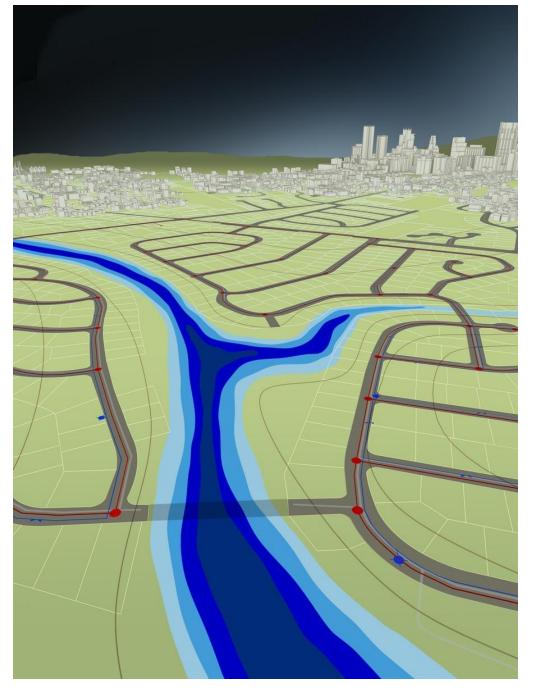# Autodesk® Storm & Sanitary Analysis 2012

Technical Capabilities and Functionalities

Autodesk

Contents

Autodesk SSA Technical Capabilities and Functionalities

Autodesk[®] Storm and Sanitary Analysis 2012 Extension is a comprehensive hydrology and hydraulic analysis application for planning and designing urban drainage systems, highway drainage systems, storm sewers, and sanitary sewers.

Major technical capabilities and functionalities include:

- Bidirectional exchange of data with AutoCAD[®] Map 3D 2012 and AutoCAD[®] Civil 3D[®] 2012 software
- Widely accepted hydrology analysis methods, including NRCS (SCS) TR-20/TR-55, Rational Method, Modified Rational, HEC-1, EPA SWMM, UK Modified Rational (Wallingford Procedure), and more
- Hydraulic analysis of pipes, open channels, streams, bridges, culverts, roadway inlet catch basins, and more
- Advanced hydrodynamic routing that can handle backwater, surcharging, flow splits, and more
- Support of networks of unlimited size
- Compare pre- and post-development conditions
- Analysis and design of detention and retention ponds
- Analysis of stormwater best management practices (BMPs)
- Water quality modeling
- Sophisticated graphical output providing detailed plan view plots, profile plots, and time series plots
- Customizable reports and templates for agency regulatory review
- Geographic information system (GIS) interoperability
- Support for both U.S. and metric (SI) units

Overall Capabilities

Autodesk Storm and Sanitary Analysis can simultaneously model complex hydrology, hydraulics, and water quality. This software can be used for designing and analyzing:

- Highway drainage systems (including curb and gutter stormwater inlets)
- Stormwater sewer networks and interconnected detention ponds
- Subdivision drainage systems
- Design and sizing of detention ponds and outlet structures
- Bridges and culverts, including roadway overtopping
- Water quality studies
- Sanitary sewers, lift stations, force mains, combined sewer overflows (CSOs), and sanitary sewer overflows (SSOs)

Typical Applications

The software has been used in thousands of sewer and stormwater studies throughout the world. Typical applications include:

- Design and sizing of drainage system components for better flood control
- Design and sizing of detention facilities for better flood control and water quality protection
- Floodplain mapping of natural channel systems
- Designing control strategies for minimizing CSOs
- Evaluating the impact of inflow and infiltration on SSOs
- Generating non–point source pollutant loadings for waste load allocation studies
- Evaluating the effectiveness of BMPs for reducing wet weather pollutant loadings

AutoCAD Map 3D and Civil 3D Support

Autodesk Storm and Sanitary Analysis easily shares data with AutoCAD Map 3D and Civil 3D software. Using GIS shapefiles, Hydraflow Storm Sewers Extension, or LandXML files, the software can share subbasin, sewer pipe, and structure entity data with AutoCAD Map 3D. In addition, AutoCAD drawings can be loaded as a background layer, enabling you to quickly digitize a network model, confirm the network layout, or enhance the output modeling results.

The software can automatically create plan and profile drawings. This helps speed up the creation of final deliverables associated with your engineering project. Profile sheets include:

- Maximum HGL and EGL
- Critical depth
- Maximum discharge
- Maximum flow depth
- Maximum flow velocity
- Pipe dimensions (sizes, inverts, etc.)
- Minimum pipe cover
- Sump and rim elevations

All elements are stored on their own individual drawing layers, so you can quickly change colors, line styles, text styles, and more. You can change the default settings, such as colors and annotations, to fit your corporate CAD standards.

Model Development

Simulation models can be more quickly developed using a variety of different sources. Network components can be directly imported from CAD and GIS. The network model can be interactively created using a mouse by pointing and clicking. Graphical symbols are used to represent network elements, such as manholes, pipes, pumps, weirs, ditches, channels, catch basin inlets, and detention ponds. The software enables you, at any time, to interactively add, insert, delete, or move any network element, automatically updating the model. For example, selecting and moving a manhole automatically moves all connected pipes, ditches, channels, and pumps.

Pipes can be curvilinear and lengths can be automatically computed. Scanned aerial orthophoto TIFF images and maps, as well as GIS and CAD files of streets, parcels, and buildings can be imported and displayed as background images. This enables you to more quickly digitize a network model, confirm the network layout, or enhance the output modeling results. Moreover, you can point to or click any greaphical symbol from the plan view to quickly determine the defined input data and output modeling results.

Network Modeling Elements

Autodesk Storm and Sanitary Analysis provides a variety of network elements that can construct a model containing:

- Watershed subbasins (catchments)
- Stormwater and wastewater sewers
- Manholes and junctions
- Inlets and catch basins
- Rivers, streams, and ditches
- Culverts and bridges
- Detention ponds, underground storage structures, and wet wells
- Complex outlet structures
- Pumps and lift stations
- Spillways, weirs, flow dividers, standpipes, orifices, inflatable rubber dams, and valves

Model Representation

Autodesk Storm and Sanitary Analysis is a subbasin-node-link–based model that performs hydrology, hydraulic, and water quality analysis of stormwater and wastewater drainage systems, including sewage treatment plants and water quality control devices. Subbasins contribute rainfall runoff and water quality pollutants, which then enter nodes. A node can represent the junction of two or more links (a manhole, for example), a storm drain catch basin inlet, the location of a flow or pollutant input into the system, or a storage element (such as a detention pond, retention pond, treatment structure, or lake). From nodes, flow is then routed (or conveyed) along links. A link represents a hydraulic element (for example, a pipe, open channel stream, swale, pump, standpipe, culvert, or weir) that transports water and water quality pollutants.

Analysis Output

Autodesk Storm and Sanitary Analysis software's graphical capabilities can provide detailed plan view plots, profile plots, and time series plots. On the plan view, the software provides automatic color-coding of links and nodes based on any input or output property, allowing the network to be color-coded based on pipe sizes, pipe slope, flow rates, velocities, capacity, water quality concentrations, or any other attribute. Directional flow arrows can be plotted on top of pipes to show the flow direction for any time step. Furthermore, pipes can be plotted with variable width and nodes with variable radius, so you can more quickly identify those areas of the network experiencing the most surcharge, flooding, pollutant concentration, and so on.

The software will automatically generate graphical animations for plan view plots and profile plots to show output result values that change with respect to time. These animations can be recorded as AVI or WMV movie files that can be viewed independent of the software.

Multiple time-series plots can be generated for various network elements, such as pipe flow, velocity, junction water surface elevation, pollutant concentration, or any other analysis output attribute. In addition, you can display and compare multiple result files simultaneously, enabling direct comparison between different simulation models.

Custom Reports

Comprehensive input data and output analysis reports can be generated using the built-in report generator. The software allows full customization of input and output reporting. This gives you greater flexibility and functionality in developing specialized user-defined reports. These reports can be fully customized to help meet any combination of modeling criteria.

GIS Support

Autodesk Storm and Sanitary Analysis can share spatial data and visual representation of the stormwater and wastewater sewer network with most GIS spatial databases, allowing the software to be part of the stormwater and wastewater management and planning system. These capabilities can greatly assist the decision-making processes for network asset inventory, rehabilitation requirements, and financial planning.

The software can intelligently import most any GIS database structure, using attribute mapping and geocoding. Also, the analysis solution results can be exported back to the GIS database, so locations of CSO and SSO spills, manhole overflows, pipe surcharging, and floodplain flooding can be more quickly identified.

Hydrology Modeling Capabilities

Autodesk Storm and Sanitary Analysis accounts for various hydrologic processes that produce runoff from urban areas, including:

- 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

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.

The following matrix correlates typical project types with available hydrology methods.

Project Type	Hydrology Method Supported						
	EPA SWMM	NRCS (SCS) TR-20/TR-55	Rational Method	Modified Rational Method	US Army Corps HEC-1	UK Modified Rational	Continuous Simulation
Highway Drainage	X	X	X			X	
Municipal Stormwater	X	X	X	X		X	X
Regional Stormwater	X	X			X		X
Land Development	X	X	X	X	X	X	
Stormwater BMPs	X	X	X	X	X	X	X
Water Quality	X						X
FEMA Flood Control	X						
Detention Facilities	X	X		X	X		
Culverts & Bridges	X	X	X	X	X		
Rivers, Streams, and Ditches	X	X	X	X	X		
Muncipal Wastewater	X						X
CSOs & SSOs	X						X
Pumps & Force Mains	X	X		X	X		X
CMOM Compliance	X						X

The software includes the following hydrology models to help determine surface drainage runoff:

- USEPA SWMM 5.0 (also imports and exports XPSWMM models)

- NRCS (SCS) TR-20 and TR-55
- U.S. Army Corps HEC-1
- Rational Method
- Modified Rational Method
- U.K. Modified Rational (Wallingford Procedure)
- DeKalb Rational Method
- Santa Barbara Unit Hydrograph
- Delmarva Unit Hydrograph
- Long-Term Continuous Simulation
- Maricopa and Pima Counties (Arizona) Papadakis-Kazan Method
- Harris County (Texas) Method

The following hydrology methods and models are not directly supported. However, the output from these different hydrology models may be imported into the software and then hydraulically routed.

- SWMHYMO—Used in some parts of Canada
- CUHP—Colorado Urban Hydrograph Procedure
- WWHM—Western Washington Hydrology Method (King County and western Washington State)
- MODRAT (LA County, CA, for drainage areas greater than 40 acres)
- Kern County, CA
- Orange County, CA
- Riverside County, CA
- Sacramento County, CA
- San Bernardino County, CA
- Ventura County, CA
- Wallingford Hydrograph Method (used in U.K. for catchments larger than 150 ha)

Time of Concentration

The software includes the following time of concentration methods:

- SCS TR-55
- Carter
- FAA
- Kirpich
- User-defined
- Kinematic Wave (EPA SWMM only)
- Eagleson
- Harris County, TX
- Papadakis-Kazan (Maricopa and Pima Counties, AZ)

Time of concentration is only selectable when the EPA SWMM subbasin hydrology method is not selected. When the EPA SWMM subbasin hydrology method is used, the time of concentration is computed by the Kinematic Wave method and cannot be changed.

EPA SWMM Infiltration Methods

The following infiltration methods are provided with the EPA SWMM hydrology method:

- SCS Curve Number
- Horton
- Green-Ampt

HEC-1 Unit Hydrograph Methods

The following unit hydrograph methods are provided with the HEC-1 hydrology method:

- Clark (default)
- Kinematic Wave
- User Defined
- SCS Dimensionless
- Snyder

HEC-1 Loss Methods

The following loss methods are provided with the HEC-1 hydrology method:

- Uniform (default)
- Exponential
- Green Ampt
- Holtan
- SCS Curve Number

Rainfall Designer

Autodesk Storm and Sanitary Analysis includes a Rainfall Designer which, after you select any location within the U.S., will provide the design rainfall for the specified storm frequency. Alternatively, a user-defined rainfall can be specified. Then the appropriate storm distribution can be selected and the design storm is created. Multiple design storms can be created and analyzed.

- Site-specific storm distribution database with over 3,500 up-to-date rainfall recording stations across the United States
- Automatically determines design rainfall (based on study location) for 1, 2, 5, 10, 25, 50, and 100 year return frequencies
- Define any storm duration, multiple storm events
- Numerous storm distributions, including SCS, Huff, Eastern Washington, Florida, Chicago Storm, Hurricane Hazel, and user-defined

For Rational and Modified Rational methods, rainfall intensity data can be defined by either:

- IDF—Table Rainfall intensity data is defined by a table of storm durations (in minutes) versus return periods (in years). This is the most commonly used option.
- BDE—Table Rainfall intensity data is defined by a table of B-D-E coefficients versus return periods (in years) using the FWHA intensity equation.
- ABCD—Table Rainfall intensity data is defined by a table of third-degree polynomial coefficients A, B, C, and D versus return periods (in years).
- Intensity Direct Entry—A single rainfall intensity is defined that is to analyzed.

Hydraulic Modeling Capabilities

Autodesk Storm and Sanitary Analysis contains a flexible set of hydraulic modeling capabilities used to help route runoff and external inflows through the drainage system network of pipes, channels, storage/treatment units, and diversion structures. The software can simultaneously simulate dual drainage networks (stormwater sewer network and city streets as separate but connected conveyance pathways) and inlet capacity. It can quickly determine the amount of stormwater flow that is intercepted by the stormwater network inlets and the amount of stormwater flow that bypasses and is then routed further downstream to other inlets. Hydraulic network modeling is performed by the Kinematic Wave or Hydrodynamic (in other words, Saint Venant equation) routing methods. The software can account for:

- Storm sewers, sanitary sewers, and combined sewers
- Open channels
- Streams
- Bridges and culverts
- Roadway storm drain inlets
- Detention ponds and outlet structures
- Force mains (using either Hazen-Williams or Darcy-Weisbach equations)
- Flood overflow routing

Kinematic wave routing provides a nonlinear reservoir formulation for channels and pipes, including translation and attenuation effects that assume the water surface is parallel to the invert slope. This method cannot simulate backwater or reverse flow. Hydrodynamic routing solves the complete St. Venant equation throughout the drainage network and includes modeling of backwater effects, flow reversal, surcharging, looped connections, pressure flow, tidal outfalls, and interconnected ponds. Flow can also be routed through a variety of different storage elements, such as detention ponds, settling ponds, and lakes.

The hydraulic grade line (HGL) is computed at nodal points (manholes, junctions, storage structures, etc.) and along each hydraulic link element (pipes, ditches, open channel reaches). The hydraulic link HGL profile is computed by segmenting the link into ten smaller segments and then using the direct step solution method to compute the HGL for each segment. This allows the software to simulate a more accurate water surface profile along the hydraulic link, which can better account for subcritical and supercritical flow, partial and complete surcharging, moving hydraulic jumps, critical depth locations, flow transitions, and other hydraulic flow phenomena.

After computing the HGL for a pipe, the maximum entrance and exit velocities are calculated. From these velocities, the corresponding velocity heads are determined which are then used to compute the energy grade line (EGL).

The software can model simple to complex networks, including the ability to:

- Handle networks of unlimited size
- Simultaneously account for dual drainage pathways and networks

The software can model various flow regimes, such as:

- Subcritical, critical, and supercritical flow regimes
- Gravity and pressurized (surcharged) flow
- Flow reversals
- Flow splits and combines
- Branched, dendritic, and looped systems
- Tailwater submergence (backwater) effects
- Interconnected ponds
- Surface ponding at manholes
- Tidal outfalls

In addition to standard network elements, the software can model special elements such as:

- Storage and treatment units
- Flow dividers
- Curb openings, gutter inlets, and median inlets
- Pumps (including user-defined controlled pumps)
- Weirs (including compound weirs and spillways)
- Orifices and standpipes
- Inflatable rubber dams (including user-defined controlled rubber dams)
- Valves (including user-defined controlled valves)

Finally, the software is capable of:

- Using a wide variety of standard pipe shapes, custom pipe shapes, open channel shapes, as well as natural channel (HEC-RAS like) cross section geometry
- Applying external flows and water quality inputs from surface runoff, groundwater interflow, rainfall-dependent infiltration/inflow (RDII), dry weather sanitary flow, and user-defined inflows
- Applying user-defined dynamic control rules to simulate the operation of pumps, orifice openings, and weir crest levels

Supported Pipe Shapes

The following pipe shapes are supported (derived from EPA SWMM):

- Circular
- Rectangular-Triangular
- Vertical Ellipse
- Basket Handle
- Horseshoe
- Semi-Circular

- Rectangular
- Rectangular-Circular
- Semi-Elliptical
- Modified Basket Handle
- Gothic
- User-Defined Custom

- Filled Circular (sediment)
- Horizontal Ellipse
- Arch
- Egg
- Catenary

Supported Open Channel Cross Section Shapes

The following open channel cross section shapes are supported (derived from EPA SWMM):

- Rectangular
- Parabolic

- Trapezoidal
- Power Function

- Triangular
- User-Defined (HEC-RAS like)

Culvert Modeling

Autodesk Storm and Sanitary Analysis incorporates the culvert hydraulic equations from the FHWA (Federal Highway Administration) Hydraulic Design Series No. 5 (HDS-5) publication *Hydraulic Design of Highway Culverts* (2005). The following standard culvert shapes are supported:

- Circular
- Vertical Ellipse
- Box
- Arch
- Horizontal Ellipse
- Pipe arch

Flow Routing Assumptions

The software computes the cross sectional area, hydraulic radius, and free surface width as a function of flow depth for each time step in the hydraulic routing. The primary dependent variable for hydraulic routing in a link is flow discharge, which is assumed to be constant (averaged) during a time step.

Surface Ponding

When flow into a junction (for example, a manhole) exceeds the capacity of the system to transport it further downstream, the excess volume overflows the system and is lost. If the surface ponding storage area is defined at a junction, the excess volume be stored atop the junction, in a ponded fashion, and will be reintroduced into the system as capacity permits.

Interconnected Detention Pond Modeling

Autodesk Storm and Sanitary Analysis enables more accurate routing in complex detention pond situations. In some situations, downstream conditions can cause backwater effects that influence the performance of a detention pond outlet structure. For example, an upstream pond may discharge to another downstream pond that is similar in elevation or influenced by downstream flooding. Such situations can result in a decrease in outlet discharges or flow reversal back into the upstream pond and can be difficult to model properly. Most approaches attempt to simplify the problem using overly conservative assumptions about the downstream water surface conditions that result in oversized detention facilities and increased costs. Other methods ignore the downstream effects, thereby resulting in overtopping of the resulting undersized ponds. However, the comprehensive network definition and fully hydrodynamic solution method provided by Autodesk Storm and Sanitary Analysis enables you to more easily model these complex situations with greater confidence.

The software can handle simple to complex detention pond designs:

- Handles single pond, multiple ponds, and interconnected ponds
- Provides constant feedback on how the detention pond design is progressing
- Uses industry-standard FHWA Hydraulic Design Series (HDS-5) equations for outlet calculations
- Handles variable tailwater conditions, including tailwater submergence effects
- Models ponds with multiple outlets and flow diversions

For detention pond structures, both simple and complex outlet structures can be considered, including:

- Inlet boxes
- Multiple orifices
- Compound spillways
- Culverts

- User-defined outflow structures

Orifice Outlet Structures

Both circular and rectangular shaped orifices are supported, and can be located vertically (like in a riser pipe) or horizontally (like in a side-flow orifice). In addition, an orifice can have a flap gate to prevent backflow. The software provides a lookup table of typical orifice coefficients.

An unsubmerged orifice will initially act as a weir until the top of the orifice is submerged. The discharge through the orifice for unsubmerged orifice flow is computed using the weir equation. The flow then transitions to a fully submerged orifice flow using the classical orifice equation. When the orifice is not completely submerged, a modified weir equation is employed that considers the orifice fraction that is submerged.

Spillway and Weir Outlet Structures

The following shapes are supported for spillways and weirs:

- Sharp-crested and broad-crested trapezoidal and rectangular
- V-notch triangular
- Side flow rectangular
- Transverse rectangular

The software provides a lookup table of typical weir coefficients. The weir crest elevation can be controlled dynamically through user-defined control rules, allowing it to simulate inflatable rubber dams.

When a weir structure is highly submerged due to high tailwater conditions, the flow over the weir no longer acts like weir flow and the carrying capacity is reduced. In this situation, the weir flow computations automatically switch to the Villemonte equation. The submergence coefficient is automatically computed, taken from *Roessert's Handbook of Hydraulics* (German).

Pumps

Pumps can be represented as either an in-line lift station or an off-line node representing a wet well, from which water is pumped to another node in the system according to a programmed rule curve or step function. Pump performance can be defined by either:

- Volume vs. Flow
- Flow vs. Depth
- Flow vs. Head Difference
- Variable Speed Pump vs. Depth
- Design Mode (all inflow is pumped out at the same flow rate, useful for preliminary design)

The on/off status of pumps can be controlled dynamically by specifying startup and shutoff water depths at the inlet node or through user-defined control rules. Rules can also be used to simulate variable speed pumps that modulate pump flow.

Underground Stormwater Detention

On-site, underground stormwater retention/detention can be incorporated into your network model. Subsurface vaults or systems of large diameter interconnected storage pipes, arched pipes, or

manufactured storage chambers can be modeled. The software includes standard storage chambers from leading manufacturers. Choose the storage chamber model from a selection list, define the quantity, backfill dimensions, stone void space, and the software will automatically compute the available storage capacity.

Storage Node Exfiltration (Infiltration)

The software can model exfiltration (sometimes called infiltration) at storage nodes, accounting for both stormwater reduction and pollutant removal. Wet and dry retention ponds can be modeled.

The following exfiltration methods are provided:

- None (default)
- Constant Rate, Free Surface Area
- Constant Rate, Projected Area
- Constant Rate, Wetted Area
- Constant Flow
- Horton, Free Surface Area
- Horton, Projected Area
- Horton, Wetted Area

Highway Drainage Design

Autodesk Storm and Sanitary Analysis helps simplify your highway drainage design work. The software can simulate dual drainage systems (stormwater sewer network and city streets as dual conveyance pathways) and inlet capacity. It can more quickly determine the amount of stormwater flow that is intercepted by the stormwater network inlets and the amount of stormwater flow that bypasses and is then routed further downstream to other inlets.

Highway drainage capabilities include:

- Select from standard curb openings, grated inlets, slotted inlets, median ditch inlets, and combination inlets
- Account for on-sag and on-grade conditions
- Look up standard curb openings and grated inlets from major foundry manufacturers and agencies
- Use industry-standard FHWA (Federal Highway Administration) HEC-22 computations
- Compute gutter spread, depth of flow, inlet efficiency, inlet spacing, velocity of flow for gutter and pavement sections

The hydraulics of storm drain inlets is computed based on the procedures and equations defined in the FHWA Hydraulic Engineering Circular No. 22 (HEC-22), *Urban Drainage Design Manual*, Third Edition, 2009. The capacity of storm drain inlets on roadway sag is computed by both the weir and orifice equations (FHWA Report, *Hydraulic Characteristics of Slotted Drain Inlets*, 1980). Flow into the storm drain inlet initially operates as a weir having a crest length equal to the length of drain perimeter that the flow crosses. The storm drain inlet operates under weir conditions to a depth of about 4 inches (100 mm) and then begins to switch to orifice flow.

Water Quality Modeling Capabilities

Autodesk Storm and Sanitary Analysis can perform urban stormwater water quality modeling:

- Account for rain gardens, green roofs, rain barrels, bioswales, dry detention ponds, wet ponds, retention ponds, wetlands, and more
- Model particulate settling, water cleanup, water quality best management practices, and total maximum daily loads (TMDL)

Water quality routing within channel and pipe links assume that the link behaves as a continuously stirred tank reactor (CSTR). Although a plug flow reactor assumption is more realistic, the differences are small if the travel time through the link is on the same order as the routing time step.

Water quality routing through detention ponds follows the same approach used for links. For nodes that have no storage volume (for example, junctions, inlets, and flow diversions), the water quality exiting the node is simply the mixture concentration of all water entering the node.

NPDES

As part of the NPDES (National Pollutant Discharge Elimination System) permitting process, modeling of stormwater quality and quantity may be required. The software can model all aspects of stormwater quality and quantity, and can incorporate best management practices directly within the model.

The following processes can be modeled for any number of user-defined water quality constituents:

- Dry-weather pollutant buildup over different land uses
- Pollutant washoff from specific land uses during storm events
- 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

Pollutant Buildup and Washoff

Water quality pollutant buildup accumulates within a land type by either a mass per unit of subbasin area or per unit of street curb length. The amount of buildup is a function of the number of preceding dry weather days (prior to the storm event) and can be computed using one of the following numerical functions.

None	-	No pollutant buildup occurs
Power	-	Pollutant buildup accumulates proportionally to time, until a maximum limit is achieved
Exponential	-	Pollutant buildup follows an exponential growth curve that approaches a maximum limit asymptotically

Saturation   -   Pollutant buildup begins at a linear rate that continuously declines with time until a saturation value is reached

Water quality pollutant washoff from a given land type occurs during wet weather periods (for example, during a storm event) and can be described in one of the following methods. Note that buildup is continuously depleted as washoff proceeds, and washoff ceases when there is no more pollutant buildup available.

None   -   No pollutant washoff occurs

Exponential   -   Pollutant washoff is proportional to the runoff and the amount of buildup remaining

Rating
Curve   -   Pollutant washoff is proportional to the runoff

Event Mean   -   Pollutant washoff is proportional to the runoff based on an event mean concentration

Typical pollutant buildup and washoff parameters are provided in a lookup table within the software.

Washoff loads for a given pollutant and land type can be reduced by a fixed percentage by specifying a BMP removal efficiency that reflects the effectiveness of any BMP controls associated with the land type.

Pollutant Treatment

Water quality pollutant treatment and removal is simulated by assigning a treatment function to a node. A treatment function is defined using a mathematical expression that describes the pollutant reduction. The mathematical expression can be simple, such as a direct concentration reduction, or sophisticated, in which various process variables can be analyzed (for example, flow rate into the node, node water depth, node surface area, hydraulic resident time, and so on).

Sanitary Sewer Modeling Capabilities

Autodesk Storm and Sanitary Analysis can analyze both simple and complex sanitary and combined sewer systems.

- Use for master planning, rehabilitation, new design, and include future growth in your sewer model
- Model looped networks, flow splits, combines, overflows, and storage capacity
- Analyze sanitary or combined sewer systems
- Include manholes, inlets, sewer networks, pumps, lift stations, storage structures, control structures, force mains, inverted siphons, overflow diversions, relief sewers, and other elements within a single model
- Construct network sewer models from CAD drawings or GIS geodatabases
- Check CMOM (Capacity, Management, Operation, and Maintenance) capacity requirements for compliance
- Find and fix sewer bottlenecks, optimize control rules, and help reduce overflow occurrences
- Regulate flow to treatment facilities by determining storage within the sewer system and designing new storage structures
- Perform CSO and SSO mitigation studies while accounting for RDII (rainfall derived inflows and infiltration)

The software includes a comprehensive table of typical daily average flows, which can be used to determine sanitary loadings based on land use and population density.