and a value greater then zero opens the pipe.

Float Valves

A Float valve is a Throttle valve with the opening controlled automatically by the water elevation of the tank specified in the TankID field. The opening vs. level relationship is specified with a Valve Data record. A Valve Data record contains a Valve Data ID and any number of Valve Pts (level, opening).

The base elevation added to the level for each Valve Pt is specified within the Grade field for the valve. If the tank water elevation meets or exceeds a Valve Pt elevation, the valve opening is set to the corresponding Valve Pt opening. If the tank water elevation is below the lowest Valve Pt elevation, the valve opening is set the value specified in the valve Amount Open field.

Emitters

An emitter allows outflow from a network based on the available pressure and the emitter coefficient. An emitter record specifies the junction node where the outflow occurs based on the available pressure at that node. Emitter outflow is calculated based on the equation:

flow = coefficient * pressure^exponent

The normal value for exponent is 0.5 (appropriate for an orifice). The units for coefficient are flow units divided by pressure units.

Extended Period Simulation

An Extended Period Simulation (EPS) consists of a series of network solutions which include the simulated fluctuation of storage levels and hydraulically activated controls. Storage tanks can be specified at any fixed grade node. The flow to or from the fixed grade node, along with the diameter of the tank, will determine the water level and the value of the fixed grade node for the next simulation.

Network solutions will be performed at a user specified interval and whenever a tank reaches a full or empty level, a rule limit or a maximum level change (See Eps Data). Change Data can be incorporated at any time.

An EPS can be cycled (repeated) any number times, with the results at the end of a cycle carried to the start of the next cycle. The first cycle will start at the user specified start time and all subsequent cycles will start at time 0. All cycles will end at the specified end time. Change situations with an EPS time less than zero will be incorporated before the start of the first cycle. All other change situations will be incorporated at the specified time for every cycle.

Rules and Control Sets

Rules are if-then-else statements that allow some network parameters to be determined by solutions results. For example, a rule can be used to set the open/closed status of a pipe, pump or valve based on the hydraulic grade at any node in the network.

A rule has the following components:

Rule ID: required
If <conditional statement>: required
Then <action statement>: required
Else <action statement>: optional
Priority #: optional
Inactive Flag: optional

Action statements are implemented only if the action would result in a change in a parameter and the rule has not already been implemented for the current solution. The action statement after Then is implemented if the conditional statement is true and the action after Else is implemented if the conditional statement is false.

Rules are implemented within the following constraints:

- Action statements are implemented in order of priority for the rule, from higher to lower priority. If a rule is implemented, no rule with a lower priority will be implemented until a solution is obtained and all rules are again evaluated. The default priority is 0 and priority values may be less than zero.
- A rule will not be implemented more then once for given time period. This is to prevent an infinite loop of conflicting rules.

The allowable parameters within the conditional and action statements can be viewed and accessed using the Build options in the Rule editor.

All Rules belong to a Control Set and there may be any number of Control Sets. Any number of Control Sets may be active for a solution sequence and the active values for Control Sets and Rules may be be changed during a solution sequence. For a Rule to be evaluated and implemented during a solution sequence, both the Rule and the Control Set to which it belongs must be active.

Pump Speed Controls

Pump Speed Controls allow the speed of a pump to be controlled automatically to maintain a target hydraulic grade at a specified junction node. In addition to specifying the pump, the junction node and the target grade, pump speed controls can specify the maximum speed, the minimum speed, the tolerance for the target grade agreement, whether the control is active and the priority for the control.

Pump Speed Controls are implemented before any Rules are implemented for a particular solution. (If a Rule is implemented, a new solution will be obtained with Pump Speed Controls implemented again.) Pump Speed Controls do not belong to either a ControlSet or a ChangeSet.

A Pump Speed Control can be specified for any pump and any node, however, caution should be used with regard to:

- The pump should have a strong influence on the grade at the specified node and the pump should be able to operate in a range that can meet the target grade. The pump will not operate above the specified maximum speed. The minimum speed may be positive or negative:
  - if positive, the pump will operate at the minimum speed even if that speed will exceed the target grade.
  - If negative and the (absolute) minimum speed will exceed the target grade, the pump will be turned off.
- A specified node should not be influenced by multiple Pump Speed Controls. If this is a possibility, the priority specification can be used to determine which control will have the final say. Pump Speed Controls will be implemented in order from highest numerical priority to lowest.

A Pump Speed Control can be created by selecting Edit | Control Data | Pump Speed from the main menu. A Pump Speed Control will be implemented only if Use Pump Controls in the active ChangeSet is set to yes and the Active field for the control is set to yes.

The target grade and the active status for a Pump Speed Control can be changed at any time during an extended period simulation by using Change Values within a Change Situation for the active ChangeSet.

Water Quality Analysis

The water quality analysis capabilities are taken from Epanet2, public domain software published by the USEPA. For further information, see the Epanet2.hlp file. The water analysis capabilities include:

- The ability to calculate the age of water after it leaves one or more sources.

- The ability to trace water (or other substances) and calculate concentrations as it moves through the network and mixes with water from other sources.

- The ability to calculate the decay or growth of substances in the water based on decay or growth coefficients specified by input data.

The options for controlling a Water Quality Analysis are selected using the Quality Data edit window. The water quality input values for individual pipes and nodes can be set using the Editing windows for pipes and nodes.

A water quality source is a node where the quality of external flow entering the network is specified along with one of the following types:

A concentration source fixes the concentration of any external inflow entering the network at a node, 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).

Pumping Cost Analysis

The purpose of a Pumping Cost Analysis is to provide information regarding the energy costs of pump operation and the peak power usage of a given pump or pumping facility. A Pumping Cost Analysis can be performed during either an EPS or a regular simulation. During a regular simulation energy and total cost cannot be calculated but power and cost per volume will be calculated.

Required input data for this analysis includes pump efficiency data, per unit power costs, the composition of pumps within pumping stations and the relationships of pumping stations within the system. Information calculated for after each simulation for each pump, pumping station and the system includes: the instantaneous hydraulic power, the instantaneous electrical power, the hydraulic energy imparted during the period, the electrical energy consumed energy during the period, the power cost per volume pumped at the moment, the peak electrical power from the start, the volume pumped from the start, the cumulative power cost from the start and the hours the operation from the start.

The required pump efficiency input data consists of two efficiency points corresponding to the second and third flow points in the pump operating data. Using these two efficiency points, the pump efficiency for any other flow is calculated based on the following equation:

eff = a*flow - b*flow**2;

where:

1. a and b are calculated based on the two input efficiency points.

Also included in the pump input data is a field for a pumping station id. For a pump to be included in a Pumping Cost Analysis, it must be included in a pumping station by entering the station id in the pump data.

Pumping Station records specify per unit cost data and whether the station contains pumps or describes a relationship between two other pumping stations. A pumping station which contains pumps may contain any number of pumps and the pumps are always assumed to operate in parallel.

A pumping station which describes the relationship between two other stations specifies whether the stations operate in parallel or series. The stations described by the relationship may be either stations containing pumps or stations describing the relationship between two other stations. Therefore, any number of stations can be described as operating in parallel, series or any combination of the two.

When a per unit power cost (> 0) is specified for a station which describes the relationship between other stations, that cost is applied for all stations which comprise the relationship, either directly or indirectly, unless such stations have a power cost specified. A system wide power cost can be specified (in EPS Data) that will be applied to any stations with no other cost applied.

An example of the use of these concepts would be a water treatment facility which includes a raw water pumping station, a main zone finished water pumping station and a high zone finished water pumping station. The three physical pumping stations would be represented by stations which contain pumps, named RAW, MFW and HFW. To get the total cost of pumping from the treatment plant to the Main Zone, a station describing a serial relationship between RAW and MFW, would be named MZ. Similarly, a station describing a serial relationship between RAW and HFW would be named HZ. This would allow the per volume pumping cost from the treatment facility to the Main and High Zones to be compared with the per volume pumping costs from other sources throughout the day. A station describing a parallel relationship between MZ and HZ would provide instantaneous power usage information for the entire treatment facility. This would also allow per unit power costs for all pumps within the treatment facility to be specified within a single Pumping Station record.

Queries and Charts provide some of the best tools for analyzing pumping cost results.

Dual Solution Engines

TdhNet now incorporates 2 solution engines, the TdhNet solution engine and the Epanet solution engine. For a detailed discussion on the implications of using either engine, see the tdhnet.com website.

The TdhNet solution engine is the default. To switch to the Epanet solution engine, from the Main menu select System | System Data and change the Engine field to Epanet. The solution engine selection is saved with the network.

Data Management

Data can be read into RAM (random access memory) using the File | Open | From File option from the Main menu. The types of files that can be read are Tdhnet and Epanet (ascii format). Once the data is contained in RAM, it may edited, used for a solution and written to the original file using the File | Save | To File option from the Main menu or to a different file using the File | Save | To File As option from the Main menu. Files may be written in any of the types that can be read.

A network is the complete collection of data associated with a modeled water system (or hydraulic network). The data associated with a network includes base data (the pipes, nodes, pumps, etc.), ControlSets and Change Data. Using either File | Open to read a network from a file or File | New to create an empty network will delete any existing data in RAM, so File | Save should be used if data changes are to be preserved.

When viewing data pertaining to a particular entity, whether editing the data, viewing a map or inspecting results, the user can switch to other available views of that entity or related entities by right clicking and selecting the desired view from a popup menu.

Import and Export

The File | Import item of the Main menu is used to add data from a file to the current network. Existing data will not be deleted or overwritten by imported data; key conflicts will prevent the imported data from being added.

After selecting Import or Export, the user will first be prompted for the type of data and then for a file name. The choices for the type of data are:

1. Tdhnet ChangeSet
2. Tdhnet ControlSet
3. Tdhnet DemandPts
4. Epanet Scenario

When exporting ChangeSets and ControlSets, only data for the set currently referenced in the respective edit window will be written.

Import/Export can be used to archive data, to exchange data between different networks or to duplicate a network within a file. When duplicating, it will be necessary to change either the network name in the existing data or the network name in the exported data before importing.

The ability to import DemandPts will allow the convenient use of demand data generated from an external source such as a spreadsheet or GIS. For a description of the data format and options for importing DemandPts, please see Tdhnet Data Files. Junction Group data may be likewise imported.

When importing an Epanet Scenario, the user will be prompted for one name to be used for both a new ChangeSet and a new ControlSet to own data contained in the scenario. This name should not already be used in the existing data for either a ChangeSet or a ControlSet.

To make the imported ChangeSet the active set used for a solution sequence, use the editor to navigate to the imported set. For ControlSets, the normal options apply for using the set, i.e. the ControlSet Use Flag and the options in Change Data.

Change Data

Change Data allows the user to vary many input conditions without having to modify the base data. Change data can be used to:

- Open or close pipes;
- Change any pipe parameter;
- Change demands by a global factor;
- Change demands by a factor for a group of nodes or a group of demand pts;
- Change the demand for a single node;
- Change the grade for a fixed grade;
- Change pump speed;
- Change power cost for a pumping station and all stations which comprise a relationship described by the station;
- Change water quality parameters for nodes;

- Change the Active status of Control Sets and Rules;

Change Data is edited by selecting the Edit | Change Data items from the Main menu.

A Change Situation contains the changes desired for a particular solution. A ChangeSet is a set of any number of Change Situations to be solved in sequence. A Network may have any number of ChangeSets, but only one ChangeSet is active for a solution sequence. The user selects the active ChangeSet using the Edit | Change Data | ChangeSets item from the Main menu.

Multiple Change Situations within a ChangeSet will be incorporated and solved sequentially. Situations are sequenced according the Situation Number. Change situations for an EPS must be sequenced according to the EPS Time, the time the situation takes effect (multiple situations may have the same time).

In addition to Change Situations, EPS data, Quality data and a ChangeSet Description apply to each ChangeSet. These data are accessed from the Edit | Change Data item in the Main menu. ChangeSet Descriptions allow the entry of any text that helps document the purpose and methods for the ChangeSet.

Change Situations

Change situations allow the application of demand factors and change values. Change situations are identified and ordered by a unique situation number which is provided by the user.

The Renumber button in the Change Situation Edit window allows the automatic renumbering of change situations as may be desired when situations are added or deleted. The renumbering begins by asking for an offset which will be applied starting at the situation being edited. All situations up to the current will be renumbered sequentially. Starting at the current situation the sequence will be increased by the offset, and all remaining situations will be renumbered sequentially.

- To insert a new situation before the current, select Renumber and specify an offset of 1.
- After a situation has been deleted, the remaining situations can be renumbered sequentially by specifying an offset of 0.

A Change Situation may have any number of Change Values, with each value changing a parameter for an entity. Change Values are edited by right clicking on the edit grid for the Change Situation to which they belong.

Demand Changes

All Change Data except demand changes (unless otherwise specified) remain in effect for subsequent situations in a change sequence. A specific demand change for a junction overrides any other demand data and demand factors, only the specific demand value is used. During an EPS, demand changes and EPS Interval changes remain in effect until a subsequent change situation is incorporated.

Demand factors may be applied universally or to specific junction groups. Universal demand factors, included in the change set and/or change situation data, are applied to demands included in all junction records and demand pts for either the entire solution sequence or the particular situation, respectively . Group demand factors, specified by selecting Edit | Change Data | Node Data | Group Demand Factor from the Main menu, apply to the demand and demand pts associated with any junction belonging to the specified group and to demand pts belonging to the group. Thus, a single demand pt could have 4 different demand factors applied to it within a single change situation:

1. a universal factor used for the entire solution sequence
2. a universal factor applied to a particular situation
3. a factor applied, for a situation, to the group containing the junction to which the demand pt belongs,.
4. a factor applied, for a situation, to the group containing the demand pt itself.

When a junction group status is off, the demand pts belonging to the group will not be used. A junction group status can be set off either within the junction group data or by selecting the AllPtsOff option within a Changeset, which will set the status off for all groups. A group status will be set on when a group demand factor change is specified for the group. The group status will remain on, using the demands multiplied by any factor for each subsequent change situation (i.e. factors are reset to a default of 1).

If the universal demand factor for a situation is -1, that situation will use all demand changes from the previous situation. Any new group demand factors are ignored if this option is specified, but new specific demand changes will be incorporated, overriding any other demand values.

Editing Data

The user may edit input data using the Edit item from the Main menu. A separate edit window can be opened for each data type. Fields with black lettering can be edited by entering new values, fields with red lettering can be editing by entering new values or by selecting from a drop down box. A selection from a drop down box doesn't become effective until the user moves to a different field, even if the user otherwise saves the record.

The user may navigate to different records by using the First, Prev, Next, Last or Goto buttons. New records are created by clicking on the Insert button. Records are deleted by using clicking on the Delete button.

Some data types (e.g. junctions, pumps, tanks, changesets and change situations) have associated sub-types (demand pts, pump points, tank points, change situations and change values, respectively). Sub-types can be accessed by right clicking on the edit grid for the owing entity/ Only the sub-type records for the current record (e.g. the pump points for the pump record being edited) are available for editing. Sub-type records are automatically deleted if the record is deleted.

Right clicking on the edit grid also allows the entity to be displayed on the map and the results for the entity to be displayed, if available. Related entities may also be accessed, such as the tank associated with a fixed grade node.

Right clicking on the edit grid for a pipe allows the pipe points to be edited in the PipePts dialog.

- A point may be selected by right clicking on a points grid cell or by clicking on the map near the point and clicking on the Find Pt button in the dialog.
- Points may be modified by entering new values in the grid or by clicking on the Drag Pt button and dragging on the map.
- A single point can be inserted prior to the selected point by clicking on the Insert Btn and dragging on the map. A series of points, terminated with a right click on the map, can be inserted by clicking on the Insert button while holding a shift key.
- A single point can be deleted by clicking on the Delete button. A range of points can be selected and deleted by clicking on a second grid cell while holding a shift key or clicking

on the Find Pt button whild holding a shift key.
- A single point or a range of points can be copied and pasted from a different pipe. Holding a shift key while pasting will paste the points in the reverse order.

If a Junction Group id is changed in the Junction Group Edit form, the change will appear in all uses of the group for junction nodes and demand pts. If a Junction Group is deleted, all demand pts belonging to the group will also be deleted.

Some Edit dialogs include a Calc button which performs a calculation pertinent to the data type:
- For Tanks, the Calc Volume button will calculate the volume at any specified water elevation. The top water elevation is provided as the default when the user is prompted to enter the elevation.
- For Pumps, the Calc Speed button will calculate the speed needed to modify the pump curve so that it intersects a specified flow, head point. (This value is copied to the clipboard.) The resulting pump curve can be displayed by:
  - entering the speed value in the pump edit dialog.
  - right clicking within the edit grid and selecting Chart.
- On the Junction Group Edit window, selecting the Calc Demand button will bring up a box showing the sum of demands associated with the current group. If any demand pts belong to the current group, the total will include demands from those demand pts. If any junction nodes belong to the current group, the total will include demands for those junction nodes including demands from demand pts belonging to the junction and any active group.

Notes

A Note may contain text of any length and may be attached to any data that appears in an edit window. Notes are created and displayed using the Note edit window. If the entity most recently navigated to has a note, it will be displayed in the Note edit window. To create a note for the data entity most recently navigated to, type text in the Note edit window and click on the Save button. To share the displayed note with another entity of the same type, click on the Share button and enter or select the id of the target entity. To remove a note from the entity most recently navigated to, click on the Remove button. If a note is removed from all entities it was associated with, it will be deleted.

Notes may be navigated using the typical navigation buttons or searched based on partial strings (case insensitive). The first entity associated with the displayed note may be found by clicking the Find Entity **|** First button and any subsequent entities sharing the note may found with the Find Entity **|** Next button.

Projects

A project specifies the use of a particular directory containing a Tdhnet database file. All data used by and produced by Tdhnet for a particular project are contained in a set of SQLite3 database files, named TdhnetData.sqlite and TdhnetResults.sqlite. Any number of projects may be created.

The Projects dialog is accessed by selecting File **|** Projects from the Main menu. This dialog provides a list of the available Projects and the following options:
- Select **-** determines the Project to be used and directory where a Tdhnet_Data database can be accessed. (Double clicking on the list item also accomplishes this).
- Add **-** prompts for a new Project name and a directory to be used for the project

database. If the specified directory does not already contain a Tdhnet_Data database, one will be created. If the Project name already exists, the directory will be changed as specified.
- Describe – provides a dialog for entering description text and for finding projects based on text contained within the project description.
- Delete - removes the Project from the Tdhnet_config database, does not delete the Tdhnet_Data database in the Project directory.
- Rename – allows renaming a project.
- Compress – reduces the file size after deletion of data and deletes any orphan records. Compression should be run after deletion of a large amount of data or the improper deletion of records by external operations.
- Below the Project list, the directory for the highlighted Project is displayed.
- Below the Project list, the Description for the highlighted Project is displayed.

Information about Projects is contained in a database file named Tdhnet_config.sqlite. By default, the program looks for this file in the sub-directory Tdhnet_Data, off the directory containing the executable file. A different directory can be specified as a command argument when starting the program.

A project named “Default” is automatically created in the same directory containing the Projects database. This project can not be deleted and the database files should not be removed. The Default directory can be changed by adding a project named Default.

Tip – to setup the program so that each user has a separate Tdhnet_config database:

- Create a program Shortcut for each user. For the Shortcut Target, specify the directory to contain the user’s Tdhnet_config database as a command argument.

The Tdhnet_Data directory is not installed and will not be overwritten during the installation of Tdhnet. If the Tdhnet_Data directory doesn't already exist, it will be created when the program is started. The Tdhnet_Structure directory will be installed and overwritten (if it exists) during the installation of Tdhnet. The purpose of the Tdhnet_Structure directory is to contain the most recent database structure for Tdhnet. The files within the Tdhnet_Data directory and any other database files used will automatically be updated, if necessary, based on the files in the Tdhnet_Structure directory. User created data should NOT be stored in the Tdhnet_Structure directory.

Tip – to copy a network to a different project, select File | Don’t Clear before changing Projects. If Don’t Clear is checked, the current network will be retained when the project is changed. The network may then be saved to the new project. (It will be necessary to change the network name if the name is already used in the new project).

Solving the Network

For a non-EPS, a solution sequence consists of a solution for the base data and a solution for each change situation in the current Changeset. For an EPS, a solution sequence consists of all the time stepped solutions between the start and stop times.

A solution sequence is begun by selecting System | Solve Control from the Main menu and then clicking on the Solve button in the Solve dialog. When a solution sequence is no longer needed the user should click on the Reset button so that another solution sequence can begin. This will wipe out the current results.

Information provided on the Solve form about the solution process includes the status, the current change situation, EPS time, and the current total demand. Other details, including the accuracy and path error for each iteration and the status change of any hydraulically controlled valve, will be provided in the main window if the Info box is checked in the Solve Control window.

Pausing

The user may select to pause a solution sequence after each solution, at a specified interval or at a specified time. During a pause, the user may continue the solution sequence to completion or to the next pause by clicking on the Solve button.

To pause after each solution, for either an eps sequence or a non-eps sequence, check the Pause box on the Solve dialog. The user may toggle between pausing and not pausing after each solution by checking/uncheicking the Pause box (to resume pausing after each solution, the sequence would need to have been paused based on times, as explained below).

To pause an eps solution sequence at a specified time, select Edit | Change Data | Eps Data from the Main menu and enter the desired time in the Time Pause field. This field maybe updated during a pause to specify the next desired time to pause.

To pause an eps solution sequence at specified intervals, greater than the solution interval, select Edit | Change Data | Eps Data from the Main menu and enter the desired interval in the Pause Interval field. This field maybe updated during a pause to specify a new pause interval. A value of zero indicates no interval pausing.

Viewing Results

The current results after completion of a solution sequence or during any pause in a solution sequence may be viewed selecting Results | Inspect | <entity type> from the Main menu. If a results window is left open, the displayed results for the selected entity will be automatically updated to the current solution at each pause.

Right clicking on the results grid produces a popup menu that allows the display of the input data for the current entity or the display of the current entity on the map. For pipes and rules, the popup menu also includes the ability to dynamically change the open/closed status or the active status, respectively.

System Data

System Data can be edited by selecting the System | System Data item of the Main menu. The first 4 fields, Units, Viscosity, Specific Gravity and Friction Formula are self-explanatory.

Required Accuracy and Error Limit: iterations for a particular solution will stop when the relative accuracy is equal to or less than this value and the greatest path error is less than the error limit (default = 1 ft or 0.3 meter) . Relative accuracy is the the sum of the change in flows divided by the sum of flows for all paths in the network.

Maximum Trials: iterations for a solution will stop if the this number of iterations is reached, whether or not the Required Accuracy is obtained. If the Required Accuracy has not be obtained, the Solution Status in the Solve Control window will be noted as Not Accurate.

Limited Output allow controlling whether output includes specified pipes and node, as follows:

- Off, all pipes and nodes are included in output.
- Default No, a pipe or node is not included in output unless it's Output_Flag is “yes” (1).
- Default Yes, a pipe or node is included in output unless it's Output_Flag is “no” (0).

The remaining fields control aspects of the mathematical solution routines. Changing these parameters may adversely affect solution performance and/or accuracy. Consultation with the program author is recommended.

ChkVlv Factor: Check valves will be closed before the desired accuracy is obtained if the change in flow for the pipe is less than the flow by this factor. A lower value will cause check valves to be adjusted earlier in the solution iterations, with the possible consequence of needing a further adjustment later in the iterations.

Reset Valves: This option determines whether all valves are reset to an open status before each solution in a sequence.

Reset Flows: This option determines whether pipe flows are reset to 0 before each solution in a sequence.

Relax Mode: relaxation refers to the method of achieving an initial estimate of flows. The options determines when this function is performed. Newpaths provides the most conservative option, i.e. the best chance of convergence, while ModInit provides the fastest solution, if convergence is not adversely affect.

ChangeSet Data

ChangeSets allow the user to create and manage the data necessary to accomplish almost any modeling objective without affecting the original data or any other ChangeSet. This is sometimes referred to as scenario management. Other type of records belong to a ChangeSet, including Eps Data, Water Quality Data, Change Situation Data and a ChangeSet Description. The fields below are contained in a ChangeSet record.

ChangeSet Name: text which uniquely identifies a changeset from all other changesets.

Demand Factor: a universal demand factor to be applied throughout the solution sequence.

All Demands Off: allows the user to ensure that only the desired Demand Pts are used by starting with all Demand Pts off and then activating the desired Demand Pts by applying Group Demand factors. also see Demand Changes.

All Controls Off: allows the user to ensure that only the desired Control Sets are used by starting with all Controls Sets off and then activating the desired Controls Sets by setting the status in a change value for any change situation.

The Number of Changes Before Solution, the number of Change Situations (see Change Data) to be incorporated before the first solution during non-EPS runs.

EPS Data

EPS Data allows the user to control Extended Period Simulations.

The EPS flag determines whether or not a simulation is EPS.

The Start and End times determine the duration of the simulation and should be coordinated with Change Situation times.

The Solve Interval is the default interval between solutions; intermediate solutions may occur if a tank status changes.

A Maximum Tank Change value can be set to limit the maximum tank level change between solutions. If a tank level change reaches this value, a solution will be performed before a regular interval. This value can be overridden for individual tanks. See Tank Data.

A Timed Pause can be set so that the simulation is paused a specified time. If the EPS is repeated, the simulation will be paused at the specified time for every cycle.

A Pause Interval can be set so the simulation is paused after every specified interval.

An Output Interval can be specified so that output doesn't occur at every solution.

An EPS can be repeated (with tank levels maintained) any number of times by specifying a Repeat value.

Fire Flows

When a solution has been obtained during a solution sequence the user may then impose a fire flow at any junction node and see the hydraulic results. The user may specify a residual pressure and have the flow calculated or specify the flow and have the pressure calculated. All the normal tools are available to examine results throughout the network. Any number of fire flow conditions may be examined for a given solution and the fire flow analysis will not affect the solution sequence, by changing tank levels, for example. (If you want to determine the effect on tank levels of a fire flow of a given duration, that can be accomplished through Change Data.)

To conduct a fire flow analysis a solution sequence should be paused at the desired solution. All conditions for that solution, such as demands, tank levels and pump status will apply to the fire flow analysis. Right click on the map at any junction node and select FireFlow, then select CalcFlow or CalcPressure.

- If CalcFlow is selected the user specifies the desired residual pressure and the analysis will calculate the available flow.
- If CalcPressure is selected the user specifies the desired flow and the analysis will calculate the available pressure.

Quality Data

Quality Data allows the user to select options for a Water Quality Analysis.

Analysis Type - Age, Trace or Chemical; as explained in Water Quality Analysis.

Trace Parameters define the node used for a Trace analysis.

Time Step - Time interval between routing of water quality constituent. Normal default is 5 minutes (0:05 hours).

Quality 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.

Limit - 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. Leave blank if not applicable.

Bulk Coefficient - Default bulk reaction rate coefficient (Kb) assigned to all pipes. This global coefficient can be overridden by editing this property for specific pipes. Use positive number for growth, 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 reaction order.

Wall Coefficient - Default wall reaction rate coefficient (Kw) assigned to all pipes. Can be overridden by editing this property for specific pipes. Use positive number for growth, 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.

Diffusivity - Molecular diffusivity of the chemical being modeled relative 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. Only used when modeling mass transfer for pipe wall reactions. Set to zero to ignore mass transfer effects.

Chemical Name - user assigned name.

Bulk 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. See Bulk Flow Reaction Rates.

Wall Order - Power to which concentration is raised when computing a pipe wall reaction rate. Choices are FIRST (1) for first-order reactions or ZERO (0) for constant rate reactions. See Pipe Wall Reaction Rates.

Roughness Factor - factor correlating wall reaction coefficient to pipe roughness. see Epanet2.hlp, wall - roughness correlation.

Maps

A Map window is created when the program is started and is deleted when the program is terminated. The user may maximize, minimize or resize the Map window, but may not delete it.

Schematic maps of a network can be generated by selecting Draw | Draw from the Map window menu. In addition to pipe and node id's, pipe diameters will be shown in smaller type under the pipe id. Using options available from the Map window menu, the user can locate a pipe or node, zoom or pan. Zooming also can be accomplished using the mouse wheel; panning to a point can be accomplished by double clicking.

Using options from the popup menu, the user can select different actions relating to the selected element including editing input data, displaying live results, moving the entity and changing pipe length (see Automatic Data Entry). After selecting Move from the popup menu, a move is accomplished by clicking and dragging the selected entity.

For pipes, the diameter is displayed on the map just below the pipe id. If a pipe contains a pump or valve, a "P" or "V", respectively, will be displayed just above the pipe id. Likewise, a fixed grade node with a tank has a "T" and a junction node with an emitter has an "E". Right clicking on these superscripts allows views for the related data to be accessed.

(Suggestion: Using dual monitors with the map maximized on one has been found to be a convenient technique for using the model).

Overview Window

An Overview window will be displayed after selecting Pan | Overview from the Map menu. The overview window floats over the map window and initially has ¼ dimension of the map window, but can be resized and placed anywhere. The overview window will have a duplicate of the drawing in the map window plus a pan ring.

- Dragging the pan ring results in panning within the map window such that the center coordinate in the map window corresponds to the center coordinate of the pan ring.
- When Zoom | Extents is selected from the main menu of the overview window, the entire drawing is displayed within the overview window and the pan ring is placed corresponding to the current center coordinates of the map window.
- If a new network is selected, Zoom | Extents should be selected in the overview window to refresh the overview drawing.
- The user may zoom and pan within the overview window to suit a given purpose.

Creating a Map

To create a map, the data must include coordinates for all junction and fixed grade nodes. Coordinates for pipes are optional; if no coordinates are provided, midpoint coordinates will be calculated and stored with the pipe record. Parallel pipes will be offset to prevent overwriting. If a node is moved, the coordinates for the connecting pipes will be set to zero so that the midpoints will be recalculated.

The elements in these maps are automatically scaled to prevent overwriting other elements. The user must specify a default node radius, which will be used if space is available, and may specify a minimum radius, if desired (overwriting may occur when using a minimum radius). These and other options may be specified by selecting Edit | Map Data from the Map window menu.

For a new network, creating the map for the first time will take several times longer than subsequent drawings. This is because the pipe coordinates and node radii will be stored and need not be calculated again. Radii and pipe coordinates will be recalculated if set to zero. All node radii can be reset to zero by selecting Draw | Recalc Radii from the Map window menu.

Pipes and nodes added during editing will not appear on a map until the map is redrawn, either by redrawing to full extents with Draw | Draw or redrawing with the current zoom with Draw | Draw_CurrentZoom from the Map window menu.

A map can be exported for use in third party software by selecting Draw | Export | [format] from the Map window. The following formats are available: svg, png, pdf, dxf and TdhCad.

Automatic Data Entry

The Automatic Data Entry mode can be entered by selecting Edit | AutoEntry from the

Map window menu. The AutoEntry options will appear in a dialog window and will remain available until the dialog is closed.

AutoEntry actions are implemented by right clicking or double clicking on the map. The AutoEntry action to be implemented is selected from radio buttons in the AutoEntry options.

With the exception of Distance (see below), the first AutoEntry action must be one that creates or selects a node. "Create Fgn" creates a new fixed grade node at the selected location. "Start Branch" selects an existing junction node. "Split Pipe" inserts a new junction node at the selected location for an existing pipe, creating a new pipe and modifying the existing pipe.

Subsequent actions can create new pipes. "Create Pipe/Jun" creates a new pipe from the previous node to a newly created junction node at the selected location. "Close Loop" creates a new pipe from the previous node to the selected, existing node. After Close Loop, the next action must create or select a node.

When adding a pipe, intermediate points are set with a single left click and the end node point is set with a double left or single right click. Intermediate points can be undone prior to setting the end node point.

All actions can be undone in reverse order by selecting Undo and redone by selecting Redo.

Other options within the AutoEntry panel affect the data created. If the Prompt for ID's box is not checked, pipe and node id's will be created automatically, in numerical sequence; otherwise, the user will be prompted to confirm or edit the suggested id's. An entry in the ID Prefix box will be used as a prefix for any automatically created id. Newly created pipes will be given the diameter and roughness specified in the corresponding boxes, which can be modified at any time.

The Distance option can be used to calculate the distance of a multi segment line and to change the length of a pipe to that distance. With the Calc Distance radio button selected, the line to be calculated is begun by double clicking, vertices are added by single clicking and the line is ended by double clicking. After the second double click, the calculated distance is displayed. If the user then right clicks on a pipe, the popup menu will include an option allowing the pipe length to be changed to the calculated distance.

Cell Grids

When creating a network map, it is possible to include a cell grid, with the cell dimensions and labeling specified by the user. The dimensions and labels are specified by selecting Draw | Map Data from the Map window menu.

When a map is created with a grid cell, the options available in the map window include the ability to locate any cell according to its label. The user may snap to the dimensions of the cell where the center of the display is located. When snapped to a cell, pans (up, down, left, right) will be in increments of the cell dimension.

The label specifications may contain the following fields, in any order, separated by commas:

1. A String Literal enclosed in quotes. Up to 10 alphanumeric characters to be included in the cell label. Multiple String Literals are allowed.

2. x<number>. The number of digits to be used for the grid position along the x-axis.

3. y<number>. The number of digits to be used for the grid position along the y-axis.

4. E. A single character with a value of E or W is used to signify whether the x coordinates are positive or negative, respectively.

5. N. A single character with a value of N or S is used to signify whether the y coordinates are positive or negative, respectively.

For example, given the following cell label specification:

CellLabel = "2",y2,N,E,x2

the label 237NW09 would be translated as follows:

The leading 2 is a string literal and has no meaning with regard to cell position.

The next two digits indicate the cell is in the 37th position from the origin along the y-axis.

The next character indicates the cell is above the origin.

The next character indicates the cell is to the left of the origin.

The next two digits indicate the cell is in the 9th position from the origin along the x-axis.

Note that leading zeros must be used to maintain the correct number of digits for the x and y positions. If the label description does not contain an E/W or N/S field and negative positions are possible, the number of x or y digits, respectively, should include space for a minus sign.

The grid positions in any direction begin with 1. There is no 0 grid position. The Cell Origin fields may be used to specify the coordinates at which cell counting begins.

Raster Images

Any number of raster images can be displayed as a background for a network drawing by selecting Draw **|** Images the Map window menu. The following file types can read:

BMP, PNG, JPEG, GIF, PCX, PNM, TIFF, TGA, IFF, XPM, ICO, CUR, ANI.

After an image is read, 2 coordinates defining the extents of the image are needed. These extents will be read from a World file, if available, or they may be entered by the user directly or through the image calibration process. Note that the extents coordinates (i.e. the minimum x, minimum y and the maximum x, maximum y) will not lie within the raster image if the image does not align with drawing coordinate system.

From the Draw **|** Images menu, the user may also:

- Change the extents coordinates.
- Rotate the image (used when the image does not align with drawing coordinate system).
- Specify a transparency value for the image, from 0 (no transparency) to 255 (full transparent).
- Set the Maximum Scale, beyond which the image will automatically be suppressed.
- Write World File, savings the extents and rotation information to a World File, which can be use by many other graphics software.
- Delete the image.

- Suppress the image, such that it will not be displayed but all information will be retained.

If zooming would require scaling the image beyond the Maximum Scale , the image will not be displayed.

The image parameters current when a network is saved will be saved with the network and used when the network is selected again. Scaling beyond 1 may start to result in display delays.

Tip => for tips on how to obtain and align a background map, see the TdhGIS_tutorials.pdf in the TdhGIS Library at TdhGIS.com (TdhGIS is free software),

Image Calibration

When an Image file is read, if a corresponding World file is not found, the user will be given the option to Calibrate the image. Calibration requires the user to right click on 3 points on the image and specify the coordinates in the desired units. The program will then calculate the image extents and rotation. The points selected by the user should be well separated, particularly along the x axis.

Prior to entering points, the user will be given the opportunity to specify an assumed aspect ratio (width to height). This aspect ratio is important only in displaying the image during the calibration process, it has no effect on the calibration results.

Tip => After obtaining extents and rotation from calibration, the user may save this information to a World file for future use by selecting File | Image | Write World File.

TdhCad Drawing

TdhCad is a free vector graphics program. A TdhCad Drawing can be displayed as a layer by selecting Draw | Import | TdhCad Drawing from the Map window. A directory containing a TdhCad database and drawing within that database must be selected. The drawing will be displayed when the network is subsequently opened until Cad data is modified by selecting Edit | Map Data from the Map window. A TdhCad drawing can be useful for displaying labels and annotations.

Tip – To add labels and annotations, export the network map to a TdhCad drawing, then create a new layer to add labels and annotations. Before importing the drawing into the TdhNet map, hide the network layer in TdhCad, so that it is not imported.

Legend

The Legend dialog allows the automatic creation and editing of a map legend. Legends are created on a Legend layer by selecting Edit | Legend | Create/Edit from the Map window, The dialog provides the following options:

- Title: the text used for the legend title.
- Items Box: each line in the Items box creates a Legend Item. Each line consists of 2 comma separated fields:
    - Color Field: can be specified as either a standard color text (e.g. “red”) or a 4 byte hexadecimal (alpha, red, green, blue) preceded by “#”. The color field can be automatically entered by clicking on the Color button and using the standard color dialog.
    - Description Field: the description text to be associated with the color.
- Position: when creating a new legend, the position information is set to approximate the current window but may be modified as follows:

    - Left, Lower: specifies the x,y coordinates for the left lower corner of the bounding parallelogram.
    - Height: specifies the height of the bounding parallelogram (the width will be determined by the text).
- Create: creates a legend as specified within the dialog. if an existing legend is being edited, the previous version will be deleted and the new version created.

A Legend can be saved to a TdhCad drawing by selecting either Edit | Legend | Save or Draw | Export |TdhCad Drawing from the Map window. The TdhCad drawing used to save a Legend will be automatically displayed upon project startup, unless the map option Cad Flag is turned off. The entities created by the Legend dialog may be manipulated by any of the applicable tools available in TdhCad.

Results and Reports

For viewing live results as they are calculated, use the Results item on the Main menu.

Reports are generated through the Reports item in the Main menu and most send output to the text editor in the Main window. The user has the ability to modify this text as desired and to send the text to a file using the Reports | Save To File item in the Main menu. The entire contents of the text editor will be erased by selecting the Reports | Clear item.

Reports for input data include a data check report and an input summary report sent to the text editor. Reports for results include a standard report sent to the text editor and the ability to create CSV (comma separated variable) files for results associated with FGNs, Junctions and Pipes. These files can be easily imported into other software such as spreadsheets and database managers, allowing the user many options for results analysis and presentation.

Queries

Queries may be executed on any saved results by using the Results | Query | Saved option from the Main menu or on the current results by using the Results | Query | Current option from the Main menu. The options in the Query Dialog may used to prepare text displayed in the SQL text box or any SQL text can be entered and edited in this box. The SQL text will be executed by clicking on the Execute button. The same SQL text will be executed on the next or previous solution in a solution sequence by clicking the spin control to the right of the Execute button.

The options in the Query Dialog are normally used from left to right.

- The Results radio box is used to select the type of data to be queried.
- The Sort option may be used to select a field to sort on, either ascending or descending.
- The Filter Options radio box includes options for filtering on input data or results data and a Series option. The Series option allows a query over all solutions in a solution sequence for a selected entity (specified in the Filter Criteria box).
- The Filter Field choice box and Filter Criteria edit box allow the selection of a field for filtering based on the specified criteria.
- The Row Limit option allows a limit on the results presented, which defaults to 100. A value of 'none' or 0 means all results will be presented.

Clicking on the Prepare button will generate Sql text based on the options selected in the Query Dialog. This text may used as is or modified to perform a customized query.

Query Sql text, along with descriptive text, can be saved within the database by clicking the Save button. Saved queries can be retrieved by clicking on the Retrieve button, which also allows a search based on strings within the descriptive text. Saved queries do not belong to a network and therefore may be used for networks other than the network for which they are originally generated. (Note that the query fields selecting the network, run and solution are parameterized so that the currently selected network, run and solution will be queried regardless of which were selected when the query was generated.)

Map Colors

The color and line width of any map element can be modified based on input data and/or results by selecting Draw | Colors from the Map window menu. The Colors Dialog options generate a script that determines how map colors will be modified. This script is displayed in the Script edit box and can be directly edited by the user.

The options in the Colors Dialog are normally used from left to right:

- The Populate buttons select the entities to be acted on by subsequent options, any previously selected entities are cleared.
- The Filter options allows entities to be filtered based on input data, current results or saved results. The filter options are implemented by clicking on the Apply Filter button.
- The Color and Width options allow a color and line width to be applied to the current entities.

Gradients can be accomplished by applying successive filters and colors. For example:

- Start with all nodes and apply the default color.
- Filter pressure < 50 and apply color orange.
- Filter pressure < 30 and apply color yellow.
- Filter pressure < 20 and apply color red.

In addition to the colors provided in the selection box, any RGB (3 byte) color can be selected by using the More... option and the displayed color wheel.

By default, all elements on a map are drawn with a line width of 1 pixel. The line width option allows the line for the selected entities to be any specified number of pixels.

The script shown in the Script Text box can be executed at any time by clicking the Execute button. The spin control below the Execute button allows the script to be executed for the previous or next solution in a saved solution sequence. This could be used, for example, to graphically display the changes in water quality parameters calculated during an EPS.

The colors and line widths will be used for elements in the network until they are set to some other value, such as the defaults.

Color scripts can be saved and retrieved in the same manner as queries.

Contours

Contour lines can be drawn on the map based on current or saved results for the following parameters:

- Hydraulic Grade
- Pressure
- Water Quality

Contour lines can also be drawn based on junction node elevations. It should be noted that all data for contour lines is accessed from the database; therefore changes to input data, such as node coordinates, must be saved to the database before they are reflected in the contour lines. Right clicking on a contour line will display the contour value.

The Contours Dialog can be accessed by selecting Draw | Contours from the Map window menu. This dialog generates a script that determines how the contours will be drawn. This script is displayed in the Script edit box and can be directly edited by the user.

The options in the Contour Dialog are normally accessed from left to right:

- The Filter options allows entities to be filtered based on input data. The filter options are implemented by clicking on the Apply Filter button.
- The Source Data option determines whether the parameter data is obtained from current results or saved results and the parameter field.
- The Lines options control the appearance of the contour lines, including the interval between lines, the number of lines between major lines, the minimum and maximum values and the line color. It is possible to generate different line colors for different contour ranges or different data points, e.g. for different zones.
- Label Size – determines the size of value labels for major lines.
    - A value greater than 0 specifies the text height in distance units (text size will change will zoom factor).
    - A value less than 0 specifies the text hight in pixels (text size won't change with zoom factor).
    - A value of 0 results in no label text.

The script shown in the Script Text box can be executed at any time by clicking the Execute button. The spin control below the Execute button allows the script to be executed for the previous or next solution in a saved solution sequence. This could be used, for example, to graphically display the changes in water quality parameters calculated during an EPS.

Contour scripts can be saved and retrieved in the same manner as queries.

Charts

A chart showing a selected parameter from selected entities from the results of an EPS solution sequence can be produced using the Results | Charts option of the Main menu. The options in the Chart dialog are normally used from left to right.

- The Records buttons select the type of entities to be charted.
- The Add, Edit, Delete and Clear buttons are used to populate the list of entities to be charted. Optionally, a color may be specified after the entity id, comma separated.
- The Field Choice box selects the data field to be charted.
- The Title text box allows for the inclusion of text at the top of the chart.

After specifying information in the Chart dialog, the following buttons may used:

- Chart – a window is displayed with the specified chart. Menu options allow:
    - a white background
    - exporting the chart to various formats, including a TdhCad drawing
    - editing the chart through the Chart edit dialog

- Write CSV – the chart data is directed to a comma separated variable file so it may be imported into a spreadsheet or other software.
- Save – the information contained in the Chart dialog is saved using a user specified name.
- Retrieve – saved Chart dialog information is restored.

Pump performance charts, including efficiency, can be obtained by right clicking on the data grid from the Pump edit dialog. Composite performance and efficiency charts can be obtained for pumps within a pumping station by right clicking on the data grid from the Station edit dialog. To exclude a pump from the composite chart, right click on the Station edit grid, select First or Next Pump to find the desired pump belonging to the station and then edit the pump field "Station Curve".

Using the Database

The TdhNet database is contained in 2 SQLite files, TdhnetData.sqlite and TdhnetResults.sqlite. The former contains all the tables for input data and the latter contains all the tables for saved results. Multiple databases may be created and selected using the Database | New and Database | Select options of the Main menu. The directory containing the active database may be determined by using the Database | Current option of the Main menu.

Use of the database is entirely optional, the model can be used based on traditional file IO only. Use of the database allows:

- data storage and retrieval without traditional file IO.
- the querying of results
- the coloring of map elements based on input data and/or results.
- The extraction of graphically selected input data into a new network
- the merging of 2 network into a new network
- restarting an EPS based on saved results

A single database may contain data for any number of networks. The active network is selected using the File | Open | From Database option of the Main menu. Data may be saved to the database immediately after reading a file or at any other time during program operation using the File | Save | To Database option from the Main menu.

Solution results can be saved for networks contained in the database. Saved results are grouped first within a Run. A Run may contain any number of solutions within a solution sequence, such as from an EPS. A solution contains results for individual entities, such as pipes and nodes. Results may be saved at every report interval during an EPS by using the Results | Save | All option from the Main menu or at any selected time during a solution sequence by using the Results | Save | Current option from the Main menu.

The Run and Solution results to be used in Queries and Map Colors may be selected by using the Results | Database | Select Run and Results | Database | Select Solution options from the Main menu.

If Limited Output is selected within the System Data, saved results will include only pipes and junctions that have the Output Flag set > 0.

Extracting Data

Data may be extracted from a network to create a new network. Extraction is begun by selecting Draw | Extract | Begin from the Map window menu. By right clicking on a pipe, the pipe data and all associated data, including pumps, valves, nodes, tank and emitters are included in the extracted data. When a pipe is extracted, it's color is changed to red.

To stop the extraction process and save the extracted data to a new network in the database, select Draw | Extract | End from the Map window menu. The color for all extracted pipes is changed to the default color.

Merging Networks

The currently active network may be merged with any network in the database to created a new network by selecting File | Merge from the Main menu. If the 2 networks use the same ID for a given entity type, only the entity from the currently active network will be included in the new network.

Restarting an EPS

Any saved results from an EPS solution can be used to restart an EPS based on the conditions that existed when the results were saved. This includes setting the start time and tank levels to the time and levels of the saved results. The initial water quality values will be set to the values in the saved results for all FGNs and all junctions included in the results (using results with limited output is not recommended in this situation). It should be noted that water quality results in the restarted EPS will be affected somewhat by the fact that water quality within the pipes is initialized to the average quality from the restart results. The is a significant improvement from the Epanet default of initializing pipe water quality as the downstream node water quality, but will not produce precisely the same results as continuation of the original solution sequence.

Orphans and Compaction

Under certain conditions, such as an interrupted deletion of a network, some records may remain in the database when the parent record no longer exists. These records may then reappear inappropriately in subsequent networks. To detect and remove any such records, use the Database | UnOphan option from the Main menu.

If it is desired to reduce the size of the database files after deleting a large amount of data, use the Database | Compact option from the Main menu.

Detection and deletion of orphans and database compaction may required a significant amount of time, depending on the number of records in the database.

Input Data Specifications

This chapter provides a field by field description of the data used by a TDHNET file. Each description begins with the data type (e.g integer, real, alpha) and value restrictions (e.g. non-negative, <1). A data type of key is equivalent to a 10 character alpha type. A data type of flag is a single alphanumeric character, usually a numeric digit.

Tdhnet Data Files

A Tdhnet file has a segmented structure. All data is contained within a section, each section starts with a section name, and each section name is on a line where the first field contains '@'.

Sections may appear in any order and any section may be omitted or repeated. However, when records reference other records (e.g pump points records reference the pump to which they belong) the referenced record must already exist for the new record to be associated with it.

Sections may have subsections which will be read properly only if it appears within the section. Sections with unknown names will be ignored and subsequent sections will be processed.

All fields are separated by commas. The numeric fields are free-format. Repeating data records within a section or sub-section have a '+' character in the first field. Any line within a set of repeating records which does not have a '+' in the first field is ignored.

The following is a list of sections which may appear in a TDHNET Data File:

Network Data - contains subsections beginning with "Network_Name" and ending with "End_Marker". The subsections contain the data described in the Input Data Specifications Chapter.

Network Name - One data line containing the network name.

System Data - a single record of the system data described in the Input Data Specifications.

Network Description - multiple records where each record is a line of the network description.

Pipes - multiple records of the pipe data.

FGNs - multiple records of the Fixed Grade Node data.

Junctions - multiple records of the Junction data.

Demand Pts - multiple records of Demand Pts data. This sub-section can be used as a stand-alone section to load new demands into a network.

- If the second line of this sub-section is "@,Reset_Demands, all existing demands will be deleted.
- If the second line of this sub-section is "@,Archive_Demands", all junction node demands will be transferred to Demand Pts belonging to a group with the id "Archive".
- If the section name Is “Set Jun Group”, then the following line will be read as a group name to be assigned to all junctions nodes listed on subsequent lines within the section.

Pumps - multiple records of the Pump data.

Pump Points - multiple records of the Pump Point data. The second field contains the PumpID of the pump record to which the pump point belongs.

PRVs - multiple records of the PRV data.

Tanks - multiple records of the Tank data.

Tank Points - multiple records of the Tank Point data. The second field contains the TankID of the tank record to which the tank point belongs.

Switches - multiple records of the Hydraulic Switch data.

SwitchPoints - multiple records of the Switch Point data. The second field contains the SwitchID of the switch record to which the switch point belongs.

Stations - multiple records of the Pumping Station data.

Options - contains flags for options applicable to a network.

Map Data - contains the following data: default radius, minimum radius.

CellGrid Data - A section which contains five lines. The first line contains the Cell Grid flag, the cell x dimension and the cell y dimension. The second line contains the full cell name description, with each descriptor field separated by a comma. The third and fourth lines contain the x and y descriptors, respectively. The fifth line contains the Cell Origin coordinates.

Control Sets – contains sub-sections for controls and ends with "End_Marker".

Rules – sub-section containing multiple records for rules.

Change Data - contains sub-sections beginning with "Change_Set" and ends with "End_Marker". The next line contains the Changeset Name.

ChangeSet Description - A sub-section which contains multiple records, where each record is a line in the ChangeSet Description.

Options - contains data for EPS parameters and other options applicable to a changeset.

Quality - contains data for Water Quality Analysis parameters.

Change Situations - multiple records for change situation data.

New Values - multiple records for new values in the change data. The second field contains the situation number to which the new value belongs.

Notes - a set of records for each note.

System Data

System Data for a network is contain on 3 types of records, each in it's own section. The sections are System, System_Labels and Options. The fields for the System record are as follows:

Units: (int) 0 - gpm; 1- cfs; 2- mgd; 3- SI_lps, 4-SI_lpm. For SI units, pressure is in meters, length is in meters and diameter is in milimeters. For all other flow units, pressure is in psi, length is feet and diameter is in inches.

Maximum Trials: (int) the maximum number of iterations performed by the solutions engine for a single solution.

Relative Accuracy: (real, >0) the relative accuracy (change in flows over total flow) the program will attempt to achieve.

Error Limit: (real, default = 1 ft or 0.3 meter) the greatest allowable path error for a solution to be considered accurate.

Specific Gravity: (real, >0) the specific gravity of the fluid within the modeled network. A value of 0 is the same as a value of 1.0.

Kinematic Viscosity: (real, >0) the kinematic viscosity of the fluid within the modeled network. This value is used only with the Darcy-Weisbach friction formula.

Friction Formula: (flag) 1- Hazen-Williams; 2 - Darcy-Weisbach.

For the System_Labels section there can be any number of records, each starting with the multiple record character (+) followed by a label string.

The Network Options section has the following fields:

Report Flag: a non-zero value prompts the generation of a standard report after a solution.

Results to Database Flag: a non-zero value prompts results to be stored in the database after a solution.

Limited Output: allows results output to be limited to specified pipes and nodes.

Page Length: a non-zero value is used as the page length in standard reports.

Pause Flag: a non-zero value prompts the program to pause after a solution, until the solve button is clicked.

Key Fields

A key field may contain an alphanumeric string of any length.

Pipe Data

See also: Pipes

Pipe ID: (key) uniquely identifies the pipe.

NodeA: (key) the first connecting node, either a junction or a FGN.

NodeB: (key) the second connecting node, either a junction of a FGN.

FGN Flag (flag) 0-both nodes are junctions, 1- NodeA is a FGN, 2- NodeB is a FGN.

Closed Flag (flag) 0-pipe is open, 1- pipe is closed.

Check Valve Flag (flag) 0 - no check valve, 1- pipe has a check valve which allow flow only from NodeA to NodeB, 2 - pipe has a check valve which allows flow only from NodeB to NodeA.

Length: (real, 0<) length of pipe in units determined by the System Data.

Diameter: (real, 0<) inside diameter of pipe in units determined by the System Data.

Roughness: (real, 0<) when the Hazen-Williams friction formula is used, this is a unitless parameter referrred to as the c-value; when the Dary-Weisbach friction formula is used, this is in milifeet or milimeters.

Minor Loss: (real, non-negative) The sum of all minor loss coefficients which apply to the pipe.

Wall Coeffecient: (real) wall reaction rate coefficient (Kw) assigned to this pipe for Water Quality Analysis. Overrides global coeffecient.

Bulk Coeffecient: (real) bulk reaction rate coefficient (Kb) assigned to this pipe for Water Quality Analysis. Overrides global coeffecient.

X Coordinate: (real) the x coordinate for the pipe; if this value is zero, it will be calculated as the midpoint between the connecting nodes.

Y Coordinate: (real) they y coordinate for the pipe; if this value is zero, it will be calculated as the midpoint between the connecting nodes.

Output Flag: (flag) – determines whether the pipe is included in output results when Limited Output is specified in the System Data.

Color Code: (int) determines the color assigned to the pipe in a Map.

Radius: (float, non-negative) determines the size of the PipeID in a Map.

Note Code: (int) references a note.

Junction Node Data

NodeID: (key) uniquely identifies the junction node.

Demand: (real) a flow into(<0) or out of (>0) the node.

Elevation: (real) the elevation of the node.

X-Coordinate: (real) the x-coordinate of the node.

Y-Coordinate: (real) the y-coordinate of the node.

Output Flag: (flag) - determines whether the node is included in output results when Limited Output is specified in the System Data.

Group Name: (key) the node group, if any, to which the node belongs.

Quality Start: (real) the water quality value for the junction at the start of the simulation.

Quality Source: (real) a value greater than zero indicates that the junction is a source for a Water Quality Analysis. Units are determined by the Source Type field.

Source Type: (int) 0 - Concentration, 1 - Mass, 2 - Setpoint, 3 - Flowpaced.

Color Code: (int) determines the color assigned to the junction in a Map.

Radius: (float, non-negative) determines the radius of the junction in a Map.

Note Code: (int) references a note.

Demand Pt Data

Junction ID: (key) the junction to which the demand pt belongs.

Group ID: (key) the junction group to which the demand pt belongs.

Demand: (real) a flow into(<0) or out of (>0) the node.

Fixed Grade Node Data

NodeID: (key) uniquely identifies the junction node.

Grade: (real) the fixed grade for the node.

X-Coordinate: (real) the x-coordinate of the node.

Y-Coordinate: (real) the y-coordinate of the node.

Quality Start: (real) the water quality value for the FGN at the start of the simulation.

Quality Source: (real) a value greater than zero indicates that the FGN is a source for a Water Quality Analysis. Units are determined by the Source Type field.

Source Type: (int) 0 - Concentration, 1 - Mass, 2 - Setpoint, 3 - Flowpaced.

Color Code: (int) determines the color assigned to the FGN in a Map.

Radius: (float, non-negative) determines the radius of the FGN in a Map.

Note Code: (int) references a note.

Pump Data

With regard to link/node data, a pump becomes an attribute of the pipe specified with the PipeID field. A PumpID must be unique among pump records. It may or may not be the same as the PipeID to which the pump belongs. A pipe may have only one pump. Each pump record may have associated Pump Point Data.

PumpID: (key) uniquely identifies the pump record.

PipeID: (key) the pipe to which the pump belongs.

Pump Type: (flag) 1- constant power, 2- three operating points

Power: (real, non-negative) the useful horsepower to be used when modeling a constant power pump.

StationID: (key) the station in which the pump is located. Required only if the pump is to be included in a pumping cost analysis.

NPSHR: (real) the grade for the upstream node below which a warning of possible cavitation will be issued.

Note Code: (int) references a note.

Pump Point Data

This data represents repeating values associated with a Pump record. For constant power pumps, no such values are used. When simulating a pump based on the characteristic curve, three such values are required. One of the three values should have a flow of 0. The relative values for tdh must be inverse to the relative values of flow.

PumpID: (key) the pump to which the pump point belongs (does not appear in edit grid).

Flow: (real, non-negative) the flow through the pump at the specified operating point.

TDH: (real, non-negative) the total dynamic head produced by the pump at the specified operating point.

Efficiency: (real, 0< <1) the wire to water efficiency of the pump at the specified operating point. Required only if the pump is to be included in a pumping cost analysis.

Valve Data

With regard to link/node data, a PRV becomes an attribute of the pipe specified in the PipeID field. A PRVID must be unique among prv records. It may or may not be the same as the pipeid to which the prv belongs. A pipe may have only one PRV.

ValveID: (key) uniquely identifies the PRV.

PipeID: (key) the pipe to which the PRV belongs.

Valve Type: (flag) 0-relief, 1- reducing, 2- flow control, 3- general purpose, 4-throttle, 5-float

Active: if false, the valve is not used in the simulation.

Grade: (real) the hydraulic grade or flow maintained by the PRV

Valve Pt Data ID (key) specified the Valve Pt Data to be used for general and float valve.

Note Code: (int) references a note.

Cv: the Cv value for a throttle and float valve.

Amount Open: the amount open (0-close, 1-fully open) for a throttle valve and for a float valve when the tank level is less than the lowest Valve Pt.

Exponent: the exponent applied to the amount ope for a throttle and float valve.

TankID: specified the tank used to control a float valve.

Valve Pt Data

A Valve Pt record can be used for any number of valves.

Valve Pt ID (key) a unique identifier for the Valve Pt data record.

Valve Type: Selected general or float, so that TdhNet uses appropriate labels for the data.

Valve Pts Data

Any number of Valve Pts can be included for a Valve Pt record.

For a general valve, when flow is between the specified flow values, linear interpolation for loss is used. For flow above the highest entered value, linear extrapolation is used. If the minimum entered flow value is not zero, a data point of (0,0) is assumed.

Flow: (real) the scalar (non-negative) value of the flow through the valve.

Loss: (real) the loss at the specified flow.

Emitter Data

An emitter is attached to a junction node.

Emitter ID: (key) uniquely identifies the emitter.

Junction ID: (key) the junction to which the emitter is attached.

Coefficient: (real) a value that determines the flow vs. pressure relationship. The units for this value must be in the user specified flow and pressure units.

Diameter: (real) used to calculate a velocity for the emitter. This value has no effect on the flow calculations, except a value of zero will disable the emitter.

Note Code: (int) references a note.

Exponent: (real) the exponent applied in the emitter loss equation.

Tank Data

Tanks can be located only at fixed grade nodes. Each Tank must have one or more associated Tank Point records.

Tank ID: (key) uniquely identifies the tank within the tank data. It may or may not be the same as the NodeID for the fixed grade node where the tank is located.

FGN ID: (key) Node ID for the fixed grade node where the tank is located.

Maximum Elevation: (real) the maximum elevation of water in the tank. If the actual water level reaches this elevation no additional water will flow into the tank.

Minimum Elevation: (real) the minimum elevation of usable water in the tank. If the actual water level reaches this elevation, no additional water will flow out of the tank.

Dead Volume: (real) the volume of water below the minimum usable elevation (MG or ML).

Note Code: (int) references a note.

Maximum Level Change: (real) maximum level change between solutions. 0 – eps default value is used; < 0 – no maximum level change is used. If the level change reaches this value, a solution will be performed before the regular interval.

Allow OverFlow: if false, inflow will not be allowed into the tank when that is at its maximize elevation. If true, inflow will be allowed but it will change the water level.

Tank Point Data

Any number of Tank Points can be included for a Tank. In a level range between Tank Points, a trapezoidal shape is assumed. If the calculated water level goes above or below the maximum or minimum level contained within the Tank Point data, a cylinder is assumed based on the nearest diameter. Therefore, only one Tank Point value is required per tank.

TankID: (key) the tank to which the tank point belongs (does not appear in edit grid).

Diameter: (real, >0) the diameter of the tank at the corresponding level.

Level: (real) the height above the minimum water level.

Pumping Station Data

StationID: (key) uniquely identifies the station in the station data.

Station Type: (flag) 0 - station describes a parallel relationship between pumps; 1 - station describes a parallel relationship between other stations; 2 - station describes a serial relationship between other stations.

Component1 and Component2: (key) the stations in the relationship described by the current station, if applicable.

Unit Cost: (real) the cost per kilowatt hour for the station. If the station describes a relationship between other stations a non-zero value will apply to all station which comprise the relationship, either directly or indirectly.

Note Code: (int) references a note.

Changeset Options Data

Demand Factor: a factor to be applied, for the entire solution sequence, to the demand for all junction nodes, except those that are included in the Node Group 'fixed'.

Change Situations Before Solve: (int) for a non-EPS, this number of situations will be incorporated before the first solution.

All Demand Pts Off: (int) a non-zero value turns off all demand pts, which can then be selectively turned on with group demand factors in the change data.

All Control Off: (int) a non-zero value turns the status for all controls to off, which then be selectively turned on in the change data.

EPS Data

EPS data controls the execution of an Extended Period Simulation and has the following fields:

EPS Flag: (int) a value greater than zero prompts an EPS.

Interval: (real, >0) the default time increment. If a tank level reaches some action level, an intermediate solution will be performed.

Start Time: (real) the time at the start of an EPS.

End Time: (real) the time at which an EPS cycle will end.

ReportStep: (real) results for an EPS will not be output if the time elapsed since the previous output is less than this value.

Cycles: (int) the number of times an EPS will be cycled

MaxTankChange: (real) maximum tank level change between solutions, can be overridden for individual tanks.

Change Situation Data

Each Change Situation may have one or more associated Change Value records.

Situation Number: (real) uniquely identifies the change situation and determines the sequence in which the situations are incorporated.

EPS Time: (real) - the time the situation is incorporated into an Extended Period Simulation. EPS Time must be sequenced the same as the Situation Number.

Demand Factor: (real) - a factor to be applied, until the next change situation, to the demand for all junction nodes, except those that are included in the Node Group 'fixed'.

Interval: (real) - the EPS time interval to be used until the next change situation is incorporated. If a change situation does not specify an Interval, the original interval is used.

Note Code: (int) references a note.

Change Value Data

Each New Value record is associated with a Change Situation record.

Situation Number: (int) the change situation to which the new value belongs. (Does not appear in edit grid).

Value Code: (int) the code which determines the type of the new value. (Does not appear in edit grid).

0 - new demand for a junction node (overrides any factors)
1 - new grade for a fixed grade node
2 - new pipe Status; 0 - toggle current status, 1- open, 2- close
3 - new pipe length
4 - new pipe diameter
5 - new pipe roughness
6 - new pipe minor loss
7 - new pump speed factor (0-1.5)
8 - new power for a constant power pump
9 - new grade/flow for a PRV
10 - new tank inflow
11 - new power cost for a pumping station
12 - apply a factor to the original demand for all nodes in a node group
13 - new source value for a junction node
14 - new source value for a fixed grade node
15 - new water quality start value for a junction node
16 - new water quality start value for a fixed grade node
17 - new pump speed for the specified pipe
18 - new status for the specified switch
19 - new factor for the specified switch

Entity ID: (key) the key value associated with the new value. (e.g. Pipe ID, Node ID, Pump ID or Junction Group Name).

New Value: (real) the new value to be applied.

Note Code: (int) references a note.

Note: When specifying a change in the Active value for a Rule, the composite ID (controlID:ruleID) must be used. Use the new values of 0 for inactive, 1 for active.

Water Quality Data

Water Quality Data is contained in 2 records with the following fields:

First record:

Global Wall Coefficient;

Global Bulk Coefficient;

Quality Solve Type: 0-none, 1-chemical, 2-age, 3-trace

Trace Node ID - if the solve type is trace, this field specifies the starting node id.

Trace Node Type - indicates whether the trace node is a junction (0) of FGN (1).

Second record:

Tolerance;

Limit;

Wall Order;

Bulk Order;

Roughness Factor;

Quality Time Step

Notes Data

Notes data is contained in 2 types of records with the following fields:

First record:

Note Code: (int) unique id for note.

Entity Type: (string) entity type to which the notes belongs.

Date: (int) when the note was created.

Second record type:

Note Strings: can be any number of records, each one starting the multi record character (+) and limited to 120 characters;

Pressure/Demand Data

Dependency ID: (key) uniquely identifies the record.

Active: (boolean) determines whether the dependency will be used.

Pressure Tolerance: (real) the required agreement, in pressure units, between the calculated pressure and the pressured used to determine the demand.

Pressure FullQ: (real) the pressure below which demands will be adjusted downward. If zero, no demand adjustment for low pressure will be applied.

Pressure NoQ: (real) the pressure at and below which there will be no demand. Linear interpolation is used to determine the demand between Pressure FullQ and Pressure NoQ.

Pressure High: (real) the pressure above which demand will be adjusted upward. If zero, no demand adjustment for high pressure will be applied.

Slope High: (real) the increase in demand for every unit of pressure above Pressure High.

Junction Group: (string) the junction group to which the dependency will apply. If blank, the dependency will apply to all junctions.

Note Code: (int) references a note.