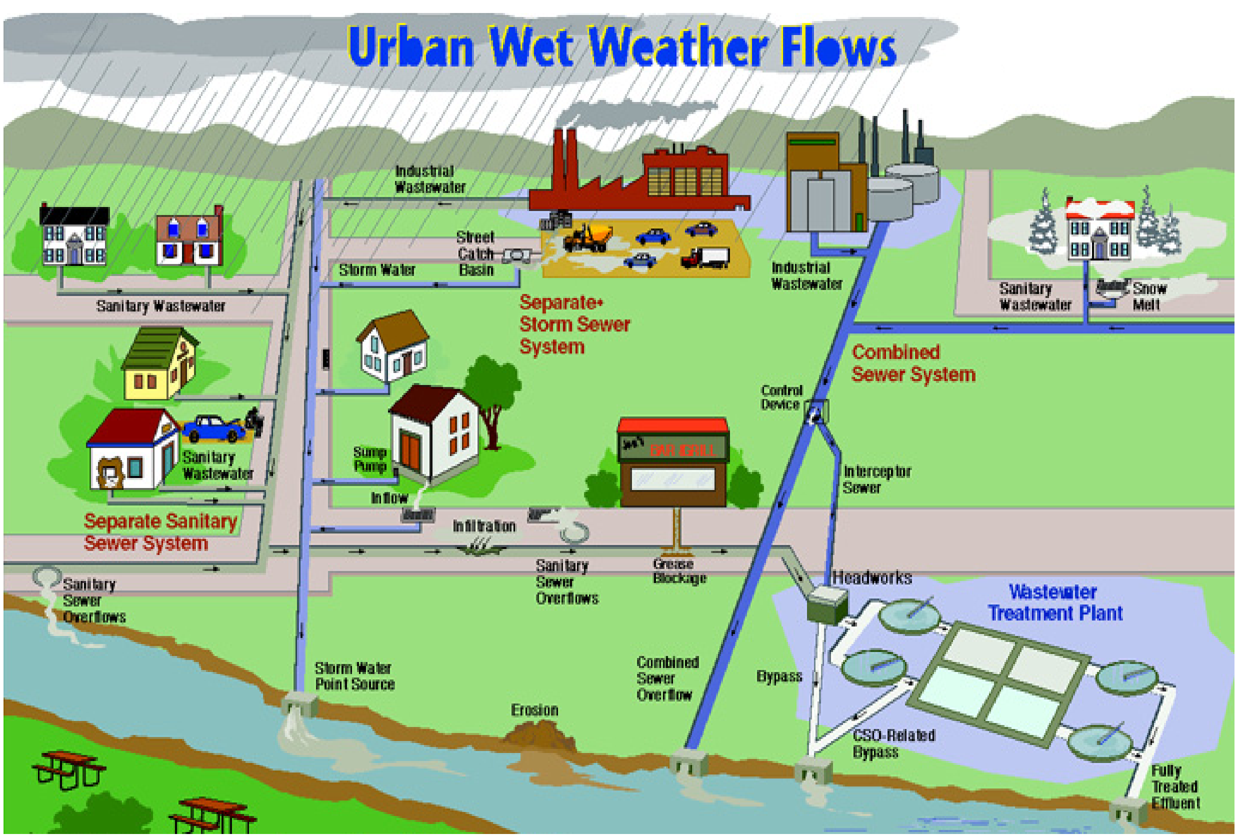
Introducing EPA SWMM

The EPA Storm Water Management Model (SWMM) is a dynamic rainfall-runoff simulation model used for single event or long-term (continuous) simulation of runoff quantity and quality from primarily urban areas. The runoff component of SWMM operates on a collection of subcatchment areas that receive precipitation and generate runoff and pollutant loads. The routing portion of SWMM transports this runoff through a system of pipes, channels, storage/treatment devices, pumps, and regulators. SWMM tracks the quantity and quality of runoff generated within each subcatchment, and the flow rate, flow depth, and quality of water in each pipe and channel during a simulation period comprised of multiple time steps.



SWMM was first developed in 1971 and has undergone several major upgrades since then. It continues to be widely used throughout the world for planning, analysis and design related to storm water runoff, combined sewers, sanitary sewers, and other drainage systems in urban areas, with many applications in non-urban areas as well. The current edition, Version 5, is a complete re-write of the previous release. Running under Windows, SWMM 5 provides an integrated environment for editing study area input data, running hydrologic, hydraulic and water quality simulations, and viewing the results in a variety of formats. These include color-coded drainage area and conveyance system maps, time series graphs and tables, profile plots, and statistical frequency analyses.

urban areas. These include:

- time-varying rainfall
- evaporation of standing surface water
- snow accumulation and melting
- rainfall interception from depression storage
- infiltration of rainfall into unsaturated soil layers
- percolation of infiltrated water into groundwater layers
- interflow between groundwater and the drainage system
- nonlinear reservoir routing of overland flow
- capture and retention of rainfall/runoff with various types of low impact development (LID) practices.

Spatial variability in all of these processes is achieved by dividing a study area into a collection of smaller, homogeneous subcatchment areas, each containing its own fraction of pervious and impervious sub-areas. Overland flow can be routed between sub-areas, between subcatchments, or between entry points of a drainage system.

Hydraulic Modeling Features

to route runoff and external inflows through a drainage system network of pipes, channels, storage/treatment units and diversion structures. These include the ability to:

- handle networks of unlimited size

natural channels

- model special elements such as storage/treatment units, curb and gutter inlets, flow dividers, pumps, weirs, and orifices
- apply external flows and water quality inputs from surface runoff, groundwater interflow, rainfall-dependent infiltration/inflow, dry weather sanitary flow, and user-defined inflows
- utilize either kinematic wave or full dynamic wave flow routing methods
- model various flow regimes, such as backwater, surcharging, reverse flow, and surface ponding
- apply user-defined dynamic control rules to simulate the operation of pumps, orifice openings, and weir crest levels.

Water Quality Modeling Features

SWMM can also estimate the production of pollutant loads associated with this runoff. The following processes can be modeled for any number of user-defined water quality constituents:

- dry-weather pollutant buildup over different land uses
- direct contribution of rainfall deposition
- reduction in dry-weather buildup due to street cleaning
- reduction in washoff load due to BMPs
- entry of dry weather sanitary flows and user-specified external inflows at any point in the drainage system
- routing of water quality constituents through the drainage system
- reduction in constituent concentration through treatment in storage units or by natural processes in pipes and channels.

Typical Applications of SWMM

stormwater studies throughout the world. Typical applications include:

water quality protection

- design and sizing of drainage system components for flood control
- flood plain mapping of natural channel systems
- designing control strategies for minimizing combined sewer overflows
- evaluating the impact of inflow and infiltration on sanitary sewer overflows
- generating non-point source pollutant loadings for waste load allocation studies
- evaluating the effectiveness of BMPs for reducing wet weather pollutant loadings.

Steps in Using SWMM

model a study area

area (see Adding Objects).

1. Specify a default set of options and object properties to use (see Setting Project Defaults).

3. Edit the properties of the objects that make up the system (see Editing Objects).

4. Select a set of analysis options (see Setting Analysis Options).

5. Run a simulation (see Initiating a Run).

6. View the results of the simulation (see Viewing Results).

For larger systems it will be more convenient to replace Step 2 by collecting study area data from various sources, such as CAD drawings or GIS files, and transferring these data into a SWMM input file whose format is described in the SWMM 5 User's Manual.

- Inconsistent reporting of Surface Runoff and Wet Weather Inflow pollutant mass in the Status Report.
- Missing limits on water flux rates between layers in several types of LID units.
- Incorrect geometry calculations for Street cross-sections with depressed gutters.
- Questionable behavior of conduit evaporation and seepage losses.
- Flickering of the Study Area Map when panning.
- Access Violation error when attempting to edit the vertices of a Storage Node polygon.
- Max/Full Depth values for orifices and weirs in the wrong column of the Summary Results Link Flow table.
- "Scrollbar property out of range" error for models with extremely long simulation periods.

Please consult the SWMM 5 Updates and Bug Fixes file for a complete listing of all program updates.

SWMM's Conceptual Model

flows between several major environmental compartments. These compartments and the SWMM objects they contain include:

- The Atmospheric compartment, which generates precipitation and deposits pollutants onto the land surface compartment. SWMM uses Rain Gage objects to represent rainfall inputs to the system.

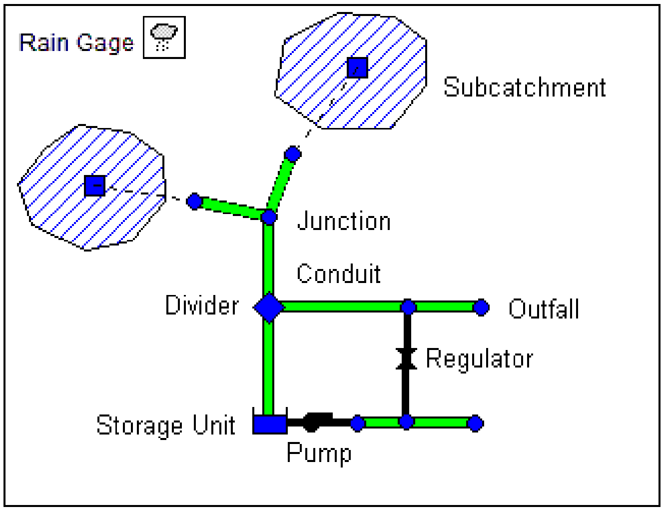
Subcatchment objects. It receives precipitation from the Atmospheric compartment in the form of rain or snow; it sends outflow in the form of infiltration to the Groundwater compartment and also as surface runoff and pollutant loadings to the Transport compartment.

- The Groundwater compartment receives infiltration from the Land Surface compartment and transfers a portion of this inflow to the Transport compartment. This compartment is modeled using Aquifer objects.
- The Transport compartment contains a network of conveyance elements (channels, pipes, pumps, and regulators) and storage/treatment units that transport water to outfalls or to treatment facilities. Inflows to this compartment can come from surface runoff, groundwater interflow, sanitary dry weather flow, or from user-defined hydrographs. The components of the Transport compartment are modeled with Node and Link objects.

Not all compartments need appear in a particular SWMM model. For example, one could model just the transport compartment, using pre-defined hydrographs as inputs.

Visual Objects

be arranged together to represent a stormwater drainage system. These objects can be displayed on a map in the SWMM workspace. Click on the name of any object to view its description.



a study region. The rainfall data can be either a user-defined time series or come from an external file. Several different popular rainfall file formats currently in use are supported, as well as a standard user-defined format.

The principal input properties of rain gages include:

- rainfall data type (e.g., intensity, volume, or cumulative volume)
- recording time interval (e.g., hourly, 15-minute, etc.)
- source of rainfall data (input time series or external file)
- name of rainfall data source

See Also

Rain Gage Properties
Rainfall Files

Subcatchments

system elements direct surface runoff to a single discharge point. The user is responsible for dividing a study area into an appropriate number of subcatchments, and for identifying the outlet point of each subcatchment. Discharge outlet points can be either nodes of the drainage system or other subcatchments.

Subcatchments are divided into pervious and impervious subareas. Surface runoff can infiltrate into the upper soil zone of the pervious subarea, but not through the impervious subarea. Impervious areas are themselves divided into two subareas - one that contains depression storage and another that does not. Runoff flow from one subarea in a subcatchment can be routed to the other subarea, or both subareas can drain to the subcatchment outlet.

Infiltration of rainfall from the pervious area of a subcatchment into the unsaturated upper soil zone can be described using five different models:

- Classic Horton infiltration
- Green-Ampt infiltration
- Modified Green-Ampt Infiltration
- SCS Curve Number infiltration

To model the accumulation, re-distribution, and melting of precipitation that falls as snow on a subcatchment, it must be assigned a Snow Pack object. To model groundwater flow between an aquifer underneath the subcatchment and a node of the drainage system, the subcatchment must be assigned a set of Groundwater parameters. Pollutant buildup and washoff from subcatchments are associated with the Land Uses assigned to the subcatchment. Capture and retention of rainfall/runoff using different types of low impact development practices (such as bio-retention cells, infiltration trenches, porous pavement, vegetative swales, and rain barrels) can be modeled by assigning a set of pre-designed LID controls to the subcatchment.

The other principal input parameters for subcatchments include:

- assigned rain gage
- outlet node or subcatchment
- total area
- percent impervious area
- average slope
- characteristic width of overland flow
- Manning's roughness (n) for overland flow on both pervious and impervious areas
- depression storage in both pervious and impervious areas
- percent of impervious area with no depression storage.

See Also

Subcatchment Properties
Infiltration
Land Uses
LID Controls
Aquifers
Snow Packs

together. There are several different categories of nodes that can be employed:

- Junctions
- Outfalls
- Flow Dividers
- Storage Units

Nodes are also the points where external inflows can enter a drainage system and where removal of pollutants through treatment can occur.

Junctions

they can represent the confluence of natural surface channels, manholes in a sewer system, or pipe connection fittings. External inflows can enter the system at junctions. Excess water at a junction can become partially pressurized while connecting conduits are surcharged and can either be lost from the system or be allowed to pond atop the junction and subsequently drain back into the junction.

The principal input parameters for a junction are:

· invert (channel or manhole bottom) elevation

· ponded surface area when flooded (optional)

· external inflow data (optional).

See Also Junction Properties

Outfalls

downstream boundaries under Dynamic Wave flow routing. For other types of flow routing they behave as a junction. Only a single link can be connected to an outfall node, and the option exists to have the outfall discharge onto a subcatchment's surface.

The boundary conditions at an outfall can be described by any one of the following stage relationships:

- the critical or normal flow depth in the connecting conduit
- a tidal stage described in a table of tide height versus hour of the day
- a user-defined time series of stage versus time.

The principal input parameters for outfalls include:

- invert elevation
- boundary condition type and stage description
- presence of a flap gate to prevent backflow through the outfall.

See Also

Outfall Properties

Flow Dividers

conduit in a prescribed manner. A flow divider can have no more than two conduit links on its discharge side. Flow dividers are only active under Steady Flow and Kinematic Wave routing and are treated as simple junctions under Dynamic Wave routing.

There are four types of flow dividers, defined by the manner in which inflows are diverted:

- **Cutoff** (diverts all inflow above a defined cutoff value)

conduit)

- **Tabular** (uses a table that expresses diverted flow as a function of total inflow)
- **Weir** (treats diverted flow as linearly proportional to the inflow above a defined cutoff value).

The principal input parameters for a flow divider are:

- junction parameters (see Junctions)
- name of the link receiving the diverted flow
- method used for computing the amount of diverted flow.

See Also Divider Properties

Storage Units

Physically they could represent storage facilities as small as a catch basin or as large as a lake. The volumetric properties of a storage unit are described by a function or table of surface area versus height. In addition to receiving inflows and discharging outflows to other nodes in the drainage network, storage nodes can also lose water from surface evaporation and from seepage into native soil.

The principal input parameters for storage units include:

- invert (bottom) elevation
- depth-surface area data
- evaporation potential
- seepage parameters (optional)
- external inflow data (optional).

See Also Storage Properties

between a pair of nodes. Types of links include:

- Conduits
- Pumps
- Regulators

Conduits

in the conveyance system. Their cross-sectional shapes can be selected from a variety of standard open and closed geometries. Irregular natural cross-section shapes are also supported, as are user-defined closed shapes.

The principal input parameters for conduits are:

- names of the inlet and outlet nodes
- offset height or elevation of the conduit above the inlet and outlet node inverts
- conduit length
- Manning's roughness (n)
- cross-sectional geometry
- entrance/exit losses (optional)
- seepage rate (optional)
- presence of a flap gate to prevent reverse flow (optional)
- culvert type code number if the conduit acts as a culvert (optional)
- name of any inlet structure placed in a street or channel conduit (optional)

Conduits designated as culverts are checked continuously during dynamic wave flow routing to see if they operate under Inlet Control as defined in the Federal Highway Administration's publication *Hydraulic Design of Highway Culverts* (Publication No. FHWA-NHI-01-020, May 2005).

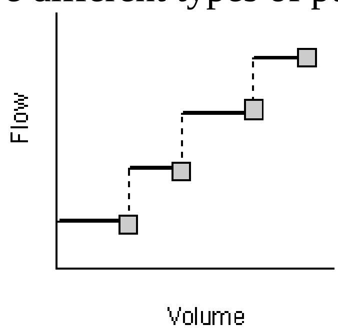
Street and channel conduits with inlet structures use the methods described in the Federal Highway Administration's publication *Urban Drainage Design Manual - HEC-22 (*Publication No. FHWA-NHI-10-009, August 2013) to determine the amount of flow they capture.

See Also Conduit Properties
Cross-Section Editor
Culvert Code Numbers
Inlets

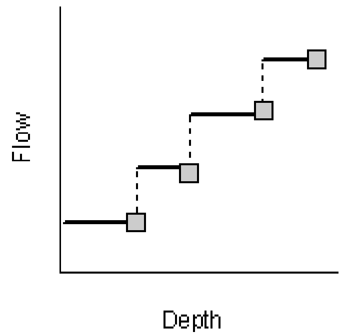
Pumps

describes the relation between a pump's flow rate and conditions at its inlet and outlet nodes. Five different types of pumps are supported:

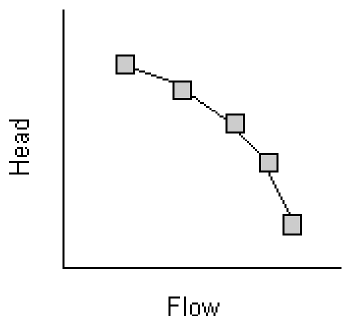
Type1
An off-line pump with a wet well where flow increases incrementally with available wet well volume.



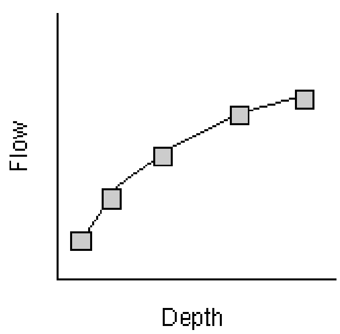
Type2
An in-line pump where flow increases incrementally with inlet node depth.



Type3
An in-line pump where flow varies continuously with head difference between the inlet and outlet nodes.



Type4
A variable speed in-line pump where flow varies continuously with inlet node depth.



Type5

Flow Regulators

within a conveyance system. They are typically used to:

- control releases from storage facilities
- prevent unacceptable surcharging
- divert flow to treatment facilities and interceptors.

SWMM can model the following types of flow regulators:

- Orifices
- Weirs
- Outlets

Orifices

systems which are typically openings in the wall of a manhole, storage facility, or control gate. They are internally represented in SWMM as a link connecting two nodes. An orifice can have either a circular or rectangular shape, be located either at the bottom or along the side of the upstream node, and have a flap gate to prevent backflow.

Orifices can be used as storage unit outlets under all types of flow routing. If not attached to a storage unit node, they can only be used in drainage networks that are analyzed with Dynamic Wave flow routing.

The flow through an orifice is computed based on the area of its opening, its discharge coefficient, and the head difference across the orifice.

The height of an orifice's opening can be controlled dynamically through user-defined Control Rules. This feature can be used to model gate openings and closings.

The principal input parameters for an orifice include:

- names of its inlet and outlet nodes
- configuration (bottom or side)
- shape (circular or rectangular)
- height above the inlet node invert
- discharge coefficient
- time to open or close (optional).

See Also

Orifice Properties
Control Rules

Weirs

drainage system. Weirs are typically located in a manhole, along the side of a channel, or within a storage unit. They are internally represented in SWMM as a link connecting two nodes, where the weir itself is placed at the upstream node. A flap gate can be included to prevent backflow.

Five varieties of weirs are available, each incorporating a different formula for computing flow as a function of area, discharge coefficient and head difference across the weir:

- **Transverse** (rectangular shape)
- **V-notch** (triangular shape)
- **Trapezoidal** (trapezoidal shape)
- **Roadway** (broad crested rectangular weir used to model roadway crossings).

Weirs can be used as storage unit outlets under all types of flow routing. If not attached to a storage unit, they can only be used in drainage networks that are analyzed with Dynamic Wave flow routing.

The height of the weir crest above the inlet node invert can be controlled dynamically through user-defined Control Rules. This feature can be used to model inflatable dams.

Weirs can either be allowed to surcharge or not. A surcharged weir will use an equivalent orifice equation to compute the flow through it. Weirs placed in open channels would normally not be allowed to surcharge while those placed in closed diversion structures or those used to represent storm drain inlet openings would be allowed to.

The principal input parameters for a weir include:

- names of its inlet and outlet nodes
- shape and geometry
- crest height above the inlet node invert
- discharge coefficient.

See Also

Weir Properties

Control Rules

from storage units. They are used to model special head-discharge relationships that cannot be characterized by pumps, orifices, or weirs. Outlets are internally represented in SWMM as a link connecting two nodes. An outlet can also have a flap gate that restricts flow to only one direction.

Outlets attached to storage units are active under all types of flow routing. If not attached to a storage unit, they can only be used in drainage networks analyzed with Dynamic Wave flow routing.

A user-defined rating curve determines an outlet's discharge flow as a function of either the freeboard depth above the outlet's opening or the head difference across it. Control Rules can be used to dynamically adjust this flow when certain conditions exist.

The principal input parameters for an outlet include:

- names of its inlet and outlet nodes
- height above the inlet node invert
- function or table containing its head (or depth) - discharge relationship.

See Also Outlet Properties
Control Rules

Map Labels

help identify particular objects or regions of the map. The labels can be drawn in any Windows font, freely edited and be dragged to any position on the map.

See Also

Map Label Properties

Non-visual Objects

SWMM utilizes several classes of non-visual data objects to describe additional characteristics and processes within a study area.

- Climatology Data
- Hydrology Data
- Hydraulic Data
- Water Quality Data
- Tabular Data

related variables used for computing runoff and snowmelt:

- Temperature
- Evaporation
- Wind Speed
- Snowmelt
- Areal Depletion
- Climate Adjustments

Temperature

processes during runoff calculations. They are also needed if the option to base evaporation rates on temperature is selected. If these processes are not being simulated then temperature data are not required. Air temperature data can be supplied to SWMM from one of the following sources:

- a user-defined time series of point values (values at intermediate times are interpolated)

(SWMM fits a sinusoidal curve through these values depending on the day of the year).

For user-defined time series, temperatures are in degrees F for US units and degrees C for metric units. The external climate file can also be used to supply evaporation and wind speed as well.

See Also

Climate Files

Climatology Editor

subsurface water in groundwater aquifers, for water traveling through open channels, and for water held in storage units. Evaporation rates can be stated as:

- a user-defined time series of values
- values computed from the daily temperatures contained in an external climate file
- a single constant value
- daily values read from an external climate file.

These values represent potential rates. The actual amount of water evaporated will depend on the amount available.

If rates are read directly from a climate file, then a set of monthly pan coefficients should also be supplied to convert the pan evaporation data to free water-surface values. An option is also available to allow evaporation only during periods with no precipitation.

See Also

Climate Files

Climatology Editor

Wind Speed

calculations. SWMM can use either a set of monthly average speeds or wind speed data contained in the same climate file used for daily minimum/maximum temperatures.

See Also

Climatology Editor

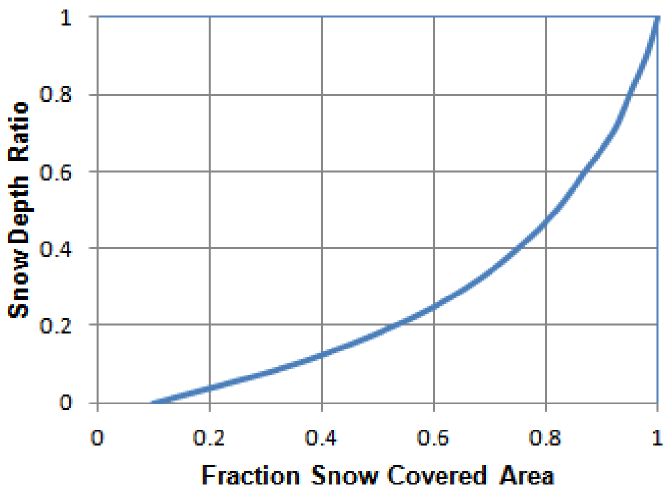
study area when simulating snowfall and snowmelt. They include:

- the air temperature at which precipitation falls as snow
- heat exchange properties of the snow surface
- study area elevation, latitude, and longitude correction.

See Also

Climatology Editor

uniformly over the surface of a subcatchment. As the melting process proceeds, the area covered by snow gets reduced. This behavior is described by an Areal Depletion Curve that plots the fraction of total area that remains snow covered against the ratio of the actual snow depth to the depth at which there is 100% snow cover. A typical ADC for a natural area is shown below.



Two such curves can be supplied to SWMM, one for impervious areas and another for pervious areas.

See Also

Climatology Editor

Climate Adjustments

Climate adjustments are optional modifications applied to the temperature, evaporation rate, and rainfall intensity that SWMM would otherwise use at each time step of a simulation. Separate sets of adjustments that vary periodically by month of the year can be assigned to these variables. They provide a simple way to examine the effects of future climate change without having to modify the original climatic time series.

In a similar manner, a set of monthly adjustments can be applied to the hydraulic conductivity used in computing rainfall infiltration on all pervious land surfaces, including those in all LID units, and exfiltration from all storage nodes and conduits. These can reflect the increase of hydraulic conductivity with increasing temperature or the effect that seasonal changes in land surface conditions, such as frozen ground, can have on infiltration capacity. They can be overridden for individual subcatchments (and their LID units) by assigning a monthly infiltration adjustment Time Pattern to a subcatchment. Monthly adjustment time patterns for depression storage and pervious surface roughness coefficient (Mannings n) can also be specified for individual subcatchments (see Subcatchment Properties).

objects are used by SWMM:

- Aquifers
- Snow Packs
- Unit Hydrographs
- LID Controls

Snow Packs

a subcatchment.

removal, and melting of snow over three types of sub-areas within a subcatchment:

- The **Pervious** snow pack area encompasses the entire pervious area of a subcatchment.
- The **Plowable** snow pack area consists of a user-defined fraction of the total impervious area. It is meant to represent such areas as streets and parking lots where plowing and snow removal can be done.

Each of these three areas is characterized by the following parameters:

- minimum and maximum snow melt coefficients
- minimum air temperature for snow melt to occur
- snow depth above which 100% areal coverage occurs
- initial snow depth
- initial and maximum free water content in the pack.

In addition, a set of snow removal parameters can be assigned to the Plowable area. These parameters consist of the depth at which snow removal begins and the fractions of snow moved onto various other areas.

Subcatchments are assigned a snow pack object through their Snow Pack property. A single snow pack object can be applied to any number of subcatchments. Assigning a snow pack to a subcatchment simply establishes the melt parameters and initial snow conditions for that subcatchment. Internally, SWMM creates a "physical" snow pack for each subcatchment, which tracks snow accumulation and melting for that particular subcatchment based on its snow pack parameters, its amount of pervious and impervious area, and the precipitation history it sees.

See Also

Snow Pack Editor

movement of water infiltrating from the subcatchments that lie above them. They also permit the infiltration of groundwater into the drainage system, or exfiltration of surface water from the drainage system, depending on the hydraulic gradient that exists. Aquifers are only required in models that need to explicitly account for the exchange of groundwater with the drainage system or to establish baseflow and recession curves in natural channels and non-urban systems.

The parameters of an aquifer object can be shared by several subcatchments but there is no exchange of groundwater between subcatchments. A drainage system node can exchange groundwater with more than one subcatchment.

Aquifers are represented using two zones - an un-saturated zone and a saturated zone. Their behavior is characterized using such parameters as soil porosity, hydraulic conductivity, evapotranspiration depth, bottom elevation, and loss rate to deep groundwater. In addition, the initial water table elevation and initial moisture content of the unsaturated zone must be supplied.

Aquifers are connected to subcatchments and to drainage system nodes as defined in a subcatchment's Groundwater Flow property. This property also contains parameters that govern the rate of groundwater flow between the aquifer's saturated zone and the drainage system node.

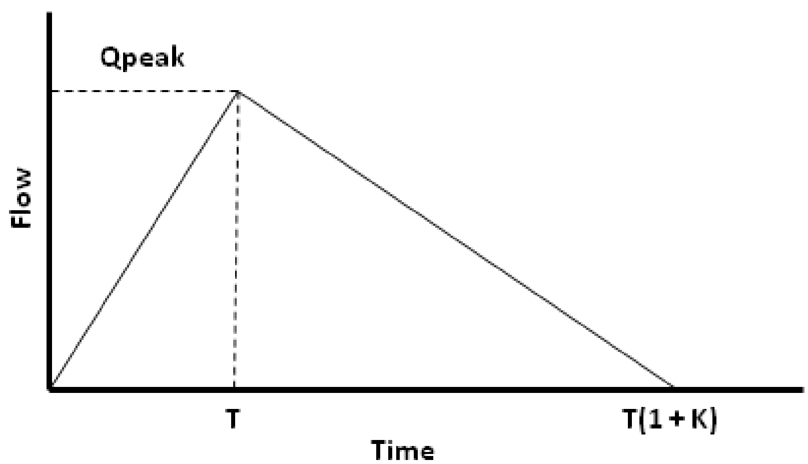
See Also

Aquifer Editor
Groundwater Flow Editor

(RDII) into a sewer system. A UH set contains up to three such hydrographs, one for a short-term response, one for an intermediate-term response, and one for a long-term response. A UH group can have up to 12 UH sets, one for each month of the year. Each UH group is considered as a separate object by SWMM, and is assigned its own unique name along with the name of the rain gage that supplies rainfall data to it.

Each unit hydrograph is defined by three parameters:

- **R**: the fraction of rainfall volume that enters the sewer system
- **T**: the time from the onset of rainfall to the peak of the UH in hours
- **K**: the ratio of time to recession of the UH to the time to peak



A unit hydrograph can also have a set of Initial Abstraction (IA) parameters associated with it. These determine how much rainfall is lost to interception and depression storage before any excess rainfall is generated and transformed into RDII flow by the hydrograph.

To generate RDII into a drainage system node, the node must identify (through its Inflows property) the UH group and the area of the surrounding sewershed that contributes RDII flow.

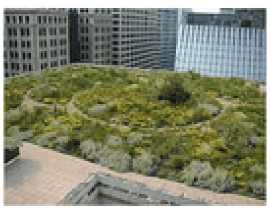
An alternative to using unit hydrographs to define RDII flow is to create an external RDII interface file, which contains RDII time series data.

LID Controls

surface runoff and provide some combination of detention, infiltration, and evapotranspiration to it. They are considered as properties of a given subcatchment, similar to how Aquifers and Snow Packs are treated. SWMM can explicitly model the following generic types of LID controls:

Bio-retention Cells are depressions that contain vegetation grown in an engineered soil mixture placed above a gravel drainage bed. They provide storage, infiltration and evaporation of both direct rainfall and runoff captured from surrounding areas.

Rain Gardens are a type of bio-retention cell consisting of just the engineered soil layer with no gravel bed below it.



Green Roofs are another variation of a bio-retention cell that have a soil layer laying atop a special drainage mat material that conveys excess percolated rainfall off of the roof.

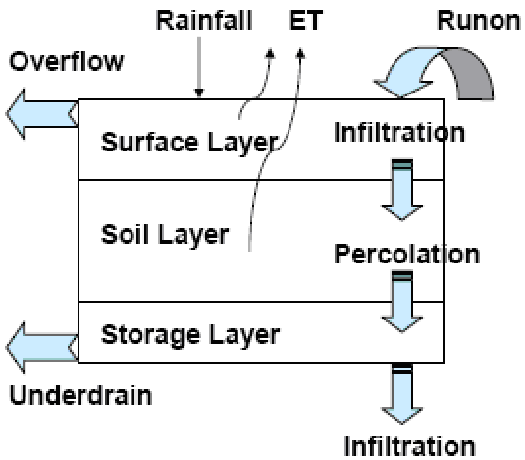
Infiltration Trenches are narrow ditches filled with gravel that intercept runoff from upslope impervious areas. They provide storage volume and additional time for captured runoff to infiltrate the native soil below.

Continuous Permeable Pavement systems are excavated areas filled with gravel and paved over with a porous concrete or asphalt mix. ***Block Paver*** systems consist of impervious paver blocks placed on a sand or pea gravel bed with a gravel storage layer below

Rain Barrels (or ***Cisterns***) are containers that collect roof runoff during storm events and can either release or re-use the rainwater during dry periods.

Rooftop Disconnection has downspouts discharge to properties are defined on a per-unit-area basis. This allows LIDs of the same design but differing areal coverage to easily be placed within different subcatchments in a study area.

During a simulation SWMM performs a moisture balance that keeps track of how much water moves between and is stored within each LID layer. As an example, the layers used to model a bio-retention cell and the flow pathways between them are shown below:



The following table indicates which combination of layers applies to each type of LID (**x** means required, **o** means optional):

LID Type	**Surface**	**Pavement**	**Soil**	**Storage**	**Drain**	**Drainage Mat**
Bio-Retention Cell	x		x	x	o	
Rain Garden	x		x			
Green Roof	x		x			x
Infiltration Trench	x			x	o	
Permeable Pavement	x	x	o	x	o	
Rain Barrel				x	x	
Rooftop Disconnection	x				x	

Vegetative	x					

LID Utilization

subcatchments.

1. creates a set of scale-independent LID controls that can be deployed throughout the study area,

Bear in mind that when LIDs are added to a subcatchment, the subcatchment's *Area* property is the total area of the subcatchment (both non-LID and LID portions) while the *Percent Imperviousness* and *Width* parameters apply only to the non-LID portion of the subcatchment.

To implement the first phase, one selects the **Hydrology | LID Controls** category from the Project Browser to add, edit or delete individual LID control objects. The LID Control Editor is used to edit the properties of the various component layers that comprise each LID control object.

For the second phase, for each subcatchment that will utilize LIDs, one selects the **LID Controls** property in the subcatchment's Property Editor to launch the LID Group Editor. This editor is used to add or delete individual LID controls from the subcatchment. For each control added the LID Usage Editor is used to specify the size of the control and what fraction of the subcatchment's impervious and pervious areas it captures.

subcatchment:

1. place one or more controls in an existing subcatchment that will displace an equal amount of non-LID area from the subcatchment
2. create a new subcatchment devoted entirely to just a single LID practice.

The first approach allows a mix of LIDs to be placed into a subcatchment, each treating a different portion of the runoff generated from the non-LID fraction of the subcatchment. Note that under this option the subcatchment's LIDs act in parallel -- it is not possible to make them act in series (i.e., have the outflow from one LID control become the inflow to another LID). Also, after LID placement the subcatchment's **Percent Impervious** and **Width** properties may require adjustment to compensate for the amount of original subcatchment area that has now been replaced by LIDs (see the figure below). For example, suppose that a subcatchment which is 40% impervious has 75% of that area converted to permeable pavement. After the LID is added the subcatchment's percent imperviousness should be changed to the percent of impervious area remaining divided by the percent of non-LID area remaining. This works out to (1 - 0.75)*40 / (100 - 0.75*40) or 14.3 %.

LID Results

The performance of the LID controls placed in a subcatchment is reflected in the overall runoff, infiltration, and evaporation rates computed for the subcatchment as normally reported by SWMM. SWMM's Summary Report also contains a section entitled **LID Performance Summary** that provides an overall water balance for each LID control placed in each subcatchment. The components of this water balance include total inflow, infiltration, evaporation, surface runoff, drain flow and initial and final stored volumes, all expressed as inches (or mm) over the LID's area.

Optionally, the entire time series of flux rates and moisture levels for a selected LID control in a given subcatchment can be written to a tab delimited text file for easy viewing and graphing in a spreadsheet program.

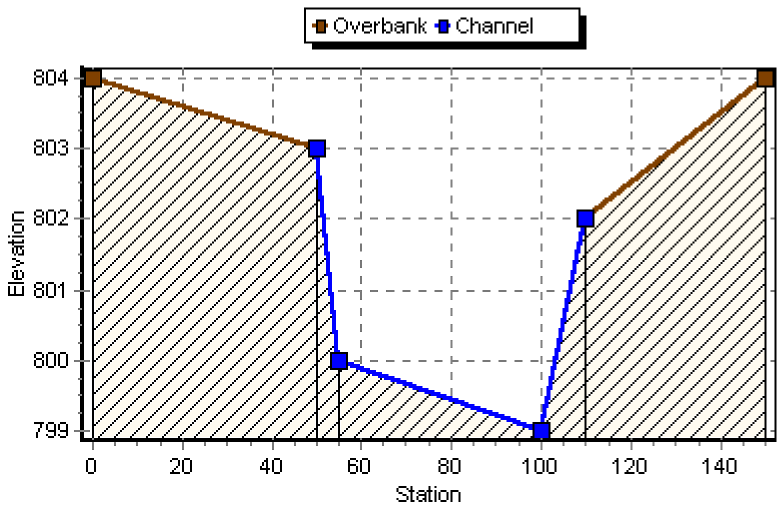
Hydraulics

a drainage system in a SWMM model, the following data objects can be used to augment the hydraulic description of the system:

- Transects
- Streets
- Inlets
- Inflows
- Controls

Transects

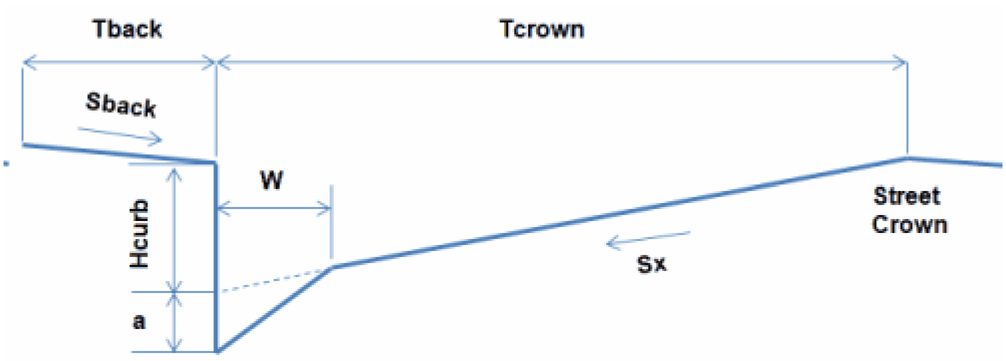
Transects refer to the geometric data that describe how bottom elevation varies with horizontal distance over the cross-section of a natural channel or irregular-shaped conduit. The figure below displays an example of a transect for a natural channel.



Each transect must be given a unique name. Conduits refer to that name to represent their shape. A special Transect Editor is available for editing the station-elevation data of a transect. SWMM internally converts these data into tables of area, top width, and hydraulic radius versus channel depth. In addition, as shown in the diagram above, each transect can have a left and right overbank section whose Manning's roughness coefficient can be different from that of the main channel. This feature can provide more realistic estimates of channel conveyance under high flow conditions.

Streets

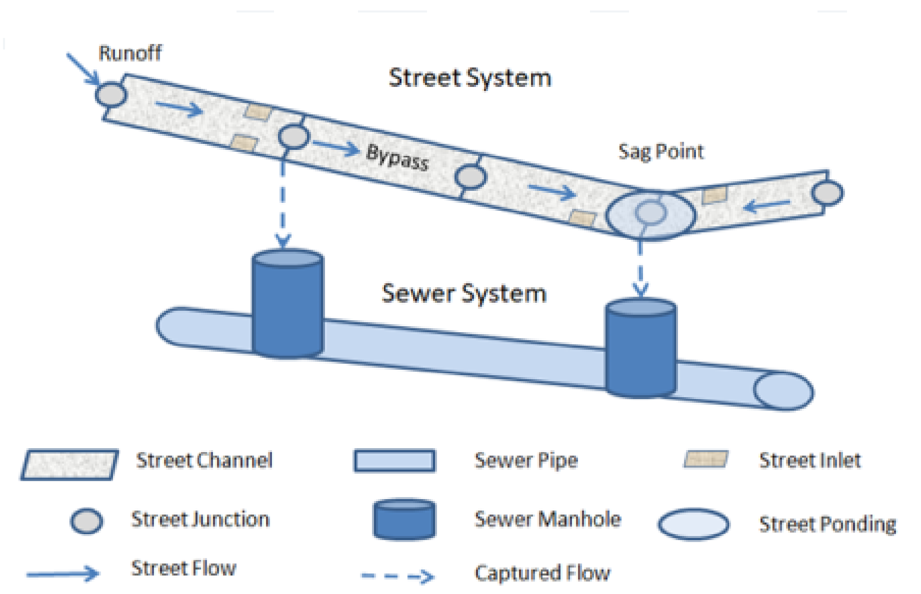
Streets are a specialized form of transect that describes the typical cross-section geometry of a street or roadway. The figure below shows a half-street layout along with the dimensions a user needs to provide.



Each street section object is assigned an ID name that a conduit can refer to for describing its cross section geometry. A Street Section Editor is available for providing a street section's dimensions and whether it is one- or two-sided.

into below-ground sewers. **Drop inlets** serve a similar purpose for trapezoidal channels. SWMM can compute the amount of flow captured by inlets and sent to designated sewer nodes using the FHWA HEC-22 methodology. The type, sizing, and spacing of street inlets will determine if the spread and depth of water on roadways can be maintained at acceptable levels.

To analyze street drainage with SWMM a site is represented as a dual drainage system consisting of both street conduits along the ground surface and sewer conduits below it. An inlet structure will divert some portion of the street flow it sees into a designated node of the sewer system with the rest being bypassed to downstream streets. When an inlets sewer node reaches its full depth any excess flow that would cause it to flood is sent back through the inlet and onto the street.



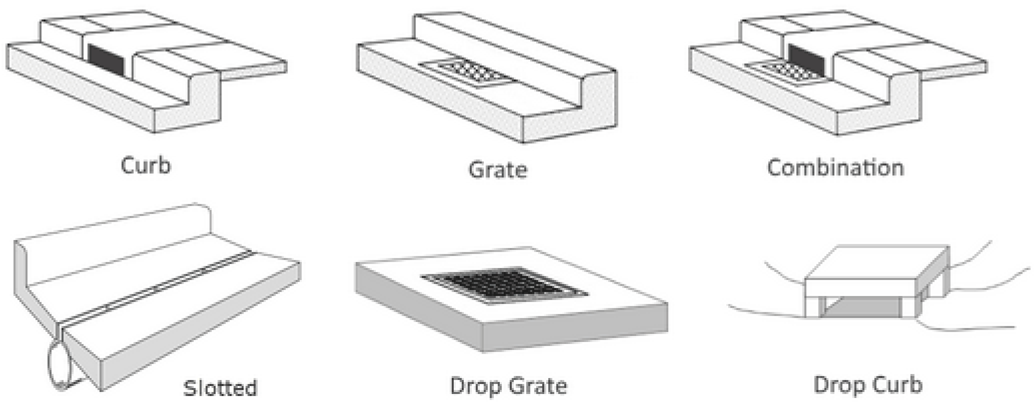
Inlet Types

Inlet Usage

Inlet Features

Inlet Types

SWMMs HEC-22 inlet capture equations support the inlet types shown below:



Drop inlets can only be used with rectangular or trapezoidal channels while the other curb and gutter inlets can only be placed in conduits with Street cross-sections. An additional **Custom** type of inlet can be used in both streets and channels. Its capture efficiency is described by either a user-supplied Diversion curve (captured flow v. approach flow) or Rating curve (captured flow v. flow depth).

Inlet Usage

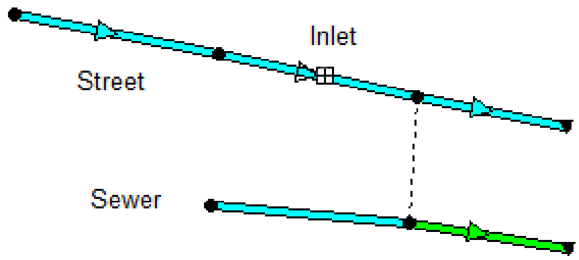
- Create one network layout for streets and another for sewers.
- For each street conduit, set its Shape property to one of the available street sections.
- Create a set of inlet structure design objects.
- Place a particular inlet structure design into a selected street conduit, assigning it a sewer node that receives its captured flow.
- Assign surface runoff from subcatchments or other external inflows to street conduit nodes.

A similar set of steps would be used to add drop inlets into rectangular or trapezoidal channels.

A summary of results for each street conduit (maximum flow depth and pavement spread) and for each inlet (percent capture at peak flow, frequency of bypass flow and frequency of sewer system backflow) will appear as a separate **Street Flow** table in SWMM's Summary Results report.

Inlet Features

- Conduits with inlets will be displayed on the Study Area Map with a ⊞ symbol near their midpoint and show their downstream node connected to the inlet's capture node with a dotted line when the Map Option to display link symbols is turned on.



- The rim elevations of nodes that receive captured inlet flow do not have to match the invert elevations of the end node of the conduit containing the inlet.
- Two-sided street conduits (that are symmetric about the street crown) use pairs of inlets placed on each curb side of the street.
- Multiple inlets of the same design can be assigned to a conduit (as pairs for two-sided streets). For on-grade placement the flow captured by each inlet is determined sequentially, so that the approach flow to the next inlet in line is the bypass flow from the inlet before it.
- Flow captured by inlets is limited by the amount that its sewer node can receive before it floods. If the node has no such capacity remaining then any excess flow that would cause it to flood is sent back through the inlet and onto the street.
- Users can stipulate whether an inlet operates on-grade or on-sag or have SWMM decide based on the slopes of the conduits adjoining it. (On-sag refers to a sump or low point that all adjoining conduits slope towards.)
- Inlets can have a degree of clogging and a flow capture restriction assigned to them.
- For Kinematic Wave and Steady Flow routing it is recommended that storage nodes be used at the end of inlet conduits that converge at sag

points since otherwise any non-captured flow will simply exit the groundwater, drainage system nodes can receive three other types of external inflows:

Direct Inflows
These are user-defined time series of inflows added directly into a node. They can be used to perform flow and water quality routing in the absence of any runoff computations (as in a study area where no subcatchments are defined).

Dry Weather Inflows
These are continuous inflows that typically reflect the contribution from sanitary sewage in sewer systems or base flows in pipes and stream channels. They are represented by an average inflow rate that can be periodically adjusted on a monthly, daily, and hourly basis by applying Time Pattern multipliers to this average value.

Rainfall-Dependent Inflow/Infiltration (RDII) These are stormwater flows that enter sanitary or combined sewers due to "inflow" from direct connections of downspouts, sump pumps, foundation drains, etc. as well as "infiltration" of subsurface water through cracked pipes, leaky joints, poor manhole connections, etc. RDII can be computed for a given rainfall record based on set of triangular unit hydrographs (UH) that determine a short-term, intermediate-term, and long-term inflow response for each time period of rainfall. Any number of UH sets can be supplied for different sewershed areas and different months of the year. RDII flows can also be specified in an external RDII Interface file.

Direct, Dry Weather, and RDII inflows are properties associated with each type of drainage system node (junctions, outfalls, flow dividers, and storage units) and can be specified when nodes are edited. They can be used to perform flow and water quality routing in the absence of any runoff computations (as in a study area where no subcatchments are defined). It is also possible to make the outflows generated from an upstream drainage system be the inflows to a downstream system by using interface files.

See Also

External Inflows Editor

Control Rules

system will be adjusted over the course of a simulation. The use of control rules is explained in the following topics:

- Example Rules
- Rule Format
- Condition Clauses
- Action Clauses
- Modulated Controls
- Named Variables
- Arithmetic Expressions

Example Rules

; Simple time-based pump control

```
RULE R1
IF SIMULATION TIME > 8
THEN PUMP 12 STATUS = ON
ELSE PUMP 12 STATUS = OFF
```

; Multi-condition orifice gate control

```
RULE R2A IF NODE 23 DEPTH > 12
AND LINK 165 FLOW > 100
THEN ORIFICE R55 SETTING = 0.5

RULE R2B
IF NODE 23 DEPTH > 12
AND LINK 165 FLOW > 200
THEN ORIFICE R55 SETTING = 1.0

RULE R2C
IF NODE 23 DEPTH <= 12
OR LINK 165 FLOW <= 100
THEN ORIFICE R55 SETTING = 0
```

; Pump station operation

```
RULE R3A IF NODE N1 DEPTH > 5
THEN PUMP N1A STATUS = ON

RULE R3B
IF NODE N1 DEPTH > 7
THEN PUMP N1B STATUS = ON

RULE R3C
IF NODE N1 DEPTH <= 3
THEN PUMP N1A STATUS = OFF
AND PUMP N1B STATUS = OFF
```

; Modulated weir height control

RULE R4

Rule Format

```
RULE ruleID

IF condition_1
AND condition_2
OR condition_3
AND condition_4
Etc.

THEN action_1
AND action_2
Etc.

ELSE action_3
AND action_4
Etc.

PRIORITY value
```

where keywords are shown in boldface and **`ruleID`** is an ID label assigned to the rule, **`condition_n`** is a Condition Clause, **`action_n`** is an Action Clause, and **`value`** is a priority value (e.g., a number from 1 to 5).

Each rule clause must begin with one of the boldface keywords shown above, and only one clause per line is allowed.

Only the **`RULE, IF`** and **`THEN`** portions of a rule are required; the **`ELSE`** and **`PRIORITY`** portions are optional.

Blank lines between clauses are permitted and any text to the right of a semicolon is considered a comment.

When mixing AND and OR clauses, the OR operator has higher precedence than AND, i.e.,

```
IF A or B and C
```

is equivalent to

Condition Clauses

```
object id attribute relation value
object id attribute relation object id attribute
```

where

`object`	=	a category of object
`id`	=	the object's ID label
`attribute`	=	an attribute or property of the object
`relation`	=	a relational operator (=, <>, <, <=, >, >=)
`value`	=	an attribute value

Some examples of condition clauses are:

```
GAGE G1 6-HR_DEPTH > 0.5
NODE N23 DEPTH > 10
NODE N23 DEPTH > NODE 25 DEPTH
PUMP P45 STATUS = OFF
LINK P45 TIMEOPEN >= 6:30
SIMULATION CLOCKTIME = 22:45:00
```

The objects and attributes that can appear in a condition clause are as follows:

Object	**Attributes**	**Value**
GAGE	INTENSITY n-HR_DEPTH	numerical value numerical value
NODE	DEPTH MAXDEPTH HEAD VOLUME INFLOW	numerical value numerical value numerical value numerical value numerical value

Action Clauses

PUMP id **STATUS** = **ON/OFF**

CONDUIT id **STATUS** = **OPEN/CLOSED**

PUMP/ORIFICE/WEIR/OUTLET id **SETTING** = value

The meaning of **SETTING** depends on the object being controlled:

- for Pumps it is a multiplier applied to the flow computed from the pump curve (or relative pump speed for a TYPE5 pump).

control is accomplished by lowering or raising a horizontal gate from the top of the orifice),

- for Weirs it is the fractional amount of the original freeboard that exists (i.e., weir control is accomplished by moving the crest height up or down),
- for Outlets it is a multiplier applied to the flow computed from the outlet's rating curve.

Some examples of action clauses are:

```
PUMP P67 STATUS = OFF
ORIFICE O212 SETTING = 0.5
```

of control applied to a pump or flow regulator as determined by the value of some controller variable, such as water depth at a node, or by time. The functional relation between the control setting and the controller variable can be specificed by using a Control Curve, a Time Series, or a PID controller. Some examples of modulated control rules are:

```
RULE MC1
IF NODE N2 DEPTH >= 0
THEN WEIR W25 SETTING = CURVE C25

RULE MC2
IF SIMULATION TIME > 0
THEN PUMP P12 SETTING = TIMESERIES TS101

RULE MC3
IF LINK L33 FLOW <> 1.6
THEN ORIFICE O12 SETTING = PID 0.1 0.0 0.0
```

Note how a modified form of the action clause is used to specify the name of the control curve, time series or PID parameter set that defines the degree of control. A PID parameter set contains three values -- a proportional gain coefficient, an integral time (in minutes), and a derivative time (in minutes). Also, by convention the controller variable used in a Control Curve or PID Controller will always be the object and attribute named in the last condition clause of the rule. As an example, in rule `MC1` above `Curve C25` would define how the fractional setting at `Weir W25` varied with the water depth at `Node N2`. In rule `MC3`, the PID controller adjusts the opening height of `Orifice O12` to maintain a flow of 1.6 in `Link L33`.

closed-loop control scheme that tries to maintain a desired set-point on some process variable by computing and applying a corrective action that adjusts the process accordingly. In the context of a hydraulic conveyance system a PID controller might be used to adjust the opening on a gated orifice to maintain a target flow rate in a specific conduit or to adjust a variable speed pump to maintain a desired depth in a storage unit. The classical PID controller has the form:

$$m(t) = Kp\left[e(t) + \frac{1}{Ti}\int e(\tau)d\tau + Td\frac{de(t)}{dt}\right]$$

where:

m(t)	=	controller output
Kp	=	proportional coefficient (gain)
Ti	=	integral time (minutes)
Td	=	derivative time (minutes)
e(t)	=	error (difference between setpoint and observed variable value)
t	=	time.

The performance of a PID controller is determined by the values assigned to the coefficients *Kp*, *Ti*, and *Td*.

The controller output *m(t)* has the same meaning as a link setting used in a rule's Action Clause while *dt* is the current flow routing time step in minutes. Because link settings are relative values (with respect to either a pump's standard operating curve or to the full opening height of an orifice or weir) the error *e(t)* used by the controller is also a relative value. It is defined as the difference between the control variable setpoint *x** and its value at time *t*, *x(t)*, normalized to the setpoint value:

$e(t) = (x^* - x(t)) / x^*$

Note that for direct action control, where an increase in the link setting causes an increase in the controlled variable, the sign of *Kp* must be positive. For reverse action control, where the controlled variable decreases as the link setting increases, the sign of *Kp* must be negative. The user must recognize whether the control is direct or reverse action and use the proper sign on *Kp* accordingly. For example, adjusting an orifice opening to maintain a desired downstream flow or downstream water level

Named Variables

object name | object attribute> (or a doublet for Simulation times) that appear in the condition clauses of control rules. They allow condition clauses to be written as:

`variable relation value` `variable relation variable`

where `variable` is defined on a separate line before its first use in a rule using the format:

`VARIABLE` `name = object id attribute`

Here is an example of using this feature:

```
VARIABLE Dabc = NODE abc DEPTH
VARIABLE Defg = NODE efg DEPTH
VARIABLE P45 = PUMP 45 STATUS

RULE 1
IF Dabc > Defg
AND P45 = OFF
THEN PUMP 45 STATUS = ON

RULE 2
IF Dabc < 1
THEN PUMP 45 STATUS = OFF
```

A variable is not allowed to have the same name as an object attribute.

Aside from saving some typing, named variables are required when using arithmetic expressions in rule condition clauses.

Arithmetic Expressions

condition clause can also contain an arithmetic expression formed from several variables whose value is compared against. Thus the format of a condition clause can be extended as follows:

```
expression relation value
expression relation variable
```

where `expression` is defined on a separate line before its first use in a rule using the format:

```
EXPRESSION name = f(variable1, variable2, ...)
```

The function `f(...)` can be any well-formed mathematical expression containing one or more named variables as well as any of the following math functions (which are case insensitive) and operators:

§ abs(x) for absolute value of x

§ step(x) which is 0 for x <= 0 and 1 otherwise

§ sqrt(x) for the square root of x

§ log(x) for logarithm base e of x

§ log10(x) for logarithm base 10 of x

§ exp(x) for e raised to the x power

§ the standard trig functions (sin, cos, tan, and cot)

§ the inverse trig functions (asin, acos, atan, and acot)

§ the hyperbolic trig functions (sinh, cosh, tanh, and coth)

§ the standard operators +, -, *, /, ^ (for exponentiation) and any level of nested parentheses.

Here is an example of using this feature:

```
VARIABLE P1_flow = LINK 1 FLOW
VARIABLE P2_flow = LINK 2 FLOW
VARIABLE O3_flow = Link 3 FLOW
EXPRESSION Net_Inflow = (P1_flow + P2_flow)/2 -
O3_flow
RULE 1
IF Net_Inflow > 0.1
THEN ORIFICE 3 SETTING = 1
ELSE ORIFICE 3 SETTING = 0.5
```

following types of objects:

- Pollutants
- Land Uses
- Treatment

Pollutants

user-defined pollutants. Required information for each pollutant includes:

- pollutant name
- concentration in rainfall
- concentration in groundwater
- concentration in inflow/infiltration
- concentration in dry weather flow
- initial concentration throughout the conveyance system
- first-order decay coefficient.

Co-pollutants can also be defined in SWMM. For example, pollutant X can have a co-pollutant Y, meaning that the runoff concentration of X will have some fixed fraction of the runoff concentration of Y added to it.

Pollutant buildup and washoff from subcatchment areas are determined by the land uses assigned to those areas. Input loadings of pollutants to the drainage system can also originate from external time series inflows as well as from dry weather inflows.

See Also

Pollutant Editor
Land Uses
Pollutant Buildup
Pollutant Washoff
External Inflows Editor

Land Uses

- Street Cleaning

characteristics assigned to subcatchments. Examples of land use activities are residential, commercial, industrial, and undeveloped. Land surface characteristics might include rooftops, lawns, paved roads, undisturbed soils, etc. Land uses are used solely to account for spatial variation in pollutant buildup and washoff rates within subcatchments.

The SWMM user has many options for defining land uses and assigning them to subcatchment areas. One approach is to assign a mix of land uses for each subcatchment, which results in all land uses within the subcatchment having the same pervious and impervious characteristics. Another approach is to create subcatchments that have a single land use classification along with a distinct set of pervious and impervious characteristics that reflects the classification.

The following processes can be defined for each land use category:

- Pollutant Buildup

See Also

Land Use Editor

Subcatchments

described (or "normalized") by either a mass per unit of subcatchment area or per unit of curb length. Mass is expressed in pounds for US units and kilograms for metric units. The amount of buildup is a function of the number of preceding dry weather days and can be computed using one of the following functions:

Power Function

Pollutant buildup (**B**) accumulates proportional to time (*t*) raised to some power, until a maximum limit is achieved,

$$B = Min(C_1, C_2 t^{C_3})$$

where C_1 = maximum buildup possible (mass per unit of area or curb length), C_2 = buildup rate constant, and C_3 = time exponent.

Exponential Function

Buildup follows an exponential growth curve that approaches a maximum limit asymptotically,

$$B = C_1\left(1 - e^{-C_2 t}\right)$$

where C_1 = maximum buildup possible (mass per unit of area or curb length) and C_2 = buildup rate constant (1/days).

Saturation Function

Buildup begins at a linear rate that continuously declines with time until a saturation value is reached,

$$B = \frac{C_1 t}{C_2 + t}$$

where C_1 = maximum buildup possible (mass per unit area or curb length) and C_2 = half-saturation constant (days to reach half of the maximum buildup).

External Time Series

This option allows one to use a Time Series to describe the rate of buildup per day as a function of time. The values placed in the time series would have units of mass per unit area (or curb length) per day. One can also provide a maximum possible buildup (mass per unit area or curb length) weather periods and can be described in one of the following ways:

Exponential Washoff
The washoff load (W) in units of mass per hour is proportional to the product of runoff raised to some power and to the amount of buildup remaining, i.e.,

$$W = C_1 q^{C_2} B$$

where C_1 = washoff coefficient, C_2 = washoff exponent, q = runoff rate per unit area (inches/hour or mm/hour), and B = pollutant buildup in mass units. The buildup here is the total mass (not per area or per curb length) and both buildup and washoff mass units are the same as used to express the pollutant's concentration (milligrams, micrograms, or counts).

Rating Curve Washoff
The rate of washoff W in mass per second is proportional to the runoff rate raised to some power, i.e.,

$$W = C_1 Q^{C_2}$$

where C_1 = washoff coefficient, C_2 = washoff exponent, and Q = runoff rate in user-defined flow units.

Event Mean Concentration
This is a special case of Rating Curve Washoff where the exponent is 1.0 and the coefficient C_1 represents the washoff pollutant concentration in mass per liter. The conversion between user-defined flow units used for runoff and liters is handled internally by SWMM. (Typical EMC's for selected constituents).

Note that in each case buildup is continuously depleted as washoff proceeds, and washoff ceases when there is no more buildup available. It is also possible to use the Event Mean Concentration option by itself, without having to model any pollutant buildup at all.

BMP Removal Efficiency

Street Sweeping

reduce the accumulated buildup of specific pollutants. The parameters that describe street sweeping include:

- days between sweeping
- the fraction of buildup of all pollutants that is available for removal by sweeping
- the fraction of available buildup for each pollutant removed by sweeping.

These parameters can be different for each land use and the last parameter can vary also with pollutant.

Treatment

node is modeled by assigning a set of treatment functions to the node. A treatment function can be any well-formed mathematical expression involving:

· the pollutant concentration (use the pollutant name to represent its concentration) for non-storage nodes this is the mixture concentration of all flow streams entering the node while for storage nodes it is the pollutant concentration within the nodes stored volume

represent removal)

any of the following process variables:

§ FLOW for flow rate into node (in user-defined flow units)

§ DEPTH for water depth above node invert (ft or m)

§ AREA for node surface area (ft2 or m2)

§ DT for routing time step (sec)

§ HRT for hydraulic residence time (hours)

· Any of the following math functions (which are case insensitive) can be used in a treatment expression:

§ abs(x) for absolute value of x

§ sgn(x) which is +1 for x >= 0 or -1 otherwise

§ step(x) which is 0 for x <= 0 and 1 otherwise

§ sqrt(x) for the square root of x

§ log(x) for logarithm base e of x

§ log10(x) for logarithm base 10 of x

§ exp(x) for e raised to the x power

§ the standard trig functions (sin, cos, tan, and cot)

§ the inverse trig functions (asin, acos, atan, and acot)

§ the hyperbolic trig functions (sinh, cosh, tanh, and coth)

§ along with the standard operators +, -, *, /, ^ (for exponentiation) and any level of nested parentheses.

The result of the treatment function can be either a concentration (denoted by the letter `C`) or a fractional removal (denoted by `R`). For example, a first-order decay expression for BOD exiting from a storage node might be expressed as: `C = BOD * exp(-0.05*HRT)` or the removal of some trace pollutant that is proportional to the removal of total suspended solids (TSS) could be expressed as: `R = 0.75 * R_TSS`

Care must be taken to avoid circular references when specifying treatment functions. For example, the above expression would not be computable if it were used to compute fractional removal of TSS.

various objects. These include:

- Curves
- Time Series
- Time Patterns

quantities. The following types of curves are used in SWMM:

Storage	describes how the surface area of a Storage Unit node varies with water depth.
Shape	describes how the width of a customized cross-sectional shape varies with height for a Conduit link.
Diversion	relates diverted outflow to total inflow for a Flow Divider node or a Custom Inlet.
Tidal	describes how the stage at an Outfall node changes by hour of the day.
Pump	relates flow through a Pump link to the depth or volume at the upstream node or to the head delivered by the pump.
Rating	relates flow through an Outlet link to the freeboard depth or head difference across the outlet; relates flow captured by a Custom Inlet drain to the depth of water above it.
Control	determines how the control setting of a pump or flow regulator varies as a function of some control variable (such as water level at a particular node) as specified in a Modulated Control rule; can also be used to adjust the flow from an LID unit's underdrain based on head.
Weir	allows a weir's discharge coefficient to vary with the hydraulic head across it.

Each curve must be given a unique name and can be assigned any number of data points.

See Also

Curve Editor

Time Series

with time. Time series can be used to describe:

- rainfall data
- temperature data
- water stage at outfall nodes
- external inflow hydrographs at drainage system nodes
- external inflow pollutographs at drainage system nodes
- control settings for pumps and flow regulators.

Each time series must be given a unique name and can be assigned any number of time-value data pairs. Time can be specified either as hours from the start of a simulation or as an absolute date and time-of-day. Time series data can either be entered directly into the program or be accessed from a user-supplied Time Series file.

For rainfall time series, it is only necessary to enter periods with non-zero rainfall amounts. SWMM interprets the rainfall value as a constant value lasting over the recording interval specified for the rain gage that utilizes the time series. For all other types of time series, SWMM uses interpolation to estimate values at times that fall in between the recorded values.

For times that fall outside the range of the time series, SWMM will use a value of 0 for rainfall and external inflow time series, and either the first or last series value for temperature, evaporation, and water stage time series.

See Also

Time Series Editor
Time Series Files

fashion. They consist of a set of adjustment factors applied as multipliers to a baseline DWF flow rate or pollutant concentration. The different types of time patterns include:

Monthly	one multiplier for each month of the year
Daily	one multiplier for each day of the week
Hourly	one multiplier for each hour from 12 AM to 11 PM
Weekend	hourly multipliers for weekend days

Each time pattern must have a unique name and there is no limit on the number of patterns that can be created. Each dry weather inflow (either flow or quality) can have up to four patterns associated with it, one for each type listed above.

Monthly time patterns can also be used to adjust the baseline values of the following hydrological parameters:

- subcatchment depression storage
- soil infiltration recovery rate
- groundwater evaporation rate.

See Also Time Pattern Editor
Inflows
Subcatchment Properties
Climatology Editor

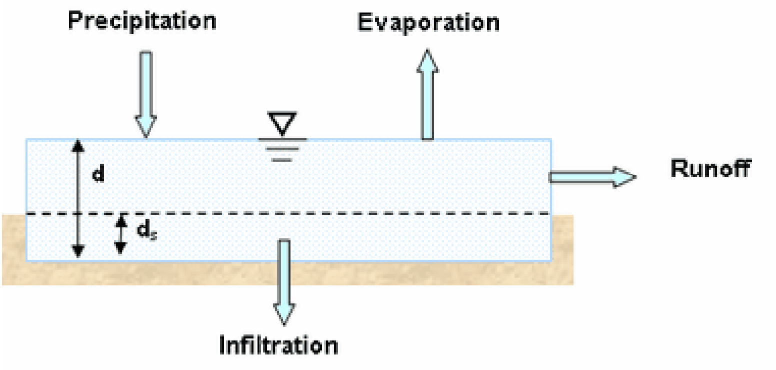
Computational Methods

principles of conservation of mass, energy, and momentum wherever appropriate. This section briefly describes the methods SWMM uses to model stormwater runoff quantity and quality through the following physical processes:

- Surface Runoff
- Groundwater
- Snowmelt
- Flow Routing
- Surface Ponding
- Water Quality Routing

More detailed descriptions of SWMMs computational procedures can be found in a series of three reference manuals available on EPAs SWMM web site.

figure below.



Each subcatchment surface is treated as a nonlinear reservoir. Inflow comes from precipitation and the runoff from any designated upstream subcatchments. Outflows consist of infiltration, evaporation, and surface runoff. The capacity of this "reservoir" is the maximum depression storage, which is the maximum surface storage provided by ponding, surface wetting, and interception. Surface runoff, Q, occurs only when the depth of water d in the "reservoir" exceeds the maximum depression storage, d_s, in which case the outflow is given by Manning's equation. Depth of water over the subcatchment (d) is continuously updated with time by solving numerically a water balance equation over the subcatchment.

unsaturated soil zone of pervious subcatchments areas. SWMM offers four choices for modeling infiltration:

Classical Horton Method
This method is based on empirical observations showing that infiltration decreases exponentially from an initial maximum rate to some minimum rate over the course of a long rainfall event. Input parameters required by this method include the maximum and minimum infiltration rates, a decay coefficient that describes how fast the rate decreases over time, and the time it takes a fully saturated soil to completely dry (used to compute the recovery of infiltration rate during dry periods).

Modified Horton Method
This is a modified version of the classical Horton Method that uses the cumulative infiltration in excess of the minimum rate as its state variable (instead of time along the Horton curve), providing a more accurate infiltration estimate when low rainfall intensities occur. It uses the same input parameters as does the traditional Horton Method.

Green-Ampt Method
This method for modeling infiltration assumes that a sharp wetting front exists in the soil column, separating soil with some initial moisture content below from saturated soil above. The input parameters required are the initial moisture deficit of the soil, the soil's hydraulic conductivity, and the suction head at the wetting front. The recovery rate of moisture deficit during dry periods is empirically related to the hydraulic conductivity.

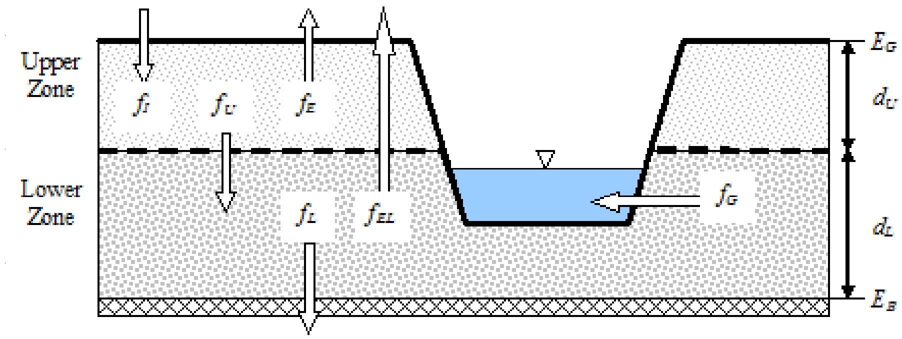
Modified Green-Ampt Method
This method modifies the original Green-Ampt procedure by not depleting moisture deficit in the top surface layer of soil during initial periods of low rainfall as was done in the original method. This change can produce more realistic infiltration behavior for storms with long initial periods where the rainfall intensity is below the soils saturated hydraulic conductivity.

Curve Number Method

This approach is adopted from the NRCS (SCS) Curve Number method unsaturated zone evaporation

that is used in SWMM. The upper zone is unsaturated with a variable moisture content of ?. The lower zone is fully saturated and therefore its moisture content is fixed at the soil porosity ϕ.

f_U percolation from the upper to lower zone which depends on the upper zone moisture content ? and depth d_U



ction of the depth

ch depends on the

rk which depends

eiving channel or node.

The fluxes shown in the figure, expressed as volume per unit area per unit time, consist of the following:

f_I infiltration from the surface

After computing the water fluxes that exist during a given time step, a mass balance is written for the change in water volume stored in each zone so that a new water table depth and unsaturated zone moisture content can be computed for the next time step.

Snowmelt

Updates the state of the snow packs associated with each subcatchment by accounting for snow accumulation, snow redistribution by areal depletion and removal operations, and snow melt via heat budget accounting. Any snowmelt coming off the pack is treated as an additional rainfall input onto the subcatchment.

At each runoff time step the following computations are made:

1. Air temperature and melt coefficients are updated according to the calendar date.

3. Any excess snow depth on the plowable area of the pack is redistributed according to the removal parameters established for the pack.
4. Areal coverages of snow on the impervious and pervious areas of the pack are reduced according to the Areal Depletion Curves defined for the study area.
5. The amount of snow in the pack that melts to liquid water is found using:
   - a heat budget equation for periods with rainfall, where melt rate increases with increasing air temperature, wind speed, and rainfall intensity
   - a degree-day equation for periods with no rainfall, where melt rate equals the product of a melt coefficient and the difference between the air temperature and the pack's base melt temperature.
6. If no melting occurs, the pack temperature is adjusted up or down based on the product of the difference between current and past air temperatures and an adjusted melt coefficient. If melting occurs, the temperature of the pack is increased by the equivalent heat content of the melted snow, up to the base melt temperature. Any remaining melt liquid beyond this is available to runoff from the pack.
7. The available snow melt is then reduced by the amount of free water holding capacity remaining in the pack. The remaining melt is treated the same as an additional rainfall input onto the subcatchment.

Flow Routing

conservation of mass and momentum equations for gradually varied, unsteady flow (i.e., the Saint Venant flow equations). The SWMM user has a choice on the level of sophistication used to solve these equations:

- Steady Flow Routing
- Dynamic Wave Routing

Each of these routing methods employs the Manning equation to relate flow rate to flow depth and bed (or friction) slope. For user-designated Force Main conduits, either the Hazen-Williams or Darcy-Weisbach equation can be used when pressurized flow occurs.

Steady Flow Routing

Steady Flow routing represents the simplest type of routing possible (actually no routing) by assuming that within each computational time step flow is uniform and steady. Thus it simply translates inflow hydrographs at the upstream end of the conduit to the downstream end, with no delay or change in shape. The normal flow equation is used to relate flow rate to flow area (or depth).

This type of routing cannot account for channel storage, backwater effects, entrance/exit losses, flow reversal or pressurized flow. It can only be used with dendritic conveyance networks, where each node has only a single outflow link (unless the node is a divider in which case two outflow links are required). This form of routing is insensitive to the time step employed and is really only appropriate for preliminary analysis using long-term continuous simulations.

form of the momentum equation in each conduit. The latter assumes that the slope of the water surface equal the slope of the conduit.

The maximum flow that can be conveyed through a conduit is the full normal flow value. Any flow in excess of this entering the inlet node is either lost from the system or can pond atop the inlet node and be re-introduced into the conduit as capacity becomes available.

Kinematic wave routing allows flow and area to vary both spatially and temporally within a conduit. This can result in attenuated and delayed outflow hydrographs as inflow is routed through the channel. However this form of routing cannot account for backwater effects, entrance/exit losses, flow reversal, or pressurized flow, and is also restricted to dendritic network layouts. It can usually maintain numerical stability with moderately large time steps, on the order of 1 to 5 minutes. If the aforementioned effects are not expected to be significant then this alternative can be an accurate and efficient routing method, especially for long-term simulations.

flow equations and therefore produces the most theoretically accurate results. These equations consist of the continuity and momentum equations for conduits and a volume continuity equation at nodes.

With this form of routing it is possible to represent pressurized flow when a closed conduit becomes full, such that flows can exceed the full normal flow value. Flooding occurs when the water depth at a node exceeds the maximum available depth, and the excess flow is either lost from the system or can pond atop the node and re-enter the drainage system.

Dynamic wave routing can account for channel storage, backwater, entrance/exit losses, flow reversal, and pressurized flow. Because it couples together the solution for both water levels at nodes and flow in conduits it can be applied to any general network layout, even those containing multiple downstream diversions and loops. It is the method of choice for systems subjected to significant backwater effects due to downstream flow restrictions and with flow regulation via weirs and orifices. This generality comes at a price of having to use much smaller time steps, on the order of thirty seconds or less (SWMM can automatically reduce the user-defined maximum time step as needed to maintain numerical stability).

Ponding and Pressurization

Normally in flow routing, when the flow into a junction exceeds the capacity of the system to transport it further downstream, the excess volume overflows the system and is lost. An option exists to have instead the excess volume be stored atop the junction, in a ponded fashion, and be reintroduced into the system as capacity permits. Under Kinematic Wave flow routing, the ponded water is stored simply as an excess volume. For Dynamic Wave routing, which is influenced by the water depths maintained at nodes, the excess volume is assumed to pond over the node with a constant surface area. This amount of surface area is an input parameter supplied for the junction.

Alternatively, the user may wish to represent the surface overflow system explicitly. In open channel systems this can include road overflows at bridges or culvert crossings as well as additional floodplain storage areas. In closed conduit systems, surface overflows may be conveyed down streets, alleys, or other surface routes to the next available stormwater inlet or open channel. Overflows may also be impounded in surface depressions such as parking lots, back yards or other areas.

In sewer systems with pressurized pipes and force mains the hydraulic head at junction nodes can at times exceed the ground elevation under Dynamic Wave routing. This would normally result in an overflow which, as described above, can either be lost or ponded. SWMM allows the user to specify an additional "surcharge" depth at junction nodes that lets them pressurize and prevents any outflow until this additional depth is exceeded. If both ponding and pressurization are specified for a node ponding takes precedence and the surcharge depth is ignored. Ponding does not apply to storage nodes.

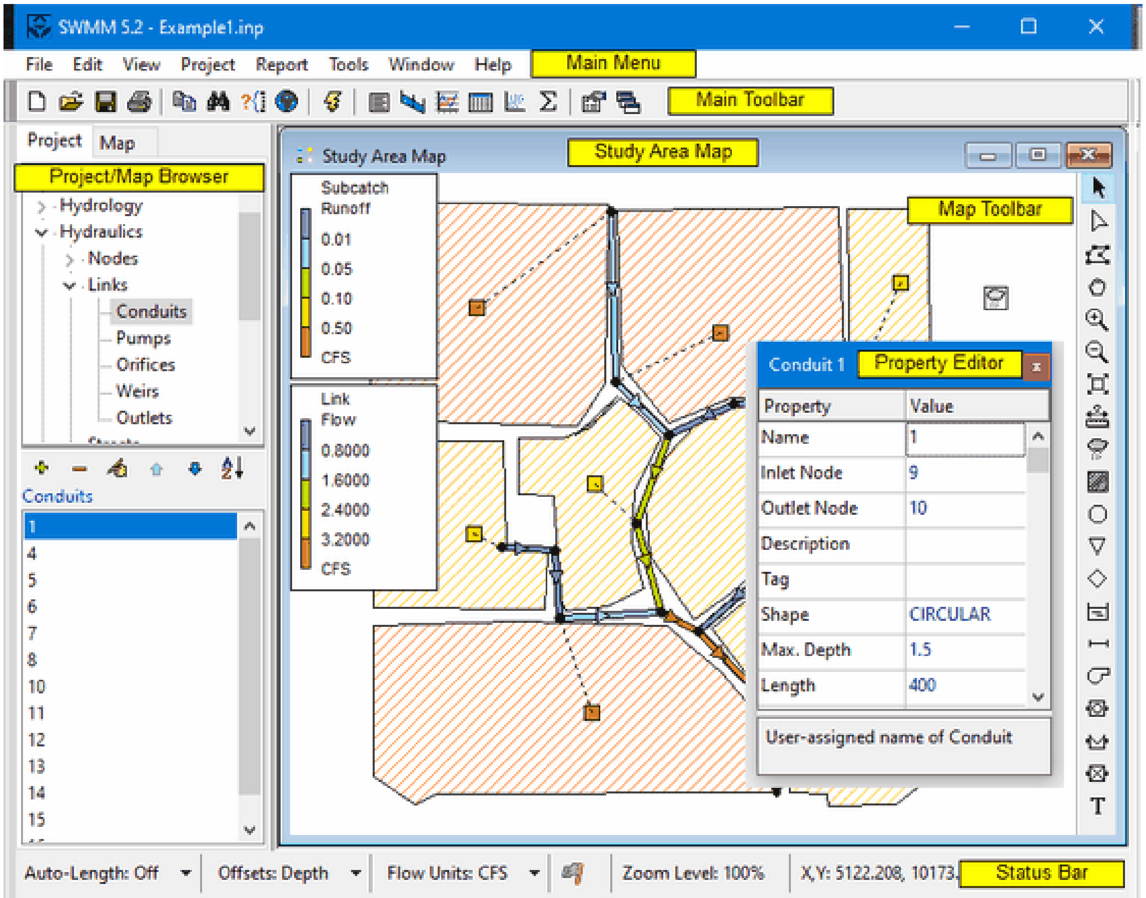
as a continuously stirred tank reactor (CSTR). Although a plug flow reactor assumption might be more realistic, the differences will be small if the travel time through the conduit is on the same order as the routing time step. The concentration of a constituent exiting the conduit at the end of a time step is found by integrating the conservation of mass equation, using average values for quantities that might change over the time step such as flow rate and conduit volume.

Water quality modeling within storage unit nodes follows the same approach used for conduits. For other types of nodes that have no volume, the quality of water exiting the node is simply the mixture concentration of all water entering the node.

The pollutant concentration in both a conduit and a storage node will be reduced by a first-order decay reaction if the pollutants first-order decay coefficient is not zero.

SWMM's Main Window

learn more about it.



Main Menu

contains a collection of menus used for working with the program. These include:

- File Menu
- Edit Menu
- View Menu
- Project Menu
- Report Menu
- Tools Menu
- Window Menu
- Help Menu

printing:

Command	Description
New	Creates a new SWMM project
Open	Opens an existing project
Reopen	Reopens a recently used project
Save	Saves the current project
Save As	Saves current project under a different name
Export	Exports the Study Area Map to a file; Exports current results to a Hot Start file; Exports the current result's Status/Summary reports
Combine	Combines two Routing Interface files together
Page Setup	Sets page margins and orientation for printing
Print Preview	Previews a printout of the current active view (map, report, graph, or table)
Print	Prints the current view
Exit	Exits SWMM

Edit Menu

Command	Description
Copy To	Copies the currently active view (map, report, graph or table) to the clipboard or to a file
Select Object	Enables the user to select an object on the Study Area Map
Select Vertex	Enables the user to select a vertex of a subcatchment or link displayed on the Map
Select Region	Enables the user to delineate a region on the Map for selecting multiple objects
Select All	Selects all objects when the Map is the active window or all cells of a table when a tabular report is the active window
Find Object	Locates a specific object by name on the Map
Edit Object	Edits the properties of the currently selected object
Delete Object	Deletes the currently selected object
Group Edit	Edits a property for the group of objects that fall within the outlined region of the Map
Group Delete	Deletes a group of objects that fall within the outlined region of the Map

the program's toolbars.

Command	Description
Dimensions	Sets reference coordinates and distance units for the study area map
Backdrop	Allows a backdrop image to be added, positioned, and viewed behind the map
Pan	Pans across the map
Zoom In	Zooms in on the map
Zoom Out	Zooms out on the map
Full Extent	Redraws the map at full extent
Query	Highlights objects on the map that meet specific criteria
Overview	Toggles the display of the Overview Map
Layers	Toggles display of object layers on the Map
Legends	Controls display of the Map legends
Toolbars	Toggles display of the toolbar

The mouse wheel can also be used to pan, zoom in or zoom out of the map at any time without having to select the Pan, Zoom In or Zoom Out commands.

analyzed.

Command	Description
Summary	Lists the number of each type of object in the project
Details	Shows a detailed listing of all project data
Defaults	Edits a project's default properties
Calibration Data	Registers files containing calibration data with the project
Add a New Object	Adds a new object to the project
Run Simulation	Runs a simulation

different formats.

Command	Description
Status	Displays a status report for the most recent simulation run
Summary	Displays summary results in tabular form
Graph	Displays simulation results in graphical form
Table	Displays simulation results in tabular form
Statistics	Displays a statistical analysis of simulation results
Customize	Customizes the display of the currently active graph

preferences, Study Area Map display options, and external add-in tools.

Command	Description
Program Preferences	Sets program preferences, such as font size, confirm deletions, number of decimal places displayed, etc.
Map Display Options	Sets appearance options for the Map, such as object size, object annotation, flow direction arrows, and back-ground color
Configure Tools	Adds, deletes, or modifies external add-in tools

windows within the SWMM workspace.

Command	Description
Cascade	Arranges windows in cascaded style, with the Study Area Map filling the entire display area
Tile	Minimizes the map and tiles the remaining windows vertically in the display area
Close All	Closes all open windows except for the map
Window List	Lists all open windows; the currently selected window has the focus and is denoted with a check mark

Help Menu

Command	Description
User Guide	Displays the User Guide's Table of Contents
How do I	Displays a list of topics covering the most common operations
Keyboard Shortcuts	Displays a list of keyboard shortcuts for main menu commands
Measurement Units	Shows measurement units for all of SWMM's parameters
Error Messages	Lists the meaning of all error messages
Tutorials	Lists tutorials that show how to use EPA SWMM
Welcome Screen	Displays SWMM's Welcome screen
About	Displays information about the version of EPA SWMM being used

to select them. They are listed below.

Menu Command	Shortcut Key
File | New	Ctrl-N
File | Open	Ctrl-O
File | Save	Ctrl-S
File | Save As	Ctrl-Alt-S
File | Exit	Alt-F4
Edit | Copy To	Ctrl-C
Edit | Select All	Ctrl-A
Edit | Find Object	Ctrl-F
Edit | Edit Object	**F2**
Edit | Delete Object	Ctrl-Delete
Edit | Group Edit	Shift-F2
View | Query	Ctrl-Q
Project | Add a New <object>	Ctrl-Insert
Project | Run Simulation	**F9**
Report | Graph | Time Series	Ctrl-G
Window | Cascade	Shift-F5
Window | Tile	Shift-F4
Window | Close All	Shift-Ctrl-F4
Help | User Guide	Ctrl-F1

In addition the F1 key can be used to bring up context-sensitive Help in most of SWMM's data editing windows.

Main Toolbar

commands:

- Creates a new project
- Opens an existing project
- Saves the current project
- Prints the currently active window
- Copies the current selection to the clipboard or to a file
- Finds a specific object on the Study Area Map
- Makes a visual query of the Study Area Map
- Toggles the display of the Overview Map
- Runs a simulation
- Displays a run's Status and Summary reports
- Creates a profile plot of simulation results
- Creates a time series plot of simulation results
- Creates a time series table of simulation results
- Creates a scatter plot of simulation results
- Performs a statistical analysis of simulation results
- Modifies display options for the currently active view
- Arranges windows in cascaded style, with the Study Area Map filling the entire display area

The toolbar can be made visible or invisible by selecting **View >> Toolbar** from the Main Menu.

Map Toolbar

Study Area Map:

- Selects an object on the map
- Selects link or subcatchment vertex points
- Selects a region on the map
- Pans across the map
- Zooms in on the map
- Zooms out on the map
- Draws the map at full extent
- Measures a length or area on the map

The mouse wheel can also be used to pan, zoom in or zoom out of the map at any time without having to select the Pan, Zoom In or Zoom Out buttons.

The Map Toolbar also contains buttons used to add objects to a project via the Study Area Map:

- Adds a rain gage to the map
- Adds a subcatchment to the map
- Adds a junction node to the map
- Adds an outfall node to the map
- Adds a flow divider node to the map
- Adds a storage unit node to the map
- Adds a conduit link to the map
- Adds a pump link to the map
- Adds an orifice link to the map
- Adds a weir link to the map
- Adds an outlet link to the map
- Adds a text label to the map

Status Bar

divided into six sections:

Auto-Length

Indicates whether the automatic computation of conduit lengths and subcatchment areas is turned on or off. The setting can be changed by clicking the drop down arrow.

Offsets

Indicates whether the positions of links above the invert of their connecting nodes are expressed as a Depth above the node invert or as the Elevation of the offset. Click the drop down arrow to change this option. If changed, a dialog box will appear asking if all existing offsets in the current project should be changed or not (i.e., convert Depth offsets to Elevation offsets or Elevation offsets to Depth offsets, depending on the option selected).

Flow Units

Displays the current flow units that are in effect. Click the drop down arrow to change the choice of flow units. Selecting a US flow unit means that all other quantities will be expressed in US units, while choosing a metric flow unit will force all quantities to be expressed in metric units. The units of previously entered data are not automatically adjusted if the unit system is changed.

Run Status

- results are not available because no simulation has been run yet
- results are up to date
- results are out of date because project data have changed.
- results are not available because the last simulation had errors

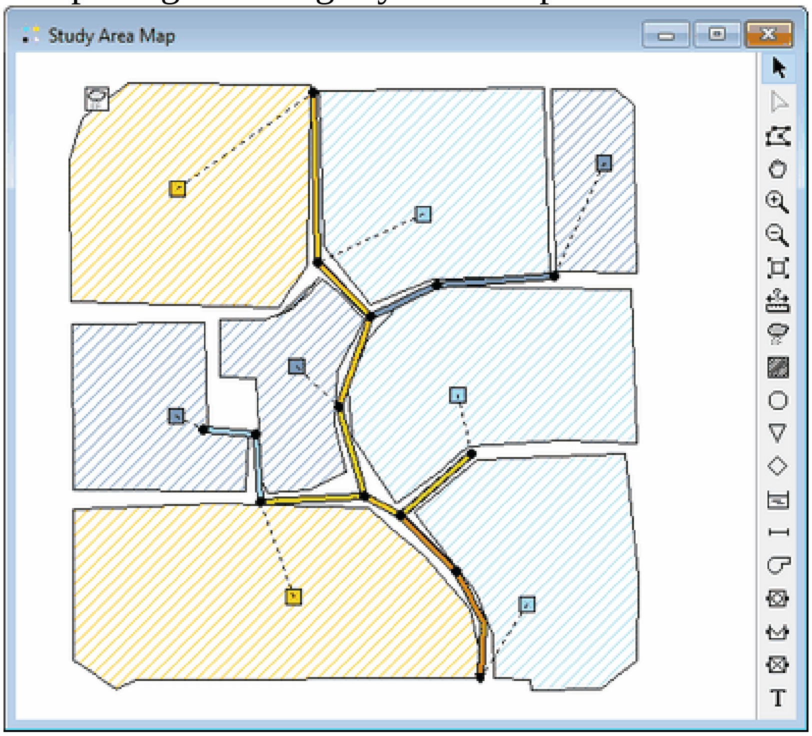
Zoom Level

Displays the current zoom level for the Study Area Map (100% is full scale).

XY Location

Displays the Study Area Map coordinates of the current position of the mouse pointer.

comprising a drainage system. Its pertinent features are as follows:

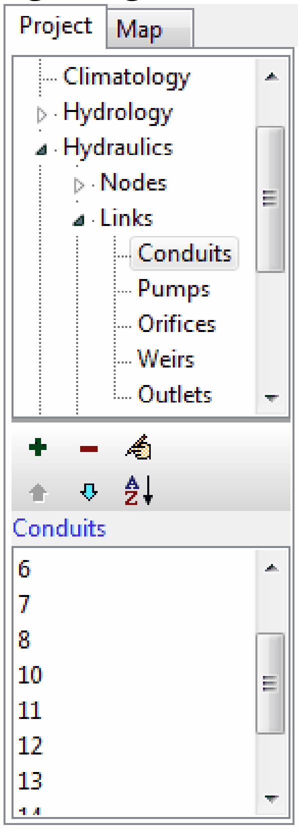


- The location of objects and the distances between them do not necessarily have to conform to their actual physical scale.
- Selected properties of these objects, such as water quality at nodes or flow velocity in links, can be displayed by using different colors. The color-coding is described in a Legend, which can be edited.
- New objects can be directly added to the Map and existing objects can be selected for editing, deleting, and repositioning.
- A backdrop drawing (such as a street or topographic map) can be placed behind the Map for reference.
- The Map can be zoomed to any scale and panned from one position to another.
- Nodes and links can be drawn at different sizes, flow direction arrows added, and object symbols, ID labels and numerical property values displayed.
- The Map can be printed, copied onto the Windows clipboard, or exported as a DXF file or Windows metafile.

See Also Working with the Map

Project Browser

SWMM's Main Window is pressed. It provides access to all of the data in a project. The vertical sizes of the list boxes in the browser can be adjusted by using the splitter bar located just below the upper box. The width of the Browser panel can be adjusted by using the splitter bar located along its right edge



The upper list box displays the various categories of data objects available to a SWMM project. The lower list box lists the name of each individual object of the currently selected data category.

The buttons between the two list boxes are used as follows:

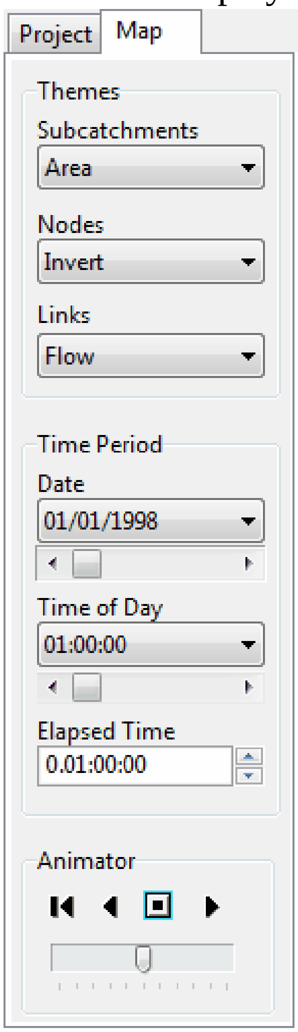
adds a new object,

deletes the selected object, edits the selected object, moves the selected object up one position, moves the selected object down one position, sorts the objects in ascending order.

Selections made in the Project Browser are coordinated with objects highlighted on the Study Area Map, and vice versa. For example, selecting a conduit in the Browser will cause that conduit to be highlighted on the map, while selecting it on the map will cause it to become the selected

Map Browser

SWMM's Main Window is selected. . It controls the mapping themes and time periods viewed on the Study Area Map. The width of the Map Browser panel can be adjusted by using the splitter bar located along its right edge. The Map Browser consists of the following three panels that control what results are displayed on the map:



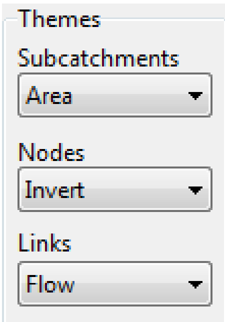
The Themes panel selects a set of variables to view in color-coded fashion on the Map.

The Time Period panel selects which time period of the simulation results are viewed on the Map.

The Animator panel controls the animated display of the Study Area Map and all Profile Plots over time.

Map Browser - Themes

to view in color-coded fashion on the Study Area Map.



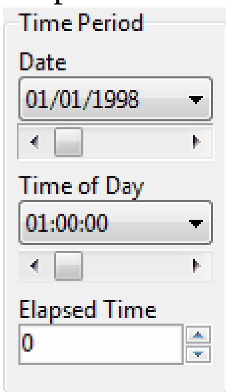
Subcatchments - selects the theme to display for the subcatchment areas shown on the Map.

Nodes - selects the theme to display for the drainage system nodes shown on the Map.

Links - selects the theme to display for the drainage system links shown on the Map.

Map Browser - Time Period

in which to view computed results in thematic fashion on the Study Area Map.

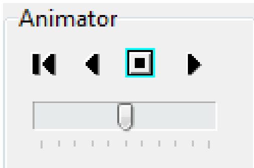


Date - selects the day for which simulation results will be viewed.

Time of Day - selects the hour of the current day for which simulation results will be viewed.

Elapsed Time - selects the elapsed time from the start of the simulation for which results will be viewed.

the Study Area Map and all Profile Plots through time i.e., updating map color-coding and hydraulic grade line profile depths as the simulation time clock is automatically moved forward or back. The meaning of the control buttons are as follows:



Returns to the starting period.

Starts animating backwards in time.

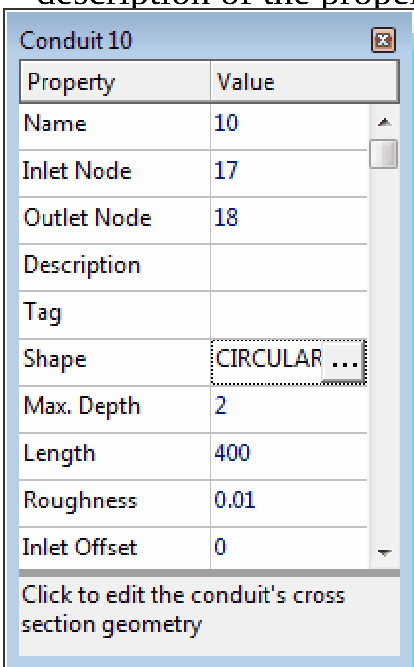
Stops the animation.

Starts animating forwards in time.

The slider bar is used to adjust the animation speed.

Property Editor

objects that can appear on the Study Area Map. It is invoked when one of these objects is selected (either on the map or in the Project Browser) and double-clicked or when the Project Browser's Edit button is clicked.



Key features of the Property Editor include:

- The Editor is a grid with two columns - one for the property's name and the other for its value.

Editor with the mouse

A hint area is displayed at the bottom of the Editor with an expanded description of the property being edited. The size of this area can be … splitter bar located just above it.

… be moved and re-sized via the normal Windows …

…rty, the value field can be one of the following:

…ou enter a value

…ox from which you select a value from a list of

…ox in which you can enter a value or select from a

…ch you click to bring up a specialized editor

…hich currently has focus will have a focus rectangle drawn around it.

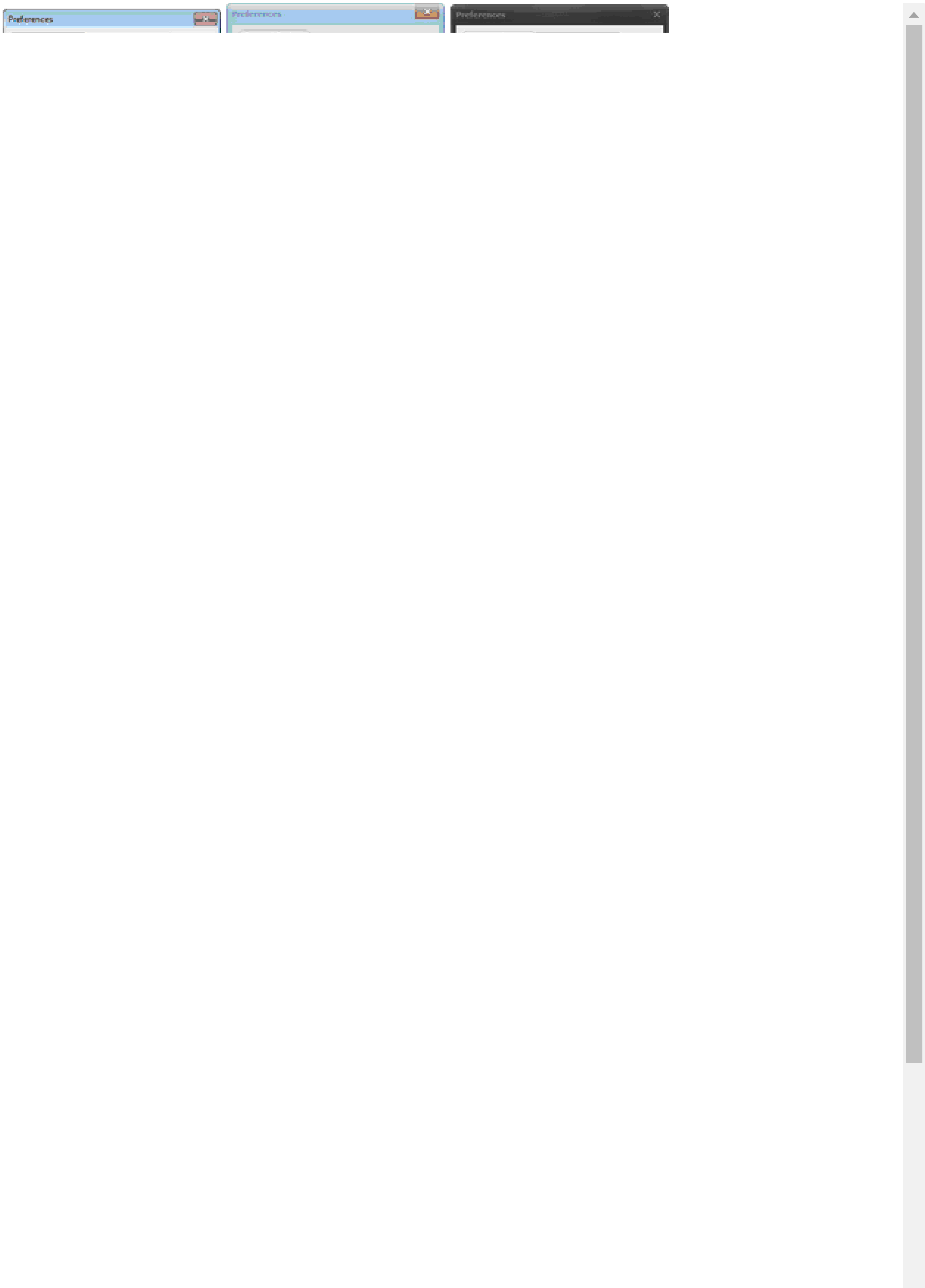
- Both the mouse and the Up and Down arrow keys on the keyboard can be used to move between fields.
- To begin editing the field with the focus, either begin typing a value or hit the Enter key.
- To have the program accept edits made in a property field, press the Enter key or move to another field. To cancel an edit, press the Esc key.
- The Property Editor can be hidden by clicking the button in the upper right corner of its title bar.

Setting Program Preferences

program preferences, select **Program Preferences** from the **Tools** menu. A Preferences dialog will appear containing two tabbed pages - one for General Preferences and one for Numerical Precision.

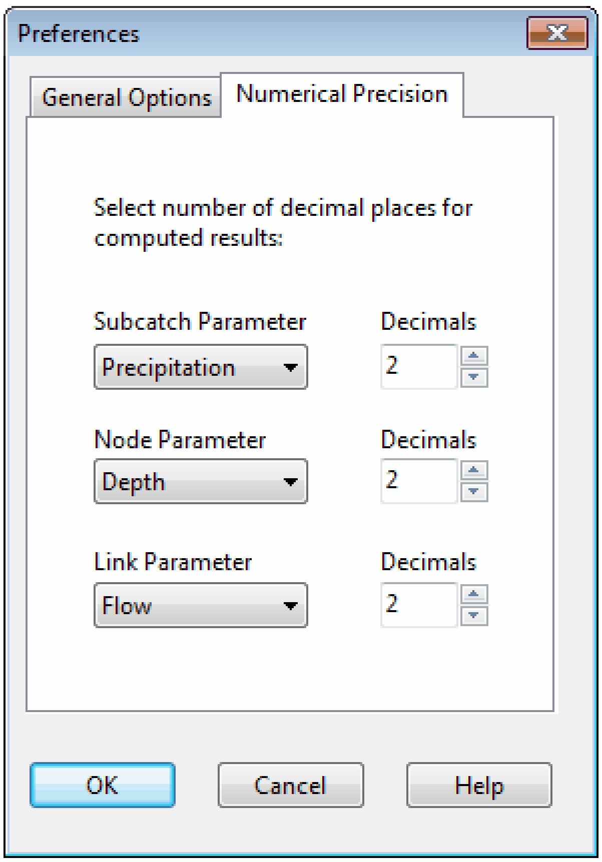
the **Preferences** dialog:

Blinking Map Highlighter	Check to make the selected object on the Study Area Map blink on and off.
Flyover Map Labeling	Check to display the ID name and current theme value in a hint-style box whenever the mouse is placed over an object on the Map.
Confirm Deletions	Check to display a confirmation dialog box before deleting any object.
Automatic Backup File	Check to save a backup copy of a newly opened project to disk named with a .bak extension.
Report Elapsed Time by Default	Check to use elapsed time (rather than date/time) as the default for time series graphs and tables.
Prompt to Save Results	If left unchecked then simulation results are automatically saved to disk when the current project is closed. Otherwise the user will be asked if results should be saved.
Show Welcome Screen	Check to have SWMM display a welcome screen when started.
Clear Recent Project List	Check to clear the list of most recently used files appearing when File >> Reopen is selected from the Main Menu.
Style Theme	Selects a color theme to use for SWMM's user interface (see below)


Preferences
Preferences
Preferences

Numerical Precision

The **Numerical Precision** page of the Preferences dialog controls the number of decimal places displayed when simulation results are reported. Use the dropdown list boxes to select a specific Subcatchment, Node or Link variable, and then use the edit boxes next to them to select the number of decimal places to include when displaying computed results for the variable.



Note that the number of decimal places displayed for any particular input design parameter, such as slope, diameter, length, etc. is whatever the user enters.

Working with Projects

are usually named with a **.INP** extension. This section describes how to create, open, and save SWMM projects and how to set their default properties.

- Creating a new project
- Opening an existing project
- Saving a project
- Setting project defaults
- Units of measurement
- Registering calibration data
- Viewing all project data

Creating a New Project

1. Select **File >> New** or click the button on the Main Toolbar.

to it) before the new project is created.

3. A new, unnamed project is created with all options set to their default values.

A new project is automatically created whenever SWMM first begins.

If you are going to use a backdrop image with automatic area and length calculation, then it is recommended that you set the map dimensions immediately after creating the new project (see Setting the Map's Dimensions).

Opening an Existing Project

1. Either select **File** >> **Open** or click the  button on the Main Toolbar.
2. You will be prompted to save the current project (if changes were made to it).
3. Select the file to open from the Open File dialog that appears.
4. Click **Open** to open the selected file.

To open a project that was worked on recently:

1. Select **File** >> **Reopen**.

Saving a Project

To save a project under its current name either select **File >> Save** or click the button on the Main Toolbar.

To save a project using a different name, select **File >> Save As**. A standard Save File dialog will appear from which you can select the folder and name that the project should be saved under.

Setting Project Defaults

the SWMM user. These values fall into three categories:

- Default node/link properties (e.g., node invert, conduit length, routing method).
- Default ID labels (labels used to identify nodes and links when they are first created)

To set default values for a project:

1. Select **Project >> Defaults**.
2. A Project Defaults dialog will appear with three pages, one for each category listed above.
3. Check the box in the lower left of the dialog form if you want to save your choices for use in all new future projects as well.
4. Click **OK** to accept your choice of defaults.

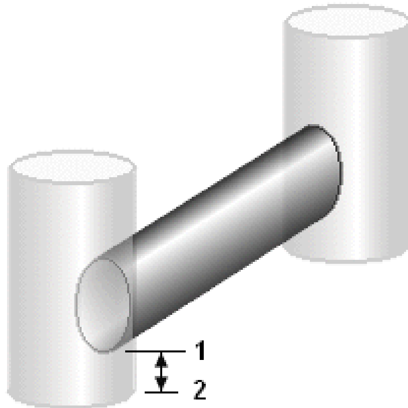
Units of Measurement

flow units determines what unit system is used for all other quantities:

- selecting **CFS** (cubic feet per second), **GPM** (gallons per minutes), or **MGD** (million gallons per day) for flow units implies that US units will be used throughout
- selecting **CMS** (cubic meters per second), **LPS** (liters per second), or **MLD** (million liters per day) as flow units implies that SI units will be used throughout.
- pollutant concentration and Mannings roughness coefficient (n) are always expressed in metric units.

Flow units can be selected directly on the main window's Status Bar or by setting a project's default values. In the latter case the selection can be saved so that all new future projects will automatically use those units. The units of previously entered data are not automatically adjusted if the unit system is changed.

distance above the invert of their connecting end nodes.



There are two different conventions available for specifying the location of these offsets. The **Depth** convention uses the offset distance from the node's invert (distance between **1** and **2** in the figure above). The **Elevation** convention uses the absolute elevation of the offset location (the elevation of point **1** in the figure).

The choice of convention can be made on the Status Bar of SWMM's main window or on the Node/Link Properties page of the Project Defaults dialog. When this convention is changed, a dialog will appear giving one the option to automatically re-calculate all existing link offsets in the current project using the newly selected convention.

Registering Calibration Data

its Time Series Plots. Before SWMM can use such calibration data they must be entered into a specially formatted text file and be registered with the project.

To register calibration data residing in a Calibration File:

1. Select **Project >> Calibration Data.**

parameter (e.g., node depth, link flow, etc.) whose calibration data will be registered.

3. Then click the **Add** button to select a Calibration File from a standard Windows file selection dialog box.

4. Click the **Edit** button if you want to open the Calibration File in Windows NotePad for editing.

5. Click the **Delete** button if you wish to remove the Calibration File from the form.

6. Repeat steps 2 - 4 for any other parameters that have calibration data.

7. Click **OK** to accept your selections.

Viewing All Project Data

A listing of all project data (with the exception of map coordinates) can be viewed in a non-editable window, formatted for input to SWMM's computational engine. This can be useful for checking data consistency and to make sure that no key components are missing. To view such a listing, select **Project >> Details** from the Main Menu. The format of the data in this listing is the same as that used when the file is saved to disk. It is described in detail in Appendix D of the SWMM 5 Users Manual.

Working with Objects

conveyance system. This section describes how these objects can be created, selected, edited, deleted, and repositioned.

- Types of objects
- Adding objects
- Selecting an object
- Moving an object
- Editing an object
- Converting an object
- Copying and pasting objects
- Deleting an object
- Shaping a link
- Shaping a subcatchment
- Selecting a group of objects
- Deleting a group of objects
- Editing a group of objects

Types of Objects

Map, and non-physical objects that encompass design, loading, and operational information. These objects, which are listed in the Project Browser, consist of the following:

- Project Title/Notes
- Simulation Options
- Climatology
- Rain Gages
- Subcatchments
- Aquifers
- Snow Packs
- Unit Hydrographs
- LID Controls
- Pollutants
- Land Uses
- Nodes
- Links
- Transects
- Streets
- Inlets
- Control Rules
- Curves
- Time Series
- Time Patterns
- Map Labels

Adding an Object

page of the Project Browser and either select **Project >> Add a New ...** from the Main Menu or click the Browser's button. If the object has a button on the Map Toolbar you can simply click the button instead.

If the object is a visual object that appears on the Study Area Map (a Rain Gage, Subcatchment, Node, Link, or Map Label) it will automatically receive a default ID name and a prompt will appear in the Status Bar telling you how to proceed. The steps used to draw each of these objects on the map are detailed below:

Rain Gages

Move the mouse to the desired location on the Map and left-click.

Subcatchments

Use the mouse to draw a polygon outline of the subcatchment on the Map:

- Left-click at each vertex

- Press the <**Esc**> key if you wish to cancel the action.

Nodes (Junctions, Outfalls, Flow Dividers, and Storage Units) Move the mouse to the desired location on the Study Area Map and left-click.

Links (Conduits, Pumps, Orifices, Weirs, and Outlets)

- Left-click the mouse on the link's inlet (upstream) node.
- Move the mouse (without pressing any button) in the direction of the link's outlet (downstream) node, clicking at all intermediate points necessary to define the link's alignment.
- Left-click the mouse a final time over the link's outlet (downstream) node. (Pressing the right mouse button or the <**Esc**> key while drawing a link will cancel the operation.)

Map Labels

- Left-click the mouse on the map location where the top left corner of the label should appear.
- Enter the text for the label.
- Press <**Enter**> to accept the label or <Esc> to cancel.

For all other non-visual types of objects, an object-specific dialog form will appear that allows you to name the object and edit its properties.

Selecting an Object

1. Make sure that the map is in Selection mode (the mouse cursor has the shape of an arrow pointing up to the left). To switch to this mode, either click the Select Object button on the Map Toolbar or choose **Edit >> Select Object** from the Main Menu.

To select an object using the Project Browser:

1. Select the object's category from the upper list in the Browser.
2. Select the object from the lower list in the Browser.

Moving an Object

location on the Study Area Map. To move an object to another location:

location:

1. Select the object on the map.

3. Release the mouse button.

The following alternative method can also be used:

1. Select the object to be moved from the Project Browser (it must be either a rain gage, subcatchment, node, or map label).
2. With the left mouse button held down, drag the item from the Items list box of the Project Browser to its new location on the map.
3. Release the mouse button.

Note that the second method can be used to place objects on the map that were imported from a project file that had no coordinate information included in it.

Editing an Object

1. Select the object on the map.
   - double click on the object
   - or right click on the object and select **Properties** from the pop-up menu that appears
   - or click in the Project Browser
   - or select **Edit >> Edit Object** from the Main Menu.

3. Edit the object's properties in the Property Editor.

To edit an object listed in the Project Browser:

1. Select the object in the Project Browser.
2. Either:
   - click in the Project Browser,
   - or select **Edit >> Edit Object** from the Main Menu,
   - or double-click the item in the Objects list,
   - or press the <**Enter**> key.

Depending on the class of object selected, a special property editor will appear in which the object's properties can be modified.

The unit system in which object properties are expressed depends on the choice of units for flow rate. Using a flow rate expressed in cubic feet, gallons or acre-feet implies that US customary units will be used for all quantities. Using a flow rate expressed in liters or cubic meters means that SI metric units will be used. Flow units are selected either from the project's default Node/Link properties (see Setting Project Defaults) or directly from the main window's Status Bar.

Converting an Object

having to first delete the object and add a new one in its place. An example would be converting a Junction node into an Outfall node, or converting an Orifice link into a Weir link.

To convert a node or link to another type:

1. Right click the object on the Study Area Map.

3. Select the new type of node or link to convert to from the sub-menu that appears.
4. Edit the object to provide any data that was not included with the previous type of object.

Only properties that are common to both types of objects will be preserved after an object is converted to a different type. For nodes this includes its name, position, description, tag, external inflows, treatment functions, and invert elevation. For links it includes just its name, end nodes, description, and tag. Non-preserved properties are assigned their default values.

Copying and Pasting Objects

and pasted into another object from the same category.

To copy the properties of an object to SWMM's internal clipboard:

1. Right click the object on the Map.

To paste copied properties into an object:

1. Right click the object on the Map.
2. Select **Paste** from the pop-up menu that appears.

Only data that can be shared between objects of the same type can be copied and pasted. Properties not copied include the object's name, coordinates, end nodes (for links), Tag property and any descriptive comment associated with the object. For Map Labels, only font properties are copied and pasted.

Deleting an Object

To delete an object:

1. Select the object on the Study Area Map or from the Project Browser.
2. Either click the button on the Project Browser, or press the <**Delete**> key on the keyboard, or select **Edit >> Delete Object** from the Main Menu, or right-click the object on the map and select **Delete** from the pop-up menu that appears.

You can require that all deletions be confirmed before they take effect. See the General Preferences page of the Program Preferences dialog box.

Shaping a Link

segments that define the alignment or curvature of the link. Once a link has been drawn on the Study Area Map, interior points that define these line segments can be added, deleted, and moved.

To edit the interior points of a link:

1. Select the link to edit on the map and put the map in **Vertex Selection** mode by either
   - or selecting **Edit >> Select Vertex** from the Main Menu,
   - or right-clicking on the link and selecting **Vertices** from the popup menu.
2. The mouse pointer will change shape to an arrow tip, and any existing vertex points on the link will be displayed as small open squares. The currently selected vertex will be displayed as a filled square.To select a particular vertex, click the mouse over it.
3. To add a new vertex to the link, right-click the mouse and select **Add Vertex** from the popup menu (or simply press the <**Insert**> key on the keyboard).
4. To delete the currently selected vertex, right click the mouse and select **Delete Vertex** from the popup menu (or simply press the <**Delete**> key on the keyboard).
5. To move a vertex to another location, drag it with the left mouse button held down to its new position.

While in Vertex Selection mode you can begin editing the vertices for another link by simply clicking on the link. To leave Vertex Selection mode, right click on the map and select **Quit Editing** from the popup menu, or simply select one of the other buttons on the Map Toolbar.

A link can also have its direction reversed (i.e., its end nodes switched) by right clicking on it and selecting **Reverse** from the pop-up menu that appears. Normally, links should be oriented so that the upstream end is at a higher elevation than the downstream end.

Shaping a Subcatchment

Subcatchments are drawn on the Study Area Map as closed polygons. To edit or add vertices to the polygon, follow the same procedures used for links (see Shaping a Link). If the subcatchment is originally drawn or is edited to have two or less vertices, then only its centroid symbol will be displayed on the Map.

Selecting a Group of Objects

can have a common property edited or be deleted all together. To select such a group of objects:

left mouse button at each successive vertex of the polygon.

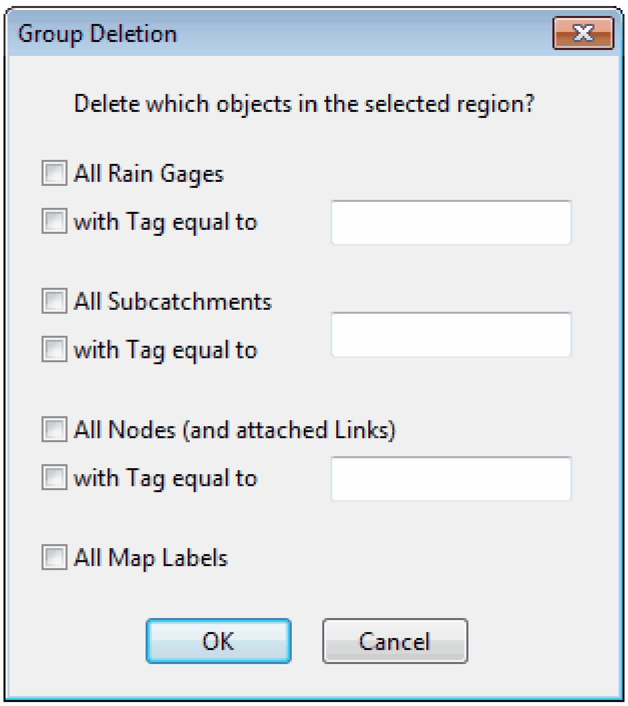
3. Close the polygon by clicking the right button or by pressing the <**Enter**> key; cancel the selection by pressing the <**Esc**> key.

1. Choose **Edit >> Select Region** from the Main Menu or click the button on the Map Toolbar.

To select all objects in the project, whether in view or not, select **Edit >> Select All** from the Main Menu.

Deleting a Group of Objects

To delete the objects located within a selected area of the Study Area Map (see Selecting a Group of Objects), select **Edit >> Group Delete** from the Main Menu. Then select the categories of objects you wish to delete from the dialog box that appears. As an option, you can specify that only objects with a specific Tag property should be deleted. Keep in mind that deleting a node will also delete any links connected to the node.



Editing a Group of Objects

Once a group of objects has been selected (see Selecting a Group of Objects), you can edit a common property shared among them:

1. Select **Edit** >> **Group Edit** from the Main Menu.
2. Use the Group Editor dialog that appears to select a property and specify its new value.

Working with the Map

section describes how you can manipulate this map to enhance your visualization of the system.

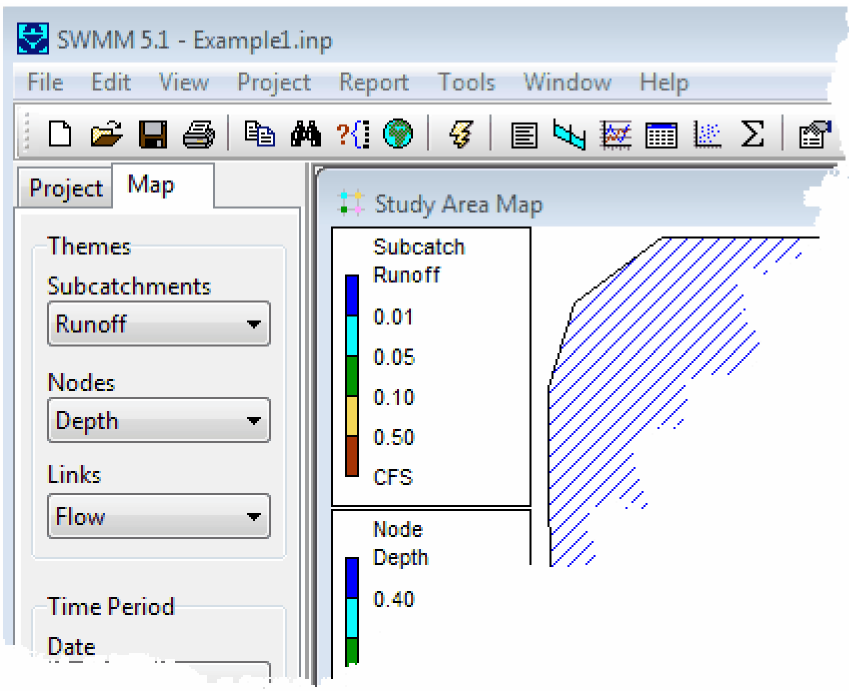
- Selecting a map theme
- Setting the map's dimensions
- Utilizing a backdrop image
- Measuring distances
- Zooming the map
- Panning the map
- Viewing at full extent
- Finding an object
- Submitting a map query
- Using the map legends
- Using the Overview Map
- Setting map display options
- Exporting the map

Viewing Map Layers

The layers that can be viewed on the Study Area consist of rain gages, subcatchments, nodes, links, labels, and the backdrop image. The display of each of these can be toggled on or off by selecting **View >> Layers** from the Main Menu or by right-clicking on the map and selecting **Layers** from the pop-up menu that appears.

Selecting a Map Theme

A map theme corresponds to a specific layer property whose value is drawn in color-coded fashion on the Study Area Map. The dropdown list boxes on the Map Browser are used for selecting a theme to display for the subcatchment, node and link layers.



Methods for changing the color-coding associated with a theme are discussed in Using the Map Legends.

can be properly scaled to the computer's video display.

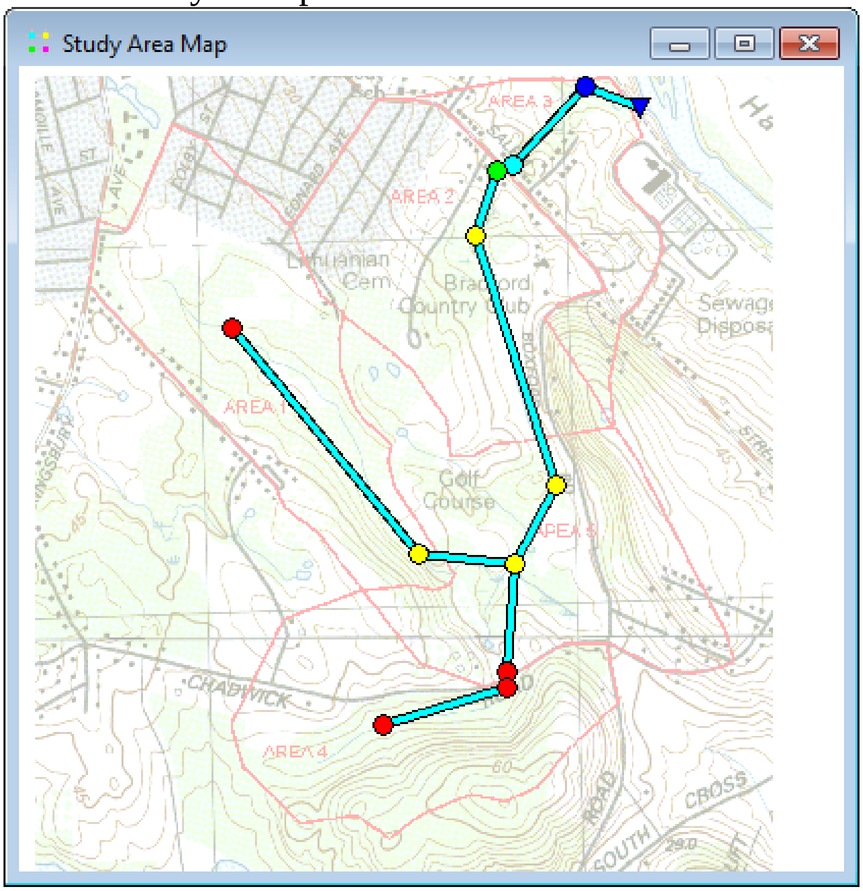
To set the map's dimensions:

1. Select **View** >> **Dimensions** from the Main Menu.
2. Enter coordinates for the lower-left and upper-right corners of the map into the Map Dimensions dialog that appears or click the **Auto-Size** button to automatically set the dimensions based on the coordinates of the objects currently included in the map.
3. Select the distance units to use for these coordinates.
4. If the Auto-Length option is in effect, check the "Re-compute all lengths and areas" box if you would like SWMM to re-calculate all conduit lengths and subcatchment areas under the new set of map dimensions.
5. Click the **OK** button to resize the map.

If you are going to use a backdrop image with the automatic distance and area calculation feature, then it is recommended that you set the map dimensions immediately after creating a new project. Map distance units can be different from conduit length units. The latter (feet or meters) depend on whether flow rates are expressed in US or metric units. SWMM will automatically convert units if necessary.

If you just want to re-compute conduit lengths and subcatchment areas without changing the map's dimensions, then just check the Re-compute Lengths and Areas box and leave the coordinate boxes as they are.

backdrop image might be a street map, utility map, topographic map, site development plan, or any other relevant picture or drawing. For example, using a street map would simplify the process of adding sewer lines to the project since one could essentially digitize the drainage system's nodes and links directly on top of it.



The Backdrop Image must be a Windows metafile, bitmap, PNG, or JPEG image created outside of SWMM. Once imported, its features cannot be edited, although its scale and viewing area will change as the map window is zoomed and panned. For this reason metafiles work better than bitmaps or JPEGs since they will not loose resolution when re-scaled. Most CAD and GIS programs have the ability to save their drawings and maps as metafiles.

Selecting **View** >> **Backdrop** from the Main Menu will display a sub-menu with the following commands: **Load** (loads a backdrop image file into the project) **Unload** (unloads the backdrop image from the project) **Align** (aligns the drainage system schematic with the backdrop) **Resize** (resizes the map dimensions of the backdrop) **Watermark** (toggles the backdrop image appearance between normal and lightened) The name of

Main Menu. A Backdrop Image Selector dialog form will be displayed. The entries on this form are as follows:

Backdrop Image File

Enter the name of the file that contains the image. You can click the  button to bring up a standard Windows file selection dialog from which you can search for the image file.

World Coordinates File

If a "world" file exists for the image, enter its name here, or click the  button to search for it. A world file contains geo-referencing information for the image and can be created from the software that produced the image file or by using a text editor. It contains six lines with the following information:

Line 1: real world width of a pixel in the horizontal direction.

Line 2: X rotation parameter (not used).

Line 3: Y rotation parameter (not used).

Line 4: negative of the real world height of a pixel in the vertical direction.

Line 5: real world X coordinate of the upper left corner of the image.

Line 6: real world Y coordinate of the upper left corner of the image.

If no world file is specified, then the backdrop will be scaled to fit into the center of the map display window.

Scale Map to Backdrop Image

This option is only available when a world file has been specified. Selecting it forces the dimensions of the Study Area Map to coincide with those of the backdrop image. In addition, all existing objects on the map will have their coordinates adjusted so that they appear within the new map dimensions yet maintain their relative positions to one another. Selecting this option may then require that the backdrop be re-aligned so that its position relative to the drainage area objects is correct.

Aligning a Backdrop Image

To align a backdrop image with the drainage system schematic on the Study Area Map:

1. Select **View** >> **Backdrop** >> **Align** from the Main Menu.
2. Move the backdrop image across the Study Area Map by moving the mouse with the left button held down until it lines up properly with the drainage system. Then release the button.

Resizing a Backdrop Image

To resize a backdrop image select **View >> Backdrop >> Resize** from the Main Menu. A Backdrop Dimensions dialog will appear that allows you to specify X,Y coordinates for the lower left and upper right corners of the backdrop image. In addition, you can specify that the backdrop be resized to fit the current dimensions of the Study Area Map or that the map have its dimensions changed to match those of the backdrop. Note that with the latter option all objects currently on the map will have their coordinates changed so that their position relative to the lower left corner of the map is maintained.

Measuring Distances

1. Click the button on the Map Toolbar.
2. Left-click on the map where you wish to begin measuring from.
3. Move the mouse over the distance being measured, left-clicking at each intermediate location where the measured path changes direction.
4. Right-click the mouse or press <Enter> to complete the measurement.
5. The distance measured in project units (feet or meters) will be displayed in a dialog box. If the last point on the measured path coincides with the first point then the area of the enclosed polygon will also be displayed.

Zooming the Map

1. Select **View >> Zoom In** from the Main Menu or click the button on the Map Toolbar.

area and click the left button.

3. To perform a custom zoom, move the mouse to the upper left corner of the zoom area and with the left button pressed down, draw a rectangular outline around the zoom area. Then release the left button.

To Zoom Out on the Study Area Map:

1. Select **View >> Zoom Out** from the Main Menu or click on the Map Toolbar.
2. The map will be returned to the view in effect at the previous zoom level.

You can also zoom in and out by rotating the mouse's scroll wheel.

Panning the Map

1. Select **View >> Pan** from the Main Menu or click the button on the Map Toolbar.

in the direction you wish to pan in.

3. Release the mouse button to complete the pan.

You can also pan by simply moving the mouse with its scroll wheel pressed down.

To pan using the Overview Map:

1. If not already visible, bring up the Overview Map by selecting **View >> Overview Map** from the Main Menu or click the button on the Main Toolbar.

2. If the Study Area Map has been zoomed in, an outline of the current viewing area will appear on the Overview Map. Position the mouse within this outline on the Overview Map.

3. With the left button held down, drag the outline to a new position.

4. Release the mouse button and the Study Area Map will be panned to an area corresponding to the outline on the Overview Map.

Viewing at Full Extent

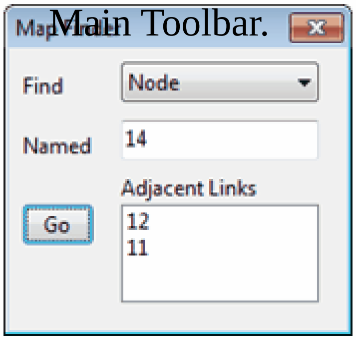
a) select **View** >> **Full Extent** from the Main Menu, or

b) click the button on the Map Toolbar.

Finding an Object

1. Select **View >> Find Object** from the Main Menu or click the on the Main Toolbar.

enter its name.



3. Click the **Go** button.

If the object exists, it will be highlighted on the map and in the Project Browser. If the map is currently zoomed in and the object falls outside the current map boundaries, the map will be panned so that the object comes into view.

User-assigned object names in SWMM are not case sensitive. E.g., NODE123 is equivalent to Node123.

After an object is found, the Map Finder dialog will also list:

- the outlet connections for a subcatchment
- the connecting links for a node
- the connecting nodes for a link.

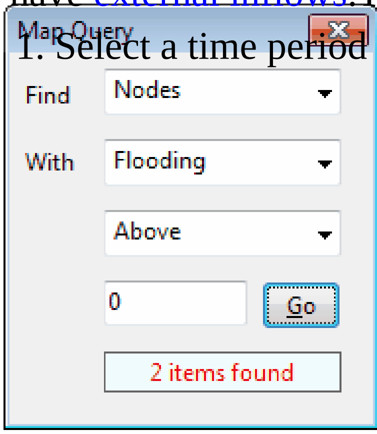
Submitting a Map Query

criterion (e.g., nodes that flood, links with velocity below 2 ft/sec, etc.). It can also identify which subcatchments have LID controls and which nodes have external inflows. To submit a map query:

1. Select a time period in which to query the map from the Map Browser.

Main Toolbar.

3. Fill in the following information in the Query dialog that appears:

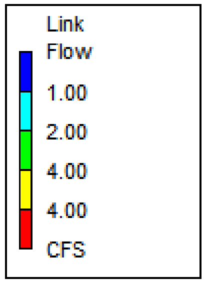


- Select whether to search for Subcatchments, Nodes, Links, LID Subcatchments, or Inflow Nodes.
- Select a parameter to query or the type of LID or inflow to locate.
- Select the appropriate operator: Above, Below, or Equals.
- Enter a value to compare against.

4. Click the **Go** button. The number of objects that meet the criterion will be displayed in the Query dialog and each such object will be highlighted on the Study Area Map.
5. As a new time period is selected in the Map Browser, the query results are automatically updated.
6. You can submit another query using the dialog box or close it by clicking the button in the upper right corner.

After the Query dialog is closed the map will revert back to its original display.

color with a range of values for the current theme being viewed. Separate legends exist for Subcatchments, Nodes, and Links. A Date/Time Legend is also available for displaying the date and clock time of the simulation period being viewed on the map.



To display or hide a map legend:

1. Select **View** >> **Legends** from the Main Menu or right click on the map and select Legends from the pop-up menu that appears.
2. Click on the type of legend whose display should be toggled on or off.

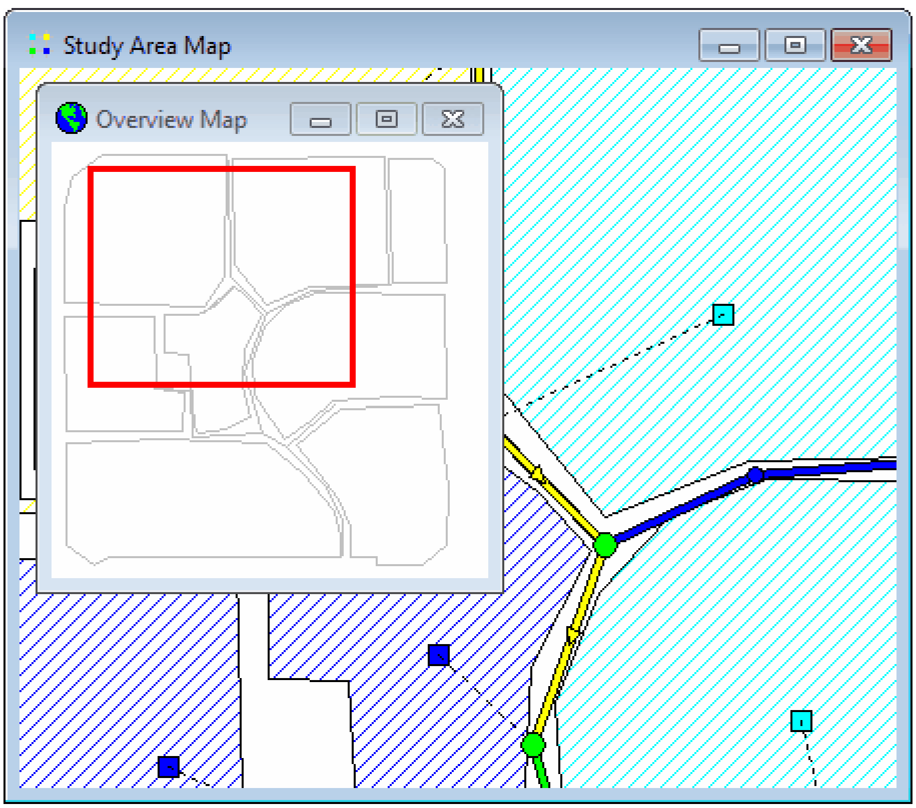
A visible legend can also be hidden by double clicking on it.

To move a legend to another location press the left mouse button over the legend, drag the legend to its new location with the button held down, and then release the button.

To edit a legend, either select **View** >> **Legends** >> **Modify** from the Main Menu or right click on the legend if it is visible. Then use the Legend Editor dialog that appears to modify the legend's colors and intervals.

Using the Overview Map

The **Overview Map**, as pictured below, allows one to see where in terms of the overall system the main Study Area Map is currently focused. The current zoom area is depicted by the rectangular outline displayed on the Overview Map. As you drag this rectangle to another position the view within the main map will be redrawn accordingly. The Overview Map can be toggled on and off by selecting **View >> Overview Map** from the Main Menu or by clicking the on the Main Toolbar. The Overview Map window can also be dragged to any position as well as be re-sized.



Setting Map Display Options

Map. There are several ways to invoke it:

- select **Tools >> Map Display Options** from the Main Menu or,
- click the Options button on the Main Toolbar when the Study Area Map window has the focus or,
- right click on any empty portion of the map and select **Options** from the popup menu that appears.

A Map Options dialog will appear where you can set various display options, such as subcatchment fill style, node and link size, flow direction arrows, and background color.

Exporting the Map

- Autodesk's DXF (Drawing Exchange Format) format,
- EPA SWMM's own ASCII text (.map) format.

The DXF format is readable by many Computer Aided Design (CAD) programs. Metafiles can be inserted into word processing documents and loaded into drawing programs for re-scaling and editing. Both formats are vector-based and will not lose resolution when they are displayed at different scales.

To export the map to a DXF, metafile, or text file:

1. Select **File >> Export >> Map** from the Main Menu.
2. In the Map Export dialog that appears select the format that you want the map saved in.
3. If you select DXF format, you have a choice of how nodes will be represented in the DXF file. They can be drawn as filled circles, as open circles, or as filled squares. Not all DXF readers can recognize the format used in the DXF file to draw a filled circle. Also note that map annotation, such as node and link ID labels, will not be exported, but map label objects will be.
4. After choosing a format, click **OK** and enter a name for the file in the Save As dialog that appears.

Running a Simulation

After a study area has been suitably described, its runoff, routing and water quality behavior can be simulated. This section describes how to specify options to be used in the analysis, how to run the simulation and how to troubleshoot common problems that may occur.

- Setting simulation options
- Initiating a run
- Troubleshooting results

Setting Simulation Options

stormwater drainage system is carried out. To set these options:

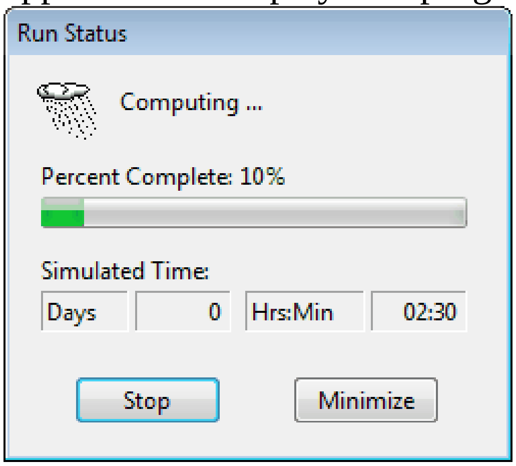
1. Select the **Options** category from the Project Browser.

- General Options
- Date Options
- Time Step Options
- Dynamic Wave Routing Options
- Interface File Options
- Reporting Options
- Event Options

3. Click the button to invoke the appropriate editor for the chosen option category (the Simulation Options dialog is used for the first 5 categories while the Reporting Options dialog and the Events editor are used for the last two, respectively).

Starting a Simulation

Main Menu or click on the Main Toolbar. A Run Status window will appear which displays the progress of the simulation.



To stop a run before its normal termination, click the **Stop** button on the Run Status window or press the <Esc> key. Simulation results up until the time when the run was stopped will be available for viewing. To minimize the SWMM program while a simulation is running, click the **Minimize** button on the Run Status window.

If the analysis runs successfully the icon will appear in the Run Status section of the Status Bar at the bottom of SWMM's main window. Any error or warning messages will appear in a Status Report window. If you modify the project after a successful run has been made, the status flag changes to indicating that the current computed results no longer apply to the modified project.

Troubleshooting Results

run was unsuccessful and direct the user to the Status Report for details. The Status Report will include an error statement, code, and description of the problem (e.g., ERROR 138: Node TG040 has initial depth greater than maximum depth).

- Drainage System Layout Errors
- Excessive Continuity Errors
- Unstable Flow Routing Results

Even if a run completes successfully, one should check to insure that the results are reasonable. The following are the most common reasons for a run to end prematurely or to contain questionable results:

- Unknown ID Errors

Unknown ID Errors

This message typically appears when an object references another object that was never defined. An example would be a subcatchment whose outlet was designated as *N29*, but no such subcatchment or node with that name exists. Similar situations can exist for incorrect references made to Curves, Time Series, Time Patterns, Aquifers, Snow Packs, Transects, Pollutants, and Land Uses.

File Errors

File errors can occur when:

- a file cannot be located on the user's computer
- a file being used has the wrong format
- a file to be written to cannot be opened because the user does not have write privileges for the directory (folder) where the file is to be stored.

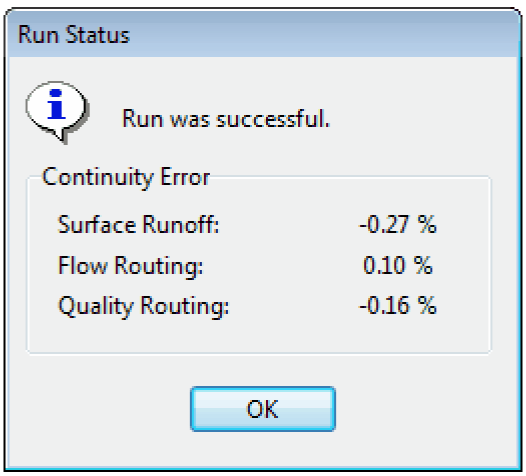
Drainage System Layout Errors

- An outfall node can have only one conduit link connected to it.
- A node can have no more than one dummy link connected to it.
- Under Kinematic Wave routing, a junction node can only have one outflow link and a regulator link cannot be the outflow link of a non-storage node.
- Under Dynamic Wave routing there must be at least one outfall node in the network.

An error message will be generated if any of these conditions are violated.

Excessive Continuity Errors

When a run completes successfully, the mass continuity errors for runoff, flow routing, and pollutant routing will be displayed in the **Run Status** window. These errors represent the percent difference between initial storage + total inflow and final storage + total outflow for the entire drainage system. If they exceed some reasonable level, such as 10 percent, then the validity of the analysis results must be questioned. The most common reasons for an excessive continuity error are computational time steps that are too long or conduits that are too short.



In addition to the system continuity error, the Status Report produced by a run will list those nodes of the drainage network that have the largest flow continuity errors. If the error for a node is excessive, then one should first consider if the node in question is of importance to the purpose of the simulation. If it is, then further study is warranted to determine how the error might be reduced.

Unstable Flow Routing Results

routing (and to a lesser extent, Kinematic Wave routing), the flows in some links or water depths at some nodes may fluctuate or oscillate significantly at certain periods of time as a result of numerical instabilities produced by the solution method. SWMM does not automatically identify when such conditions exist, so it is up to the user to verify the numerical stability of the model and to determine if the simulation results are valid for the modeling objectives. Time series plots at key locations in the network can help identify such situations as can a scatter plot between a link's flow and the corresponding water depth at its upstream node. (See Viewing Results with a Graph).

Numerical instabilities can occur over short durations and may not be apparent when time series are plotted with a long time interval. When detecting such instabilities, it is recommended that a reporting time step of 1 minute or less be used, at least for an initial screening of results.

The run's Status Report will list the links having the five highest values of a Flow Instability Index (FII). This index counts the number of times that the flow value in a link is higher (or lower) than the flow in both the previous and subsequent time periods. The index is normalized with respect to the expected number of such 'turns' that would occur for a purely random series of values and can range from 0 to 150. Flow time series plots for the links having the highest FII's should be inspected to insure that flow routing results are acceptably stable.

Numerical instabilities under Dynamic Wave flow routing can be reduced by:

- reducing the routing time step
- utilizing the variable time step option with a smaller time step factor
- selecting to ignore the inertial terms of the momentum equation
- selecting the option to lengthen short conduits.

Viewing Simulation Results

can be viewed. These include a status report, a summary report, various map views, graphs, tables, and a statistical frequency report.

Viewing a status report

Viewing a summary report

Variables that can be viewed

Viewing results on the map

Viewing results with a graph

Viewing results with a table

Viewing a statistics report

Viewing a Status Report

information on the following:

- Rain File Summary
- Analysis Options
- Error Messages
- Control Actions (if requested in the Reporting Options)
- Continuity Errors
- Stability Results
- Time Step Variation

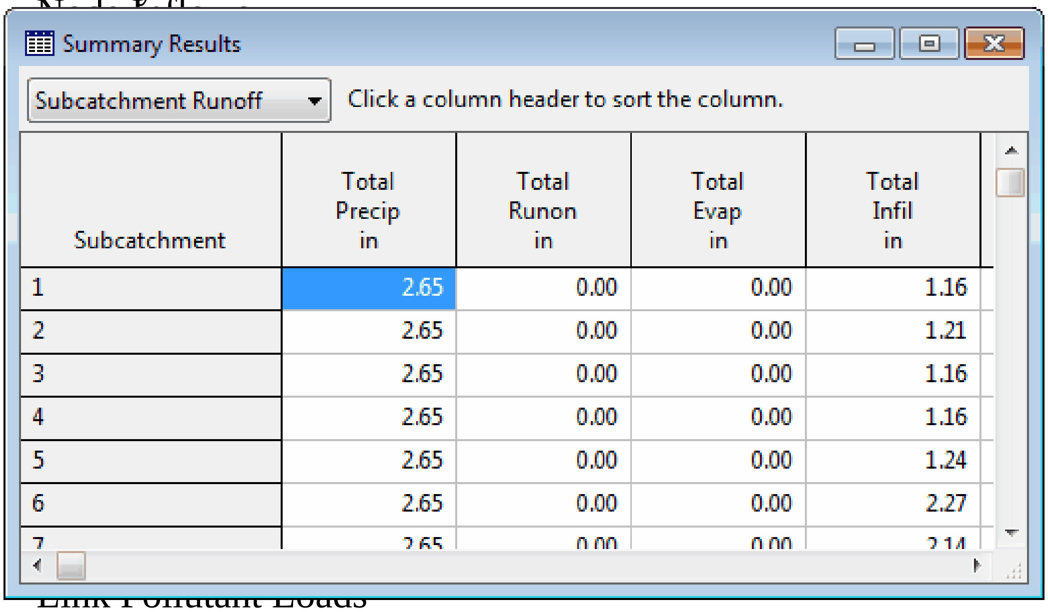
To view the Status Report select **Report >> Status** from the Main Menu or click the button on the Main Toolbar and select **Status Report** from the drop-down menu that appears.

To copy selected text from the Status Report to a file or to the Windows Clipboard, first select the text to copy with the mouse and then choose **Edit >> Copy To** from the Main Menu (or press the button on the Main Toolbar).

To save both the entire Status Report and Summary Report to file, select **File >> Export >> Status/Summary Report** from the Main Menu.

Viewing Summary Results

subcatchment, node, and link in the project through a selectable list of tables. To view the various summary results tables, select **Report >> Summary** from the Main Menu or click the button on the Main Toolbar and select **Summary Results** from the drop-down menu that appears. The Summary Results window looks as follows:



The drop-down box at the upper left allows you to choose the type of results to view. The choices are:

- Groundwater Summary
- Subcatchment Washoff
- Node Depths
- Pumping Summary
- Subcatchment Runoff

The summary results displayed in these tables are based on results found at every computational time step and not just on the results from each reporting time step.

Only summary categories relevant to the particular project and its results will be listed (e.g., Node Flooding and Pumping Summary will not appear if the run has no flooded nodes and the project has no pumps, respectively). Clicking on the name of an object in the first column of the table will locate that object both in the Project Browser and on the Study Area Map. Clicking on a column heading will sort the entries in the table by the values in that column (alternating between ascending and descending order with each click. Selecting **Edit >> Copy To** from the Main Menu or clicking on the Main Toolbar will allow you to copy the contents of the table to either the Windows Clipboard or to a file. To save both the entire Status Report and all tables of the Summary Report to a file, select **File >> Export >> Status/Summary Report** from the Main Menu.

Variables That Can be Viewed

Aside from the results presented in the Status and Summary Reports, computed results for individual subcatchments, nodes, and links can be viewed in several different formats. These variables include:

- Time series variables of values saved at every reporting time step.
- Statistical summary variables derived from the time series results (i.e., a subset of variables contained in the Summary Report).

Time Series Variables

the map and can be plotted, tabulated, and statistically analyzed:

Subcatchment Variables

- rainfall rate (in/hr or mm/hr)
- snow depth (inches or millimeters)
- evaporation loss (in/day or mm/day)
- infiltration loss (in/hr or mm/hr)
- runoff flow (flow units)
- groundwater flow into the drainage network (flow units)
- groundwater elevation (ft or m)
- soil moisture in the unsaturated groundwater zone (volume fraction)
- washoff concentration of each pollutant (mass/liter)

Link Variables

- flow rate (flow units)
- average water depth (ft or m)
- flow velocity (ft/sec or m/sec)
- volume of water (ft3 or m3)
- capacity (fraction of full area filled by flow for conduits; control setting for pumps and regulators)
- concentration of each pollutant (mass/liter)

Node Variables

- water depth (ft or m above the node invert elevation)
- hydraulic head (ft or m, absolute elevation per vertical datum)
- stored water volume (including ponded water, ft3 or m3)

System-Wide Variables

- air temperature (degrees F or C)
- potential evaporation (in/day or mm/day)
- actual evaporation (in/day or mm/day)
- total rainfall (in/hr or mm/hr)

that were selected to have detailed time series results saved for them. This normally includes all such objects in the project unless the Reporting option (under the Options category in the Project Browser) was used to select specific objects to report on. For nodes and links, results are normally point values computed (or interpolated) at the end of each reporting time step. They can also be the average of all values computed during a reporting time step if **Report Average Results** is selected as a Reporting option.

each object on the Study Area Map:

- **Subcatchment Variables**
  - o total evaporation (inches or millimeters)
  - o total infiltration (inches or millimeters)
  - o total runoff (inches or millimeters)
  - o peak runoff (in/hr or mm/hr)
  - o runoff coefficient
- **Node Variables**
  - o maximum water depth (ft or m)
  - o maximum hydraulic head (ft or m)
  - o maximum lateral inflow (flow units)
  - o total lateral inflow (million gallons or million liters)
  - o maximum flooding rate (flow units)
  - o total flooded volume (million gallons or million liters)
  - o hours flooded
- **Link Variables**
  - o maximum flow rate (flow units)
  - o maximum velocity (ft/sec or m/sec)
  - o maximum degree full
  - o hours flowing full
  - o hours capacity limited

As with time series results, these summary variables can be viewed only for those subcatchments, nodes, and links that were selected to have detailed time series results saved for them.

Viewing Results on the Map

simulation results directly on the Study Area Map:

- For the current settings on the Map Browser, the subcatchments, nodes and links of the map will be colored according to their respective Map Legends. The map's color coding will be updated as a new time period is selected in the Map Browser.

the mouse over any map object will display its ID name and the value of its current theme parameter in a hint-style box.

- ID names and parameter values can be displayed next to all subcatchments, nodes and/or links by selecting the appropriate options on the Annotation page of the Map Options dialog.

- Subcatchments, nodes or links meeting a specific criterion can be identified by submitting a Map Query.

- One can animate the display of results on the network map either forward or backward in time by using the controls on the Animator panel of the Map Browser.

- The map can be printed, copied to the Windows clipboard, or saved as a DXF file or Windows metafile.

Viewing Results with a Graph

Graphs can be printed, copied to the Windows clipboard, or saved to a text file or to a Windows metafile. The following types of graphs can be created from available simulation results:

- Time Series Plot
- Profile Plot
- Scatter Plot

You can zoom in or out of any graph by holding down the <Shift> key while drawing a zoom rectangle with the mouse's left button held down. Drawing the rectangle from left to right zooms in, drawing from right to left zooms out. The plot can also be panned in any direction by moving the mouse across the plot with the left button held down.

An opened graph will normally be redrawn when a new simulation is run. To prevent the automatic updating of a graph once a new set of results is computed you can lock the current graph by clicking the icon in the upper left corner of the graph. To unlock the graph, click the icon again.

See Also

Copying the Current View
Printing the Current View

Time Series Plots

objects and variables. Up to six time series can be plotted on the same graph. When only a single time series is plotted, and that item has calibration data registered for the plotted variable, then the calibration data will be plotted along with the simulated results.

on the Main Toolbar, or simply right-click on the graph.

3. Use the Graph Options dialog that appears to customize the plot's appearance.

To create a Time Series Plot:

1. Select **Report >> Graph >> Time Series** from the Main Menu or click the on the Main Toolbar.
2. A Time Series Plot dialog will appear. Use it to describe what objects and variables should be plotted.

To customize the appearance of a time series plot:

1. Make the plot the active window (click on its title bar).

over a connected path of drainage system links and nodes at a particular point in time. Once the plot has been created it will be automatically updated as a new time period is selected using the Map Browser.

To create a Profile Plot:

1. Select **Report >> Graph >> Profile** from the Main Menu or press on the Main Toolbar.
2. A Profile Plot dialog will appear. Use it to identify the path along which the profile plot is to be drawn.

Main Toolbar, or simply right-click on the graph.

3. Use the Profile Plot Options dialog that appears to customize the plot's appearance.

Profile plots can also be created before any simulation results are available, to help visualize and verify the vertical layout of a drainage system. Plots created in this manner will contain a refresh button in the upper left corner that can be used to redraw the plot after edits are made to any elevation data appearing in the plot.

To customize the appearance of a profile plot:

1. Make the plot the active window (click on its title bar).

Scatter Plots

flow rate in a pipe versus water depth at a node

Main Toolbar, or simply right-click on the graph.

3. Use the Graph Options dialog that appears to customize the plot's appearance.

To create a Scatter Plot:

1. Select **Report >> Graph >> Scatter** from the Main Menu or press on the Main Toolbar.
2. Specify what time interval and what pair of objects and their variables to plot using the Scatter Plot dialog that appears.

To customize the appearance of a scatter plot:

1. Make the plot the active window (click on its title bar).

Viewing Results with a Table

tabular format. There are two types of formats available:

several objects of the same type (e.g., runoff for a group of subcatchments)

- **Table by Object** - tabulates the time series of several variables for a single object (e.g., flow and water depth for a conduit).

To create a tabular report:

1. Select **Report** >> **Table** from the Main Menu or click on the Main Toolbar.
2. Choose the table format (either **By Object** or **By Variable**) from the sub-menu that appears.
3. Fill in the Table by Object or Table by Variable dialogs to specify what information the table should contain.

results. For a given object and variable this report will do the following:

- segregate the simulation period into a sequence of non-overlapping events, either by day, month, or by flow (or volume) above some minimum threshold value,

maximum, or total sum of the variable over the event's time period),

- compute summary statistics for the entire set of event values (mean, standard deviation and skewness),
- perform a frequency analysis on the set of event values.

The frequency analysis of event values will determine the frequency at which a particular event value has occurred and will also estimate a return period for each event value. Statistical analyses of this nature are most suitable for long-term continuous simulation runs.

To generate a Statistics Report:

1. Select **Report >> Statistics** from the Main Menu or click the Σ on the Main Toolbar.
2. Fill in the Statistics Selection dialog that appears, specifying the object, variable, and event definition to be analyzed.

The Statistics Report consists of four tabbed pages that contain:

- a table of event summary statistics
- a table of rank-ordered event periods, including their date, duration, and magnitude
- a histogram plot of the chosen event statistic
- an exceedance frequency plot of the event values.

The exceedance frequencies included in the Statistics Report are computed with respect to the number of events that occur, not the total number of reporting periods.

Printing and Copying

to file the contents of the currently active window in the SWMM workspace. This can include the Study Area Map, a graph, a table, or a report.

- Selecting a printer
- Setting the page format
- Previewing the page
- Printing the current view
- Copying the current view

Selecting a Printer

properties:

1. Select **File >> Page Setup** from the Main Menu.

3. Select a printer from the choices available in the combo box in the Print Setup dialog that appears.

4. Click the **Properties** button to select the appropriate printer properties (which vary with choice of printer).

5. Click **OK** on each dialog to accept your selections.

Setting the Page Format

1. Select **File** >> **Page Setup** from the Main Menu.
2. Use the **Margins** page of the Page Setup dialog box that appears to:
   - Select a printer.
   - Select the paper orientation (Portrait or Landscape).
   - Set left, right, top, and bottom margins.
3. Use the **Headers/Footers** page of the dialog box to:
   - Supply the text for a header that will appear on each page.
   - Indicate whether the header should be printed or not and how its text should be aligned.
   - Supply the text for a footer that will appear on each page.
   - Indicate whether the footer should be printed or not and how its text should be aligned.
   - Indicate whether pages should be numbered.
4. Click **OK** to accept your choices.

Previewing the Page

To preview a printout, select **File >> Print Preview** from the Main Menu. A Preview form will appear which shows how each page being printed will appear. While in preview mode, the left mouse button will re-center and zoom in on the image and the right mouse button will re-center and zoom out.

Printing the Current View

workspace, either select **File** >> **Print** from the Main Menu or click on the Main Toolbar. The following views can be printed:

- Study Area Map (at the current zoom level)
- Summary Report (for the current table being viewed)
- Graphs (Time Series, Profile, and Scatter plots)
- Tabular Reports
- Statistical Reports.

Copying the Current View

to the Windows clipboard or to a file. Views that can be copied in this fashion include the Study Area Map, summary report tables, graphs, time series tables, and statistical reports.

To copy the current view to the clipboard or to file:

1. If the current view is a time series table, select the cells of the table to copy by dragging the mouse over them or copy the entire table by selecting **Edit >> Select All** from the Main Menu.

Main Toolbar.

3. Select choices from the Copy dialog that appears and click the **OK** button.

4. If copying to file, enter the name of the file in the Save As dialog that appears and click **OK**.

include:

- Project File
- Report and Output Files
- Rainfall Files
- Climate File
- Calibration Data Files
- Time Series Files
- Interface Files

The only file required to run SWMM is the project file; the others are optional.

Project File

A SWMM **Project File** is a plain text file that contains all of the data used to describe a study area and the options used to analyze it. The file is organized into sections, where each section generally corresponds to a particular category of object used by SWMM. The contents of the file can be viewed from within SWMM while it is open by selecting **Project** >> **Details** from the Main Menu. An existing project file can be opened by selecting **File** >> **Open** from the Main Menu and be saved by selecting **File** >> **Save** (or **File** >> **Save As**).

Normally a SWMM user would not edit the project file directly, since SWMM's graphical user interface can add, delete, or modify a project's data and control settings. However, for large projects where data currently reside in other electronic formats, such as CAD or GIS files, it may be more expeditious to extract data from these sources and save it to a formatted Project file before running SWMM. The format of the project file is described in detail in Appendix D of the SWMM 5.1 Users Manual.

After a project file is saved to disk, a settings file will automatically be saved with it. This file has the same name as the project file except that its extension is **.ini** (e.g., if the project file were named **project1.inp** then its settings file would have the name **project1.ini**). It contains various settings used by SWMM's graphical user interface, such as map display options, legend colors and intervals, object default values, and calibration file information. Users should not edit this file. A SWMM project will still load and run even if the settings file is missing.

contains the contents of both the Status Report and all of the tables included in the Summary Results report. Refer to the two aforementioned topics to review its content.

The **Output File** is a binary file that contains the numerical results from a successful SWMM run. This file is used by SWMM's user interface to interactively create time series plots and tables, profile plots, and statistical analyses of a simulation's results.

Whenever a successfully run project is either saved or closed, the report and output files are saved with the same name as the project file, but with extensions of **.rpt** and **.out**. This will happen automatically if the program preference *Prompt to Save Results* is turned off. Otherwise you will be asked if the current results should be saved or not. If results are saved then the next time the project is opened, the results from these files will automatically be available for viewing.

Rainfall Files. The program currently recognizes the following formats for storing such data:

- Hourly and fifteen minute precipitation data from over 5,500 reporting stations retrieved using NOAA's National Centers for Environmental Information (NCEI) Climate Data Online service (www.ncdc.noaa.gov/cdo-web) (space delimited format only).
- The older DS-3240 and related formats used for hourly precipitation by NCEI.
- The older DS-3260 and related formats used for fifteen minute precipitation by NCEI.
- HLY03 and HLY21 formats for hourly rainfall at Canadian stations, available from Environment Canada at www.climate.weather.gc.ca.
- FIF21 format for fifteen minute rainfall at Canadian stations, also available from Environment Canada.
- a standard user-prepared format where each line of the file contains the station ID, year, month, day, hour, minute, and non-zero precipitation reading, all separated by one or more spaces.

When requesting data from NCEIs online service, be sure to specify the TEXT format option, make sure that the data flags are included, and, for 15-minute data, select the QPCP option and not the QGAG one.

An excerpt from the user-prepared format might look as follows:

```
STA01 2004 6 12 00 00 0.12
STA01 2004 6 12 01 00 0.04
STA01 2004 6 22 16 00 0.07
```

This format can also accept multiple stations within the same file.

When a rain gage is designated as receiving its rainfall data from a file, the user must supply the name of the file and the name of the recording station referenced in the file. For the standard user-prepared format, the rainfall type (e.g., intensity or volume), recording time interval, and depth units must also be supplied as rain gage properties. For the other file types these temperature, evaporation, and wind speed data. The program currently recognizes the following formats:

- Global Historical Climatology Network - Daily (GHCN-D) files (TEXT output format) available from NOAA's National Centers for Environmental Information (NCEI) Climate Data Online service at www.ncdc.noaa.gov/cdo-web.
- Older NCEI DS3200 or DS3210 files.
- Canadian climate files available from Environment Canada at www.climate.weather.gc.ca.
- A user-prepared climate file where each line contains a recording station name, the year, month, day, maximum temperature, minimum temperature, and optionally, evaporation rate, and wind speed. If no data are available for any of these items on a given date, then an asterisk should be entered as its value.

When a climate file has days with missing values, SWMM will use the value from the most recent previous day with a recorded value.

For a user-prepared climate file, the data must be in the same units as the project being analyzed. For US units, temperature is in degrees F, evaporation is in inches/day, and wind speed is in miles/hour. For metric units, temperature is in degrees C, evaporation is in mm/day, and wind speed is in km/hour.

Calibration Files

locations that can be compared with simulated values in Time Series Plots.

· Subcatchment Groundwater Elevation

Separate files can be used for each of the following:

· Subcatchment Snow Pack Depth

· Subcatchment Runoff

· Subcatchment Pollutant Washoff

· Node Depth

· Node Lateral Inflow

· Node Flooding

· Node Water Quality

· Link Flow

· Link Velocity

· Link Depth

Calibration files are registered to a project by selecting **Project >> Calibration Data** from the Main Menu (see Registering Calibration Data).

The format of the file is as follows:

1. The name of the first object with calibration data is entered on a single line.
2. Subsequent lines contain the following recorded measurements for the object:
   - measurement date (month/day/year, e.g., 6/21/2004) or number of whole days since the start of the simulation
   - measurement time (hours:minutes) on the measurement date or relative to the number of elapsed days
   - measurement value (for pollutants, a value is required for each pollutant).
3. Follow the same sequence for any additional objects.

An excerpt from an example calibration file is shown below. It contains flow values for two conduits: 1030 and 1602.Note that a semicolon can be used to begin a comment. In this example, elapsed time rather than the actual measurement date was used.

```
;Flows for Selected Conduits
;Conduit Days Time Flow ;----
-------------------------
```

1030

0 0:15 0
0 0:30 0
0 0:45 23.88
0 1:00 94.58
0 1:15 115.37

1602

```
0 0:15 5.76
0 0:30 38.51
0 1:00 67.93
0 1:15 68.01
```

series objects. Examples of time series data include rainfall, evaporation, inflows to nodes of the drainage system, and water stage at outfall boundary nodes. The file must be created and edited outside of SWMM, using a text editor or spreadsheet program. A time series file can be linked to a specific time series object using SWMM's Time Series Editor.

The format of a time series file consists of one time series value per line. Comment lines can be inserted anywhere in the file as long as they begin with a semicolon. Blank lines are also permitted. Time series values can either be in date / time / value format or in time / value format, where each entry is separated by one or more spaces or tab characters. For the date / time / value format, dates are entered as month/day/year (e.g., 7/21/2004) and times as 24-hour military time (e.g., 8:30 pm is 20:30). After the first date, additional dates need only be entered whenever a new day occurs. For the time / value format, time can either be decimal hours or military time since the start of a simulation (e.g., 2 days, 4 hours and 20 minutes can be entered as either 52.333 or 52:20). An example of a time series file is shown below:

```
;Rainfall Data for Gage G1
07/01/2003 00:00 0.00000
00:15 0.03200
00:30 0.04800
00:45 0.02400
01:00 0.0100
07/06/2003 14:30 0.05100
14:45 0.04800
15:00 0.03000
18:15 0.01000
18:30 0.00800
```

Interface Files

externally imposed inputs (e.g., rainfall or inflow/infiltration hydrographs) or the results of previously run analyses (e.g., runoff or routing results). These files can help speed up simulations, simplify comparisons of different loading scenarios, and allow large study areas to be broken up into smaller areas that can be analyzed individually. The different types of interface files that are currently available include:

- Hot Start File
- RDII Interface File
- Routing Interface Files
- Rainfall Interface File

Consult the Setting Simulation Options topic for instructions on how to specify interface files for use as input and/or output in a simulation.

by SWMM that can be saved and reused from one analysis to the next.

The rainfall interface file collates a series of separate rain gage files into a single rainfall data file. Normally a temporary file of this type is created for every SWMM analysis that uses external rainfall data files and is then deleted after the analysis is completed. However, if the same rainfall data are being used with many different analyses, requesting SWMM to save the rainfall interface file after the first run and then reusing this file in subsequent runs can save computation time.

The rainfall interface file should not be confused with a rainfall data file. The latter is an external text file that provides rainfall time series data for a single rain gage. The former is a binary file created internally by SWMM that processes all of the rainfall data files used by a project.

The runoff interface file can be used to save the runoff results generated from a simulation run. If runoff is not affected in future runs, the user can request that SWMM use this interface file to supply runoff results without having to repeat the runoff calculations again.

hydrologic, hydraulic and water quality state of the study area at the end of a run. The file saved after a run can be used to define the initial conditions for a subsequent run, producing the same results as if one continuous run were made.

Hot start files can be used to avoid the initial numerical instabilities that sometimes occur under Dynamic Wave routing. For this purpose they are typically generated by imposing a constant set of base flows (for a natural channel network) or set of dry weather sanitary flows (for a sewer network) over some startup period of time. The resulting hot start file from this run is then used to initialize a subsequent run where the inflows of real interest are imposed.

It is also possible to both use and save a hot start file in a single run, starting off the run with one file and saving the ending results to another. The resulting file can then serve as the initial conditions for a subsequent run if need be. This technique can be used to divide up extremely long continuous simulations into more manageable pieces.

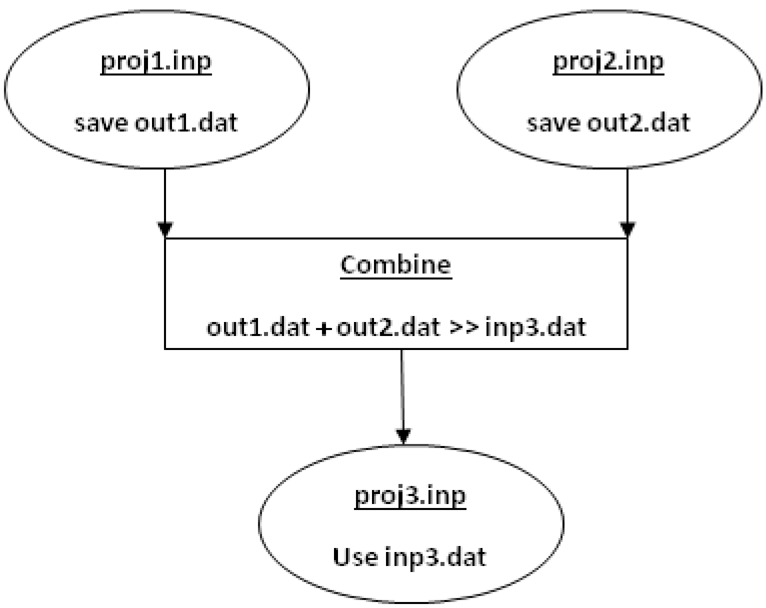
Instructions to save and/or use a hot start file can be issued when editing the Interface Files options available in the Project Browser. One can also save the results of the current run to an abridged hot start file by selecting **File >> Export >> Hot Start File** from the Main Menu (not all state variables are available to be saved after a run has been made).

RDII Files

The **RDII Interface File** contains a time series of rainfall-dependent infiltration/inflow flows for a specified set of drainage system nodes. This file can be generated from a previous SWMM run when Unit Hydrographs and nodal RDII inflow data have been defined for the project, or it can be created outside of SWMM using some other source of RDII data (e.g., through measurements or output from a different computer program). RDII files generated by SWMM are saved in a binary format. RDII files created outside of SWMM are text files with the same format used for a Routing Interface File, where Flow is the only variable contained in the file.

Routing Files

A **Routing Interface File** stores a time series of flows and pollutant concentrations that are discharged from the outfall nodes of drainage system model. This file can serve as the source of inflow to another drainage system model that is connected at the outfalls of the first model. A Combine utility is available on the File menu that will combine pairs of routing interface files into a single interface file. This allows very large systems to be broken into smaller sub-systems that can be analyzed separately and linked together through the routing interface file. The figure below illustrates this concept.



A single SWMM run can utilize an outflows routing file to save results generated at a system's outfalls, an inflows routing file to supply hydrograph and pollutograph inflows at selected nodes, or both.

RDII / Routing File Format

- the first line contains the keyword "SWMM5" (without the quotes)
- the time step used for all inflow records (integer seconds)
- the number of variables stored in the file, where the first variable must always be flow rate
- the name and units of each variable (one per line), where flow rate is the first variable listed and is always named FLOW
- the number of nodes with recorded inflow data
- the name of each node (one per line)
- a line of text that provides column headings for the data to follow (can be blank)
- for each node at each time step, a line with:

  a) the name of the node,

  b) the date (year, month, and day separated by spaces),

  c) the time of day (hours, minutes, and seconds separated by spaces),

  d) the flow rate followed by the concentration of each quality constituent.

Time periods with no values at any node can be skipped. An excerpt from an RDII / Routing interface file is shown below.

```
SWMM5
Example File 300
```

1

FLOW CFS

2
N1

N2

Node Year Mon Day Hr Min Sec
Flow N1 2002 04 01 00 20 00
0.000000
N2 2002 04 01 00 20 00 0.002549
N1 2002 04 01 00 25 00 0.000000
N2 2002 04 01 00 25 00 0.002549

Using Add-In Tools

user interface that can extend its capabilities. This section describes how such tools can be registered and share data with SWMM 5.

- What are add-in tools
- Configuring add-in tools

What are Add-In Tools

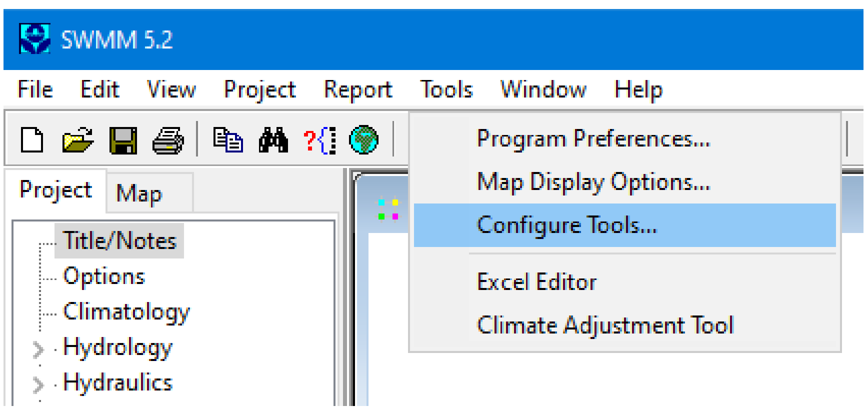
menu of SWMM's Main Menu and be launched while SWMM is still running. SWMM can interact with these applications to a limited degree by exchanging data through its pre-defined files or through the Windows clipboard. Add-in tools can provide additional modeling capabilities to what SWMM already offers. Some examples of useful add-ins might include:

- a tool that performs a statistical analysis of long-term rainfall data prior to adding it to a SWMM rain gage,

SWMM data set,

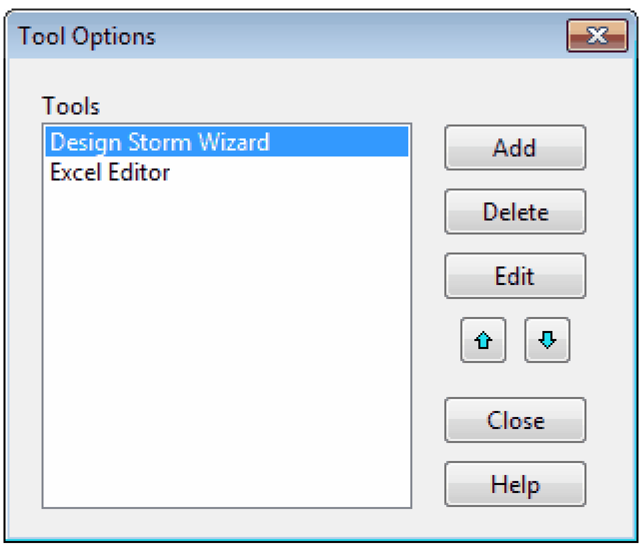
- a unit hydrograph estimator program that would derive the R-T-K parameters for a set of RDII unit hydrographs which could then be copied and pasted directly into SWMM's Unit Hydrograph Editor,
- a post-processor program that uses SWMM's hydraulic results to compute suspended solids removal through a storage unit,
- a third-party dynamic flow routing program used as a substitute for SWMM's own internal procedure.

The figure below shows what the **Tools** menu might look like after several add-in tools have been registered with it. The **Configure Tools** option is used to add, delete, or modify add-in tools. The options below this are the individual tools that have been made available (by this particular user) and can be launched by selecting them from the menu.



Configuring Add-In Tools

To configure one's personal collection of add-in tools, select **Configure Tools** from the **Tools** menu. This will bring up the Tool Options dialog as shown below. The dialog lists the currently available tools and has command buttons for adding a new tool and for deleting or editing an existing tool. The up and down arrow buttons are used to change the order in which the registered tools are listed on the **Tools** menu.



Whenever the **Add** or **Edit** button is clicked on this dialog a Tool Properties dialog will appear which is used to describe the properties of the new tool being added or the existing tool being edited.

US Customary Units

Area (Storage Unit)	square feet
Area (Ponding)	square feet
Capillary Suction	inches
Concentration	milligrams / liter (mg/L) micrograms / liter (ug/L) counts / liter (#/L)
Decay Constant (Infiltration)	1 / hours
Decay Constant (Pollutants)	1 / days
Depression Storage	inches
Depth	feet
Diameter	feet
Discharge Coefficient Orifice Weir	 dimensionless cubic feet / second / feetn (CFS/ft^n)
Elevation	feet
Evaporation	inches / day
Flow	cubic feet /second (CFS) gallons / minute (GPM) million gallons / day (MGD)
Head	feet
Hydraulic Conductivity	inches / hour
Infiltration Rate	inches / hour
Length	feet
Manning's n	seconds / $meter^{1/3}$
Pollutant Buildup	mass / acre mass / length

Rainfall Intensity	inches / hour

SI Metric Units

Area (Storage Unit)	square meters
Area (Ponding)	square meters
Capillary Suction	millimeters
Concentration	milligrams / liter (mg/L) micrograms / liter (ug/L) Counts / liter (#/L)
Decay Constant (Infiltration)	1 / hours
Decay Constant (Pollutants)	1 / days
Depression Storage	millimeters
Depth	meters
Diameter	meters
Discharge Coefficient Orifice Weir	 dimensionless cubic meters / second / meter^n (CMS/meter^n)
Elevation	meters
Evaporation	millimeters / day
Flow	cubic meters / second (CMS) liters per second (LPS) million liters / day (MLD)
Head	meters
Hydraulic Conductivity	millimeters / hour
Infiltration Rate	millimeters / hour
Length	meters
Manning's n	seconds / meter$^{1/3}$
Pollutant Buildup	mass / hectare

	mass / length

Soil Characteristics

Class					
Sand	4.74	1.93	0.437	0.062	0.024
Loamy Sand	1.18	2.40	0.437	0.105	0.047
Sandy Loam	0.43	4.33	0.453	0.190	0.085
Loam	0.13	3.50	0.463	0.232	0.116
Silt Loam	0.26	6.69	0.501	0.284	0.135
Sandy Clay Loam	0.06	8.66	0.398	0.244	0.136
Clay Loam	0.04	8.27	0.464	0.310	0.187
Silty Clay Loam	0.04	10.63	0.471	0.342	0.210
Sandy Clay	0.02	9.45	0.430	0.321	0.221
Silty Clay	0.02	11.42	0.479	0.371	0.251

SCS Curve Numbers

	Hydrologic Soil Group			
Land Use Description	**A**	**B**	**C**	**D**
Cultivated land				
Without conservation treatment	**72**	**81**	**88**	**91**
With conservation treatment	**62**	**71**	**78**	**81**
Pasture or range land				
Poor condition	**68**	**79**	**86**	**89**
Good condition	**39**	**61**	**74**	**80**
Meadow				
Good condition	**30**	**58**	**71**	**78**
Wood or forest land				
Thin stand, poor cover, no mulch	**45**	**66**	**77**	**83**
Good cover[2]	**25**	**55**	**70**	**77**
Open spaces, lawns, parks, golf courses, cemeteries, etc.				

Good condition: grass cover				

Soil Group Definitions

Source: *Hydrology National Engineering Handbook, Chapter 7*, Natural Resources Conservation Service, U.S. Department of Agriculture, January 2009.

Group	Meaning	Saturated Conductivity (in/hr)
A	Low runoff potential. Water is transmitted freely through the soil. Group A soils typically have less than 10 percent clay and more than 90 percent sand or gravel and have gravel or sand textures.	> 1.42
B	Moderately low runoff potential. Water transmission through the soil is unimpeded. Group B soils typically have between 10 percent and 20 percent clay and 50 percent to 90 percent sand and have loamy sand or sandy loam textures.	0.57 - 1.42
C	Moderately high runoff potential. Water transmission through the soil is somewhat restricted. Group C soils typically have between 20 percent and 40 percent clay and less than 50 percent sand and have loam, silt loam, sandy clay loam, clay loam, and silty clay loam textures.	0.06 - 0.57
D	High runoff potential. Water movement through the soil is restricted or very restricted. Group D soils typically have greater than 40 percent clay, less than 50 percent sand, and have clayey textures.	< 0.06

Depression Storage

Impervious surfaces	0.05 - 0.10 inches
Lawns	0.10 - 0.20 inches
Pasture	0.20 inches
Forest litter	0.30 inches

(Source: ASCE,(1992), Design & Construction of Urban Stormwater Management Systems, New York, NY)

Manning's n - Overland Flow

	n
Smooth asphalt	**0.011**
Smooth concrete	**0.012**
Ordinary concrete lining	**0.013**
Good wood	**0.014**
Brick with cement mortar	**0.014**
Vitrified clay	**0.015**
Cast iron	**0.015**
Corrugated metal pipes	**0.024**
Cement rubble surface	**0.024**
Fallow soils (no residue)	**0.05**

Manning's n - Closed Conduits

	n
Asbestos-cement pipe	0.011 - 0.015
Brick	0.013 - 0.017
Cast iron pipe	
- Cement-lined & seal coated	0.011 - 0.015
Concrete (monolithic)	
- Smooth forms	0.012 - 0.014
- Rough forms	0.015 - 0.017
Concrete pipe	0.011 - 0.015
Corrugated-metal pipe (1/2-in. x 2-2/3-in. corrugations)	
- Plain	0.022 - 0.026
- Paved invert	0.018 - 0.022
- Spun asphalt lined	0.011 - 0.015
Plastic pipe (smooth)	0.011 - 0.015
Vitrified clay	
- Pipes	0.011 - 0.015
- Liner plates	0.013 - 0.017

Source: ASCE (1982). *Gravity Sanitary Sewer Design and Construction*, ASCE Manual of Practice No. 60, New York, NY.

Manning's n - Open Channels

	n
Lined Channels	
- Asphalt	0.013 - 0.017
- Brick	0.012 - 0.018
- Concrete	0.011 - 0.020
- Rubble or riprap	0.020 - 0.035
- Vegetal	0.030 - 0.40
Excavated or dredged	
- Earth, straight and uniform	0.020 - 0.030
- Earth, winding, fairly uniform	0.025 - 0.040
- Rock	0.030 - 0.045
- Unmaintained	0.050 - 0.140
Natural channels (minor streams, top width at flood stage < 100 ft)	
- Fairly regular section	0.030 - 0.070
- Irregular section with pools	0.040 - 0.100

Source: ASCE (1982). *Gravity Sanitary Sewer Design and Construction*, ASCE Manual of Practice No. 60, New York, NY.

Water Quality Characteristics of Urban Runoff

Constituent	Concentrations
TSS (mg/L)	**180 - 548**
BOD (mg/L)	**12 - 19**
COD (mg/L)	**82 - 178**
Total P (mg/L)	0.42 - 0.88
Soluble P (mg/L)	0.15 - 0.28
TKN (mg/L)	1.90 - 4.18
NO2/NO3-N (mg/L)	0.86 - 2.2
Total Cu (ug/L)	**43 - 118**
Total Pb (ug/L)	**182 - 443**
Total Zn (ug/L)	**202 - 633**

Source: U.S. Environmental Protection Agency. (1983). *Results of the Nationwide Urban Runoff Program (NURP), Vol. 1*, NTIS PB 84-185552), Water Planning Division, Washington, DC.

2 Groove end with headwall

3 Groove end projecting

Circular Corrugated Metal Pipe 4 Headwall
5 Mitered to slope

6 Projecting

Circular Pipe, Beveled Ring Entrance 7 45 deg. bevels
8 33.7 deg. bevels

Rectangular Box; Flared Wingwalls 9 30-75 deg. wingwall flares
10 90 or 15 deg. wingwall flares
11 0 deg. wingwall flares (straight sides)
Rectangular Box;Flared Wingwalls and Top Edge Bevel 12 45 deg flare; 0.43D top edge bevel 13 18-33.7 deg. flare; 0.083D top edge bevel
Rectangular Box, 90-deg Headwall, Chamfered / Beveled Inlet Edges
14 chamfered 3/4-in.
15 beveled 1/2-in/ft at 45 deg (1:1) 16 beveled 1-in/ft at 33.7 deg (1:1.5)
Rectangular Box, Skewed Headwall, Chamfered / Beveled Inlet Edges
17 3/4" chamfered edge, 45 deg skewed headwall 18 3/4" chamfered edge, 30 deg skewed headwall 19 3/4" chamfered edge, 15 deg skewed headwall
20 45 deg beveled edge, 10-45 deg skewed headwall
Rectangular Box, Non-offset Flared Wingwalls, 3/4" Chamfer at Top of Inlet 21 45 deg (1:1) wingwall flare
22 8.4 deg (3:1) wingwall flare
23 18.4 deg (3:1) wingwall flare, 30 deg inlet skew
Rectangular Box, Offset Flared Wingwalls, Beveled Edge at Inlet Top
24 45 deg (1:1) flare, 0.042D top edge bevel 25 33.7 deg (1.5:1) flare, 0.083D top edge bevel 26 18.4 deg (3:1) flare, 0.083D top edge bevel
Corrugated Metal Box 27 90 deg headwall
28 Thick wall projecting

29 Thin wall projecting

Culvert Inlet Loss Coefficients

- Pipe, Concrete

Projecting from fill, socket end (groove-end) Projecting from fill, sq. cut end 0.2
Headwall or headwall and wingwalls: **0.5**

§ Socket end of pipe (groove-end)

§ Square-edge

Rounded (radius = D/12) 0.2
Mitered to conform to fill slope 0.5
*End-Section conforming to fill slope 0.2
Beveled edges, 33.7 deg or 45 deg bevels 0.7
Side- or slope-tapered inlet 0.5
0.2

0.2

- Pipe or Pipe-Arch, Corrugated Metal

Projecting from fill (no headwall) 0.9
Headwall or headwall and wingwalls 0.5
square-edge Mitered to conform to fill 0.7
slope, paved or unpaved slope *End- 0.5
Section conforming to fill slope Beveled 0.2
edges, 33.7 or 45 bevels Side- or slope-tapered inlet **0.2**

- Box, Reinforced Concrete

Headwall parallel to embankment (no wingwalls):

§ Square-edged on 3 edges **0.5**

§ Rounded on 3 edges to radius of D/12 or B/12 or beveled edges on 3 sides

Wingwalls at 30 deg to 75 deg to barrel: **0.2**

§ Square-edged at crown

§ Crown edge rounded to radius of D/12 0.4

Standard Elliptical Pipe Sizes

	(in)	(in)	(mm)	
1	14	23	356	584
2	19	30	483	762
3	22	34	559	864
4	24	38	610	965
5	27	42	686	1067
6	29	45	737	1143
7	32	49	813	1245
8	34	53	864	1346
9	38	60	965	1524
10	43	68	1092	1727

Standard Arch Pipe Sizes

Code	Rise (in)	Span (in)	Rise (mm)	Span (mm)
1	11	18	279	457
2	13.5	22	343	559
3	15.5	26	394	660
4	18	28.5	457	724
5	22.5	36.25	572	921
6	26.625	43.75	676	1111
7	31.3125	51.125	795	1299
8	36	58.5	914	1486
9	40	65	1016	1651
10	45	73	1143	1854
11	54	88	1372	2235
12	62	102	1575	2591
13	72	115	1829	2921
14	77.5	122	1969	3099

Corrugated Steel, 2 2/3 x 1/2" Corrugation

Code	Rise (in)	Span (in)	Rise (mm)	Span (mm)
18	13	17	330	432
19	15	21	381	533
20	18	24	457	610
21	20	28	508	711
				889
				1067
				1245
25	38	57	965	1448
26	43	64	1092	1626
27	47	71	1194	1803
				1956
				2108

Corrugated Steel, 3 x 1" Corrugation

Code	Rise (in)	Span (in)	Rise (mm)	Span (mm)
30	31	40	787	1016
				1168
				1346
				1524
34	51	66	1295	1676
35	55	73	1397	1854
36	59	81	1499	2057
37	63	87	1600	2210
38	67	95	1702	2413
39	71	103	1803	2616
40	75	112	1905	2845
41	79	117	2007	2972
42	83	128	2108	3251
43	87	137	2210	3480
44	91	142	2311	3607

Structural Plate, 18" Corner Radius

Code	Rise (in)	Span (in)	Rise (mm)	Span (mm)
45	55	73	1397	1854
46	57	76	1448	1930
47	59	81	1499	2057
48	61	84	1549	2134
49	63	87	1600	2210
50	65	92	1651	2337
51	67	95	1702	2413
52	69	98	1753	2489
53	71	103	1803	2616
54	73	106	1854	2692
55	75	112	1905	2845
56	77	114	1956	2896
57	79	117	2007	2972
58	81	123	2057	3124
59	83	128	2108	3251
60	85	131	2159	3327
61	87	137	2210	3480
62	89	139	2261	3531
63	91	142	2311	3607
64	93	148	2362	3759
65	95	150	2413	3810
66	97	152	2464	3861
67	100	154	2540	3912
68	101	161	2565	4089
69	103	167	2616	4242
70	105	169	2667	4293
71	107	171	2718	4343
72	109	178	2769	4521
73	111	184	2819	4674

74	113	186	2870	4724
75	115	188	2921	4775
76	118	190	2997	4826
77	119	197	3023	5004
78	121	199	3073	5055

Structural Plate, 31" Corner Radius

Code	Rise (in)	Span (in)	Rise (mm)	Span (mm)
79	112	159	2845	4039
80	114	162	2896	4115
81	116	168	2946	4267
82	118	170	2997	4318
83	120	173	3048	4394
84	122	179	3099	4547
85	124	184	3150	4674
86	126	187	3200	4750
87	128	190	3251	4826
88	130	195	3302	4953
89	132	198	3353	5029
90	134	204	3404	5182
91	136	206	3454	5232
92	138	209	3505	5309
93	140	215	3556	5461
94	142	217	3607	5512
95	144	223	3658	5664
96	146	225	3708	5715
97	148	231	3759	5867
98	150	234	3810	5944
99	152	236	3861	5994
100	154	239	3912	6071
101	156	245	3962	6223
102	158	247	4013	6274

Source: *Modern Sewer Design* (Fourth Edition), American Iron and Steel Institute, Washington, DC, 1999.

Rain Gage Properties

X-Coordinate	Horizontal location of the rain gage on the Study Area Map. If left blank then the rain gage will not appear on the map.
Y-Coordinate	Vertical location of the rain gage on the Study Area Map. If left blank then the rain gage will not appear on the map.
Description	Click the ellipsis button (or press Enter) to edit an optional description of the rain gage.
Tag	Optional label used to categorize or classify the rain gage.
Rain Format	Format in which the rain data are supplied: **INTENSITY**: each rainfall value is an average rate in inches/hour (or mm/hour) over the recording interval. **VOLUME**: each rainfall value is the volume of rain that fell in the recording interval (in inches or millimeters). **CUMULATIVE**: each rainfall value represents the cumulative rainfall that has occurred since the start of the last series of non-zero values (in inches or millimeters).
Rain Interval	Recording time interval between gage readings in decimal hours or hours:minutes format.
Snow Catch Factor	Factor that corrects gage readings for snowfall.
Data Source	Source of rainfall data; either **TIMESERIES** for user-defined

time series data or **FILE** for an

	name.
X-Coordinate	Horizontal location of the subcatchment's centroid on the Study Area Map. If left blank then the subcatchment will not appear on the map.
Y-Coordinate	Vertical location of the subcatchment's centroid on the Study Area Map. If left blank then the subcatchment will not appear on the map.
Description	Click the ellipsis button (or press Enter) to edit an optional description of the subcatchment.
Tag	Optional label used to categorize or classify the subcatchment.
Rain Gage	Name of the rain gage associated with the subcatchment.
Outlet	Name of the node or subcatchment that recieves the subcatchment's runoff.
Area	Area of the subcatchment, including any LID controls (acres or hectares).
Width	Characteristic width of the overland flow path for sheet flow runoff (feet or meters). (More...)
% Slope	Average percent slope of the subcatchment.
% Imperv	Percent of the land area (not including any LIDs) which is impervious.
N-Imperv	Manning's n for overland flow over the impervious portion of the

Junction Properties

X-Coordinate	Horizontal location of the junction on the Study Area Map. If left blank then the junction will not appear on the map.
Y-Coordinate	Vertical location of the junction on the Study Area Map. If left blank then the junction will not appear on the map.
Description	Click the ellipsis button (or press Enter) to edit an optional description of the junction.
Tag	Optional label used to categorize or classify the junction.
Inflows	Click the ellipsis button (or press Enter) to assign external direct, dry weather, or RDII inflows to the junction.
Treatment	Click the ellipsis button (or press Enter) to edit a set of treatment functions for pollutants entering the node.
Invert El.	Invert elevation of the junction (feet or meters).
Max. Depth	Maximum depth at the junction (i.e., the distance from the invert to the ground surface) (feet or meters). If zero, then the distance from the invert to the top of the highest connecting link will be used.
Initial Depth	Depth of water at the junction at the start of the simulation (feet or meters).
Surcharge Depth	Additional depth of water beyond the maximum depth that is allowed

	before the junction floods (feet or

Outfall Properties

X-Coordinate	Horizontal location of the outfall on the Study Area Map. If left blank then the outfall will not appear on the map.
Y-Coordinate	Vertical location of the outfall on the Study Area Map. If left blank then the outfall will not appear on the map.
Description	Click the ellipsis button (or press Enter) to edit an optional description of the outfall.
Tag	Optional label used to categorize or classify the outfall.
Inflows	Click the ellipsis button (or press Enter) to assign external direct, dry weather or RDII inflows to the outfall.
Treatment	Click the ellipsis button (or press Enter) to edit a set of treatment functions for pollutants entering the node.
Invert El.	Invert elevation of the outfall (feet or meters).
Tide Gate	**YES** - tide gate present to prevent backflow **NO** - no tide gate present
Route To	Optional name of a subcatchment that receives the outfall's discharge.
Type	Type of outfall boundary condition: **FREE**: outfall stage determined by minimum of critical flow depth and normal flow depth in the connecting conduit **NORMAL**: outfall stage based on normal flow depth in the connecting conduit **FIXED**: outfall stage set to a fixed

	value **TIDAL**: outfall stage given

Flow Divider Properties

X-Coordinate	Horizontal location of the divider on the Study Area Map. If left blank then the divider will not appear on the map.
Y-Coordinate	Vertical location of the divider on the Study Area Map. If left blank then the divider will not appear on the map.
Description	Click the ellipsis button (or press Enter) to edit an optional description of the divider.
Tag	Optional label used to categorize or classify the divider.
Inflows	Click the ellipsis button (or press Enter) to assign external direct, dry weather or RDII inflows to the divider.
Treatment	Click the ellipsis button (or press Enter) to edit a set of treatment functions for pollutants entering the node.
Invert El.	Invert elevation of the divider (feet or meters).
Max. Depth	Maximum depth at the divider (i.e., distance from the invert to the ground surface) (feet or meters). If zero then the distance from the invert to the top of the highest connecting link will be used.
Initial Depth	Depth of water at the divider at the start of the simulation (feet or meters)
Surcharge Depth	Additional depth of water beyond the maximum depth that is

allowed before the divider floods

Storage Unit Properties

X-Coordinate	Horizontal location of the storage unit on the Study Area Map. If left blank then the storage unit will not appear on the map.
Y-Coordinate	Vertical location of the storage unit on the Study Area Map. If left blank then the storage unit will not appear on the map.
Description	Click the ellipsis button (or press Enter) to edit an optional description of the storage unit.
Tag	Optional label used to categorize or classify the storage unit.
Inflows	Click the ellipsis button (or press Enter) to assign external direct, dry weather, or RDII inflows to the storage unit.
Treatment	Click the ellipsis button (or press Enter) to edit a set of treatment functions for pollutants within the storage unit.
Invert El.	Elevation of the bottom of the storage unit (feet or meters).
Max. Depth	Maximum depth of the storage unit (feet or meters).
Initial Depth	Initial depth of water in the storage unit at the start of the simulation (feet or meters).
Surcharge Depth	Additional depth above the maximum depth that allows the unit to pressurize before it overflows (feet or meters). Only used for covered units.
Evap. Factor	The fraction of the potential evaporation from the storage unit's

water surface that is actually

Conduit Properties

Inlet Node	Name of node on the inlet end of the conduit (which is normally the end at higher elevation).
Outlet Node	Name of node on the outlet end of the conduit (which is normally the end at lower elevation).
Description	Click the ellipsis button (or press Enter) to edit an optional description of the conduit.
Tag	Optional label used to categorize or classify the conduit.
Shape	Click the ellipsis button (or press Enter) to edit the geometric properties of the conduit's cross section.
Max. Depth	Maximum depth of the conduit's cross section (feet or meters).
Length	Conduit length (feet or meters).
Roughness	Manning's roughness coefficient. (Values for closed conduits) (Values for open channels)
Inlet Offset	Depth or elevation of the conduit invert above the node invert at the inlet end of the conduit (feet or meters).
Outlet Offset	Depth or elevation of the conduit invert above the node invert at the outlet end of the conduit (feet or meters).
Initial Flow	Initial flow in the conduit at the start of the simulation (flow units).
Maximum Flow	Maximum flow allowed in the conduit (flow units) - use 0 or leave blank if not applicable.

Pump Properties

Inlet Node	Name of node on the inlet side of the pump.
Outlet Node	Name of node on the outlet side of the pump.
Description	Click the ellipsis button (or press Enter) to edit an optional description of the the pump.
Tag	Optional label used to categorize or classify the pump.
Pump Curve	Name of the Pump Curve which contains the pump's operating data (double-click to edit the curve). Enter * for an Ideal pump.
Initial Status	Status of the pump (**ON** or **OFF**) at the start of the simulation.
Startup Depth	Depth at inlet node when pump turns on (ft or m). Enter 0 if not applicable.
Shutoff Depth	Depth at inlet node when pump shuts off (ft or m). Enter 0 if not applicable.

Orifice Properties

Inlet Node	Name of node on the inlet side of the orifice.
Outlet Node	Name of node on the outlet side of the orifice.
Description	Click the ellipsis button (or press Enter) to edit an optional description of the orifice.
Tag	Optional label used to categorize or classify the orifice.
Type	Type of orifice (**SIDE** or **BOTTOM**).
Shape	Orifice shape (**CIRCULAR** or **RECT_CLOSED**).
Height	Height of orifice opening when fully open (feet or meters). Corresponds to the diameter of a circular orifice or the height of a rectangular orifice.
Width	Width of rectangular orifice when fully opened (feet or meters)
Inlet Offset	Depth or elevation of bottom of orifice above invert of inlet node (feet or meters).
Discharge Coeff.	Discharge coefficient (unitless). A typical value is 0.65.
Flap Gate	**YES** if a flap gate exists which prevents backflow through the orifice, or **NO** if no flap gate exists.
Time to Open/Close	The time to open a closed (or close an open) gated orifice in decimal hours. Use 0 or leave blank if timed openings/closings do not apply. Use Control Rules to adjust gate position.

Weir Properties

Inlet Node	Name of node on inlet side of weir.
Outlet Node	Name of node on outlet side of weir.
Description	Click the ellipsis button (or press Enter) to edit an optional description of the weir.
Tag	Optional label used to categorize or classify the weir.
Type	Weir type: **TRANSVERSE, SIDEFLOW, V-NOTCH, TRAPEZOIDAL** or **ROADWAY**.
Height	Vertical height of weir opening (feet or meters)
Length	Horizontal length of weir opening (feet or meters)
Side Slope	Slope (width-to-height) of side walls for a V-NOTCH or TRAPEZOIDAL weir.
Inlet Offset	Depth or elevation of bottom of weir opening from invert of inlet node (feet or meters).
Discharge Coeff.	Discharge coefficient for flow through the central portion of the weir (for flow in CFS when using US units or CMS when using SI units). Typical values are: 3.33 US (1.84 SI) for sharp crested transverse weirs, 2.5 - 3.3 US (1.38 - 1.83 SI) for broad crested rectangular weirs, 2.4 - 2.8 US (1.35 - 1.55 SI) for V-notch (triangular) weirs. Discharge over Roadway weirs with a non-zero road width is computed using the FHWA HDS-5 method.

Flap Gate	**YES** if the weir has a flap gate that

Outlet Properties

Inlet Node	Name of node on inflow side of outlet.
Outlet Node	Name of node on discharge side of outlet.
Description	Click the ellipsis button (or press Enter) to edit an optional description of the outlet.
Tag	Optional label used to categorize or classify the outlet.
Inlet Offset	Depth or elevation of outlet above inlet node invert (ft or m).
Flap Gate	**YES** if a flap gate exists which prevents backflow through the outlet, or **NO** if no flap gate exists.
Rating Curve	Method of defining flow (Q) as a function of freeboard depth or head (y) across the outlet. **FUNCTIONAL/DEPTH** - uses a power function $Q = Ay^B$ where y is the freeboard depth above the outlet's opening. **FUNCTIONAL/HEAD** - uses a power function $Q = Ay^B$ where y is the head difference across the outlet. **TABULAR/DEPTH** - uses a tabulated curve of flow versus freeboard depth values. **TABULAR/HEAD** - uses a tabulated curve of flow versus head difference values.
FUNCTIONAL	

- *Coefficient*	Coefficient (A) for the functional

Map Label Properties

X-Coordinate	Horizontal location of the upper-left corner of the label on the study area map.
Y-Coordinate	Vertical location of the upper-left corner of the label on the study area map.
Anchor Node	Name of node (or subcatchment) that anchors the label's position when the map is zoomed in (i.e., the pixel distance between the node and the label remains constant). Leave blank if anchoring is not used.
Font	Click the ellipsis button (or press Enter) to modify the font used to draw the label.

or an exisitng Aquifer object is selected for editing. It contains the following data fields:

Name
User-assigned aquifer name.

Porosity
Volume of voids / total soil volume (volumetric fraction).

Wilting Point
Soil moisture content at which plants cannot survive (volumetric fraction).

Field Capacity
Soil moisture content after all free water has drained off (volumetric fraction).

Conductivity
Soil's saturated hydraulic conductivity (in/hr or mm/hr).

Conductivity Slope
Average slope of log(conductivity) versus soil moisture deficit (i.e., porosity minus moisture content) curve (unitless).

Tension Slope
Average slope of soil tension versus soil moisture content curve (inches or mm).

Upper Evaporation Fraction Fraction of total evaporation available for evapotranspiration in the upper unsaturated zone.

Lower Evaporation Depth Maximum depth below the surface at which evapotranspiration from the lower saturated zone can still occur (ft or m).

Lower Groundwater Loss Rate Rate of percolation to deep groundwater when the water table reaches the ground surface (in/hr or mm/hr).

Bottom Elevation

Elevation of the bottom of the aquifer (ft or m).

Backdrop Image imposed over the Study Area Map.

Lower Left Coordinates
Enter the X,Y coordinates of the lower left corner of the backdrop image.

Upper Right Coordinates
Enter the X,Y coordinates of the upper right corner of the backdrop image.

Resize Backdrop Image Only
Select this button if only the backdrop, and not the Study Area Map, should be resized according to the coordinates specified in the dialog.

Scale Backdrop Image to Map
Select this button to position the backdrop image in the center of the Study Area Map and have it resized to fill the display window without changing its aspect ratio. The map's lower left and upper right coordinates will be placed in the data entry fields for the backdrop coordinates, and these fields will become disabled.

Scale Map to Backdrop Image
Select this button to make the dimensions of the map coincide with the dimensions being set for the backdrop image. Note that this option will change the coordinates of all objects currently on the map so that their positions relative to one another remain unchanged.

Backdrop Image Selector Dialog

as a Backdrop Image behind the Study Area Map. It contains the following data fields:

Backdrop Image File

Enter the name of the file that contains the image. You can click the button to bring up a standard Windows file selection dialog from which you can search for the image file.

World Coordinates File

If a "world" file exists for the image, enter its name here, or click the button to search for it. A world file contains geo-referencing information for the image and can be created from the software that produced the image file or by using a text editor. It contains six lines with the following information:

```
Line 1: real world width of a pixel in the horizontal direction.
Line 2: X rotation parameter (not used).
Line 3: Y rotation parameter (not used).
Line 4: negative of the real world height of a pixel in the vertical direction.
Line 5: real world X coordinate of the upper left corner of the image.
Line 6: real world Y coordinate of the upper left corner of the image.
```

If no world file is specified, then the backdrop will be scaled to fit into the center of the map display window.

Scale Map to Backdrop Image

This option is only available when a world file has been specified. Selecting it forces the dimensions of the Study Area Map to coincide with those of the backdrop image. In addition, all existing objects on the map will have their coordinates adjusted so that they appear within the new map dimensions yet maintain their relative positions to one another. Selecting this option may then require that the backdrop be re-aligned so that its position relative to the drainage area objects is correct (see

Climatology Editor

variables required by certain SWMM simulations. The dialog is divided into six tabbed pages, where each page provides a separate editor for the following data categories:

- Temperature
- Evaporation
- Wind Speed
- Snowmelt
- Areal Depletion
- Adjustments

the source of temperature data used for snow melt computations. It is also used to select a climate file as a possible source for evaporation rates. There are three choices available:

· **No Data**

Select this choice if snowmelt is not being simulated and evaporation rates are not computed from daily temperatures.

· **Time Series**

Select this choice if the variation in temperature over the simulation period will be described by one of the project's time series. Also enter (or select) the name of the time series. Click the button to make the Time Series Editor appear for the selected time series.

· **External Climate File**

Select this choice if min/max daily temperatures will be read from an external climate file. Also enter the following information:

§ Click the button to search for a climate file or click the button to clear the file name.

different than the start date of the simulation (as specified in the Simulation Options), check off the "Start Reading File at" box and enter a starting date (month/day/year) in the date entry field next to it.

§ If using a NOAA-GHCN file, specify the units of temperature used by the file.

Use this source of temperature data if you want daily evaporation rates to be estimated from daily temperatures or be read directly from the file.

Evaporation Page

potential evaporation rates, in inches/day (or mm/day), for a study area. There are five choices for specifying these rates that are selected from the Source of Evaporation Rates combo box:

- **Constant Value**

- **Time Series**

Select this choice if evaporation rates will be specified in a time series. Enter or select the name of the time series in the dropdown combo box provided. Click the button to bring up the Time Series Editor for the selected series. Note that for each date specified in the time series, the evaporation rate remains constant at the value supplied for that date until the next date in the series is reached (i.e., interpolation is not used on the series).

- **Climate File**

This choice indicates that daily evaporation rates will be read from the same climate file that was specified for temperature. Enter values for monthly pan coefficients in the data grid provided (these are used to convert pan evaporation to actual evaporation and are typically on the order of 0.7).

- **Monthly Averages**

Use this choice to supply an average rate for each month of the year. Enter the value for each month in the data grid provided. Note that rates remain constant within each month.

- **Computed from Temperatures**

The Hargreaves' method will be used to compute daily evaporation rates from the daily air temperature record contained in the external climate file specified on the Temperature page of the dialog. This method also uses the site's latitude, which can be entered on the Snowmelt page of the dialog even if snow melt is not being simulated.

- **Evaporate Only During Dry Periods**

Select this option if evaporation can only occur during periods with no precipitation.

In addition this page allows one to specify an optional **Monthly Soil Recovery Pattern**. This is a time pattern whose factors adjust the rate at which infiltration capacity is recovered during periods with no precipitation. It applies to all subcatchments for any choice of infiltration method. For example, if the normal infiltration recovery rate was 1% during a specific time period and a pattern factor of 0.8 applied to this period, then the actual recovery rate would be 0.8%. The Soil Recovery Pattern allows one to account for seasonal soil drying rates. In principle, the variation in pattern factors should mirror the variation in evaporation rates but might be influenced by other factors such as seasonal groundwater levels. The button is used to launch the Time Pattern Editor for the selected pattern.

Wind Speed Page

temperature.

average monthly wind speeds. These are used when computing snowmelt rates under rainfall conditions. Melt rates increase with increasing wind speed. Units of wind speed are miles/hour for US units and km/hour for metric units. There are two choices for specifying wind speeds:

- **Monthly Averages**
- **Climate File Data**

Wind speed is specified as an average value that remains constant in each month of the year. Enter a value for each month in the data grid provided. The default values are all zero.

values for the following parameters related to snow melt calculations:

Dividing Temperature Between Snow and Rain Enter the temperature below which precipitation falls as snow instead of rain. Use degrees F for US units or degrees C for metric units.

ATI (Antecedent Temperature Index) Weight This parameter reflects to what degree heat transfer within a snow pack during non-melt periods is affected by prior air temperatures. Smaller values reflect a thicker surface layer of snow which result in reduced rates of heat transfer. Values must be between 0 and 1, and the default is 0.5.

Negative Melt Ratio

This is the ratio of the heat transfer coefficient of a snow pack during non-melt conditions to the coefficient during melt conditions. It must be a number between 0 and 1. The default value is 0.6.

Elevation Above MSL

Enter the average elevation above mean sea level for the study area, in feet or meters. This value is used to provide a more accurate estimate of atmospheric pressure. The default is 0.0, which results in a pressure of 29.9 inches Hg. The effect of wind on snow melt rates during rainfall periods is greater at higher pressures, which occur at lower elevations.

Latitude

Enter the latitude, in degrees North, of the study area. This number is used when computing the hours of sunrise and sunset, which in turn are used to extend min/max daily temperatures into continuous values. It is also used to compute daily evaporation rates from daily temperatures. The default is 50 degrees North.

Longitude Correction

This is a correction, in minutes of time, between true solar time and the standard clock time. It depends on a location's longitude (θ) and the standard meridian of its time zone (SM) through the expression 4 (θ-SM). This correction is used to adjust the hours of sunrise and sunset when extending daily min/max temperatures into continuous values. The default

Areal Depletion Page

The **Areal Depletion** page of the Climatology Editor dialog is used to specify points on the Areal Depletion Curves for both impervious and pervious surfaces within a project's study area. These curves define the relation between the area that remains snow covered and snow pack depth. Each curve is defined by 10 equal increments of relative depth ratio between 0 and 0.9. (Relative depth ratio is the ratio of an area's current snow depth to the depth at which there is 100% areal coverage). Enter values in the data grid provided for the fraction of each area that remains snow covered at each specified relative depth ratio. Valid numbers must be between 0 and 1, and be increasing with increasing depth ratio.

Clicking the **Natural Area** button fills the grid with values that are typical of natural areas. Clicking the **No Depletion** button will fill the grid with all 1's, indicating that no areal depletion occurs. This is the default for new projects.

Adjustments Page

set of monthly adjustments applied to the temperature, evaporation rate, and rainfall that SWMM uses at each time step of a simulation:

- The monthly **Temperature** adjustment (plus or minus in either degrees F or C) is added to the temperature value that SWMM would otherwise use in a specific month of the year.

in/day) is added to the evaporation rate value that SWMM would otherwise use in a specific month of the year.

- The monthly **Rainfall** adjustment is a multiplier applied to the precipitation value that SWMM would otherwise use in a specific month of the year.

- The monthly **Conductivity** adjustment is a multiplier applied to the soil hydraulic conductivity used compute rainfall infiltration, groundwater percolation, and exfiltration from channels and storage units.

The same adjustment is applied for each time period within a given month and is repeated for that month in each subsequent year being simulated. Leaving a monthly adjustment blank means that there is no adjustment made in that month.

Copy Dialog

Use the Copy dialog as follows to define how you want your data copied and to where:

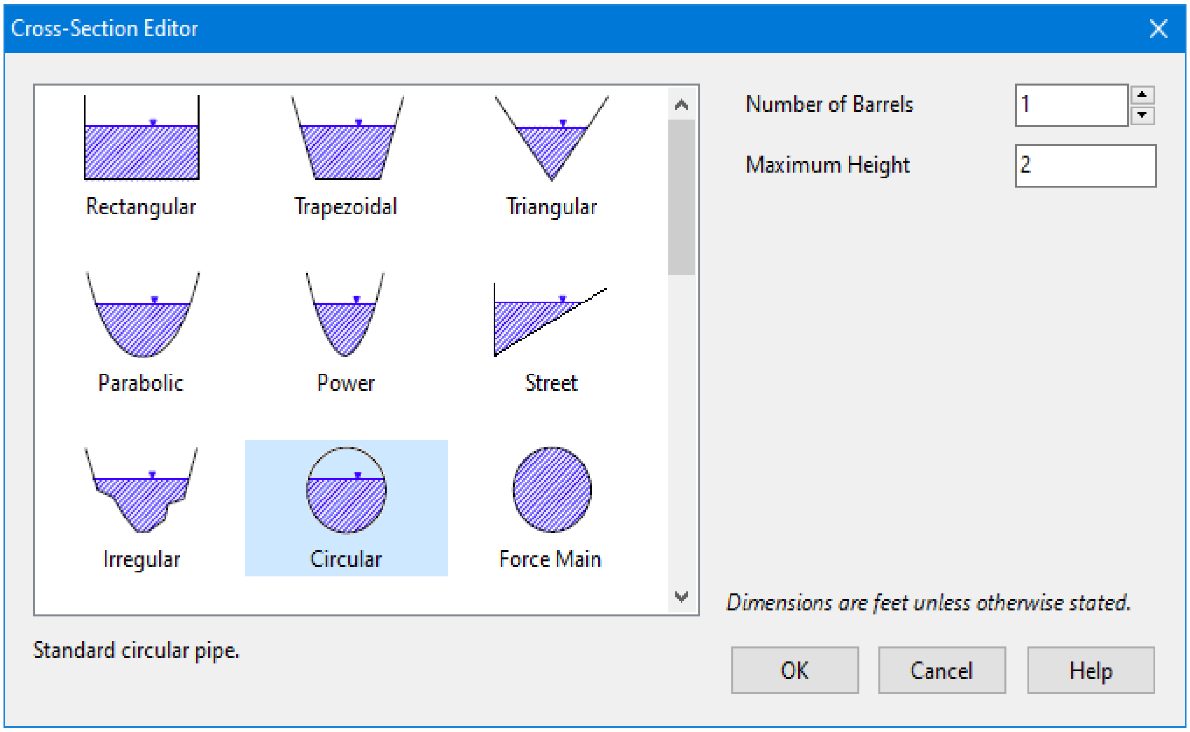
1. Select a destination for the material being copied (Clipboard or File)

- **Bitmap** (graphics only)
- **Metafile** (graphics only)
- **Data** (text, selected cells in a table, or data used to construct a graph)

3. Click **OK** to accept your selections or Cancel to cancel the copy request.

The bitmap format copies the individual pixels of a graphic. The metafile format copies the instructions used to create the graphic and is more suitable for pasting into word processing documents where the graphic can be re-scaled without loosing resolution. When data is copied, it can be pasted directly into a spreadsheet program to create customized tables or charts.

of a conduit's cross-section.



When a shape is selected from the image list an appropriate set of edit fields appears for describing the dimensions of that shape. Length dimensions are in units of feet for US units and meters for SI units. Slope values represent ratios of horizontal to vertical distance. The Barrels field specifies how many identical parallel conduits exist between its end nodes.

The **Force Main** shape option is a circular conduit that uses either the Hazen-Williams or Darcy-Weisbach formulas to compute friction losses for pressurized flow during Dynamic Wave flow routing. In this case the appropriate C-factor (for Hazen-Williams) or roughness height (for Darcy-Weisbach) is supplied as a cross-section property. The choice of friction loss equation is made on the Dynamic Wave Simulation Options dialog. Note that a conduit does not have to be assigned a Force Main shape for it to pressurize. Any of the other closed cross-section shapes can potentially pressurize and thus function as force mains using the Manning equation to compute friction losses.

If a **Custom** shaped section is chosen, a drop-down edit box will appear where you can enter or select the name of a Shape Curve that will be used to define the geometry of the section. This curve specifies how the width of the cross-section varies with height, where both width and height are scaled relative to the section's maximum depth. This allows the same shape curve to be used for conduits of differing sizes. Clicking the **Edit** button next to the

Curve Editor

created or an existing Curve object is selected for editing. The Editor adapts itself to the category of curve being edited (Storage, Shape, Tidal, Diversion, Pump, Rating, Control or Weir). To use the Curve Editor:

1. Enter values for the following dialog items:

Name	Name of the curve.
Type	(Pump Curves Only) Choice of pump curve type (see Pumps for a description of each curve type).
Description	Optional comment or description of what the curve represents. Click the button to launch a multi-line comment editor if more than one line is needed.
Data Grid	The curve's X,Y data.

2. Click the **View** button to see a graphical plot of the curve drawn in a separate window.

3. If additional rows are needed in the Data Grid, simply press the <**Enter**> key when in the last row.

4. Right-clicking over the Data Grid will make a popup Edit menu appear. It contains commands to cut, copy, insert, and paste selected cells in the grid as well as options to insert or delete a row.

5. Press **OK** to accept the curve entries or **Cancel** to cancel the edits made.

You can also click the **Load** button to load in a curve that was previously saved to file or click the **Save** button to save the current curve's data to a file.

Options is selected for editing from the Project Browser.

It is used to limit the periods of time in which a full unsteady hydraulic analysis of the drainage network is performed. For times outside of these periods, the hydraulic state of the network stays the same as it was at the end of the previous hydraulic event. Although hydraulic calculations are restricted to these pre-defined event periods, a full accounting of the system's hydrology is still computed over the entire simulation duration. During inter-event periods any inflows to the network, from runoff, groundwater flow, dry weather flow, etc., are ignored. The purpose of only computing hydraulics for particular time periods is to speed up long-term continuous simulations where one knows in advance which periods of time (such as representative or critical storm events) are of most interest.

The editor consists of a table listing the start and end date of each event, plus a blank line at the end of the list used for adding a new event. The events do not have to be entered in chronological order. There are date and time selection controls below the table used to edit the dates of a selected event. Clicking the **Replace Event** button will replace the row with the entries in these controls. The **Delete Event** button will delete the selected event and the **Delete All** button will delete all events from the table. The first column of the table contains a check box which determines if the event should be used in the analysis or not.

To identify event periods of interest, one can first run a simulation with Flow Routing turned off (see Simulation Options - General) and then perform a statistical frequency analysis on the system's rainfall record (see Viewing a Statistics Report).

When a new event occurs, the water in a storage unit node will remain at the same level it had at the end of the previous event. Therefore one may want to choose event intervals long enough to minimize the effect that storage carryover might have.

Graph Options Dialog

time series plot, a scatter plot, or a frequency plot. It is invoked by selecting **Report >> Customize** from the Main Menu when the graph window has the focus or by simply right-clicking on the graph. To use the dialog box:

1. Select from among the four tabbed pages that cover the following categories of options:
   - Axes
   - Legend
   - Styles
2. Check the **Default** box to use the current settings as defaults for all new graphs as well.
3. Select **OK** to accept your selections.

dialog box:

Option	Description
Panel Color	Color of the panel that contains the graph
Start Background Color	Starting gradient color of graph's plotting area
End Background Color	Ending gradient color of graph's plotting area
View in 3D	Check if graph should be drawn in 3D
3D Effect Percent	Degree to which 3D effect is drawn
Main Title	Text of graph's main title
Font	Click to set the font used for the main title

are drawn on a graph. One first selects an axis (Bottom, Left or Right (if present)) to work with and then selects from the following options:

Option	**Description**
Grid Lines	Displays grid lines on the graph.
Inverted	Inverts the scale of the right vertical axis.
Auto Scale	Fills in the Minimum, Maximum and Increment boxes with an automatic axis scaling.
Minimum	Sets the minimum axis value (the minimum data value is shown in parentheses). Can be left blank.
Maximum	Sets the maximum axis value (the maximum data value is shown in parentheses). Can be left blank.
Increment	Sets the increment between axis labels. If left blank or set to zero the program will automatically select an increment.
Axis Title	Text of axis title.
Font	Click to select a font for the axis title.

is displayed on the graph.

Option	Description
Position	Selects where to place the legend.
Color	Selects color to use for the legend background.
Check Boxes	If selected, check boxes will appear next to each legend entry, allowing one to make the data series visible or invisible on the graph.
Framed	Places a frame around the legend.
Shadowed	Places a shadow behind the legend's text.
Transparent	Makes the legend background transparent.
Visible	Makes the legend visible.
Symbol Width	Selects the width used to draw the symbol portion of a legend item, as a percentage of the length of the longest legend label.

Graph Options - Styles

data series (or curves) are displayed on a graph. To use this page:

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

1. Select a data series to work with from the **Series** combo box.

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

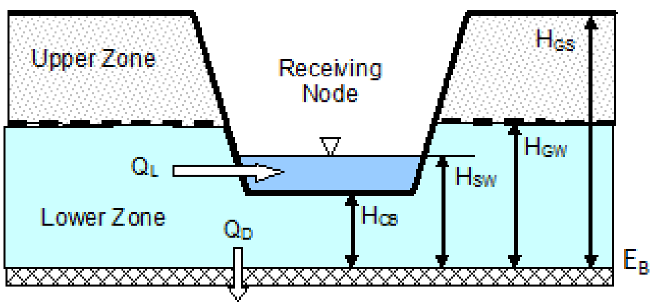
Groundwater Flow Editor

property of a Subcatchment is being edited. It is used to link a subcatchment to both a parent aquifer and to a node of the conveyance system that exchanges groundwater with the subcatchment.

The editor also specifies coefficients that determine the rate of lateral groundwater flow between the aquifer and the node. These coefficients (A1, A2, B1, B2, and A3) appear in the following equation that computes lateral groundwater flow as a function of groundwater and surface water levels:

$$Q_L = A1(H_{GW} - H_{CB})^{B1} - A2(H_{SW} - H_{CB})^{B2} + A3(H_{GW} H_{SW})$$

where Q_L = lateral groundwater flow (cfs per acre or cms per hectare), H_{GW} = height of saturated zone above bottom of aquifer (ft or m), H_{SW} = height of surface water at receiving node above aquifer bottom (ft or m), and H_{CB} = height of channel bottom above aquifer bottom (ft or m). Note that Q_L can also be expressed in inches/hr for US units.



The rate of percolation to deep groundwater, Q_D, in in/hr (or mm/hr) is given by the following equation:

$$Q_D = \mathrm{LGLR} * H_{GW} / H_{GS}$$

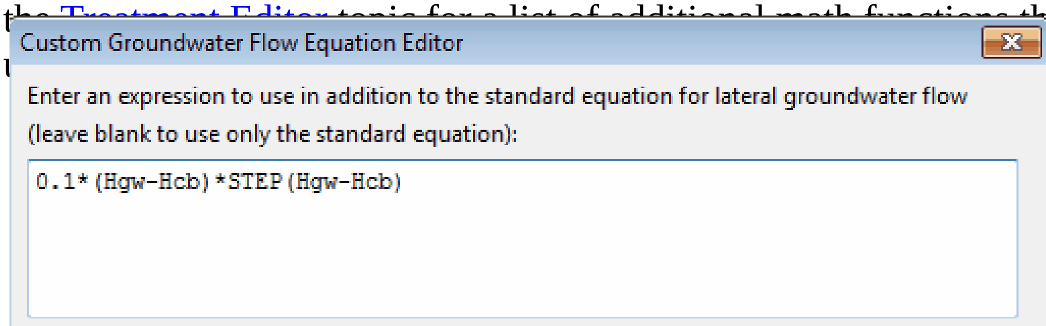
where LGLR is the lower groundwater loss rate parameter assigned to the subcatchment's aquifer (in/hr or mm/hr) and H_{GS} is the distance from the ground surface to the aquifer bottom (ft or m).

In addition to the standard lateral flow equation, the dialog allows one to

Groundwater Equation Editor

for computing groundwater flow between the saturated sub-surface zone of a subcatchment and either a node in the conveyance network (lateral flow) or to a deeper groundwater aquifer (deep flow). It is invoked from the Groundwater Flow Editor form.

The **STEP** function can be used to have flow only when the groundwater level is above a certain threshold. For example, the expression: 0.001 * (Hgw - 5) * STEP(Hgw - 5) would generate flow only when Hgw was above 5. See the Treatment Editor topic for a list of additional math functions that can be used.



For lateral groundwater flow the result of evaluating the custom equation will be added onto the result of the standard equation. To replace the standard equation completely set all of its coefficients to 0. Remember that groundwater flow units are cfs/acre for US units and cms/ha for metric units.

The following symbols can be used in the equation:

Hgw	(height of the groundwater table)
Hsw	(height of the surface water)
Hcb	(height of the channel bottom)
Hgs	(height of the ground surface)
Phi	(porosity of the subsurface soil)
Theta	(moisture content of the upper unsaturated zone)
Ks	(saturated hydraulic conductivity in inches/hr or mm/hr)
K	(hydraulic conductivity at the current moisture content in inches/hr or mm/hr)
Fi	(infiltration rate from the ground surface in inches/hr or mm/hr)
Fu	(percolation rate from the upper unsaturated zone in inches/hr or mm/hr)

objects (see Selecting a Group of Objects). To use the dialog box:

1. Select a class of objects (Subcatchments, Infiltration, Junctions, Storage Units, or Conduits) to edit.

limit the objects selected for editing to those with a specific Tag value. (For Infiltration the Tag will be that of the subcatchment to which the infiltration parameters belong.)

3. Enter a Tag value to filter on if you have selected that option.

4. Select the property to edit.

5. Select whether to replace, multiply, or add to the existing value of the property. Note that for some non-numerical properties the only available choice is to replace the value.

6. In the lower-right edit box, enter the value that should replace, multiply, or be added to the existing value for all selected objects. Some properties will have an ellipsis button displayed in the edit box which should be clicked to bring up a specialized editor for the property.

7. Click **OK** to execute the group edit.

After the group edit is executed a confirmation dialog box will appear informing you of how many items were modified. It will ask if you wish to continue editing or not. Select **Yes** to return to the Group Edit dialog box to edit another parameter or **No** to dismiss the Group Edit dialog.

Infiltration Editor

The **Infiltration Editor** dialog is used to specify the method and its parameters that model the rate at which rainfall infiltrates into the upper soil zone of a subcatchment's pervious area. It is invoked when editing the Infiltration property of a Subcatchment. The infiltration parameters depend on which infiltration model is selected for the subcatchment: Horton and Modified Horton, Green-Ampt and Modified Green-Ampt, or Curve Number. The infiltration model is normally the default one set by project's Simulation Options or its Default Properties. The dialog allows one to override the default method for the subcatchment being edited.

- Horton Infiltration Parameters
- Green-Ampt Infiltration Parameters
- Curve Number Infiltration Parameters

Horton Infiltration Parameters

Max. Infil. Rate	Maximum infiltration rate on the Horton curve (in/hr or mm/hr) (see table below)
Min. Infil. Rate	Minimum infiltration rate on the Horton curve (in/hr or mm/hr). Equivalent to the saturated hydraulic conductivity. See the Soil Characteristics Table for typical values.
Decay Const.	Infiltration rate decay constant for the Horton curve (1/hours). Typical values range between 2 and 7.
Drying Time	Time in days for a fully saturated soil to dry completely. Typical values range from 2 to 14 days.
Max. Infil. Vol.	Maximum infiltration volume possible (inches or mm, 0 if not applicable). It can be estimated as the difference between a soil's porosity and its wilting point times the depth of the infiltration zone.

Representative Values for Max. Infiltration Rate

A. DRY soils (with little or no vegetation):
   Sandy soils: 5 in/hr Loam soils: 3 in/hr Clay soils: 1 in/hr
B. DRY soils (with dense vegetation):
   Multiply values given in A. by 2

C. MOIST soils
   Soils which have drained but not dried out (i.e., field capacity): divide values from A and B by 3.
   Soils close to saturation: choose value close to min. infiltration rate.

Green-Ampt Infiltration Parameters

Suction Head	Average value of soil capillary suction along the wetting front (inches or mm)
Conductivity	Soil saturated hydraulic conductivity (in/hr or mm/hr)
Initial Deficit	Fraction of soil volume that is initially dry (i.e., difference between soil porosity and initial moisture content)

The initial deficit for a completely drained soil is the difference between the soil's porosity and its field capacity. Typical values for all of these parameters can be found in the Soil Characteristics Table.

Curve Number Infiltration Parameters

This is the SCS curve number which is tabulated in the publication SCS Urban Hydrology for Small Watersheds, 2nd Ed., (TR-55), June 1986. Consult the Curve Number Table for a listing of values by soil group, and the accompanying Soil Group Table for the definitions of the various groups. Adjustments will be needed when a subcatchment contains separate pervious and impervious fractions and a Curve Number is selected from a table where the two land uses are lumped together.

Conductivity

This property has been deprecated and is no longer used.

Drying Time

The number of days it takes a fully saturated soil to dry. Typical values range between 2 and 14 days.

Inflows Editor

The **Inflows Editor** dialog is used to assign Direct, Dry Weather, and RDII inflow into a node of the drainage system. It is invoked whenever the Inflows property of a Node object is selected in the Property Editor. The dialog consists of three tabbed pages that provide a special editor for each type of inflow:

- Direct Inflow
- Dry Weather Inflow
- RDII Inflow

history of direct external flow and water quality entering a node of the drainage system. These inflows are represented by both a constant and time varying component as follows:

Inflow at time t	=	(baseline value) * (baseline pattern factor) + (scale factor) * (time series value at time t)

The page contains the following input fields that define the properties of this relation:

Constituent
Selects the constituent (**FLOW** or one of the project's named pollutants) whose direct inflow will be described.

Baseline
Specifies the value of the constant baseline component of the constituent's inflow. For **FLOW**, the units are the project's flow units. For pollutants, the units are the pollutant's concentration units if inflow is a concentration, or can be any mass flow units if the inflow is a mass flow (see Units Factor below). If left blank then no baseline inflow is assumed.

Baseline Pattern
An optional Time Pattern whose factors adjust the baseline inflow on either an hourly, daily, or monthly basis (depending on the type of time pattern specified). Clicking the button will bring up the Time Pattern Editor dialog for the selected time pattern. If left blank, then no adjustment is made to the baseline inflow.

Time Series
The name of the time series that describes the time varying component of the constituent's inflow. If left blank then no time varying inflow is assumed. Clicking the button will bring up the Time Series Editor dialog for the selected time series. The units of the time series values obey the same convention as described above for Baseline inflow.

Dry Weather Inflow Page

continuous source of dry weather flow entering a node of the drainage system. The page contains the following input fields:

Constituent
Selects the constituent (FLOW or one of the project's specified pollutants) whose dry weather inflow will be specified.

Average Value
Specifies the average (or baseline) value of the dry weather inflow of the constituent in the relevant units (flow units for flow, concentration units for pollutants). Leave blank if there is no dry weather flow for the selected constituent.

Time Patterns
Specifies the names of the time patterns to be used to allow the dry weather flow to vary in a periodic fashion by month of the year, by day of the week, and by time of day (for both week days and week ends). One can either type in a name or select a previously defined pattern from the dropdown list of each combo box. Up to four different types of patterns can be assigned. You can click the button next to each Time Pattern field to edit the respective pattern.

More than one constituent can be edited while the dialog is active by simply selecting another choice for the Constituent property. However, if the Cancel button is clicked then any changes made to all constituents will be ignored.

RDII Inflow Page

RDII (Rainfall Dependent Infiltration/Inflow) for the node in question. The page contains the following two input fields:

Unit Hydrograph Group

Enter (or select from the dropdown list) the name of the Unit Hydrograph group that applies to the node in question. The unit hydrographs in the group are used in combination with the group's assigned rain gage to develop a time series of RDII inflows per unit area over the period of the simulation. Leave this field blank to indicate that the node receives no RDII inflow. Clicking the button will launch the Unit Hydrograph Editor for the UH group specified.

Sewershed Area

Enter the area (in acres or hectares) of the sewershed which contributes RDII to the node in question. Note this area will typically be only a small, localized portion of the subcatchment area that contributes surface runoff to the node.

Initial Buildup Editor

The **Initial Buildup** editor is invoked from the Property Editor when editing the Initial Buildup property of a Subcatchment. It specifies the amount of pollutant buildup existing over the subcatchment at the start of the simulation.

The editor consists of a data entry grid with two columns. The first column lists the name of each pollutant in the project and the second column contains edit boxes for entering the initial buildup values. If no buildup value is supplied for a pollutant, it is assumed to be 0. The units for buildup are either pounds per acre when US units are in use or kilograms per hectare when SI metric units are in use.

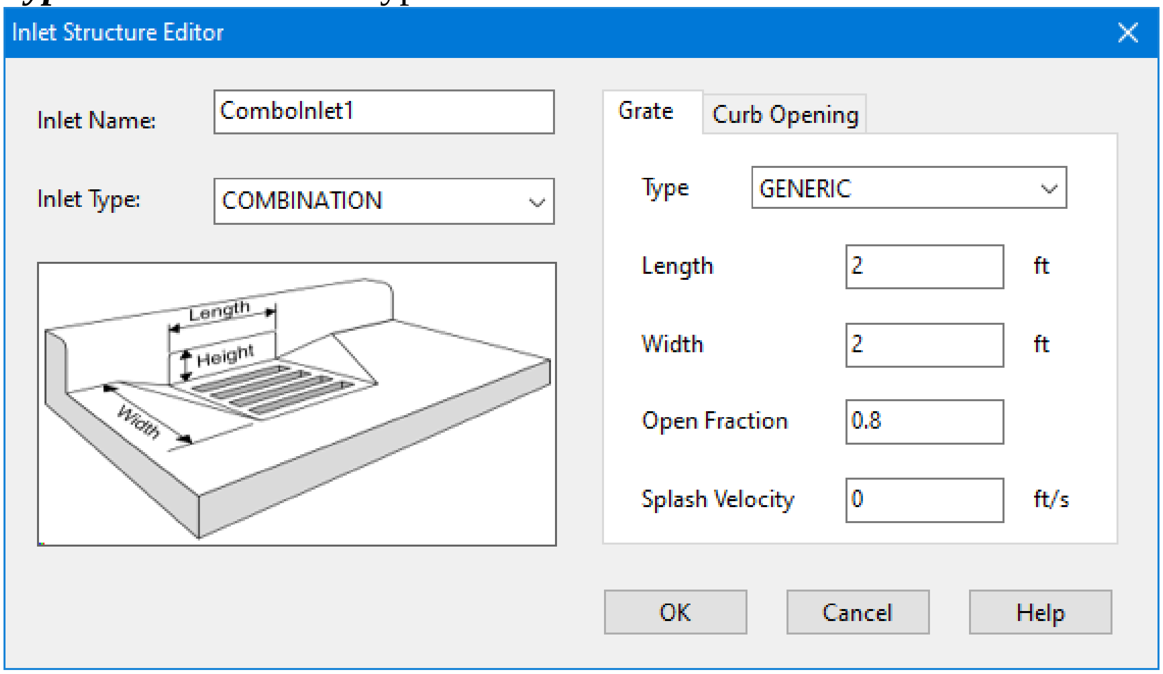
If a non-zero value is supplied for the initial buildup of a pollutant, it will override any initial buildup computed from the **Antecedent Dry Days** parameter specified on the **Dates** page of the Simulation Options dialog.

Inlet Structure Editor

is selected for editing from the Project Browser. As shown below it contains an ***Inlet Name*** field used to uniquely identify the inlet structure and an ***Inlet Type*** field to select the type of structure.

· Combination (Grated + Curb Opening) Inlet

· Slotted Drain Inlet



The design parameters shown in the data entry panel depend on the choice of inlet type:

· Grated Inlet

Grated Inlet

Length

The grate's length parallel to the street curb (feet or meters).

Width

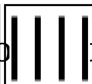
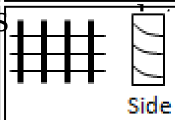
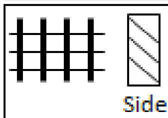
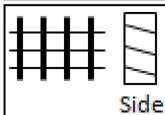
The grate's width (feet or meters).

Open Fraction (for GENERIC grates only) The fraction of the grate's area that is open. Values are predetermined for non-Generic grates.

Splash Velocity (for GENERIC grates only) The minimum velocity that causes some water to shoot over the inlet thus reducing its capture efficiency (ft/sec or m/sec). Values are predetermined for non-Generic grates.

Grate Type

One of the following types of grate designs:

P_BAR-50		Parallel bar grate with bar spacing 1-7/8-in on center
P_BAR-50X100		Parallel bar grate with bar spacing 1-7/8-in on center and 3/8-in diameter lateral rods spaced at 4-in on center
P_BAR-30		Parallel bar grate with 1-1/8-in on center bar spacing
CURVED_VANE		Curved vane grate with 3-1/4-in longitudinal bar and 4-1/4-in transverse bar spacing on center
TILT_BAR-45		45 degree tilt bar grate with 2-1/4-in longitudinal bar and 4-in transverse bar spacing on center
TILT_BAR-30		30 degree tilt bar grate with 3-1/4-in and 4-in on center longitudinal and lateral bar spacing respectively
RETICULINE		"Honeycomb" pattern of lateral bars and longitudinal bearing bars
GENERIC		A generic grate design.

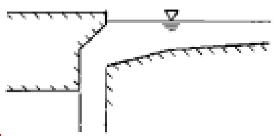
Curb Opening Inlet

Length
The length of the opening (feet or meters).

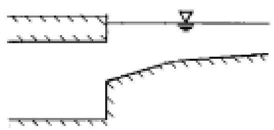
Height
The height of the opening (feet or meters).

Throat Angle
The orientation of the curb opening's throat relative to the street surface. Choices are:

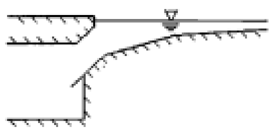
VERTICAL



HORIZONTAL



INCLINED



For combination inlets only the portion of the curb opening that extends beyond the length of the grated inlet contributes to inlet capture efficiency.

Slotted Drain Inlet

Length
The drain's length parallel to the street curb (feet or meters).

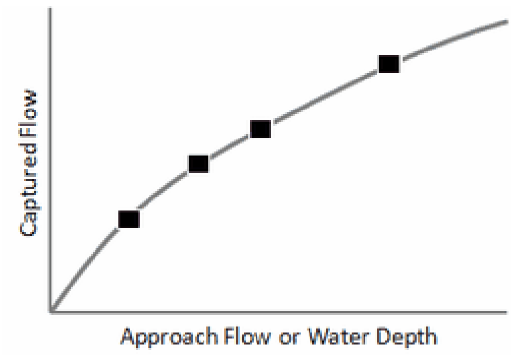
Width
The drain's width (feet or meters).

flow capture curve. Two options for this curve are available:

1. a Diversion Curve (normally used for Divider nodes) that has captured flow be a function of the inlet's approach flow

flow be a function of water depth.

Diversion curves are best suited for on-grade inlets and Rating curves for on-sag inlets.



Clicking the button next to the curve's name field to open up the Curve Editor dialog.

channel conduit. It is accessed by selecting a conduit into the Property Editor and then clicking the ellipsis button in its **Inlets** property. The following information is requested by the editor:

Inlet Structure
The name of the inlet structure to use. Select a previously defined structure from the drop-down list. The list will contain only those inlets that are compatible with the conduit's cross section (i.e., curb and gutter inlets for street sections or drop inlets for trapezoidal or rectangular channel sections). Selecting the blank first item will remove the inlet from the conduit.

Capture Node
The name of the node that receives flow captured by the inlet. You can select the node by clicking it on the Study Area Map or by selecting it from the Project Browser.

Number of Inlets
The number of identical inlets placed in the conduit. For two-sided street conduits this number refers to pairs of inlets placed on each side of the street. For example if 2 inlets are specified for a two-sided street, then a total of 4 inlets will be utilized, two on each side of the street.

Percent Clogged
The percentage to which each inlet is clogged. Suppose a value of 50% was used. Then the normal flow capture computed for the inlet would be reduced by half.

Flow Restriction
The maximum flow (in the project's flow units) that can be captured by a single inlet. A value of 0 indicates that flow capture is unrestricted.

Depression Height
The height of any local gutter depression that exists over the length of the inlet (in feet or meters). This local depression will be added onto any continuous depression that the conduit's Street section might have. A value

of 0 indicates no local depression. This parameter is ignored for drop

Interface File Combine Dialog

Combine from the Main Menu. It is used to combine two Routing Interface files into a single third file. The dialog contains a data entry grid with the following fields:

Interface File 1
Enter the name of the first interface file to be combined.

Interface File 2
Enter the name of the second interface file to be combined.

Interface File 3
Enter the name of the combined interface file to be written.

Description
Enter a descriptive line of text that will be written to the header of the combined file (optional).

Instead of typing in file names you can click the Browse button to select file names from a standard Windows File Open/Save dialog.

interface file to the project using the Interface Files page of the Simulation Options dialog.

File Type
Select the type of interface file to be specified.

Use / Save Buttons Select whether the named interface file will be used to supply input to a simulation run or whether simulation results will be saved to it.

File Name
Enter the name of the interface file.

Browse Button
Click this button to launch a standard file selection dialog from which the path and name of the interface file can be selected.

Land Use Assignment Editor

The **Land Use Assignment** editor is invoked from the Property Editor when editing the Land Uses property of a Subcatchment. Its purpose is to assign land uses to the subcatchment for water quality simulations. The percent of land area in the subcatchment covered by each land use is entered next to its respective land use category. If the land use is not present its field can be left blank. The percentages entered do not necessarily have to add up to 100.

Land Use Editor

The **Land Use Editor** dialog is used to define a category of land use for the study area and to define its pollutant buildup and washoff characteristics. The dialog contains three tabbed pages of land use properties:

- General Page (provides land use name and street sweeping parameters)
- Buildup Page (defines rate at which pollutant buildup occurs)
- Washoff Page (defines rate at which pollutant washoff occurs)

properties of a particular land use category:

Land Use Name
The name assigned to the land use.

Description
An optional comment or description of the land use. (Click the ellipsis button or press **Enter** to edit).

Street Sweeping Interval
Days between street sweeping within the land use (0 for no sweeping).

Street Sweeping Availability
Fraction of the buildup of all pollutants that is available for removal by sweeping.

Last Swept
Number of days since last swept at the start of the simulation.

If Street Sweeping does not apply to the land use, then the last three properties can be left blank.

associated with pollutant buildup over the land during dry weather periods. These consist of:

Pollutant
Select the pollutant whose buildup properties are being edited.

Function
The type of buildup function to use for the pollutant. The choices are **NONE** for no buildup, **POW** for power function buildup, **EXP** for exponential function buildup, **SAT** for saturation function buildup, and **EXT** for buildup supplied by an external time series. See the Pollutant Buildup topic for explanations of these different functions. Select **NONE** if no buildup occurs.

Max. Buildup
The maximum buildup that can occur, expressed as lbs (or kg) of the pollutant per unit of the normalizer variable (see below). This is the same as the C1 coefficient used in the buildup formulas discussed under Pollutant Buildup.

The following two properties apply to the **POW, EXP**, and **SAT** buildup functions:

> **Rate Constant**
> The time constant that governs the rate of pollutant buildup. This is the C2 coefficient in the Power and Exponential buildup formulas discussed under Pollutant Buildup. For Power buildup its units are mass / days raised to a power, while for Exponential buildup its units are 1/days.
>
> **Power/Sat. Constant**
> The exponent C3 used in the Power buildup formula, or the half-saturation constant C2 used in the Saturation buildup formula discussed under Pollutant Buildup. For the latter case, its units are days.

The following two properties apply to the **EXT** (External Time Series)

Land Use Editor - Washoff Page

washoff load associated with pollutant washoff over the land use during wet weather events. These consist of:

Pollutant

The name of the pollutant whose washoff properties are being edited.

Function

The choice of washoff function to use for the pollutant. The choices are:

NONE	no washoff
EXP	exponential washoff
RC	rating curve washoff
EMC	event-mean concentration washoff

Coefficient

This is the value of C1 in the exponential and rating curve formulas, or the event-mean concentration.

Exponent

The exponent used in the exponential and rating curve washoff formulas.

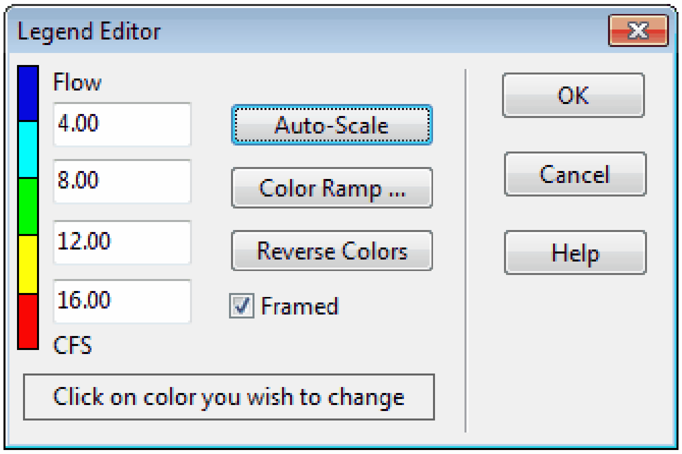
Cleaning Efficiency The street cleaning removal efficiency (percent) for the pollutant. It represents the fraction of the amount that is available for removal on the land use as a whole (set on the General page of the editor) which is actually removed.

BMP Efficiency

Removal efficiency (percent) associated with any Best Management Practice that might have been implemented. The washoff load computed at each time step is simply reduced by this amount.

As with the Buildup page, each pollutant must be selected in turn from the Pollutant dropdown list and have its pertinent washoff properties defined.

which different colors are assigned for viewing a particular parameter on the Study Area Map.



- Numerical values, in increasing order, are entered in the edit boxes to define the ranges. Not all four boxes need to have values.
- To change a color, click on its color band in the Editor and then select a new color from the Color Dialog box that will appear.
- Click the **Auto-Scale** button to automatically assign ranges based on the minimum and maximum values attained by the parameter in question at the current time period.
- The **Color Ramp** button is used to select from a list of built-in color schemes.
- The **Reverse Colors** button reverses the ordering of the current set of colors (the color in the lowest range becomes that of the highest range and so on).
- Check **Framed** if you want a frame drawn around the legend.

Changes made to a legend are saved with the project's settings and remain in effect when the project is re-opened in a subsequent session.

three different editor forms:

- The LID Control Editor is used to define re-usable LID controls, designed on a per-unit-area basis, that can be placed throughout a study area's subcatchments. It is invoked by adding a new **LID Control** object or editing an existing one from the main form's Project Browser.

specific subcatchment. It is invoked by selecting the subcatchment's **LID Controls** property from the subcatchment's Property Editor

- The LID Usage Editor is used to describe how each LID control added to an LID group is deployed within the group's subcatchment. It is invoked from the LID Group Editor to specify the areal extent of the control and the portion of the subcatchment's runoff that it treats.

LID Control Editor

control that can be deployed throughout a study area to store, infiltrate, and evaporate subcatchment runoff. The design of the control is made on a per-unit-area basis so that it can be placed in any number of subcatchments at different sizes or number of replicates.

The editor contains the following data entry fields:

Control Name
A name used to identify the particular LID control.

LID Type
The generic type of LID being defined (bio-retention cell, rain garden, green roof, infiltration trench, permeable pavement, rain barrel, or vegetative swale).

Process Layers
These are a tabbed set of pages containing data entry fields for the vertical layers and drain system that comprise an LID control. They include some combination of the following, depending on the type of LID selected:

- Surface Layer
- Soil Layer
- Storage Layer
- Drain System
- Drainage Mat
- Pollutant Removal

surface properties of all types of LID controls except rain barrels. Surface layer properties include:

Berm Height (or Storage Depth)

When confining walls or berms are present this is the maximum depth to which water can pond above the surface of the unit before overflow occurs (in inches or mm). For Rooftop Disconnection it is the roofs depression storage depth and for Vegetative Swales it is the height of the trapezoidal cross section.

Vegetation Volume Fraction

The fraction of the volume within the surface storage depth filled with vegetation. This is the volume occupied by stems and leaves, not their surface area coverage. Normally this volume can be ignored, but may be as high as 0.1 to 0.2 for very dense vegetative growth.

Surface Roughness

Manning's roughness coefficient (n) for overland flow over surface soil cover, pavement, roof surface or a vegetative swale (see this table for suggested values). Use 0 for other types of LIDs.

Surface Slope

Slope of a roof surface, pavement surface or vegetative swale (percent). Use 0 for other types of LIDs.

Swale Side Slope

Slope (run over rise) of the side walls of a vegetative swale's cross section. This value is ignored for other types of LIDs.

If either the Surface Roughness or Surface Slope values are 0 then any ponded water that exceeds the surface storage depth is assumed to completely overflow the LID control within a single time step.

the following properties of a permeable pavement LID:

Thickness
The thickness of the pavement layer (inches or mm). Typical values are 4 to 6 inches (100 to 150 mm).

Void Ratio
The volume of void space relative to the volume of solids in the pavement for continuous systems or for the fill material used in modular systems. Typical values for pavements are 0.12 to 0.21. Note that porosity = void ratio / (1 + void ratio).

Impervious Surface Fraction
Ratio of impervious paver material to total area for modular systems; 0 for continuous porous pavement systems.

Permeability
Permeability of the concrete or asphalt used in continuous systems or hydraulic conductivity of the fill material (gravel or sand) used in modular systems (in/hr or mm/hr). In the latter case the fill's nominal conductivity should be multiplied by the fraction of the total area it covers. The permeability of new porous concrete or asphalt is very high (e.g., hundreds of in/hr) but can drop off over time due to clogging by fine particulates in the runoff (see below).

Clogging Factor
Number of pavement layer void volumes of runoff treated it takes to completely clog the pavement. Use a value of 0 to ignore clogging. Clogging progressively reduces the pavement's permeability in direct proportion to the cumulative volume of runoff treated.

If one has an estimate of the number of years *Yclog* it takes to fractionally clog the system to a degree *Fclog*, then the Clogging Factor (*CF*) can be computed as:

CF = *Yclog * Pa * (1 + CR) * (1 + VR) / (VR * (1 - ISF) * T * Fclog)*

the engineered soil mixture used in bio-retention types of LIDs and the optional sand layer beneath permeable pavement. These properties are:

Thickness

The thickness of the soil layer (inches or mm). Typical values range from 18 to 36 inches (450 to 900 mm) for rain gardens, street planters and other types of land-based bio-retention units, but only 3 to 6 inches (75 to 150 mm) for green roofs.

Porosity

The volume of pore space relative to total volume of soil (as a fraction).

Field Capacity

Volume of pore water relative to total volume after the soil has been allowed to drain fully (as a fraction). Below this level, vertical drainage of water through the soil layer does not occur.

Wilting Point

Volume of pore water relative to total volume for a well dried soil where only bound water remains (as a fraction). The moisture content of the soil cannot fall below this limit.

Conductivity

Hydraulic conductivity for the fully saturated soil (in/hr or mm/hr).

Conductivity Slope

Average slope of the curve of log(conductivity) versus soil moisture deficit (porosity minus moisture content (unitless). Typical values range from 30 to 60. It can be estimated from a standard soil grain size analysis as 0.48×(%Sand) + 0.85×(%Clay).

Suction Head

The average value of soil capillary suction along the wetting front (inches or mm). This is the same parameter as used in the Green-Ampt infiltration model.

Porosity, field capacity, conductivity and conductivity slope are the properties of the crushed stone or gravel layer used in bio-retention cells, permeable pavement systems, and infiltration trenches as a bottom storage/drainage layer. It is also used to specify the height of a rain barrel (or cistern). The following data fields are displayed:

Thickness (or Barrel Height) This is the thickness of a gravel layer or the height of a rain barrel (inches or mm). Crushed stone and gravel layers are typically 6 to 18 inches (150 to 450 mm) thick while single family home rain barrels range in height from 24 to 36 inches (600 to 900 mm).

The following data fields do not apply to Rain Barrels.

Void Ratio

The volume of void space relative to the volume of solids in the layer. Typical values range from 0.5 to 0.75 for gravel beds. Note that porosity = void ratio / (1 + void ratio).

Seepage Rate

The rate at which water seeps into the native soil below the layer (in inches/hour or mm/hour).This would typically be the Saturated Hydraulic Conductivity of the surrounding subcatchment if Green-Ampt infiltration is used or the Minimum Infiltration Rate for Horton infiltration. If there is an impermeable floor or liner below the layer then use a value of 0.

Clogging Factor

Total volume of treated runoff it takes to completely clog the bottom of the layer divided by the void volume of the layer. Use a value of 0 to ignore clogging. Clogging progressively reduces the Infiltration Rate in direct proportion to the cumulative volume of runoff treated and may only be of concern for infiltration trenches with permeable bottoms and no underdrains. Refer to the Pavement Layer page for more discussion of the Clogging Factor.

The following data field applies only to Rain Barrels.

Covered

Specifies if the rain barrel is covered or not.

water entering the layer and conveys it to a conventional storm drain or other location (which can be different than the outlet of the LID's subcatchment). Drain flow can also be returned it to the pervious area of the LID's subcatchment. The drain can be offset some distance above the bottom of the storage layer, to allow some volume of runoff to be stored (and eventually infiltrated) before any excess is captured by the drain. For Rooftop Disconnection, the drain system consists of the roof's gutters and downspouts that have some maximum conveyance capacity.

The **Drain** page of the LID Control Editor describes the properties of and LID unit's drain system. It contains the following data entry fields:

Drain Coefficient and Drain Exponent
The drain coefficient *C* and exponent *n* determines the rate of flow through a drain as a function of the height of stored water above the drain's offset. The following equation is used to compute this flow rate (per unit area of the LID unit):

$$q = C\, h^n$$

where q is outflow (in/hr or mm/hr) and h is the height of saturated media above the drain (inches or mm). **If the layer has no drain then set *C* to *0*.**

A typical value for *n* would be 0.5 (making the drain act like an orifice). Note that the units of *C* depends on the unit system being used as well as the value assigned to *n*. Click here for more advice on setting drain parameters.

Drain Offset Height
This is the height of the drain line above the bottom of a storage layer or rain barrel (inches or mm).

Drain Delay (for Rain Barrels only)
The number of dry weather hours that must elapse before the drain line in a rain barrel is opened (the line is assumed to be closed once rainfall begins). A value of 0 signifies that the barrel's drain line is always open and drains continuously. This parameter is ignored for other types of LID practices.

The user specifies its height above the bottom of the unit's storage layer as well as how its volumetric flow rate (per unit area) varies with the height of saturated media above it. There are several things to keep in mind when specifying the parameters of an LID drain:

- If the drain can carry whatever flow enters the storage layer up to some specific limit then set the drain coefficient to the limit and the drain exponent to 0.

- If the drain consists of slotted pipes where the slots act as orifices, then the drain exponent would be 0.5 and the drain coefficient would be 60,000 times the ratio of total slot area to LID area. For example, drain pipe with five 1/4" diameter holes per foot spaced 50 feet apart would have an area ratio of 0.000035 and a drain coefficient of 2.

- If the storage layer that contains the drain has an impermeable bottom then it's best to place the drain at the bottom with a zero offset. Otherwise, to allow the full storage volume to fill before draining occurs, one would place the drain at the top of the storage layer.

- If the goal is to drain a fully saturated unit in a specific amount of time then set the drain exponent to 0.5 (to represent orifice flow) and the drain coefficient to 2D1/2/T where D is the distance from the drain to the surface plus any berm height (in inches or mm) and T is the time in hours to drain. For example, to drain a depth of 36 inches in 12 hours requires a drain coefficient of 1. If this drain consisted of the slotted pipes described in the previous bullet, whose coefficient was 2, then a flow regulator, such as a cap orifice, would have to be placed on the drain outlet to achieve the reduced flow rate.

media and above the roof structure. Its purpose is to convey any water that drains through the soil layer off of the roof. The **Drainage Mat** page of the LID Control Editor for Green Roofs lists the properties of this layer which include:

Thickness

The thickness of the mat or plate (inches or mm). It typically ranges between 1 to 2 inches.

Void Fraction

The ratio of void volume to total volume in the mat. It typically ranges from 0.5 to 0.6.

Roughness

This is the Manning's roughness coefficient (*n*) used to compute the horizontal flow rate of drained water through the mat. It is not a standard product specification provided by manufacturers and therefore must be estimated. Previous modeling studies have suggested using a relatively high value such as from 0.1 to 0.4.

specify the degree to which pollutants are removed by an LID control as seen by the flow leaving the unit through its underdrain system. Thus it only applies to LID practices that contain an underdrain (bio-retention cells,permeable pavement, infiltration trenches, and rain barrels).

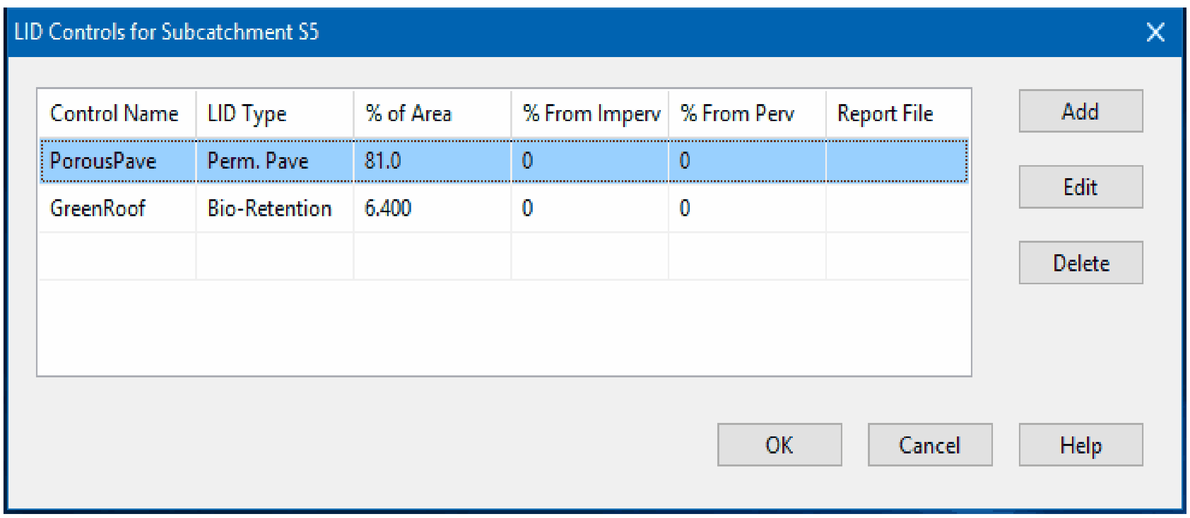
The page contains a data entry grid with the project's pollutant names listed in one column and the percent removal that each receives by the LID unit in the second editable column. If a percent removal value is left blank it is assumed to be 0.

The removals specified on this page are applied to the unit's underdrain when it sends flow onto either a subcatchment or into a conveyance system node. They do not apply to any surface flow that leaves the LID unit. As an example, if the runoff treated by the LID unit had a TSS concentration of 100 mg/L and a removal percentage of 90, then if 5 cfs flowed from its drain into a conveyance system node the mass loading contribution to the node would be 100 x (100 - 90) x 5 x 28.3 L/ft3 = 1,415 mg/sec. If in addition the unit had a surface outflow of 1 cfs into the same node, the mass loading from this flow stream would be 100 x 1 x 28.3 = 2,830 mg/sec.

LID Group Editor

Subcatchment is selected for editing. It is used to identify a group of previously defined LID controls that will be placed within the subcatchment, the sizing of each control, and what percent of runoff from the non-LID portion of the subcatchment each should treat.

The editor displays the current group of LIDs placed in the subcatchment along with buttons for adding an LID unit, editing a selected unit, and deleting a selected unit. These actions can also be chosen by hitting the <**Insert**> key, the <**Enter**> key, and the <**Delete**> key, respectively. Selecting **Add** or **Edit** will bring up an LID Usage Editor where one can enter values for the data fields shown in the Group Editor.



Note that the total **% of Area** for all of the the LID units within a subcatchment must not exceed 100%. The same applies to **% From Imperv** and **% From Perv**. Refer to the LID Usage Editor for the meaning of these parameters.

Editor to specify how a particular LID control will be deployed within the subcatchment. It contains the following data entry fields:

Control Name
The name of a previously defined LID control to be used in the subcatchment.

LID Occupies Full Subcatchment
Select this checkbox option if the LID control occupies the full subcatchment (i.e., the LID is placed in its own separate subcatchment and accepts runoff from upstream subcatchments).

Area of Each Unit
The surface area devoted to each replicate LID unit (sq. ft or sq. m). If the ***LID Occupies Full Subcatchment*** box is checked, then this field becomes disabled and will display the total subcatchment area divided by the number of replicate units. (See LID Placement for options on placing LIDs within subcatchments.) The label below this field indicates how much of the total subcatchment area is devoted to the particular LID being deployed and gets updated as changes are made to the number of units and area of each unit.

Number of Replicate Units
The number of equal size units of the LID practice (e.g., the number of rain barrels) deployed within the subcatchment.

Surface Width Per Unit
The width of the outflow face of each identical LID unit (in ft or m). This parameter applies to roofs, pavement, trenches, and swales that use overland flow to convey surface runoff off of the unit. It can be set to 0 for other LID processes, such as bio-retention cells, rain gardens, and rain barrels that simply spill any excess captured runoff over their berms.

% Initially Saturated
For bio-retention cells, rain gardens, and green roofs this is the degree to which the unit's soil is initially filled with water (0 % saturation corresponds to the wilting point moisture content, 100 % saturation has the

moisture content equal to the porosity). For units with a storage layer it

Map Dimensions Dialog

area map

- Degrees
- None

Lower Left Coordinates

Enter the X,Y coordinates of the lower left corner of the map.

Upper Right Coordinates

Enter the X,Y coordinates of the upper right corner of the map.

Map Units

Select the units used to measure distances on the map. The choices are:

- Feet

Re-Compute Length and Areas

This check box only appears if the Auto-Length option is in effect. If selected, then the lengths of all conduits and the areas of all subcatchments will be re-computed based on the map's new dimensions.

Auto-Size Button

Click this button to automatically set the dimensions based on the coordinates of the objects currently included in the map.

Map Options Dialog

Map. The dialog contains separate tabbed pages that control the appearance of the following items:

- Links (controls thickness of links and making thickness be proportional to value)
- Subcatchments (controls fill style, symbol size, and outline thickness of subcatchment areas)
- Labels (turns display of map labels on/off)
- Annotation (displays or hides node/link ID labels and parameter values)
- Symbols (turns display of storage unit, pump, and regulator symbols on/off)
- Flow Arrows (selects visibility and style of flow direction arrows)
- Background (changes the map's background color)

subcatchment areas are displayed on the Study Area Map.

Fill Style
Selects style used to fill interior of subcatchment areas.

Symbol Size
Sets the size of the symbol placed at the centroid of subcatchments.

Outline Thickness
Sets the thickness of the line used to draw the subcatchment's boundary; set to zero if no boundary should be displayed.

Display Link to Outlet
If checked then a dashed line is drawn between the subcatchment centroid and the subcatchment's outlet node (or subcatchment).

displayed on the Study Area Map.

Node Size
Selects node diameter in pixels.

Proportional to Value Select if node size should increase as the viewed parameter increases in value.

Display Border
Select if a border should be drawn around each node (recommended for light-colored backgrounds).

displayed on the Study Area Map.

Link Size
Sets thickness of links displayed on map.

Proportional to Value Select if link thickness should increase as the viewed parameter increases in value.

Display Border
Check if a black border should be drawn around each link.

map labels are displayed on the Study Area Map.

Use Transparent Text
Check to display text with a transparent background (otherwise an opaque background is used).

At Zoom Of
Selects minimum zoom at which labels should be displayed; labels will be hidden at zooms smaller than this.

Map Options - Annotation

of annotation is provided alongside of the objects on the Study Area Map.

ID Labels

Rain Gages	check to display rain gage ID names
Subcatchments	check to display subcatchment ID names
Nodes	check to display node ID names
Links	check to display link ID names

Subcatchments	check to display value of current subcatchment variable being viewed

Nodes	check to display value of current node variable being viewed

Links check to display value of current link variable being viewed

Use Transparent Text Check to display text with a transparent background (otherwise an opaque background is used).

Font Size Adjusts the size of the font used to display annotation.

At Zoom Of Selects minimum zoom at which annotation should be displayed; all annotation will be hidden at zooms smaller than this.

symbols should be used to display objects on the Study Area Map.

Display Node Symbols
If checked then special node symbols will be used.

Display Link Symbols
If checked then special link symbols will be used.

At Zoom Of
Selects minimum zoom at which symbols should be displayed; symbols will be hidden at zooms smaller than this.

direction arrows are displayed on the Study Area Map.

Arrow Style
Selects style (shape) of arrow to display (select None to hide arrows).

Arrow Size
Sets arrow size.

At Zoom of
Selects minimum zoom at which arrows should be displayed; arrows will be hidden at smaller zooms.

Flow direction arrows will only be displayed after a successful simulation has been made and a computed parameter has been selected for viewing. Otherwise the direction arrow will point from the user-designated start node to the end node.

colors used to paint the map's background with.

created or an existing pollutant is selected for editing. It contains the following fields:

Name
The name assigned to the pollutant.

Units
The concentration units (mg/L, ug/L, or #/L (counts/L)) in which the pollutant concentration is expressed.

Rain Concentration Concentration of the pollutant in rain water (concentration units).

GW Concentration
Concentration of the pollutant in ground water (concentration units).

Initial Concentration Concentration of the pollutant throughout the conveyance system at the start of the simulation.

I&I Concentration Concentration of the pollutant in any Infiltration/Inflow (concentration units).

DWF Concentration
Concentration of the pollutant in any dry weather sanitary flow (concentration units). This value can be overridden for any specific node of the conveyance system by editing the node's Inflows property.

Decay Coefficient
First-order decay coefficient of the pollutant (1/days).

Snow Only
YES if pollutant buildup occurs only when there is snow cover, **NO** otherwise (default is **NO**).

Co-Pollutant
Name of another pollutant whose runoff concentration contributes to the runoff concentration of the current pollutant.

Profile Plot Selection Dialog

drainage system links along which a water depth profile versus distance should be drawn. To define a path using the dialog:

1. Enter the ID of the upstream node of the first link in the path in the **Start Node** edit field (or click on the node on the Study Area Map and then on the button next to the edit field).

Node edit field (or click the node on the map and then click the button next to the edit field).

3. Click the **Find Path** button to have the program automatically identify the path with the smallest number of links between the start and end nodes. These will be listed in the **Links in Profile** box.

4. You can insert a new link into the **Links in Profile** list by selecting the new link either on the Study Area Map or in the Project Browser and then clicking the button underneath the **Links in Profile** list box.

5. Entries in the **Links in Profile** list can be deleted or rearranged by using the , , and buttons underneath the list box

6. Click the **OK** button to view the profile plot.

To save the current set of links listed in the dialog for future use:

1. Click the **Save Current Profile** button.

2. Supply a name for the profile when prompted.

To use a previously saved profile:

1. Click the **Use Saved Profile** button.

2. Select the profile to use from the **Profile Selection** dialog that appears.

Profile Plot Options Dialog

Profile Plot. The dialog contains five pages:

1. **Colors**:

2. **Styles**

   Lets one choose:

   - to use thick lines when drawing conduits and the ground profile
   - to display the ground profile
   - to display conduits only (which provides a closer look at water levels within conduits by removing all other details form the plot).

3. **Axes**

   Edits the main and axis titles, including their fonts, and selects to display axis grid lines.

4. **Vertical Scale**

   Lets one choose the minimum, maximum, and increment values for the vertical axis scale, or have SWMM set the scale automatically. If the increment field contains 0 or is left blank the program will automatically select an increment to use.

5. **Node Labels**

   - Selects to display node ID labels either along the plot's top axis, directly on the plot above the node's crown height, or both.
   - Selects the length of arrow to draw between the node label and the node's crown on the plot (use 0 for no arrows).
   - Selects the font size of the node ID labels.

Check the **Default** box to have these options apply to all new profile plots when they are first created.

Project Defaults Dialog

properties and certain simulation options. The dialog has three tabbed pages covering the following categories:

- Default ID Labels
- Default Subcatchment Properties
- Default Node/Link Properties

Default ID Labels

The **ID Labels** page of the **Project Defaults** dialog form is used to determine how SWMM will assign default ID labels for the visual project components when they are first created. For each type of object you can enter a label prefix in the corresponding entry field or leave the field blank if an object's default name will simply be a number. In the last field you can enter an increment to be used when adding a numerical suffix to the default label. As an example, if C were used as a prefix for Conduits along with an increment of 5, then as conduits are created they receive default names of C5, C10, C15 and so on. An object's default name can be changed by using the Property Editor for visual objects or the object-specific editor for non-visual objects.

Default Node/Link Properties

values for newly created nodes and links. These properties include:

- Node Ponded Area
- Node Invert Elevation
- Storage Surface Area
- Conduit Length
- Conduit Shape and Size
- Conduit Roughness
- Flow Units
- Link Offsets Convention
- Routing Method
- Force Main Equation

The defaults that are automatically assigned to individual objects can be changed by using the object's Property Editor. The choice of Flow Units and Link Offsets Convention can be changed directly on the main window's Status Bar.

The choice of flow units determines whether US or metric units are used for all other quantities. Default values are not automatically adjusted when the unit system is changed from US to metric (or vice versa).

Link Offsets can be specified as either depth above invert or as absolute elevation. When this convention is changed, a dialog will appear giving one the option to automatically re-calculate all existing link offsets in the current project using the newly selected convention.

Default Subcatchment Properties

values for newly created subcatchments. These properties include:

· Subcatchment Area

· Slope

· % Impervious

· Impervious Area Roughness

· Pervious Area Roughness

· Impervious Area Depression Storage

· Pervious Area Depression Storage

· % of Impervious Area with No Depression Storage

· Infiltration Method.

Explanations of these properties can be found in the Subcatchment Properties topic.

These default properties for a particular subcatchment can be modified later on by using the Property Editor.

Changing the Infiltration Method and its default parameters for an existing project will affect all subcatchments that were assigned the previous Infiltration Method. See the description of **Infiltration Model** for the General Simulation Options dialog for details.

nodes, and links that will have detailed time series results saved for viewing after a simulation has been run. The default for new projects is that all objects will have detailed results saved for them. It is also used to select what optional material appears in the Status Report and whether time series results consist of point value (the default) or values averaged over a reporting time step.

The dialog contains three tabbed pages - one each for subcatchments, nodes, and links. It is a stay-on-top form which means that you can select items directly from the Study Area Map or Project Browser while the dialog remains visible.

To include an object in the set that is reported on:

1. Select the tab to which the object belongs (Subcatchments, Nodes or Links).
2. Unselect the "**All**" check box if it is currently checked.
3. Select the specific object either from the Study Area Map or from the listing in the Project Browser.
4. Click the **Add** button on the dialog.
5. Repeat the above steps for any additional objects.

To remove an item from the set selected for reporting:

1. Select the desired item in the dialog's list box.
2. Click the **Remove** button to remove the item.

To remove all items from the reporting set of a given object category, select the object category's page and click the **Clear** button.

To include all objects of a given category in the reporting set, check the "**All**" box on the page for that category (i.e., subcatchments, nodes, or links). This will override any individual items that may be currently listed on the page.

To dismiss the dialog click the **Close** button.

In addition the following reporting options can be selected from this dialog: **Report Input Summary**

Check this option to have the simulation's Status Report list a summary of the project's input data.

Report Control Actions

Check this option to have the simulation's Status Report list all discrete

Scatter Plot Dialog

graphed against one another in a scatter plot. Use the dialog as follows:

1. Select a **Start Date** and **End Date** for the plot (the default is the entire simulation period).

the horizontal axis).

- Object Category (Subcatchment, Node or Link)
- Object ID (enter a value or click on the object either on the Study Area Map or in the Project Browser and then click the button on the dialog)
- Variable to plot (choices depend on the category of object selected).

3. Do the same for the Y-variable (the quantity plotted along the vertical axis).
4. Click the **OK** button to create the plot.

Simulation Options Dialog

how a SWMM simulation is made. The dialog consists of the following tabbed pages:

- Time Step Options
- General Options
- Dynamic Wave Routing Options
- Interface File Options

After selecting the desired options, click the **OK** button to save your choices or the **Cancel** button to abandon them.

Events and Reporting simulation options have their own specialized dialog forms (see Events Editor and Reporting Options Dialog).

following options:

Process Models

Select which process models (Rainfall/Runoff, Rainfall Dependent I/I, Snow Melt, Groundwater, Flow Routing, and Water Quality) should be included in the analysis.

Infiltration Model

This option selects the default method used to model infiltration of rainfall into the upper soil zone of subcatchments. The choices are:

- Horton
- Green-Ampt
- Modified Green-Ampt
- Curve Number

All new subcatchments added to a project will default to using the selected method. For existing subcatchments, their infiltration method will only change if they had been using the previous default option. That would require re-entering values for the infiltration parameters in each such subcatchment, unless the change was between the two Horton options or the two Green-Ampt options. A prompt is issued asking if SWMM should automatically assign a default set of parameter values to all subcatchments that switch between two incompatible types of infiltration methods.

Routing Model

This option determines which method is used to route flows through the conveyance system. The choices are:

- Steady Flow
- Kinematic Wave
- Dynamic Wave

See the Flow Routing topic for more details.

Allow Ponding
Checking this option will allow excess water to collect atop nodes and be re-introduced into the system as conditions permit. In order for ponding to actually occur at a particular node, a non-zero value for its Ponded Area attribute must be used.

Minimum Conduit Slope
This is the minimum value allowed for a conduit's slope (%). If zero (the default) then no minimum is imposed (although SWMM uses a lower limit on elevation drop of 0.001 ft (0.00035 m) when computing a conduit slope).

and ending dates/times of a simulation.

Start Analysis On
Enter the date (month/day/year) and time of day when the simulation begins.

Start Reporting On
Enter the date and time of day when reporting of simulation results is to begin. Using a date prior to the start date is the same as using the start date.

End Analysis On
Enter the date and time when the simulation is to end.

Start Sweeping On
Enter the day of the year (month/day) when street sweeping operations begin. The default is January 1.

End Sweeping On
Enter the day of the year (month/day) when street sweeping operations end. The default is December 31.

Antecedent Dry Days
Enter the number of days with no rainfall prior to the start of the simulation. This value is used to compute an initial buildup of pollutant load on the surface of subcatchments.

If rainfall or climate data are read from external files, then the simulation dates should be set to coincide with the dates recorded in these files.

length of the time steps used for runoff computation, routing computation and results reporting. Time steps are specified in days and hours:minutes:seconds except for flow routing which is entered as decimal seconds.

Reporting Time Step
Enter the time interval for reporting of computed results.

Runoff - Wet Weather Time Step Enter the time step length used to compute runoff from subcatchments during periods of rainfall, or when ponded water still remains on the surface, or when LID controls are still infiltrating or evaporating runoff.

Runoff - Dry Weather Time Step Enter the time step length used for runoff computations (consisting essentially of pollutant buildup) during periods when there is no rainfall, no ponded water, and LID controls are dry. This must be greater or equal to the Wet Weather time step.

Control Rule Time Step
Enter the time step length used for evaluating Control Rules. The default is 0 which means that controls are evaluated at every routing time step.

Routing Time Step
Enter the time step length used for routing flows and water quality constituents through the conveyance system. Note that Dynamic Wave routing requires a much smaller time step than the other methods of flow routing.

Steady Flow Periods
This set of options tells SWMM how to identify and treat periods of time when system hydraulics are not changing. The system is considered to be in a steady flow period if:

1. The percent difference between total system inflow and total system outflow is below the **System Flow Tolerance**,

the previous time step for all points in the conveyance system are below the **Lateral Flow Tolerance**.

Checking the **Skip Steady Flow Periods** box will make SWMM keep using the most recently computed conveyance system flows (instead of computing a new flow solution) whenever the above criteria are met. Using this feature can help speed up simulation run times at the expense of reduced accuracy.

Simulation Options - Dynamic Wave

parameters that control how the dynamic wave flow routing computations are made. These parameters have no effect for the other flow routing methods.

- **Slope & Froude** - both water surface slope and Froude number are checked
- **None** - no check for normal flow limitation is made.

The first two choices were used in earlier versions of SWMM while the third choice, which checks for either condition, is now the recommended one.

Inertial Terms
Indicates how the inertial terms in the St. Venant momentum equation will be handled.

- **KEEP** maintains these terms at their full value under all conditions.
- **DAMPEN** reduces the terms as flow comes closer to being critical and ignores them when flow is supercritical.
- **IGNORE** drops the terms altogether from the momentum equation, producing what is essentially a Diffusion Wave solution.

Normal Flow Criterion
Selects the basis used to determine when supercritical flow limits a conduit's maximum flow to normal flow. The choices are:

- **Slope** - water surface slope only (i.e., water surface slope > conduit slope)

Force Main Equation
Selects which equation will be used to compute friction losses during pressurized flow for conduits that have been assigned a Circular Force Main cross-section. The choices are either the Hazen-Williams equation or the Darcy-Weisbach equation.

Surcharge Method
Selects which method will be used to handle surcharge conditions. The **Extran** option uses a variation of the Surcharge Algorithm from previous versions of SWMM to update nodal heads when all connecting links become full. The **Slot** option uses a Preissmann Slot to add a small amount of virtual top surface width to full flowing pipes so that SWMM's normal procedure for updating nodal heads can continue to be used.

Variable Time Step
Check the box if an internally computed variable time step should be used at each routing time period and select an adjustment (or safety) factor to apply to this time step. The variable time step is computed so as to satisfy the Courant condition within each conduit. A typical adjustment factor would be 75% to provide some margin of conservatism. The computed variable time step will not be less than the minimum variable step discussed below nor be greater than the fixed time step specified on the Time Steps page of the dialog.

Minimum Variable Time Step
This is the smallest time step allowed when variable time steps are used. The default value is 0.5 seconds. Smaller steps may be warranted, but they can lead to longer simulations runs without much improvement in solution quality.

Time Step for Conduit Lengthening
This is a time step, in seconds, used to artificially lengthen conduits so that they meet the Courant stability criterion under full-flow conditions (i.e., the travel time of a wave will not be smaller than the specified conduit lengthening time step). As this value is decreased, fewer conduits will require lengthening. A value of 0 means that no conduits will be lengthened. The ratio of the artificial length to the original length for each conduit is listed in the Flow Classification table that appears in the simulation's Summary Report.

Minimum Nodal Surface Area
This is a minimum surface area used at nodes when computing changes in water depth. If 0 is entered, then the default value of 12.566 sq. ft (1.167 sq. m) is used. This is the area of a 4-ft diameter manhole. The value entered should be in square feet for US units or square meters for SI units.

Head Convergence Tolerance

This is the maximum difference in computed heads between successive trials of SWMMs iterative method for computing a dynamic wave hydraulic solution that determines when convergence is reached within a given time step. The default tolerance is 0.005 ft (0.0015 m).

Maximum Trials Per Time Step
This is the maximum number of trials that SWMM will use in its iterative method for computing a dynamic wave hydraulic solution within each time step. The default value is 8.

Number of Parallel Threads to Use
This selects the number of parallel computing threads to use on machines equipped with multi-core processors. The default is 1. Clicking the button will display the number of physical cores and logical processors available.

Clicking the **Apply Defaults** label will set all the Dynamic Wave options to their default values.

Simulation Options - Interface Files

specify which interface files will be used or saved during the simulation. The page contains a list box with three buttons underneath it. The list box lists the currently selected files, while the buttons are used as follows:

Add adds a new interface file specification to the list.

Edit edits the properties of the currently selected interface file.

Delete deletes the currently selected interface from the project (but not from your hard drive).

When the Add or Edit buttons are clicked, an Interface File Selection dialog appears where one can specify the type of interface file, whether it should be used or saved, and its name.

Snow Pack Editor

The **Snow Pack Editor** is invoked whenever a new Snow Pack object is created or an existing snow pack is selected for editing. The editor contains a data entry field for the snow pack's name and two tabbed pages, one for Snow Pack Parameters and one for Snow Removal Parameters.

melt parameters and initial conditions for snow that accumulates over three different types of areas: the impervious area that is plowable (i.e., subject to snow removal), the remaining impervious area, and the entire pervious area. The page contains a data entry grid which has a column for each type of area and a row for each of the following parameters:

Minimum Melt Coefficient
The degree-day snow melt coefficient that occurs on December 21. Units are either in/hr-deg F or mm/hr-deg C.

Maximum Melt Coefficient
The degree-day snow melt coefficient that occurs on June 21. Units are either in/hr-deg F or mm/hr-deg C. For a short term simulation of less than a week or so it is acceptable to use the same value for both the minimum and maximum melt coefficients.

The minimum and maximum snow melt coefficients are used to estimate a melt coefficient that varies by day of the year. The latter is used in the following degree-day equation to compute the melt rate for any particular day: Melt Rate = (Melt Coefficient) * (Air Temperature - Base Temperature).

Base Temperature
Temperature at which snow begins to melt (degrees F or C).

Fraction Free Water Capacity
The volume of a snow pack's pore space which must fill with melted snow before liquid runoff from the pack begins, expressed as a fraction of snow pack depth.

Initial Snow Depth
Depth of snow at the start of the simulation (water equivalent depth in inches or millimeters).

Initial Free Water
Depth of melted water held within the pack at the start of the simulation (inches or mm). This number should be at or below the product of the

initial snow depth and the fraction free water capacity.

snow removal occurs within the plowable area of a snow pack. The following parameters govern this process:

Depth at which snow removal begins (in or mm)
Depth which must be reached before any snow removal begins.

Fraction transferred out of the watershed
The fraction of snow depth that is removed from the system (and does not become runoff).

Fraction transferred to the impervious area
The fraction of snow depth that is added to snow accumulation on the pack's impervious area.

Fraction transferred to the pervious area
The fraction of snow depth that is added to snow accumulation on the pack's pervious area.

Fraction converted to immediate melt
The fraction of snow depth that becomes liquid water which runs onto any subcatchment associated with the snow pack.

Fraction moved to another subcatchment
The fraction of snow depth which is added to the snow accumulation on some other subcatchment. The name of the subcatchment must also be provided.

The various removal fractions must add up to 1.0 or less. If less than 1.0, then some remaining fraction of snow depth will be left on the surface after all of the redistribution options are satisfied.

analysis to be made on a computed simulation result. It contains the following data fields:

Object Category
Select the category of object to analyze (Subcatchment, Node, Link, or System).

Object Name
Enter the ID name of the object to analyze. Instead of typing in an ID name, you can select the object on the Study Area Map or in the Project Browser and then click the ✚ button to select it into the object name field.

Variable Analyzed
Select the variable to be analyzed. The available choices depend on the object category selected (e.g., rainfall, losses, or runoff for subcatchments; depth, inflow, or flooding for nodes; depth, flow, velocity, or capacity for links; water quality for all categories).

Event Time Period
Select the length of the time period that defines an event. The choices are daily, monthly, or event-dependent. In the latter case, the event period depends on the number of consecutive reporting periods where simulation results are above the threshold values defined below.

Statistic
Choose an event statistic to be analyzed. The available choices depend on the choice of variable to be analyzed and include such quantities as mean value, peak value, event total, event duration, and inter-event time (i.e., the time interval between the midpoints of successive events). For water quality variables the choices include mean concentration, peak concentration, mean loading, peak loading, and event total load.

Event Thresholds
These define minimum values that must be met for an event to occur:

- The **Analysis Variable** threshold specifies the minimum value of the variable being analyzed that must be exceeded for a time period to be included in an event.

rainfall volume that must be exceeded for a result to be counted as part of an event. Enter 0 if no volume threshold applies.

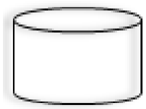
- **Separation Time** sets the minimum number of hours that must occur between the end of one event and the start of the next event. Events with fewer hours are combined together. This value applies only to event-dependent time periods (not to daily or monthly event periods).

If a particular type of threshold does not apply, then leave the field blank.

area varies with depth above the bottom of the unit. . It is invoked when the Storage Shape property of a storage node is selected for editing. There are six types of shapes one can choose from:

Cylindrical

The storage unit has vertical sides and an elliptical base. The equation for surface area is:



*Area = (π / 4) * (L * W)* where L = base major axis length and W = base minor axis width. If only the surface area is known then one can use the **Functional** storage option instead.

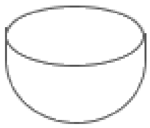
Conical

The storage unit is shaped as a truncated elliptical cone. The equation for surface area is:

*Area = π * (L * W / 4 + W * Z * Depth + (W / L) * (Z * Depth)^2)* where L = base major axis length, W = base minor axis width and Z = side slope (run / rise) of a vertical slice through the major axis.

Parabolic

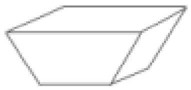
The storage unit has the shape of an elliptical paraboloid. The equation for surface area is:



*Area = (π / 4) * (L * W / H) * Depth* where L = major axis length at height H and W = minor axis width at height H. This shape can also be described using the **Functional** storage option.

Pyramidal

This is for storage units shaped as a truncated rectangular pyramid or a rectangular box. The equation for surface area is:



*Area = L * W + 2 * (L + W) * Z * Depth +(2 * Z * Depth)^2*

where L = base length, W = base width and Z = side slope (run / rise) (which would be 0 for a box).

Functional

The following general function is used to relate surface area to depth:

$$Area = a0 + a1 * Depth ^{a2}$$

where *a0*, *a1*, and *a2* are user supplied coefficients. Here are coefficient values for some particular shapes:

- Shapes with vertical sides (such as a cylinder or rectangular prism):

a0 = area of the base
a1 = *a2* = 0

- Open channel with a trapezoidal cross section and vertical ends:

$a0 = W * L$
$a1 = 2 * Z * L$
$a2 = 1$

where *W* = bottom width of cross section, *L* = channel length, and *Z* = side slope.

- Open channel with a parabolic cross section and vertical ends:

$a0 = 0$
$a1 = W * L * H^{0.5}$
$a2 = 1$

where *W* = top width, *L* = channel length and *H* = full height.

- Elliptical paraboloid:

$a0 = 0$
$a1 = \pi * L * W / H$
$a2 = 1$

where *L* is the length of the major axis and *W* the length of the minor axis at full height *H*.

- Circular non-truncated cone:

$a0 = 0$
$a1 = (\pi/4) * (W / H)^2$
$a2 = 2$

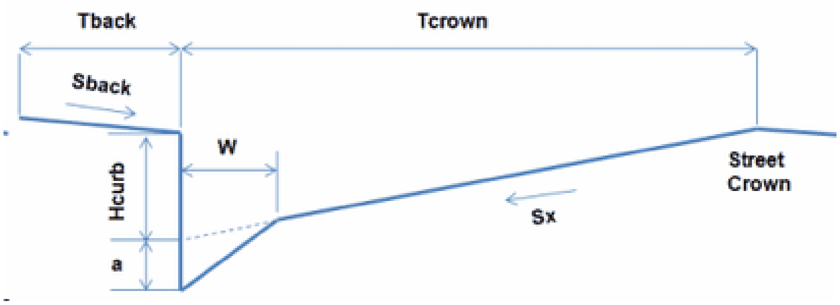
where W is the cone's diameter at height *H*.

Tabular
This method uses a tabular Storage Curve to relate surface area to depth. It can represent natural depressions with irregular shaped contour intervals, spheroid storage vessels or conventional shapes with different base sizes stacked on top of one another. The first point supplied to the curve should be the surface area of the unit's base at a depth of 0. Otherwise it will be assumed that the unit has zero surface area at its base. The curve will be extrapolated outwards to meet the unit's maximum depth if need be.

For each of these methods, depth is measured in feet and surface area in square feet for US units, while meters and square meters, respectively, are used for SI units.

Clicking the **Show Volume Calculator** label will display a panel where one can see what the surface area and stored volume will be at a specified water depth in the selected storage shape.

roadway cross-section. It is invoked when a new Street object is created or is selected for editing from the Project Browser , or when a STREET shape is chosen from the Cross- Section Editor. The editor asks that the following dimensions be provided for the portion of the street extending from the high point of the roadway to the curb and beyond to any backing that might exist (see figure below):



Street Section Name
The name assigned to the street cross section. Conduits with a STREET shape cross section will refer to this name to identify its cross section dimensions.

Road Width (Tcrown)
The distance from the curb to the high point of the street roadway (i.e., the street crown) (feet or meters). Traffic lanes are typically 10 to 12 feet (3.3 to 3.7 meters) wide with gutters being 1 to 3 feet (0.3 to 1 meter) wide.

Curb Height (Hcurb)
The height of the curb with respect to the street's cross slope (feet or meters). Typical heights are 4 to 8 inches with 6 inches (0.5 feet or 0.15 meters) being standard in the U.S..

Cross Slope (Sx)
The slope of the roadway portion of the cross section (percent). Cross slopes range between 1 to 4 percent with 2 percent being the most common value.

Street Roughness

Manning's roughness coefficient (n) for the street surface. Typical values

Table by Object Dialog

several variables for a single object. Use the dialog as follows:

1. Select a **Start Date** and **End Date** for the table (the default is the entire simulation period).

3. Choose an **Object Category** (Subcatchment, Node, Link, or System).

4. Identify a specific object in the category by clicking the object either on the Study Area Map or in the Project Browser and then clicking the button on the dialog. Only a single object can be selected for this type of table.

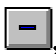
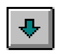
5. Check off the variables to be tabulated for the selected object. The available choices depend on the category of object selected.

6. Click the **OK** button to create the table.

Table by Variable Dialog

single variable for one or more objects. Use the dialog as follows:

1. Select a **Start Date** and **End Date** for the table (the default is the entire simulation period).
3. Choose an **Object Category** (Subcatchment, Node or Link).
4. Select a simulated variable to be tabulated. The available choices depend on the category of object selected.
5. Identify one or more objects in the category by successively clicking the object either on the Study Area Map or in the Project Browser and then clicking the  button on the dialog.
6. Click the **OK** button to create the table.

A maximum of 6 objects can be selected for a single table. Objects already selected can be deleted, moved up in the order or moved down in the order by clicking the , , and  buttons, respectively.

is created or an existing time pattern is selected for editing. The editor contains the following data entry fields:

Name
Enter the name assigned to the time pattern.

Type
Select the type of time pattern being specified. The choices are Monthly, Daily, Hourly and Weekend Hourly.

Description
Provide an optional comment or description for the time pattern. If more than one line is needed, click the button to launch a multi-line comment editor.

Multipliers
Enter a value for each multiplier. The number and meaning of the multipliers changes with the type of time pattern selected:

MONTHLY	One multiplier for each month of the year
DAILY	One multiplier for each day of the week
HOURLY	One multiplier for each hour from 12 midnight to 11 PM
WEEKEND	Same as for *HOURLY* except applied for weekend days

In order to maintain an average dry weather flow or pollutant concentration at its specified value (as entered on the Inflows Editor), the multipliers for a pattern should average to 1.0.

Time Series Editor

created or an existing time series is selected for editing. To use the Time Series Editor:

column remains empty).

For rainfall time series, it is only necessary to enter periods with non-zero rainfall amounts. SWMM interprets the rainfall value as a constant value lasting over the recording interval specified for the rain gage which utilizes the time series. For all other types of time series, SWMM uses interpolation to estimate values at times that fall in between the recorded values.

1. Enter values for the following standard items:

Name	Name of the time series.
Description	Optional comment or description of what the time series represents. Click the button to launch a multi-line comment editor if more than one line is needed.

2. Select whether to use an external file as the source of the data or to enter the data directly into the form's data entry grid.

3. If the external file option is selected, click the button to locate the file's name. The file's contents must be formatted in the same manner as the direct data entry option discussed below. See the description of Time Series Files for details.

4. For direct data entry, enter values in the data entry grid as follows:

Date Column	Optional date (in month/day/year format) of the time series values (only needed at points in time where a new date occurs).
Time Column	If dates are used, enter the military time of day for each time series value (as hours:minutes or decimal hours). If dates are not used, enter time as hours since the start of the simulation.
Value Column	Time series numerical values.

A graphical plot of the data in the grid can be viewed in a separate window by clicking the **View** button. Right clicking over the grid will make a popup Edit menu appear. It contains commands to cut, copy,

Time Series Plot Selection Dialog

variables whose computed time series will be graphed in a Time Series Plot. The dialog is used as follows:

1. Select a **Start Date** and **End Date** for the plot (the default is the entire simulation period).

3. Add up to six different data series to the plot by clicking the **Add** button above the data series list box.

4. Use the **Edit** button to make changes to a selected data series or the **Delete** button to delete a data series.

5. Use the **Up** and **Down** buttons to change the order in which the data series will be plotted.

6. Click the **OK** button to create the plot.

When you click the **Add** or **Edit** buttons a Data Series Selection dialog will be displayed for selecting a particular object and variable to plot.

Data Series Selection Dialog

Name (for system variables)

Selection dialog to select a data series for plotting in a Time Series Plot. It contains the following data fields:

Object Type	the type of object to plot (Subcatchment, Node, Link or System).
Variable	the variable whose time series will be plotted (choices vary by object type).
Legend Label	the text to use in the legend for the data series. If left blank, a default label made up of the object type, name, variable and units will be used (e.g. Link C16 Flow (CFS)).
Axis	whether to use the left or right vertical axis to plot the data series.

As you select objects on the Study Area Map or in the Project Browser their types and ID names will automatically appear in this dialog.

Click the **Accept** button to add/update the data series into the plot or click the **Cancel** button to disregard your edits. You will then be returned to the Time Series Plot Selection dialog where you can add or edit another data series.

To make a precipitation time series display in inverted fashion on a plot, assign it to the right axis and after the plot is displayed, use the Graph Options Dialog to invert the right axis and expand the scales of both the left and right axes (so it doesn't overlap another data series).

Tool Properties Dialog

tool that has been added to the Tools menu of SWMM's Main Menu bar. It contains the following data entry fields: **Tool Name**
This is the name to be used for the tool when it is displayed in the Tools Menu.
Program
Enter the full path name to the program that will be launched when the tool is selected. You can click the button to bring up a standard Windows file selection dialog from which you can search for the tool's executable file name.
Working Directory
This field contains the name of the directory that will be used as the working directory when the tool is launched. You can click the button to bring up a standard directory selection dialog from which you can search for the desired directory. You can also enter the macro symbol `$PROJDIR` to utilize the current SWMM project's directory or `$SWMMDIR` to use the directory where the SWMM 5 executable resides. Either of these macros can also be inserted into the Working Directory field by selecting its name in the list of macros provided on the dialog and then clicking the button. This field can be left blank, in which case the system's current directory will be used.
Parameters
This field contains the list of command line arguments that the tool's executable program expects to see when it is launched. Multiple parameters can be entered field as long as they are separated by spaces. A number of special macro symbols have been pre-defined, as listed in the Macros list box of the dialog, to simplify the process of listing the command line parameters. When one of these macro symbols is inserted into the list of parameters, it will be expanded to its true value when the tool is launched. A specific macro symbol can either be typed into the Parameters field or be selected from the Macros list (by clicking on it) and then added to the parameter list by clicking the button.
Disable SWMM while executing Check this option if SWMM should be minimized and disabled while the tool is executing. Normally you will need to employ this option if the tool produces a modified input file or output file, such as when the `$INPFILE` or `$OUTFILE` macros are used as

created or an existing Transect is selected for editing. It contains the following data entry fields: **Name**
The name assigned to the transect.
Description
An optional comment or description of the transect.
Station/Elevation Data Grid Values of distance from the left side of the channel along with the corresponding elevation of the channel bottom as one moves across the channel from left to right, looking in the downstream direction. The elevations can be relative to any reference point, such as the bottom of the channel, and not necessarily mean sea level. Up to 1500 data values can be entered.
Roughness
Values of Manning's roughness coeffcient (*n*)for the left overbank, right overbank, and main channel portion of the transect. The overbank roughness values can be zero if no overbank exists.
Bank Stations
The distance values appearing in the Station/Elevation grid that mark the end of the left overbank and the start of the right overbank. Use 0 to denote the absence of an overbank.
Modifiers

- The **Stations** modifier is a factor by which the distance between each station will be multiplied when the transect data is processed by SWMM. Use a value of 0 if no such factor is needed.
- The **Elevations** modifier is a constant value that will be added to each elevation value.
- The **Meander** modifier is the ratio of the length of a meandering main channel to the length of the overbank area that surrounds it. This modifier is applied to all conduits that use this particular transect for their cross section. It assumes that the length supplied for these conduits is that of the longer main channel. SWMM will use the shorter overbank length in its calculations while increasing the main channel roughness to account for its longer length. The modifier is ignored if it is left blank or set to 0.

Treatment Editor

node is selected from the Property Editor. It displays a list of the project's pollutants with an edit box next to each as shown below.

Pollutant	Treatment Expression
TSS	C = 0.523 * TSS^0.5 * FLOW^1.2
Lead	

Enter a valid treatment expression in the box next to each pollutant which receives treatment. Click the **OK** button to accept your edits or click **Cancel** to ignore them.

Any of the following math functions (which are case insensitive) can be used in a treatment expression:

· abs(x) for absolute value of x

· step(x) which is 0 for x <= 0 and 1 otherwise

· sqrt(x) for the square root of x

· the standard trig functions (sin, cos, tan, and cot)

· the inverse trig functions (asin, acos, atan, and acot)

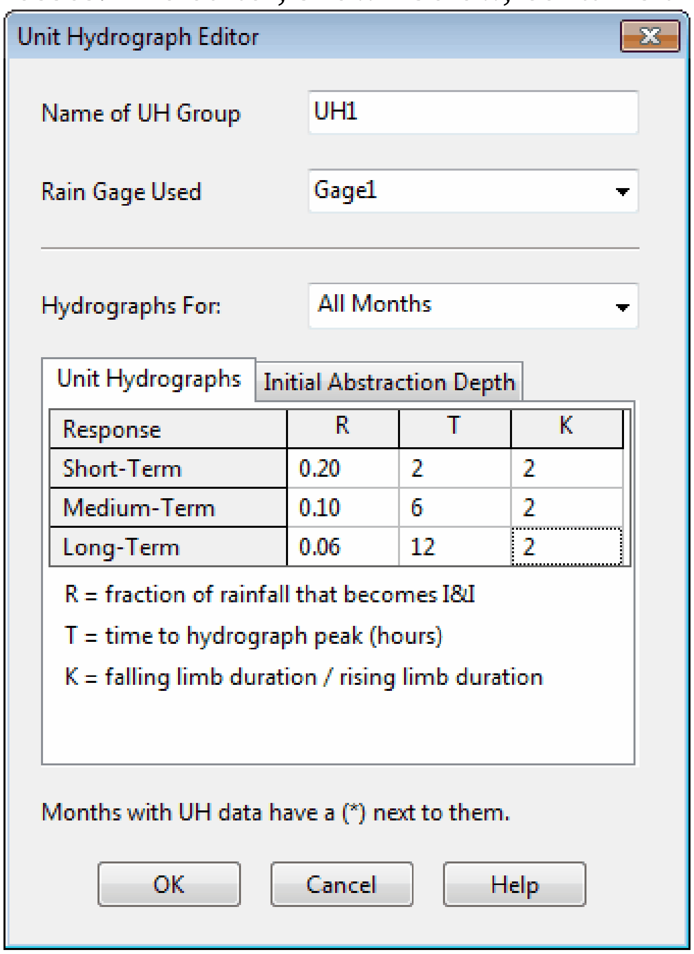
· the hyperbolic trig functions (sinh, cosh, tanh, and coth)

along with the standard operators +, -, *, /, ^ (for exponentiation) and any level of nested parentheses..

Unit Hydrograph Editor

Hydrograph object is created or an existing one is selected for editing. It is used to specify the shape parameters and rain gage for a group of triangular unit hydrographs. These hydrographs are used to compute rainfall-derived inflow/infiltration (RDII) flow at selected nodes of the drainage system.

A UH group can contain up to 12 sets of unit hydrographs (one for each month of the year), and each set can consist of up to 3 individual hydrographs (for short-term, intermediate-term, and long-term responses, respectively) as well as parameters that describe any initial abstraction losses. The editor, shown below, contains the following data entry fields:



Name of UH Group

Enter the name assigned to the UH Group.

Rain Gage Used

Type in (or select from the dropdown list) the name of the rain gage that supplies rainfall data to the unit hydrographs in the group.

Hydrographs For:

Error Messages

Run Time Errors	101 - 107
Property Errors	108 - 195
Format Errors	200 - 233
File Errors	301 - 357
Warnings	01 - 11

101:

There is not enough physical memory in the computer to analyze the study area.

ERROR 103: **cannot solve KW equations for Link xxx.**

The internal solver for Kinematic Wave routing failed to converge for the specified link at some stage of the simulation.

ERROR 105: **cannot open ODE solver.**

The system could not open its Ordinary Differential Equation solver.

ERROR 107: **cannot compute a valid time step.**

A valid time step for runoff or flow routing calculations (i.e., a number greater than 0) could not be computed at some stage of the simulation.

108:

The name of the element identified as the outlet of a subcatchment belongs to both a node and a subcatchment in the project's data base.

ERROR 109: **invalid parameter values for Aquifer xxx.**

The properties entered for an aquifer object were either invalid numbers or were inconsistent with one another (e.g., the soil field capacity was higher than the porosity).

ERROR 110: **ground elevation is below water table for Subcatchment xxx.**

The ground elevation assigned to a subcatchment's groundwater parameters cannot be below the initial water table elevation of the aquifer object used by the subcatchment.

ERROR 111: **invalid length for Conduit xxx.**

Conduits cannot have zero or negative lengths.

ERROR 112: **elevation drop exceeds length for Conduit xxx.**

The elevation drop across the ends of a conduit cannot be greater than the conduit's length. Check for errors in the length and in both the invert elevations and offsets at the conduit's upstream and downstream nodes.

ERROR 113: **invalid roughness for Conduit xxx.**

Conduits cannot have zero or negative roughness values.

ERROR 114: **invalid number of barrels for Conduit xxx.**

Conduits must consist of one or more barrels.

ERROR 115: **adverse slope for Conduit xxx.**

Under Steady or Kinematic Wave routing, all conduits must have positive slopes. This can usually be corrected by reversing the inlet and outlet nodes of the conduit (i.e., right click on the conduit and select Reverse from the popup menu

Format Errors

200:

This message appears when one or more input file parsing errors (the 200-series errors) occur.

ERROR 201: **too many characters in input line.**

A line in the input file cannot exceed 1024 characters.

ERROR 203: **too few items at line n of input file.**

Not enough data items were supplied on a line of the input file.

ERROR 205: **invalid keyword at line n of input file.**

An unrecognized keyword was encountered when parsing a line of the input file.

ERROR 207: **duplicate ID name at line n of input file.**

An ID name used for an object was already assigned to an object of the same category.

ERROR 209: **undefined object xxx at line n of input file.**

A reference was made to an object that was never defined. An example would be if node 123 were designated as the outlet point of a subcatchment, yet no such node was ever defined in the study area.

ERROR 211: **invalid number xxx at line n of input file.**

Either a string value was encountered where a numerical value was expected or an invalid number (e.g., a negative value) was supplied.

ERROR 213: **invalid date/time xxx at line n of input file.**

An invalid format for a date or time was encountered. Dates must be entered as month/day/year and times as either decimal hours or as hour:minute:second.

ERROR control rule clause out of sequence at line n of input file.

File Errors

301:

The input, report, and binary output files specified on the command line cannot have the same names.

ERROR 303: **cannot open input file.**

The input file either does not exist or cannot be opened (e.g., it might be in use by another program).

ERROR 305: **cannot open report file.**

The report file cannot be opened (e.g., it might reside in a directory to which the user does not have write privileges).

ERROR 307: **cannot open binary results file.**

The binary output file cannot be opened (e.g., it might reside in a directory to which the user does not have write privileges).

ERROR 309: **error writing to binary results file.**

There was an error in trying to write results to the binary output file (e.g., the disk might be full or the file size exceed the limit imposed by the operating system).

ERROR 311: **error reading from binary results file.**

The command line version of SWMM could not read results saved to the binary output file when writing results to the report file.

ERROR 313: **cannot open scratch rainfall interface file.**

SWMM could not open the temporary file it uses to collate data together from external rainfall files.

ERROR 315: **cannot open rainfall interface file xxx.**

SWMM could not open the specified rainfall interface file, possibly because it does not exist or because the user does not have write privileges to its directory.

Warning Messages

01: **Rain Gage xxx.**

The wet weather time step was automatically reduced so that no period with rainfall would be skipped during a simulation.

WARNING 02: **maximum depth increased for Node xxx.**

The maximum depth for the node was automatically increased to match the top of the highest connecting conduit.

WARNING 03: **negative offset ignored for Link xxx.**

The link's stipulated offset was below the connecting node's invert so its actual offset was set to 0.

WARNING 04: **minimum elevation drop used for Conduit xxx.**

The elevation drop between the end nodes of the conduit was below 0.001 ft (0.00035 m) so the latter value was used instead to calculate its slope.

WARNING 05: **minimum slope used for Conduit xxx.**

The conduit's computed slope was below the user-specified Minimum Conduit Slope so the latter value was used instead.

WARNING 06: **dry weather time step increased to wet weather time step.**

The user-specified time step for computing runoff during dry weather periods was lower than that set for wet weather periods and was automatically increased to the wet weather value.

WARNING 07: **routing time step reduced to wet weather time step.**

The user-specified time step for flow routing was larger than the wet weather runoff time step and was automatically reduced to the runoff time step to prevent loss of accuracy.

WARNING 08: **elevation drop exceeds length for Conduit xxx.**

The elevation drop across the ends of a conduit exceeds its

<u>Subcatchment Width</u> An initial estimate of the characteristic width is given by the subcatchment area divided by the average maximum overland flow length. The maximum overland flow length is the length of the flow path from the outlet to the furthest drainage point of the subcatchment. Maximum lengths from several different possible flow paths should be averaged. These paths should reflect slow flow, such as over pervious surfaces, more than rapid flow over pavement, for example. Adjustments should be made to the width parameter to produce good fits to measured runoff hydrographs.