FLO-2D©

PRO VERSION

TWO-DIMENSIONAL

FLOOD ROUTING MODEL

STORM DRAIN MANUAL

NOVEMBER 2019 - BUILD NO. 19

FLO-2D Software, Inc.

Table of Contents

List of Figures

List of Tables

Chapter 1

FLO-2D Storm Drain Overview

Initially, the FLO-2D PRO two-dimensional flood routing model was integrated with the Environmental Protection Agency (EPA) Storm Water Management Model (SWMM) Version 5.0.022 (Rossman, 2005; Rossman, Lewis A., 2006; Rossman, Lewis A., 2007) starting in 2013. The FLO-2D storm drain engine has evolved into a completely new and unique model component. The FLO-2D storm drain engine simulates the exchange of surface water flow with a storm drain system as a flow continuum (one body of water). Over the years, as the storm drain code expanded, the original SWMM source code became obsolete, and the FLO-2D storm drain system represents a significant advancement in storm drain detail, accuracy and speed.

In the coupled model system FLO-2D hosts the closed conduit storm drain system and both models run simultaneously as one engine. FLO-2D calculates all hydrologic and hydraulic flood routing while the closed conduit component computes the pipe hydraulics. The integration process involves allowing both systems to share data on a computational timestep basis controlled by the FLO-2D surface water engine. The storm drain inlet discharge and the potential return flow to the surface is a function of the water surface elevation (WSE) and the storm drain pressure requiring a seamless sharing of data. Both systems must have the same coordinate data base. The FLO-2D model will compute the storm drain inflow discharge based on the predicted grid element headwater depth and inlet geometry type. This inlet-controlled discharge will then be routed as storm drain pipe discharge. The storm drain return flow to the surface water system is exchanged through storm drain inlets/outlets and outfalls. The complete conceptualized flood routing system is shown in **Error! Reference source not found.**.

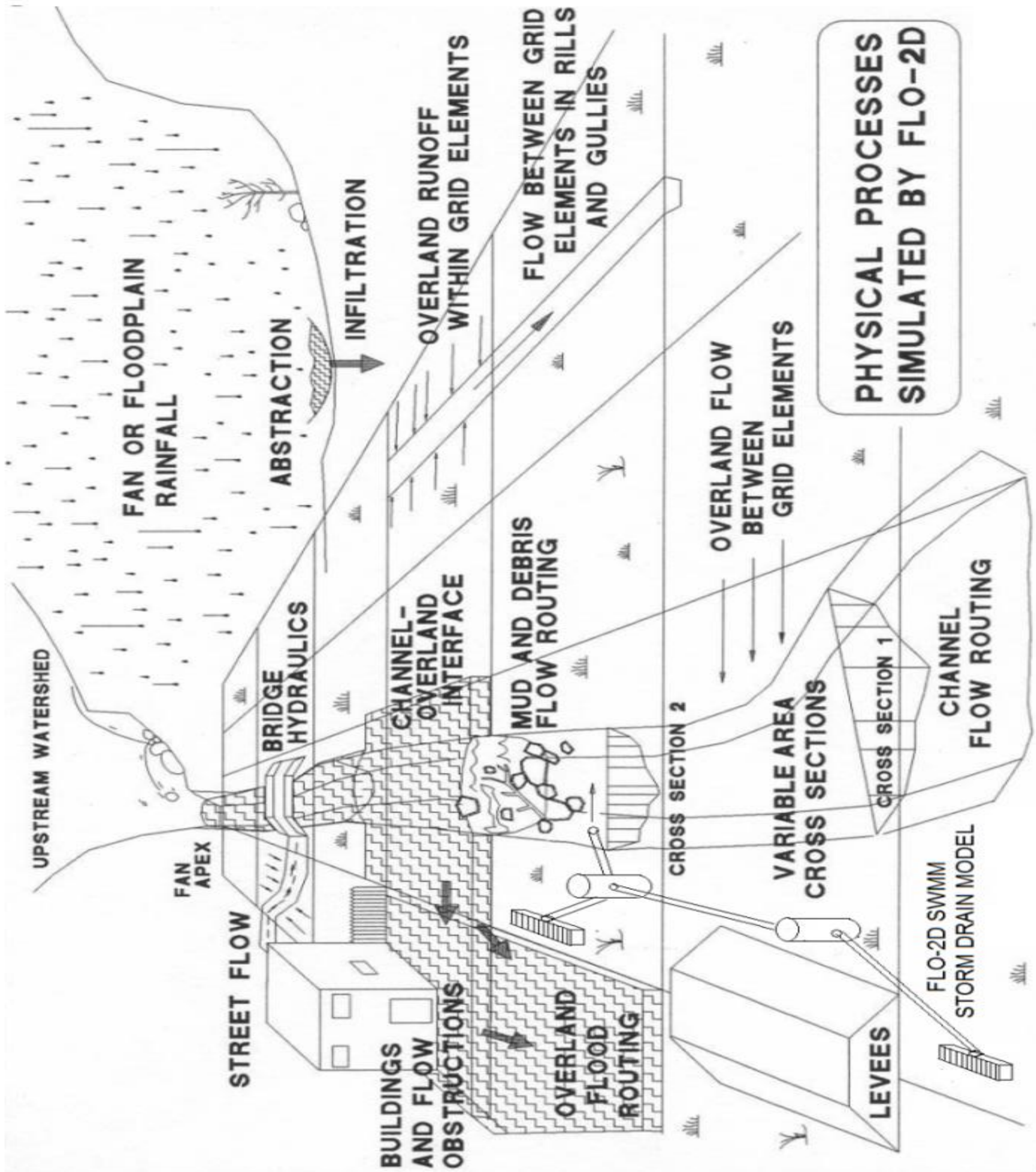


Figure 1. Conceptualized FLO-2D Model System with a Storm Drain Component.

The FLO-2D storm drain component can be visualized in layers. The surficial layer represents all the surface water flood movement which is connected to the subsurface pipe layer through the pipe junctions defined as inlets (or outfalls). Figure 2 illustrates the layered system.

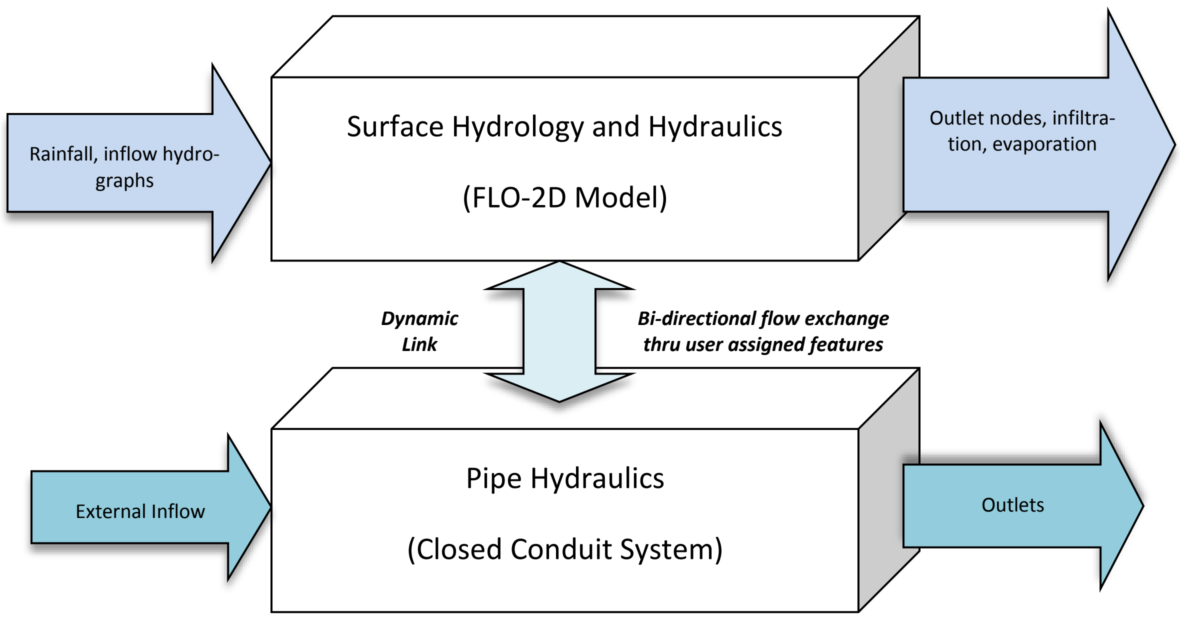


Figure 2. Volume Exchange between the Surface Water and Storm Drain System

The FLO-2D storm drain component can be applied to a variety of different storm water projects including:

- Assessment of the storm drain capacity;
- Existing storm drain system response to floods events;
- Design (sizing) of flood control and drainage system features;
- Analysis of urban surface features (levees, walls, streets, channels) and their impact on the volumes captured by the piping system;
- Design strategies to minimize storm drain overflow.

The FLO-2D storm drain system data base can be developed using the FLO-2D QGIS plug-in tool, the EPA SWMM Graphical User Interface (GUI) or other external SWMM software programs. The SWMM GUI is installed with the FLO-2D Pro Model installation package. The only requirement is to be able to generate the SWMM.inp Version 5.0 data file. To run the storm drain model, the switch in the FLO-2D model data base must be activated or turned 'on' (FLO-2D Control Variables Dialog). This can be accomplished in the FLO-2D model Grid Developer System (GDS) or in the QGIS plug-in. Once the storm drain system is built and the run switch is set, the flow into and through the pipe system is simulated automatically when the FLO-2D model is started. Following a successful simulation of the surface water and storm water integrated model, the storm drain results can be reviewed in either the QGIS, GDS (Figure 3) or in the EPA SWMM GUI. Junctions are shown in green and Inlets are blue.

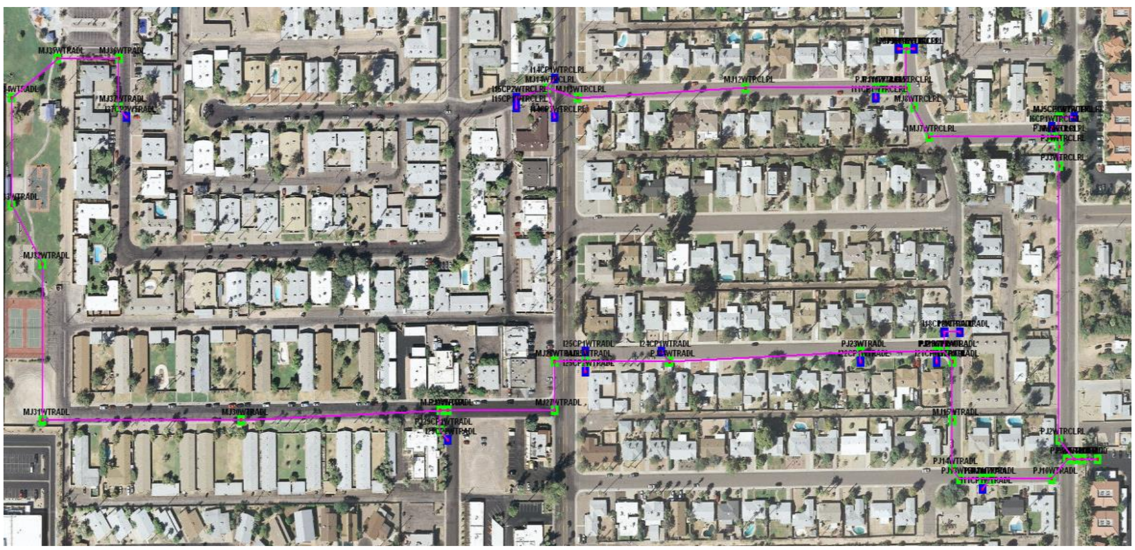


Figure 3. A Typical Storm Drain System as Viewed in the GDS

FLO-2D Storm Drain Model Enhancements

The enhancements to the original SWMM model are extensive. Some of the enhancements are related to the water surface head on the storm drain system. The original SWMM model did not predict or utilize surface water elevation. The FLO-2D model computes the storm drain inlet inflow discharge based on the inlet geometry and the head on the inlet and shares the discharge with the storm drain engine. The inlet and outfall exchange with the surface water including return flow to the storm drain system are based on the hydraulic head not just the rim elevation. There are a number of enhancements to the outfall functions. Finally, there have been a number of enhancements made to both model codes that include timestep management, inlet geometry, manhole covers, storm drain return flows, reporting output, and feature options.

Computational Timesteps

The storm drain routing timestep in the original SWMM model was based on one of several methods that the user selected. The FLO-2D surface water engine computational timesteps were typically smaller than the storm drain model timestep requiring several successive FLO-2D computational timesteps to be completed to match the simulation time of the storm drain model. During periods of successive FLO-2D timesteps, the volume of water entering the storm drain system from the surface water model was accumulated. Once storm drain simulation time was exceeded, the host FLO-2D model activated the storm drain-surface water interface and the block of surface water was distributed into the storm drain system.

The update interval between the models was essentially the storm drain dry timestep (DRY_STEP) which was hardwired to 60 seconds. The FLO-2D discharge to the storm drain for each computational timestep was accumulated as a volume over the update interval and then was divided by the update interval time to derive a steady discharge exchange from FLO-2D surface water to the storm drain. The flow between the models could only be in one direction, either inflow to the storm drain system or return flow to the surface water. Oscillations occurred when the surface water and storm drain conditions force the inflow and return flow to alternate with large blocks of water. In addition, a storm drain feature may fill and drain repeatedly even though there may be only inflow to the storm drain system and or only return flow to the surface water.

A new timestep synchronization method was implemented and the surface water and the storm drain systems now communicate with each FLO-2D computational timestep. The storm drain system uses the FLO-2D computational timestep for the water volume exchange as well as for the flow routing through the pipe system. With the more frequent water volume exchange at each FLO-2D computational timestep, the inconsistencies between the volumes leaving the surface water and entering the storm drain system were eliminated. The storm drain routing timestep is set up as the FLO-2D timestep for the dynamic wave solution throughout the simulation for all conditions, if the user sets up a fixed timestep or if the user assigns a variable timestep.

In most of the simulations, the FLO-2D timesteps are small enough for the storm drain solution to converge. For all conditions, the computed variable timestep is equal to the FLO-2D computational timestep.

Inlet Geometry

In the original SWMM model, pipe discharge was based on the system conveyance capacity, ignoring the inlets discharge capacity (control). The FLO-2D storm drain model computes the storm drain inlet discharge based on the storm drain inlet geometry and the predicted water surface elevations. An inlet can be assigned to a floodplain, channel or street element. Three inlet options represent typical storm drain inlet designs. A fourth option enables a stage-discharge rating table for a unique inlet conditions (INTYPE= 4) and a fifth option will simulate a manhole (INTYPE=5).

The EPA SWMM5 model introduced a ponding feature to enhance the flooding approach used by the EPA SWMM 4 model. The purpose of this feature was to emulate surface water that would both keep the excess storm drain water under pressure and return flow to the system when storage capacity became available. The ponding feature is activated when the inlet storm drain pressure head exceeds rim elevation. Two overflow conditions were evaluated by the SWMM model when the pressure head exceeds the rim elevation:

Flooding: Originally developed in EPA SWMM Version 4, excess volume in the storm drain overflows and is lost because there is no inlet temporary storage. Hydraulic head at the node will not rise above the inlet rim + surcharge depth. This approach is a simplification that ignores the additional surface water head that may force more water to move through the pipe system.

Ponding: This routine was created to represent a surface water layer in the EPA SWMM 5 model. Inlet overflow is stored at the node until such time when the pressure head is reduced below the rim elevation and the stored volume is released back to the storm drain system. In this case, the overflow volume is never lost from the storm drain system. Ponded nodes do not contribute any volume to the surface water and in the flow routing continuity report any remaining ponded volume at the end of the simulation is reported as Final Stored Volume. The ponded volume continues to increase with more inlet overflow (return flow). To enable return flow, the catch basin pressure head must exceed the rim elevation plus surcharge depth (manhole cover). The rate of the rise in the pressure head depends on multiple factors including the available ponded area. There is no maximum ponded volume.

- The global ponding options was off and none of the nodes could store volume. The flooding option was applied for this case.

- The Allow Ponding option was selected, and those nonzero ponded area nodes would receive ponded water. Nodes with a zero-ponded area would receive flooding water.

When an inlet is flooded, the computed depth will decrease to the rim elevation and the overflow lost from the system is considered excess inflow into the node. For a ponded node, the depth is based on volume divided by the ponded area. The smaller the assigned ponded area, the higher the water rises and drains. The ponded area affects how quickly ponded water reenters the storm drain system because it defines the pressure head at node. The program accounts for the ponded volume so the pressure head can be computed for the next time step.

EPA SWMM5 ponded routine represents an unrealistic condition where the storm drain water under pressure exceeding the rim elevation is accumulated at an inlet but does not return to the surface water to flow away from the inlet. This is because the SWMM model cannot route surface water. Since the FLO-2D model routes the surface water and predicts flood hydraulics, the SWMM ponding feature was modified. Consideration was given to the fact that no inflow should occur during flooding conditions in the storm drain. If the inflow to inlet is ceased at the moment there is no storm drain capacity, discharge oscillations can physically occur. This is an actual response of the storm drain system to the surface water.

There are two pressure conditions that had to be assessed in the revised ponding feature:

FLO-2D WSE > storm drain pressure head > rim elevation:

The conduit water volume under pressure stays in the pipe and the return volume is assigned to the drop basin pipe based on its flow area. No volume is lost from the storm drain system to the surface water and the volume stays in the pipe. The pressure head in the storm drain is correctly represented for this condition. For the second condition:

FLO-2D WSE < storm drain pressure head > rim elevation

When the storm drain pressure exceeds the surface water elevation, the water should overflow the inlet and join the surface water to be routed away from the inlet and there is no storage volume in the pipe system.

Surface Water – Storm Drain Exchange Conditions:

When the storm drain capacity is exceeded and is under pressure, the return flooding volume becomes part of the surface water in the FLO-2D model if the pressure head exceeds the FLO-2D water surface elevation. When an inlet has orifice flow and the flow depth is higher than the curb height, water is pouring into the storm drain pipe. As the storm drain fills and the pipe water surface approaches the inlet rim elevation and there is a rapid increase in pressure head in the catch basin. When this occurs, oscillations may be noted in the storm drain pressure and discharge plots. With this modification to the storm drain code, the conditions that control the inlet flow direction described below and shown in Figure 4 through Figure 10:

Storm drain pressure head < rim elevation:

- Inflow discharge is passed from FLO-2D to the storm drain;
- Pipe is not full;
- No return flow.

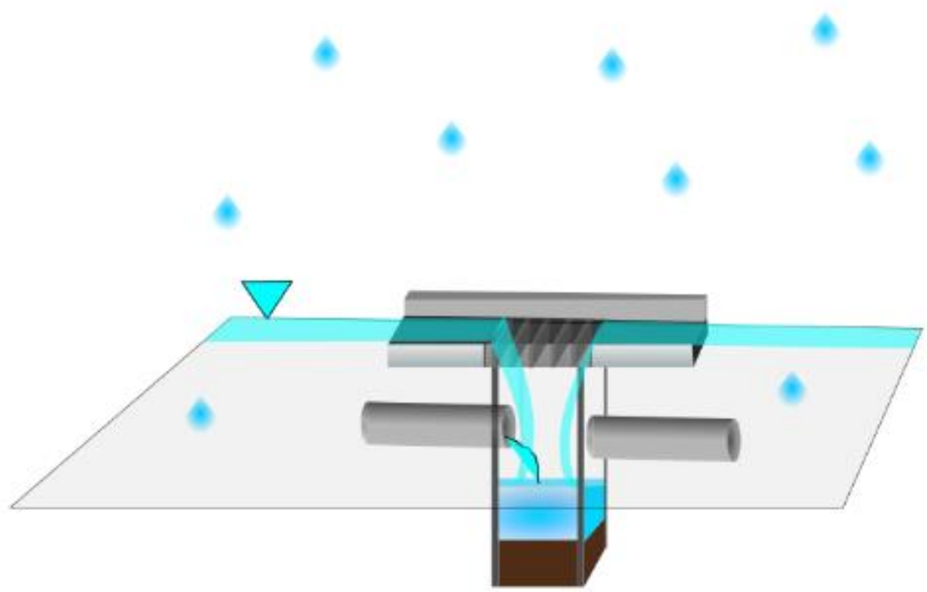


Figure 4. Inlet No Return Flow.

Storm drain pressure head > FLO-2D WSE > rim elevation:

- No inflow discharge is passed from FLO-2D surface to stormdrain;
- Pipe capacity is full;
- Return flow is exchanged to the surface;
- Water leaves the storm drain system.

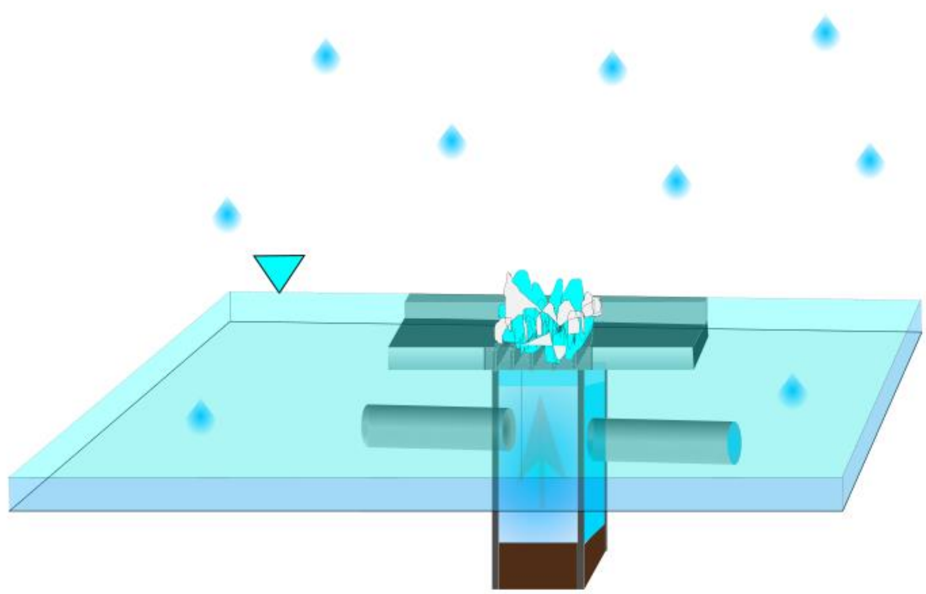


Figure 5. Inlet with Return Flow

FLO-2D WSE > Storm drain pressure head> rim elevation:

- No inflow discharge is passed from FLO-2D
- Pipe capacity is full;
- No return flow.

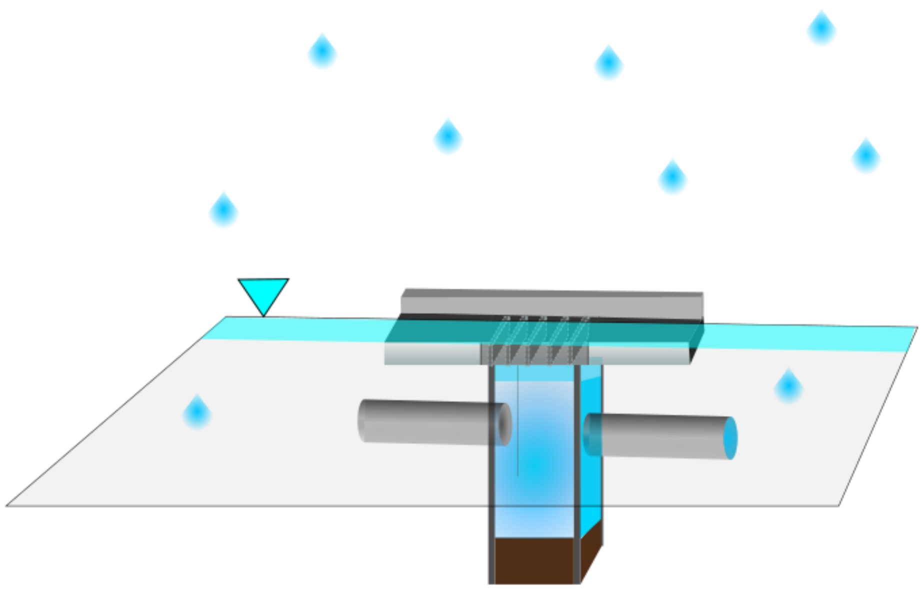


Figure 6. No Return Flow No Inlet Flow

Inlet pressure head and the FLO-2D WSE

- The inlet pressure head increases;
- Pressure head is less than the FLO-2D WSE;
- The head exceeds the rim elevation;
- Pressure head in storm drain network increases;
- No return flow;
- The volume of water column above the rim will not return to surface until it exceeds the WSE.

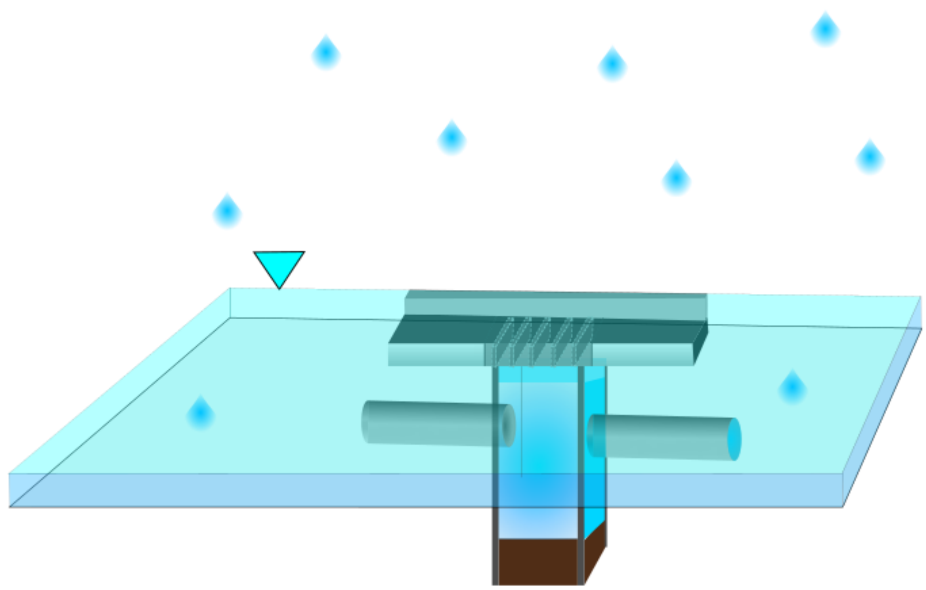


Figure 7. WSE Greater than Inlet Pressure Head.

Inlet pressure head and the pipe flow

- Higher inlet pressure head from surface water;
- Surface water head forces more water through the downstream conduits at higher velocities;
- Reduced or no return flow to the surface;
- Higher head may force the flow to the outfalls instead of overflowing the inlets and manholes.

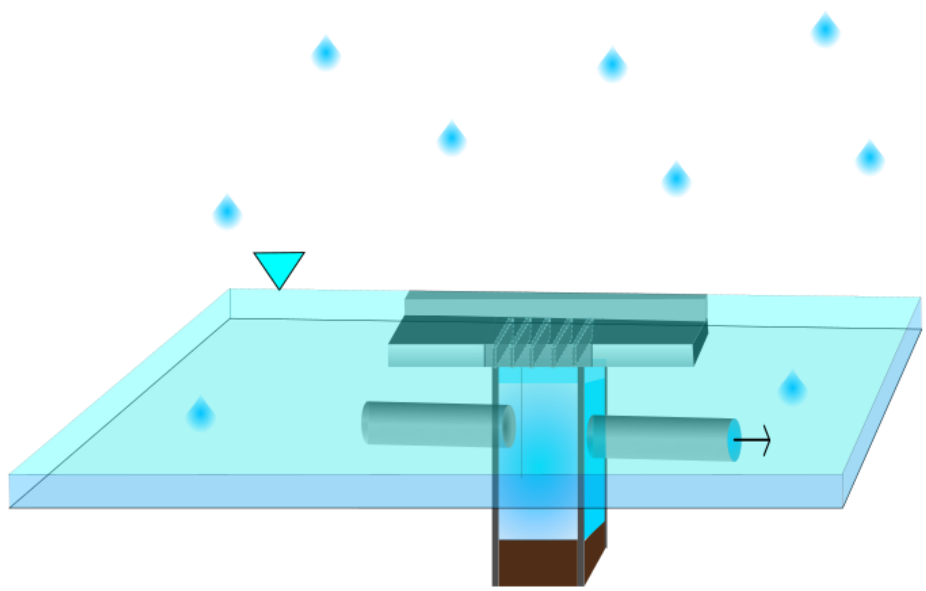


Figure 8. High Pressure Head at an Inlet.

Return flow to the surface

- The return flow to the surface water is equal to the flow volume in the timestep minus the volume in the vertical pipe (catchment basin or drop box);
- The volume fills the vertical pipe to the rim elevation;
- All available return volume is exchanged to the surface water.

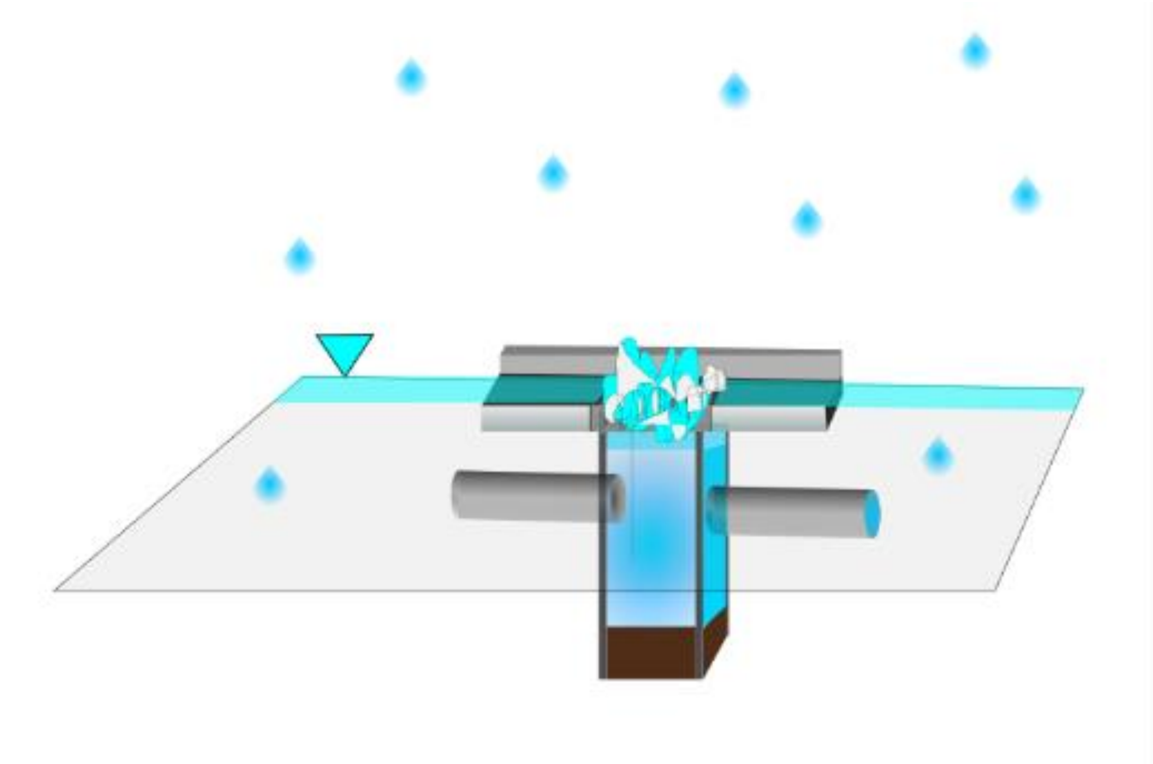
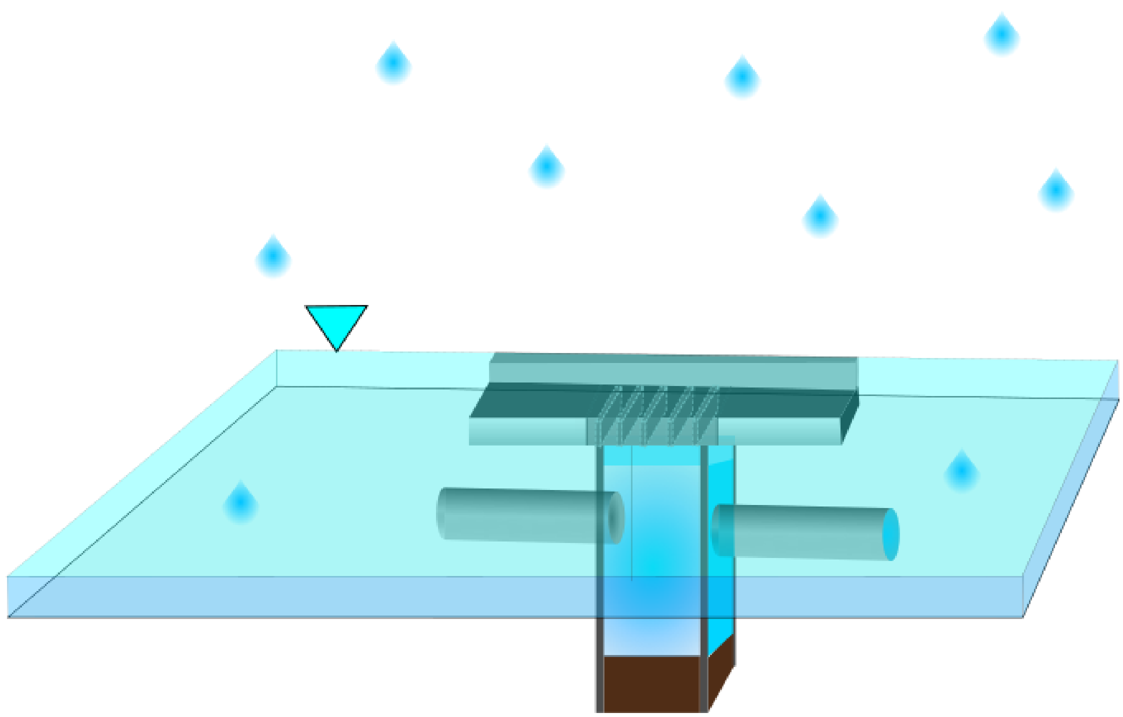


Figure 9. Return Flow.

Surface water and the pipe capacity

When an inlet is connected to a downstream pipe that is at full capacity with zero pipe velocity (backwater effect), the inflow from the surface water is zero, the inlet pressure head will exceed the crown elevation of the horizontal pipe and the FLO-2D surface water elevation may be greater than zero.

- Pipe capacity full;
- Pipe velocity zero;
- Inlet flow zero;
- No return flow.



Pressure head and manholes

Flooding will occur at manholes when the pressure head exceeds manhole rim elevation plus surcharge depth plus FLO-2D water depth.

- Pipe capacity full;
- Pressure head greater than water surface elevation;
- Return flow to system.

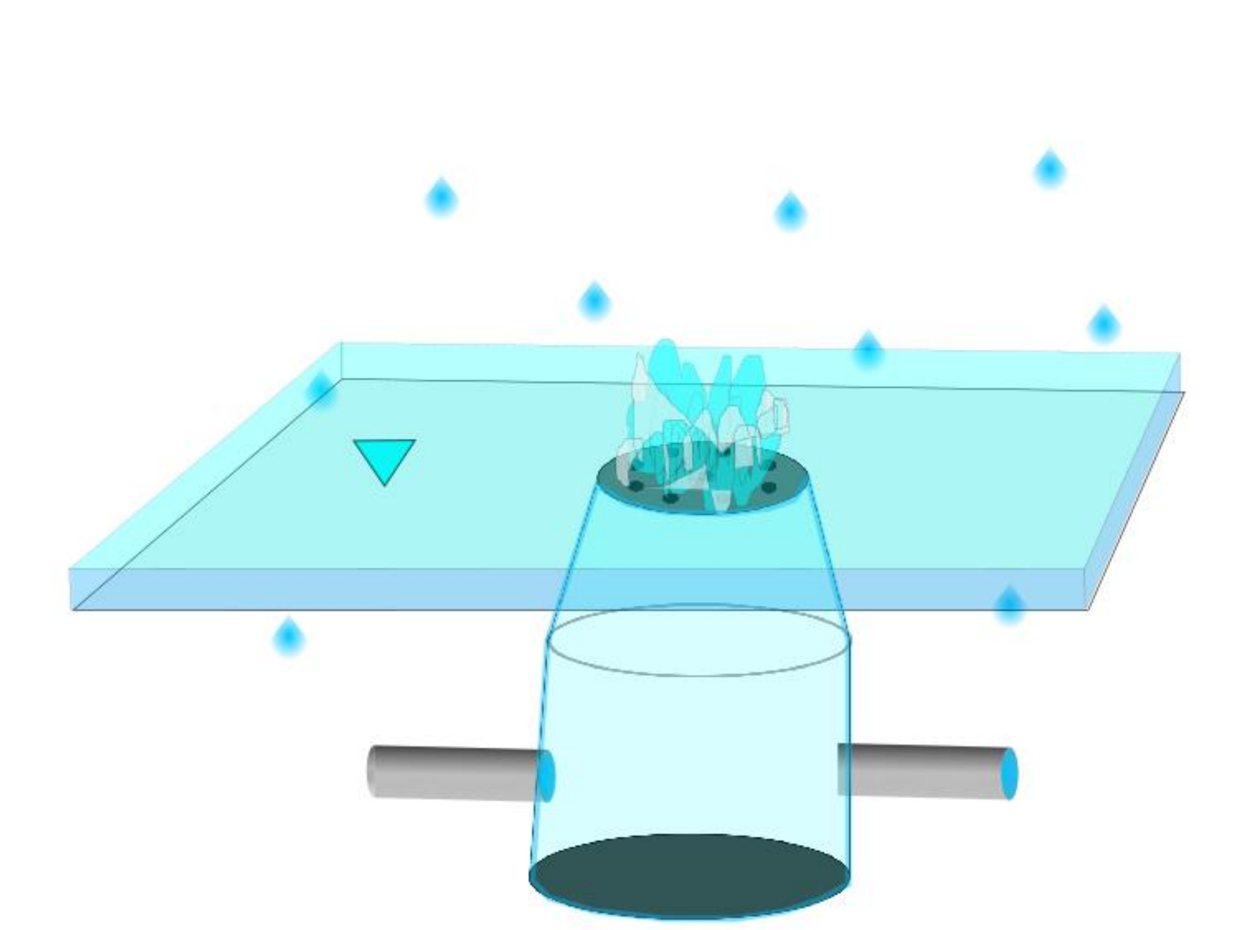


Figure 10. Manhole under Pressure with Return Flow.

Pressure head variability

This approach for the water exchange between the surface and the storm drain may result in pressure head variability, conduit velocity fluctuations and different return flow results for inlets and manholes under pressure when compared with SWMM models or early FLO-2D storm drain models. The following response can be observed in the FLO-2D storm drain results:

1. The head on the inlet continuously increases when the PH is less than the FLO-2D WSE even though the head exceeds the rim elevation. Since the inlets are flooded, this results in higher storm drain pressure. The volume above the rim is not released to the surface until the PH exceeds the FLO-2D WSE.
2. For underwater inlets, the higher pressure head pushes more water through the downstream conduits at higher velocity.
3. Higher velocities in downstream conduits may result in higher discharges in various locations in the storm drain with a possible corresponding reduction in the return flow to the surface water for some inlets and manholes. Maintaining continuity in the storm drain system, there may be sufficient head to force the flow to the outfalls instead of overflowing the inlets and manholes.

Summarizing, higher upstream pressure head on inlets (higher FLO-2D WSE) may result in a change in the distribution between the return flow from a popped manhole or inlet compared the downstream conduit flow through the outfall nodes. This is a physical process that was not simulated in the original SWMM storm drain engine.

Outfall Discharge

The GDS or QGIS will create the SWMMOUTF.DAT containing the outfall nodes that are defined in the SWMM.inp file. The outfall discharge to the surface water can be turned 'on' = 1 or 'off' = 0 in the GDS dialog window (Tools | Storm Drain | View Outfall Nodes Dialog). Setting the outfall switch to 'off" will force the outfall discharge out of the model system. The outfall nodes listed in the SWMMOUTF.DAT file should be in the same order as they appear in the SWMM.inp file. When the outfall order is modified in the SWMM.inp, because either a new outfall node was created or deleted from the pipeline system, the SWMMOUTF.DAT should be regenerated with the GDS. If the outfall switch is 'on', the FLO-2D water surface elevation and storm drain pressure head are compared and the outfall will discharge to the surface water until the FLO-2D water surface elevation is equal to or greater than the pressure head. Potential backflow into the outfall pipe depends on the flapgate assignment in the SWMM.inp file. Outfall discharge from storm drain to the FLO-2D surface water is reported to the SWMMOUTFIN.OUT file. This file lists the grid element (or channel element if applicable) in the first line followed by the hydrograph with time and discharge pairs.

The invert elevation of outfalls can be less than the floodplain, channel or street elevations. This may occur for a ponded surface water condition that is assigned as a ground elevation because it would not contribute to downstream flooding. An outfall invert underground (or underwater) is imposed for this condition (Figure 11). An artificial head equal to the ground elevation is assigned to the outfall node (for the entire simulation). This artificial head causes the pipe to fill and the artificial volume is accounted for in the storm drain model. When the model runs, the inflow may be added to either the outfall grid element or the upstream storm drain pipe network and the flow can go either in or out of the outfall pipe based on the pressure head.

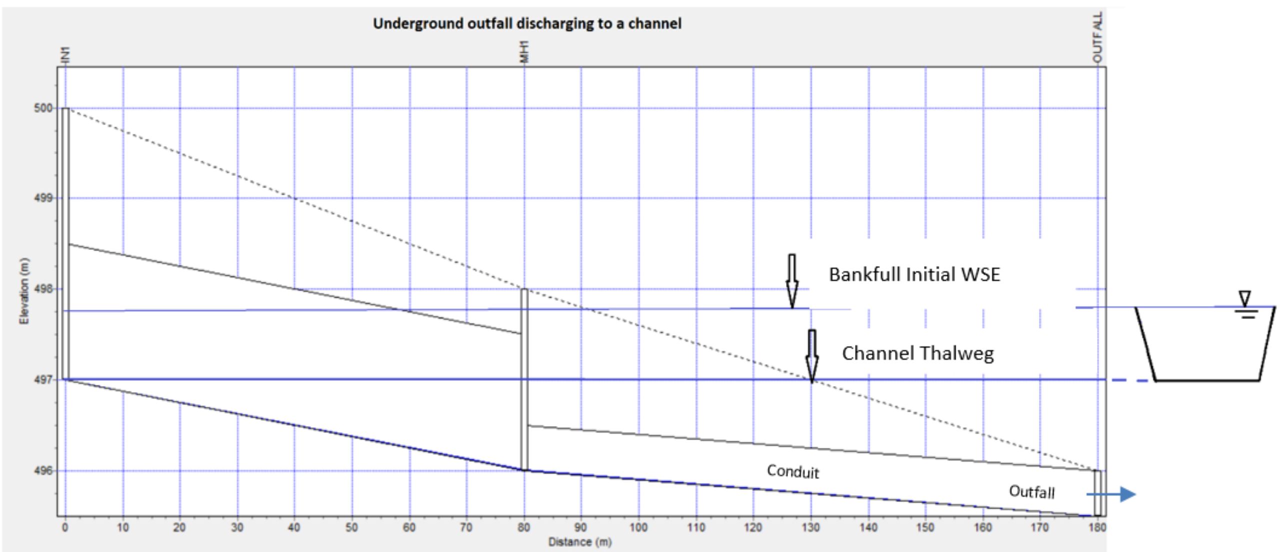


Figure 11. Initial Condition for an Underground (Underwater) Storm Drain Outfall

Typically, an outfall has an invert elevation equal to or greater than the floodplain, channel or street elevations. To account for volume conservation, the storm drain outflow that represents inflow volume to a FLO-2D channel is reported in the CHVOLUME.OUT file. Water will flow in or out of the outfall pipe based on the head comparison. Water can enter the storm drain when the water surface elevation is greater than the invert, but it can also be evacuated from the storm drain if the invert is above the water surface elevation.

Flapgate Option

A storm drain inlet can be simulated as an outlet with a flapgate to stop the surface water from entering the storm drain system (Type 4 inlet). The flapgate switch in SWMMFLO.DAT has the following settings:

Feature = 0, No flapgate – horizontal inlet opening
Feature = 1, No flapgate – vertical inlet opening
Feature = 2, Flapgate 'on' for fake outfalls

A "fake outfall" can be set up as an inlet that will discharge flow from the storm drain to the surface water.

Manhole Covers

The manhole cover lift-off (popping) is simulated by assigning the surcharge depth in the SWMMFLO.DAT file (Type 5 inlet). When the cover is in place there is no flow exchange. Discharge exchange between FLO-2D and the manhole cover junction box is calculated only after the manhole cover has been popped. To pop the cover, the storm drain pressure plus surcharge depth is compared to the surface water. Once the cover is off, the surcharge depth is set to 0, the cover stays off and inlet discharge or return flow can be calculated. Flooding occurs at manholes when the pressure head at node is above manhole invert + maximum depth + surcharge depth.

A file SDManholePopUp.OUT is created when at least one manhole pops in the storm drain system. This file contains the following information:

- Manhole ID.
- Time of occurrence
- Pressure head
- Rim elevation + Surcharge Elevation
- FLO-2D WSE.

The following is an example of the information that is reported to the output file:

MANHOLE: I5-37-27-28
POPPED AT TIME (hrs): 3.93
PRESSURE HEAD: 1374.07 > RIM + SURCH: 1371.44 > FLO-2D WSE: 1370.95

Curb Inlet Flow Adjustment

For each timestep, the FLO-2D grid element water surface elevation (flow depth) is used to calculate the discharge that passes through the inlets using the weir and orifice equations as well as the geometry of the inlets defined by the user in the SWMMFLO.DAT file. This uniform water surface elevation over the grid element does not take into consideration a street cross slope and thus will under predict the flow into the drain. Using an assumed 2% street cross slope results in a higher depth (more head) on the storm drain inlet (see Figure 12 and Figure 13). The curb height can be entered in the SWMMFLO.DAT file to make this adjustment automatically for each inlet. The curb inlet flow assignment is the same concept as the Street Gutter Flow feature (requires GUTTER.DAT file) that can be applied to gutters in streets without storm drain inlets.

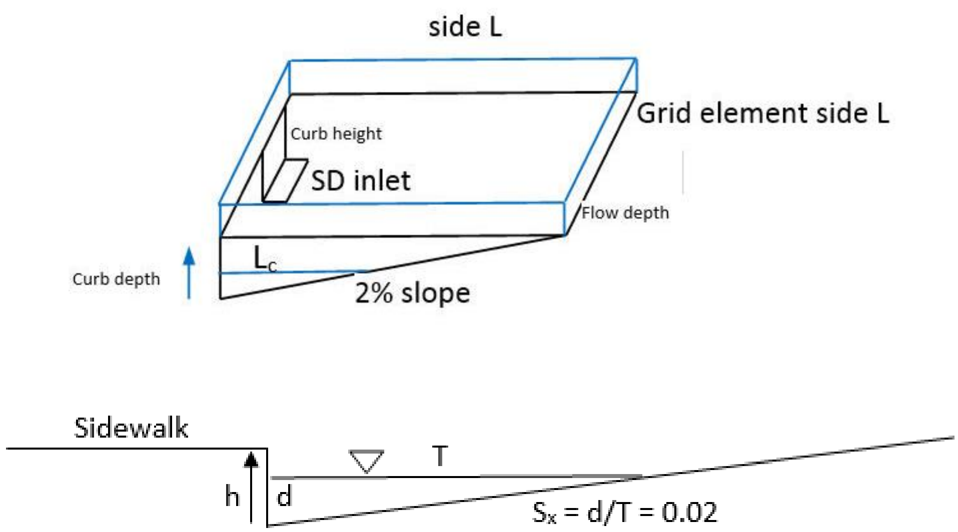


Figure 12. Curb Inlet Water Depth Profile Adjustment

Definitions:

Grid depth = flow depth on conventional grid element

Curb depth = depth on the storm drain

Flow depth = flow depth above the curb height

L_c = length of street away from curb that is inundated by the curb depth

Volume = total water volume on a grid element = side L x side L x Grid Depth

VOLCurb = volume equal to the curb height =

1/2 base (L) x height (0.02 x L) x side L = 0.5 x 0.02 x L^3 = 0.01 L^3

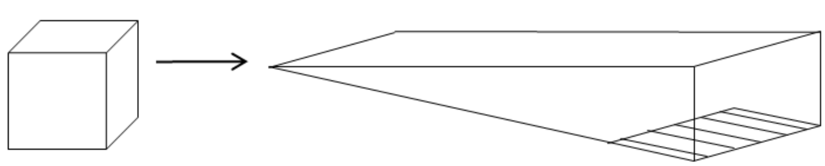


Figure 13. Volume Conversion - Square Floodplain Grid Element to Right Triangle at 2% Slope

To calculate flow depth on the storm drain inlet:

IF Volume < VOLCurb:

$$\text{Volume} = 0.5 \times \text{Curb depth} \times L_c \times L = 0.5 \times \text{Curb depth} \times \text{Curb depth}/0.02 \times L$$

$$\text{Curb depth} = (\text{Volume}/(25. \times L))^{0.5}$$

Flow depth = 0.

If VOLCurb ≤ Volume:

$$\text{Volume} - \text{VOLCurb} = L \times L \times \text{Flow depth}$$

$$\text{Flow depth} = (\text{Volume} - 0.01\, L^3)/\, L^2$$

Curb depth = Curb height + Flow depth

The curb depth is used to compute the discharge into the storm drain. This inlet discharge volume is removed from the grid element and the model continues to route the remaining volume down the street.

Storm Drain Pressure Head Variation Dampening

In the original SWMM model, to avoid rapid pressure fluctuation that induces discharge oscillations in the drop boxes, the storm drain engine had a pressure dampening algorithm. This algorithm used the surface area of the lateral conduit connected to the drop box. As the conduit water surface elevation approached the soffit, the algorithm applied a decreasing water surface area. Once the flow reached the soffit, the pressure head dampening method is applied for a distance above the invert of 1.25 times the conduit pipe diameter. The surface area was then exponentially reduced to the drop box diameter as the flow filled the catch basin over the prescribed distance. The justification for this dampening routine is that the pressure head change in one computational timestep may be sufficient to fill a four-foot drop box causing both oscillation and volume conservation error. In reality, during a storm this may occur as evidenced by manhole popping or spraying of water from inlets.

To more effectively represent the physical system, the pressure head dampening routine was reviewed and several options to revise the dampening algorithm were evaluated including allowing the pressure head variation up to be exponentially reduced over to entire drop box to the rim elevation. The Figure 16 through Figure 19 display water profiles and the pressure head versus time for an upstream inlet that has an inflow condition that fills the vertical pipe above the rim elevation. This example shows how the pressure head calculation is affected for three different dampening methods when the pressure head exceeds the soffit elevation. The selected method allows the pressure head to be exponentially reduced over to entire drop box to the RIM elevation.

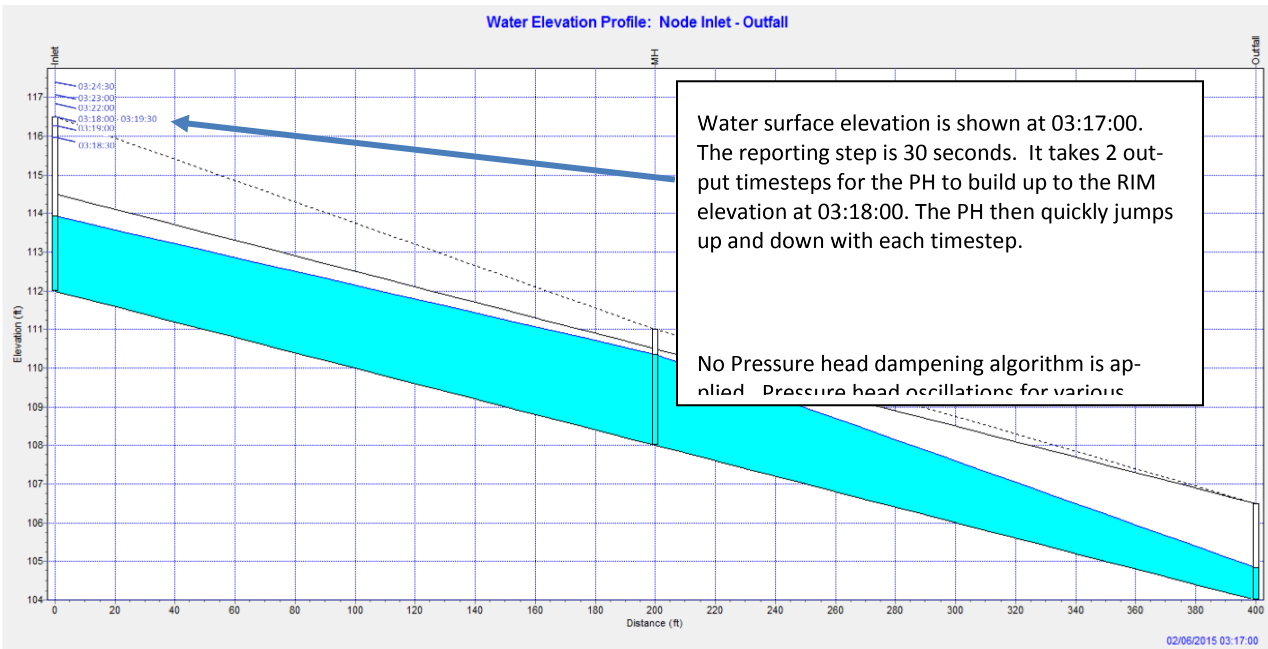


Figure 14. Water Elevation Profile at 03:17:00.

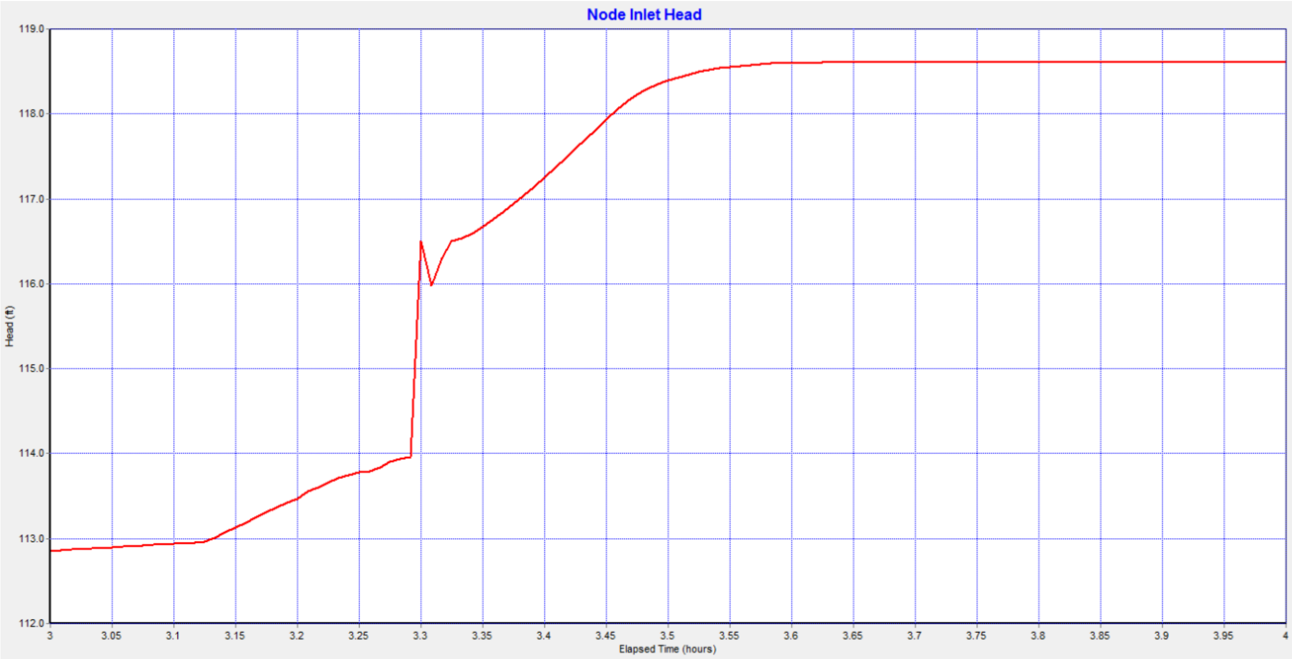


Figure 15. Inlet Pressure Head - No Pressure Head Dampening is Applied

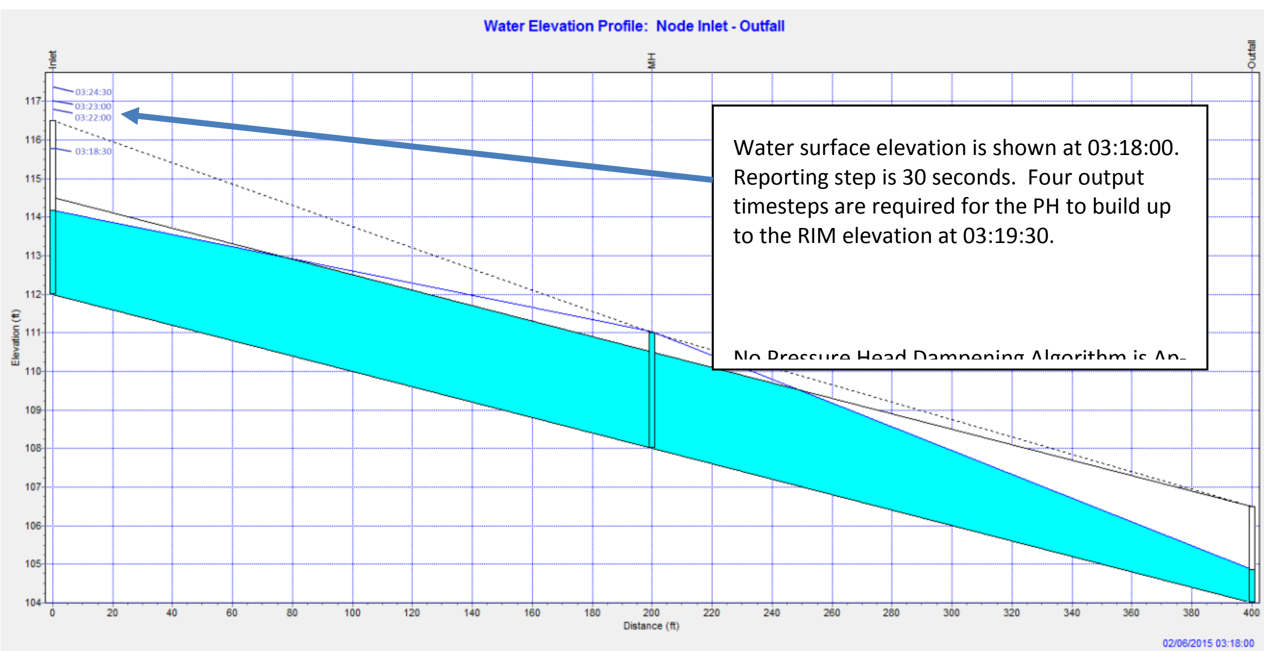


Figure 16. Water Elevation Profile at 03:18:00.

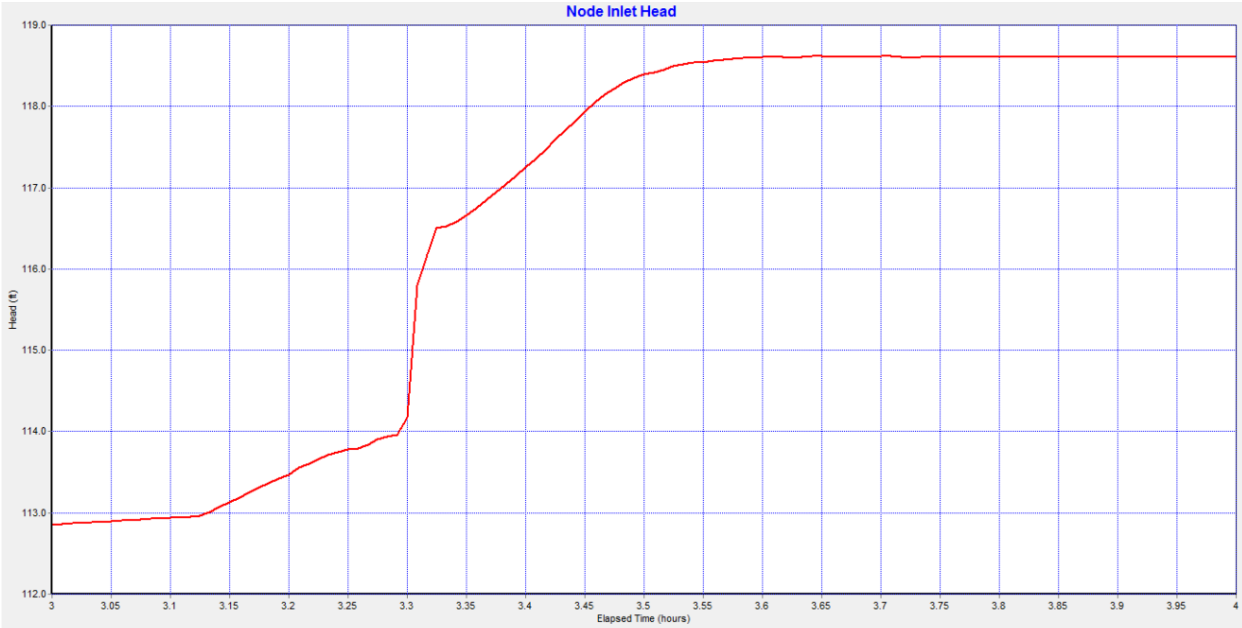


Figure 17. Inlet Pressure Head with Dampening up to 1.25 Times the Lateral Pipe Diameter.

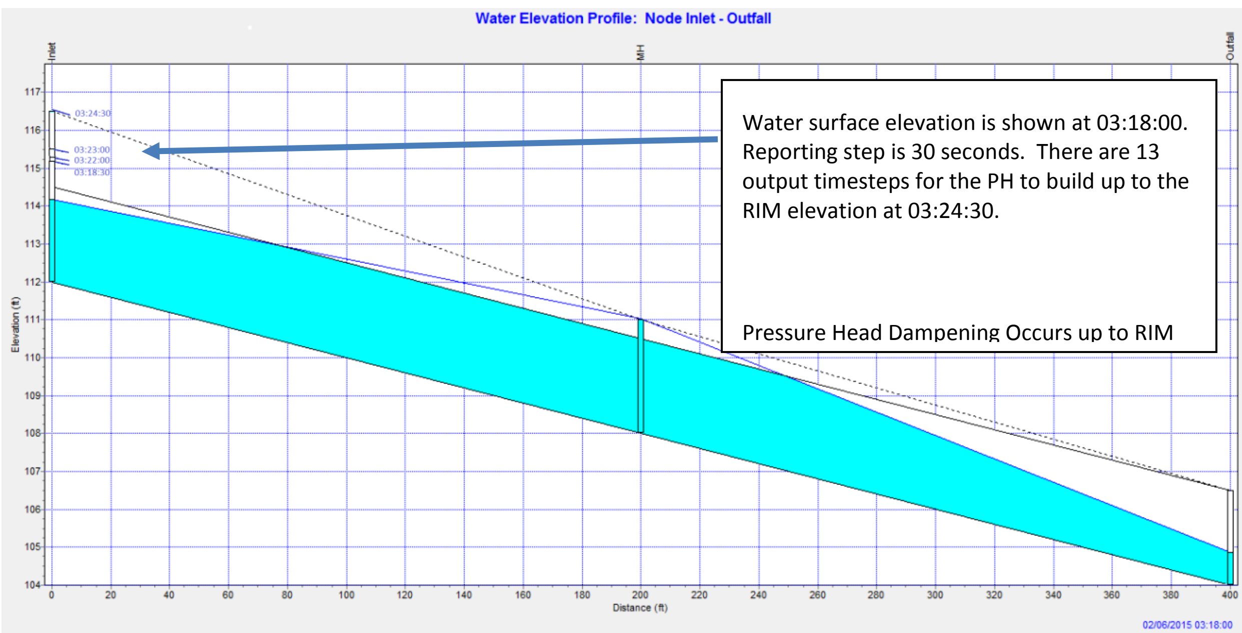


Figure 18. Water Elevation Profile at 03:18:00.

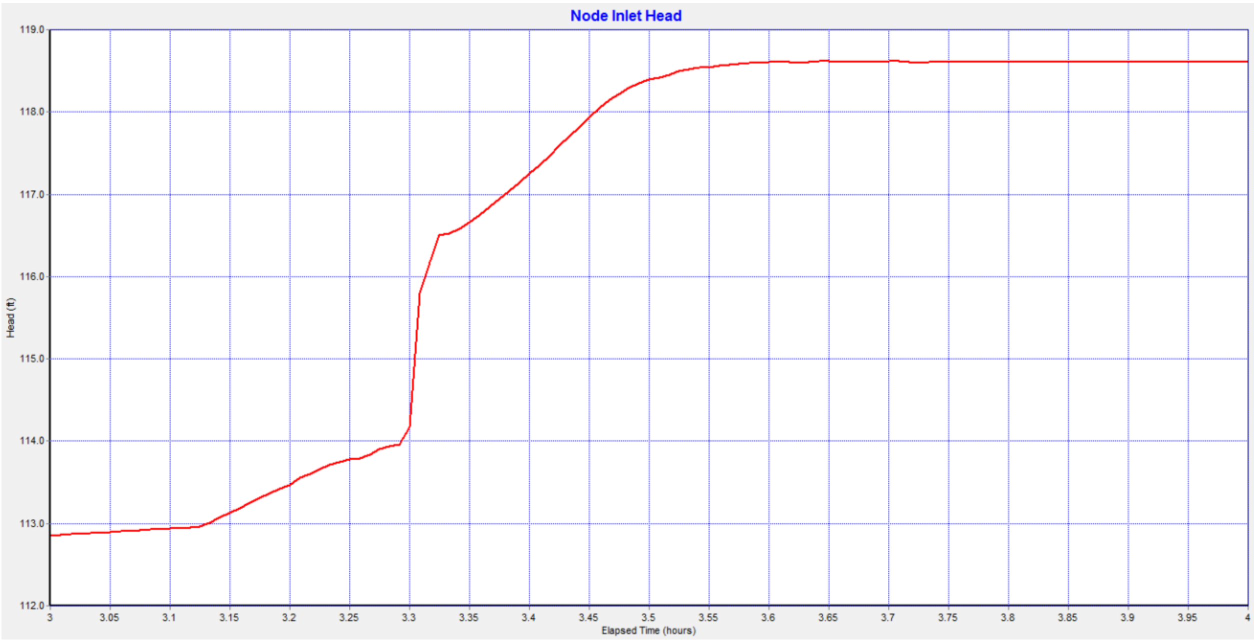


Figure 19. Inlet Pressure Head with Dampening up to RIM Elevation

Storm Drain Blockage Method

A clogging factor was created to simulate a reduction in inlet capacity. The SDCLOGGING.DAT file has the following format:

ID	Grid Cell	Inlet ID	Clogging Factor - C_f (%)	Time for clogging -T_c (hr)
D	2694	I1	25	0.50
D	2409	I2	50	3.25

The inlet discharge calculated using either the orifice or weir equations is subject to a blockage reduction that is specified by the user. The inlet discharge is calculated and then reduced using the clogging factor in the following equation:

$$Q_R = (1-C_f)\ Q_c$$

where:

Q_R = reduced inflow discharge

C_f = clogging factor

Q_c= calculated discharge using the orifice/weir equations.

This methodology is recommended for single inlets by entities such as the Colorado Department of Transportation and the cities of Denver and Las Vegas. Figure 20 and Figure 21 show the reduced discharge for a Type 2 inlet using a clogging factor of 50% at time 0.5 hrs.

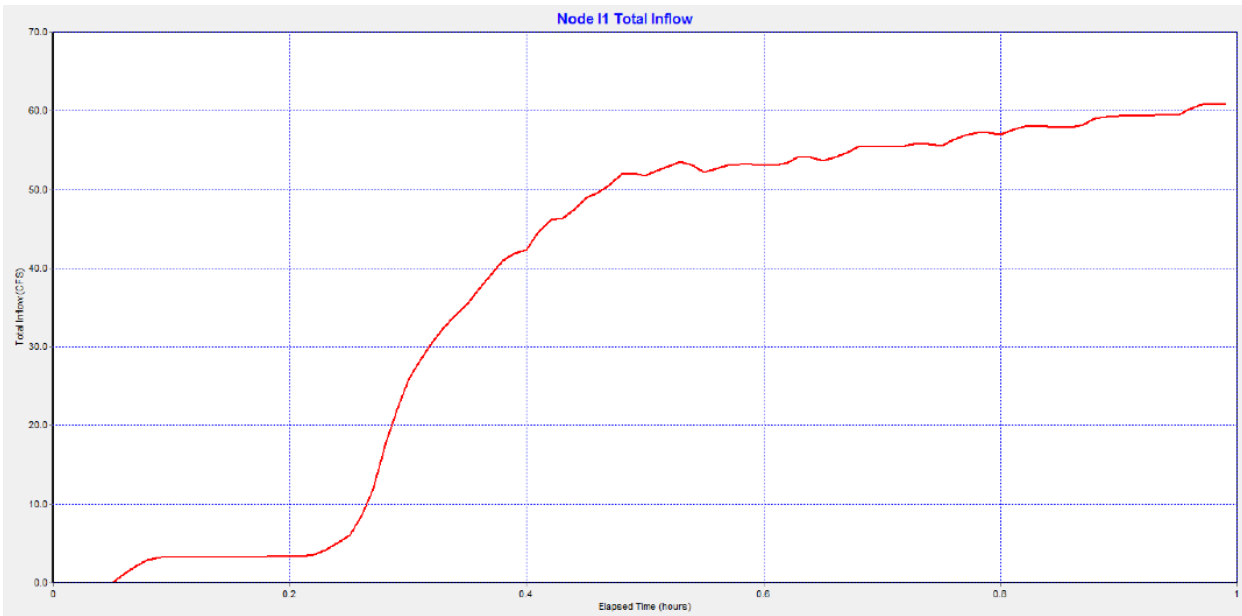


Figure 20. Type 2 Inlet Discharge versus Time

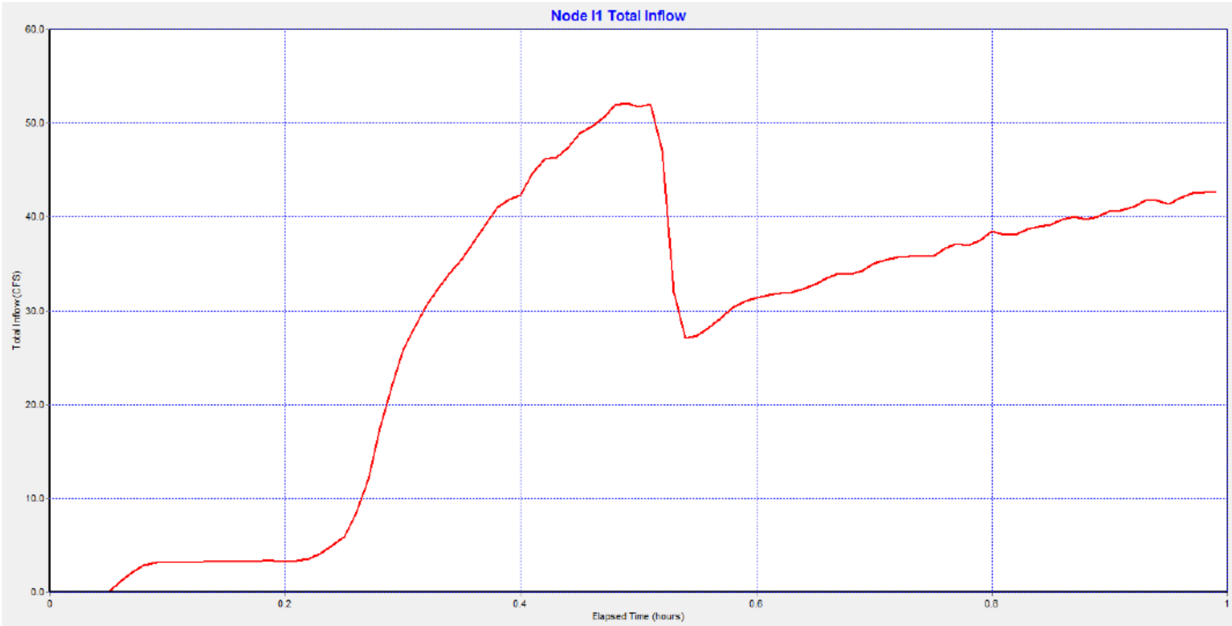


Figure 21. Type 2 Inlet Discharge versus Time Using a Clogging Factor of 50% at Time 0.5 hrs

It is noted that the Flood Control District of Maricopa County (FCDMC) in Phoenix, Arizona recommends this approach for flooding and drainage studies, but it should not be applied for storm drain design. In a design project the storm drain features are oversized to provide enough capacity for clogging. Table 1 shows the FCDMC catch basin clogging factors for predicting inlet discharge (FCDMC, 2018). The clogging factor data file can be created in the SWMMFLO.DAT data dialog for all types of inlets.

Table 1. FCDMC Catch Basin Clogging Factors (FCDMC, 2018)

Condition	Inlet Type	Clogging Factor
Sump	Curb Opening[18]	1.25L
Sump	Grate[19, 20]	2.0P
Sump	Combination[21]	1.25L and 2.0P
Continuous Grade	Curb Opening[22]	$1.25L_t$
Continuous Grade	Longitudinal Bar Grate[23]	$0.75R_f$ and 1.25L
	with recessed transverse bars[23]	$0.60R_f$ and 1.5L
	with transverse bars[23]	$0.40R_f$ and 2.0L
	Reticuline Grate[23]	$0.35R_f$ and 2.25L
Continuous Grade	Combination[24]	Apply factor $1.25L_t$ to curb opening only
Shallow Sheet Flow[25]	Slotted Drains	1.25L[26]

18 Applied to total length, L, per Example 5 in Chapter 3 of Hydraulics volume
19 Grated inlets in sump condition should be avoided whenever possible.
20 Applied to total grate perimeter, P, per Example 6 in Chapter 3 of Hydraulics volume
21 Apply clogging factors to both curb opening and grate
22 Applied to Lt per Example 2 in Chapter 3 of Hydraulics volume
23 Applied to Rf and L per Example 3 in Chapter 3 of Hydraulics volume
24 Applied to Lt per Example 4 in Chapter 3 of Hydraulics volume
25 Slotted drains are most effective for shallow sheet flow conditions or sumps. With greater depths and flows, a different type of inlet should be used.
26 Applied to total length of slotted drain.

Reduction of Return Flow to Surface

Flow energy losses are experienced when a conduit or conveyance facility has change in size or geometry. There is contraction in the flow area between the catch basin of an inlet (vertical pipe) and the inlet. The sudden contraction at the inlet from the drop box pipe diameter results in an energy loss in the return flow from the storm drain to the surface water. The energy loss for a contraction in pressure flow can be calculated from the following equation (DOT Urban Drainage Design Manual):

$$H_c=K_c(V_2^2/2g)$$

where:

K_c = contraction coefficient (see **Error! Reference source not found.**)

V = velocity downstream of transition

G = acceleration due to gravity 9.81 m/s^2 (32.2 ft/s^2)

Table 2. Values of K for Contraction. (Dept. of Transportation, 2013)

Table 7-4e (SI Units). Values of K_e for Determining Loss of Head due to Sudden Contraction.													
	Velocity, V_1, in Meters Per Second												
D_2/D_1	0.6	0.9	1.2	1.5	1.8	2.1	2.4	3.0	3.7	4.6	6.1	9.1	12.2
1.1	0.03	0.04	0.04	0.04	0.04	0.04	0.04	0.04	0.04	0.04	0.05	0.05	0.06
1.2	0.07	0.07	0.07	0.07	0.07	0.07	0.07	0.08	0.08	0.08	0.09	0.10	0.11
1.4	0.17	0.17	0.17	0.17	0.17	0.17	0.17	0.18	0.18	0.18	0.18	0.19	0.20
1.6	0.26	0.26	0.26	0.26	0.26	0.26	0.26	0.26	0.26	0.26	0.25	0.25	0.24
1.8	0.34	0.34	0.34	0.34	0.34	0.34	0.33	0.33	0.32	0.32	0.32	0.29	0.27
2.0	0.38	0.38	0.37	0.37	0.37	0.37	0.36	0.36	0.35	0.34	0.33	0.31	0.29
2.2	0.40	0.40	0.40	0.39	0.39	0.39	0.39	0.38	0.37	0.37	0.35	0.33	0.30
2.5	0.42	0.42	0.42	0.41	0.41	0.41	0.40	0.40	0.39	0.38	0.37	0.34	0.31
3.0	0.44	0.44	0.44	0.43	0.43	0.43	0.42	0.42	0.41	0.40	0.39	0.36	0.33
4.0	0.47	0.46	0.46	0.46	0.45	0.45	0.45	0.44	0.43	0.42	0.41	0.37	0.34
5.0	0.48	0.48	0.47	0.47	0.47	0.46	0.46	0.45	0.45	0.44	0.42	0.38	0.35
10.0	0.49	0.48	0.48	0.48	0.48	0.47	0.47	0.46	0.46	0.45	0.43	0.40	0.36
∞	0.49	0.49	0.48	0.48	0.48	0.47	0.47	0.47	0.46	0.45	0.44	0.41	0.38
D_2/D_1	= ratio of diameter of larger pipe to smaller pipe												
V_1	= velocity in smaller pipe (downstream of transition)										(Source: Reference 8)		

Table 7-4e (English Units). Values of K_e for Determining Loss of Head due to Sudden Contraction.													
	Velocity, V_1, in feet Per Second												
D_2/D_1	2.0	3.0	4.0	5.0	6.0	7.0	8.0	10.0	12.0	15.0	20.0	30.0	40.0
1.1	0.03	0.04	0.04	0.04	0.04	0.04	0.04	0.04	0.04	0.04	0.05	0.05	0.06
1.2	0.07	0.07	0.07	0.07	0.07	0.07	0.07	0.08	0.08	0.08	0.09	0.10	0.11
1.4	0.17	0.17	0.17	0.17	0.17	0.17	0.17	0.18	0.18	0.18	0.18	0.19	0.20
1.6	0.26	0.26	0.26	0.26	0.26	0.26	0.26	0.26	0.26	0.26	0.25	0.25	0.24
1.8	0.34	0.34	0.34	0.34	0.34	0.34	0.33	0.33	0.32	0.32	0.32	0.29	0.27
2.0	0.38	0.38	0.37	0.37	0.37	0.37	0.36	0.36	0.35	0.34	0.33	0.31	0.29
2.2	0.40	0.40	0.40	0.39	0.39	0.39	0.39	0.38	0.37	0.37	0.35	0.33	0.30
2.5	0.42	0.42	0.42	0.41	0.41	0.41	0.40	0.40	0.39	0.38	0.37	0.34	0.31
3.0	0.44	0.44	0.44	0.43	0.43	0.43	0.42	0.42	0.41	0.40	0.39	0.36	0.33
4.0	0.47	0.46	0.46	0.46	0.45	0.45	0.45	0.44	0.43	0.42	0.41	0.37	0.34
5.0	0.48	0.48	0.47	0.47	0.47	0.46	0.46	0.45	0.45	0.44	0.42	0.38	0.35
10.0	0.49	0.48	0.48	0.48	0.48	0.47	0.47	0.46	0.46	0.45	0.43	0.40	0.36
∞	0.49	0.49	0.48	0.48	0.48	0.47	0.47	0.47	0.46	0.45	0.44	0.41	0.38
D_2/D_1	= ratio of diameter of larger pipe to smaller pipe												
V_1	= velocity in smaller pipe (downstream of transition)										(Source: Reference 8)		

Storm Drain Model Governing Equations

Unsteady Flow in a Pipe Network

The storm drain engine solves the 1-D Saint Venant equations for the conservation of mass and momentum that governs the unsteady flow of water through a network of pipes (Rossman, 2006).

Continuity Equation:

$$\frac{\partial A}{\partial t} + \frac{\partial Q}{\partial x} = 0$$

Momentum equation for x-direction:

$$\frac{\partial Q}{\partial t} + \frac{\partial (Q^2/A)}{\partial x} + gA\frac{\partial H}{\partial x} + gAS_f + gAh_L = 0$$

where:

x = distance along the conduit
t = computational timestep
A = cross sectional area of the pipe
Q = pipe discharge
H = hydraulic head of water in the conduit (sum of WSE plus pressure head)
S_f = friction slope or head loss per unit length of pipe
h_L = local energy loss per unit length of pipe
g = gravitational acceleration

These equations are solved for the discharge Q and head H in each pipe by setting the initial conditions for H and Q at the beginning of the simulation as well as setting the boundary conditions at the beginning and end of each conduit for all timesteps. For each pipe, the geometry (flow area A) is known as a function of the flow depth y and head H. Unsteady flows are routed through a network of closed conduits. Unsteady flow with backwater effects, flow reversals, pressurized flow with entrance/exit energy losses and other conditions can be simulated (Rossman, 2005). The momentum equation inertial terms are reduced as flow comes closer to being critical and are ignored when the flow is supercritical based on the following options:

- Damping option (KEEP) - inertial terms of the St. Venant equation solution are included;
- Ignore option (IGNORE) - inertial terms are ignored;

- Dampen option (DAMPEN) - implements Local Partial Inertial modification (LPI).

For the FLO-2D model the LPI damping option is always applied. The simulation of unsteady flows with subcritical/supercritical mixed flow regimes is accomplished by neglecting varying portions of the inertial terms in the unsteady momentum equations according to the local Froude number. A weighting factor σ which ranges between 0 and 1 is utilized. This parameter damps out the contribution of the inertial terms as the Froude number Fr increases and approaches 1.0 and ignores them completely when the Froude number is greater than 1 (supercritical flow). The weighting factor σ varies as:

$$\sigma = 1.0 \qquad\qquad for\ Fr < 0.5$$

$$\sigma = 2.*(1.-Fr) \qquad\qquad for\ 0.5 \leq Fr \leq 1.0$$

$$\sigma = 0. \qquad\qquad for\ Fr > 1.0$$

The inertial terms are multiplied by σ when they are added into the solution of the momentum equation for each timestep and conduit. The Froude number is calculated at the midpoint depth in the conduit. This solution (DAMPEN) produces more stable results around the critical stage of the flow, but retains the essential accuracy of the fully dynamic solution at sub-critical flow conditions.

The friction slope component S_f is based on Manning's equation:

$$S_f = \frac{n^2 V|V|}{k^2 R^{4/3}}$$

where:

n	=	Manning roughness coefficient
V	=	average flow velocity (Q/A)
R	=	hydraulic radius
k	=	1.486 for English units or 1.0 for metric units

The local head loss term h_L is caused by an energy loss that is proportional to the velocity head and it can be expressed as:

$$h_L = \frac{KV^2}{2gL}$$

where:

K = loss coefficient for each pipe
V = velocity
L = conduit length
g = gravitational acceleration

To calculate the change in pressure head at each node that connects two or more conduits an additional equation is necessary (Figure 22):

$$\frac{\partial H}{\partial t} = \frac{\sum Q}{A_{store} + \sum As}$$

where:

H	=	flow depth (difference between the node head and the pipe invert elevation)
A_{store}	=	node surface area
$\sum As$	=	surface area contributed by the conduits connected to the node.
$\sum Q$	=	net flow at Node J contributed by all connected conduits plus external inflows

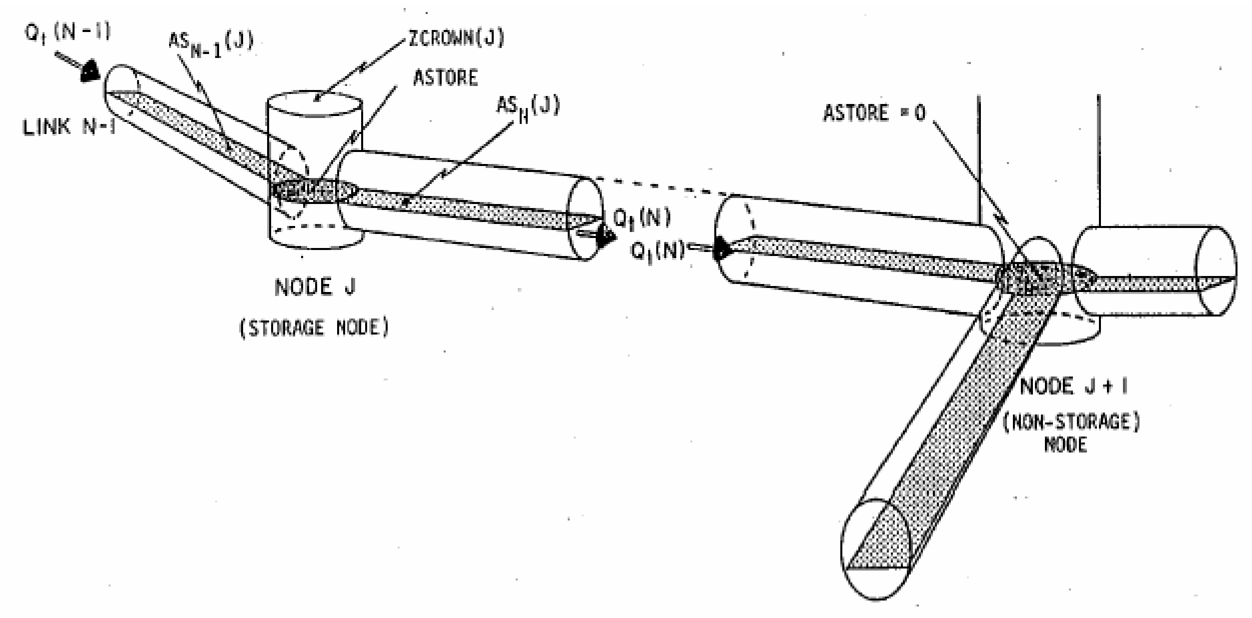


Figure 22. Node-Link Representation of a Drainage System (Roesner et al, 1992)

Solution Algorithm – How the Model Works

The differential form of the continuity and momentum equations for the storm drain component are solved by converting them into an explicit set of finite difference formulas that compute the flow Q in each conduit and head at each node for time T + Δt. Explicit schemes for these type of solutions are limited to minor steps by strict numerical stability criteria. The following discussion has been extracted and summarized from the SWMM documentation (Rossman et al, 2005).

The flow equation solved for each conduit is given by:

$$Q_{t+\Delta t} = \frac{Q_t + \Delta Q_{gravity} + \Delta Q_{inertial}}{1 + \Delta Q_{friction} + \Delta Q_{losses}}$$

The ΔQ in each conduit corresponds to the different force terms expressed as:

$$\Delta Q_{gravity} = g\overline{A}(H_1 - H_2)\,\Delta t/L$$

$$\Delta Q_{inertial} = 2\overline{V}(\overline{A} - A_t) + \overline{V}^2(A_2 - A_1)\Delta t/L$$

$$\Delta Q_{friction} = \frac{gn^2|\overline{V}|\Delta t}{k^2\overline{R}^{4/3}}$$

$$\Delta Q_{losses} = \frac{\sum_i K_i|V_i|\Delta t}{2L}$$

where:

$\overline{A}$ = conduit average cross-sectional flow area
$\overline{R}$ = average conduit hydraulic radius
$\overline{V}$ = conduit average flow velocity
V_i = local flow velocity at location i along the conduit
K_i = local loss coefficient at location i along the conduit
H_1 = head at upstream node of conduit
H_2 = head at downstream node of conduit
A_1 = cross-sectional area at the upstream end of the conduit
A_2 = cross-sectional area at the downstream end of the conduit
n = Manning roughness coefficient
L = conduit length

g = gravitational acceleration
t = time
ΔT = timestep

The average area $\bar{A}$, hydraulic radius $\bar{R}$, and velocity $\bar{V}$ are computed using the heads H_1 and H_2 at either end of the conduit from which corresponding flow depth values y_1 and y_2 can be derived. An average depth *y* is then computed by averaging these values and is used with the cross-section geometry of the conduit to compute $\bar{A}$ and $\bar{R}$. The average velocity $\bar{V}$ is determined by dividing the most current discharge by the average flow area. A limitation on this velocity is coded to prevent unbounded frictional flow adjustments. As a consequence, the velocity cannot be higher than 50 ft/sec.

For a conduit with free fall discharge into either of its end nodes, the depth at the end of the conduit for the node below the invert elevation of the conduit is set equal to the smaller of the critical depth and the normal flow depth for the conduit flow. The equation to calculate the head adjustment term for each timestep at each node is:

$$H_{t+\Delta t} = H_t + \frac{\Delta \mathrm{Vol}}{(A_{store} + \sum A_s)_{t+\Delta t}}$$

where ΔVol is the net volume flowing through the node over the timestep. The net volume is computed as:

$$\Delta \mathrm{Vol} = 0.5\left[\left(\sum Q\right)_t + \left(\sum Q\right)_{t+\Delta t}\right]\Delta t$$

The conduit surface area (A_{store}) depends on the flow condition within the conduit as follows:

1. Under normal conditions the pipe surface area equals half of the conduit length times the average of the top width at the end and mid points of the conduit. These widths are evaluated before the next updated timestep using the flow depths y_1, y_2, and y.
2. If the inflow of the conduit to a node is in free-fall (conduit invert elevation is above the water surface of the node), then the conduit does not contribute to the node surface area.
3. For conduits with closed shapes such as circular pipes that are greater than 96 percent full, a constant top width equal to the width when 96 percent full is used. This prevents the head adjustment term H_t from creating numerical instability as the top width and corresponding surface area approach zero when the conduit reaches a full condition. A

minimum surface area for A_{store} is assigned to all nodes, including junctions that normally have no storage volume, preventing H_t from becoming unbounded. Under normal conditions A_{store} equals half the conduit's length times the average of the top width at the end- and mid-points of the conduit. These widths are evaluated before the next updated flow solution is found, using the flow depths y1, y2, and y discussed previously. The default value for this minimum area is 12.57 ft^2 which corresponds to the area of a 4-foot diameter manhole.

To calculate the discharge Q and the head H, the equations are solved for each timestep using a method of successive approximations with under-relaxation (Rossman, 2005). The solution algorithm involves the following steps:

1. A first estimate of discharge Q in each conduit at time *t+Δt* is calculated by solving for $Q_{t+\Delta t}$ using the heads, areas, and velocities determined at the current time *t*.

2. A first estimate of the head (H) in each conduit at time *t + Δt* is calculated by evaluating $H_{t+\Delta t}$ using the discharge Q just computed. The results are denoted as:

$$Q_{last} \text{ and } H_{last}$$

3. The equation $Q_{t+\Delta t}$ is solved once again, using the head, area, and velocity based on the Q_{last} and H_{last} values just computed. A relaxation factor Ω is used to combine the new flow estimate Q_{new} with the previous estimate Q_{last} to generate a new Q_{new} according to the equation:

$$Q_{new} = (1-\Omega)\, Q_{last} + \Omega\, Q_{new}$$

4. The equation for $H_{t+\Delta t}$ is solved again for heads using Q_{new}. As with discharge, this new solution for head, H_{new} is weighted with H_{last} to produce an updated estimate for heads:

$$H_{new} = (1-\Omega)\, H_{last} + \Omega\, H_{new}$$

5. If H_{new} is close enough to H_{last} then the process stops with Q_{new} and H_{new} as the solution for time t + Δt. Otherwise H_{last} and Q_{last} are replaced with H_{new} and Q_{new}, respectively and the process returns to step 2.

The procedure uses the following parameters and conditions for this iterative procedure:

- A constant relaxation factor Ω equal to 0.5.
- A convergence tolerance of 0.005 feet on nodal heads.
- Number of trials is limited to four.

The flow depth in conduits that are not surcharged is limited not to exceed the normal flow depth for the discharge at the upstream end of the conduit whenever the flow regime is super-critical. FLO-2D Storm Drain model uses the water surface slope and Froude number to determine when supercritical flow occurs in a conduit.

Surcharge conditions

A node is defined to be in a surcharged condition when its water level exceeds the crown of the highest conduit connected to it. Under this condition the surface area of any closed conduits would be zero and equation for the change in the pressure head would no longer be applicable. Additional criteria include:

- An alternative nodal continuity condition is used where the total rate of outflow from a surcharged node must equal the total rate of inflow $\Sigma Q = 0$. This equation only contains flow and it is insufficient to update nodal heads at the new time step.
- Since the flow and head updating equations for the system are not solved simultaneously, there is no guarantee that the condition will hold at the surcharged nodes after a flow solution has been reached.
- Flow continuity condition is enforced in the form of a perturbation equation:

$$\Sigma\left[Q + \frac{\partial Q}{\partial H}\Delta H\right] = 0$$

where:

ΔH = node head that must be made to achieve flow continuity.

Solving for ΔH:

$$\Delta H = \frac{-\sum Q}{\sum \partial Q/\partial H}$$

where:

$$\frac{\partial Q}{\partial H} = \frac{-g\bar{A}\,\Delta t/L}{1 + \Delta Q_{friction} + \Delta Q_{losses}}$$

$\frac{\partial Q}{\partial H}$ has a negative sign because when evaluating $\sum Q$ because the flow directed out of a node is considered negative while flow into the node is positive. If surcharge (return flow to the surface water) is computed, the pressure head is considered in the total node adjustment for the successive approximation scheme.

Boundary conditions – FLO-2D inlet discharge

Floodplain runoff discharges from the surface layer typically only enters the pipeline system at inlets. Weir and orifice equations are used to calculate an inflow discharge under inlet control. In the original SWMM model there was no inlet control and all the water in the subcatchment was made available to the storm drain system capacity. With inlet control, the inlet discharge is based on the inlet geometry and on the comparison between the FLO-2D water surface elevation and the storm drain pressure head. The inlet discharge is imposed as surface water boundary conditions (BC) and is passed to the storm drain layer for routing. The following equations (Johnson and Fred, 1984) are used:

Weir Flow:

$$Q_w = CLH^m$$

where:

Q_w = weir discharge
C = weir coefficient, enter in the "Inlet Weir Coeff." field in the SWMMFLO.DAT
L = crest length; enter in the "Length (1 or 2)" field in the SWMMFLO.DAT
H = FLO-2D grid element water depth that contains the inlet structure
m = 1.5 for a broad crested weir. This is hardcoded.

Orifice Flow:

$$Q_o = C_d A\sqrt{2gH}$$

where:

Q_o = orifice flow rate at depth H
C_d = discharge coefficient hardcoded to 0.67
A = Lh; cross-sectional orifice area, computed from inlet opening length (L) and inlet opening height (h) fields in the SWMMFLO.DAT
g = gravitational acceleration
H = FLO-2D grid element water depth that contains the inlet structure

The discharges are calculated based on the physical behavior of the inlet as a weir or an orifice for a given timestep and the smaller of the two discharges is used in the surface water exchange to the storm drain system. Using orifice flow accounts for the gutter velocity that would reduce the weir flow discharge.

Surface Water – Storm Drain Model Integration

The FLO-2D model moves around blocks of water on a discretized grid system. Grid elements assigned as inlets connect the surface layer with the closed conduit storm drain system. A comparison of the grid element water surface elevation with the pressure head from the closed conduit system node in a given cell determines the direction of the flow exchanged between the two systems. The models are fully integrated on a computational timestep basis.

The advantages of the FLO-2D storm drain component over the original SWMM model are:

- Complete surface water hydrology and hydraulics including rainfall runoff, infiltration, and flood routing in channels, streets or unconfined overland flow are simulated by FLO-2D surface water model.
- The storm drain component solves the pipe hydraulics and flow routing but integrates the inlet/outlets and outfalls with the surface water for each computational timestep.
- FLO-2D computes the storm drain inlet discharge based on the water surface head and the inlet geometry. The original SWMM model did not consider inlet control.
- Only those junctions set up as inlets/outfalls in the storm drain model is recognized for system exchange. Pipe junctions without an inlet will not receive a surface runoff discharge.
- The inlet locations digitized in storm drain data files are automatically read by the FLO-2D GDS to establish the storm drain inlet connections.
- Inlets can become outlets if the storm drain pressure head exceeds the grid element water surface elevation at a given node. The potential return flow to the surface water is based on the water surface elevation not the rim elevation as in the original SWMM model.
- Manhole covers can pop and allow return flow based on a surcharge depth representing the manhole cover weight. Once popped the manhole surcharge is turned 'off' and the manhole functions as an inlet/outlet for the rest of the simulation. This is an improvement over the original SWMM model.
- For outfall nodes in the closed conduit system network, pipe discharge can be removed from the storm drain system or returned to the surface water as a user defined option. The outfall can function as an inlet to the storm drain system based on the surface water elevation. A flapgate can be used to prevent inflow to the outfall. The integration of the outfall boundary conditions with surface water represents an enhancement over the original SWMM model.

To integrate the surface water and storm drain models, the first task is to develop a running FLO-2D surface water flood model. Then the storm drain model can be built with the assigned inlets/manholes/outfalls for surface water exchange.

Storm Drain Model Features and Modifications

Some of the original SWMM model data and functions have been modified or simplified to enable the flow exchange with the FLO-2D model. The objective has been to eliminate nonfunctional storm drain variables.

Rain gage

Rain gages are not required in the FLO-2D storm drain model. The FLO-2D surface model simulates the hydrology. The model is backward compatible and will run simulations that have a rain gage.

Subcatchment

No subcatchments need to be assigned. The watersheds are represented by the FLO-2D grid elements. Junctions with an ID that starts with 'I' will identify the storm drain inlets and collect water from the surface model.

Junctions

Junctions function as pipe connection nodes. FLO-2D can only exchange flow with those junctions defined as inlets (see Inlets). The junction will not receive FLO-2D surface inflow if it serves as a simple pipe connection. The required input data is:

- Name
- X and Y Coordinates
- Invert elevation
- Maximum depth is the distance from invert to the rim
- Initial depth (optional)
- Surcharge Depth (optional)

Inlets

Storm drain inlets will exchange flow between the FLO-2D surface water and the pipe system. An inlet is a junction that captures surface inflow and must be connected to the grid system. To be recognized as inlets the junctions ID in the SWMM.inp file have to start with an 'I'. FLO-2D computes surface water inflow to the inlet using inlet geometry and water surface head. Inlets can be assigned to a FLO-D floodplain, channel or street grid cell. Inlets become outlets when the storm drain system pressure head exceeds the water surface elevation. Manholes are covered inlets that capture flow from the surface when the cover is popped. The manhole ID name starts with an 'I'. The required inlet data is:

- Name: Starts with an 'I' to be identified as Inlets
- X and Y Coordinates
- Invert elevation
- Maximum depth is the distance from invert to the rim
- Initial depth (optional)

Conduits

Conduits convey flow through the storm drain system. Slope is calculated internally based on inlet and outlet node invert elevation. Required input data is:

- Conduit name
- Name of connecting feature inlet and outlet
- Cross-sectional Geometry
- Length - between nodes
- Pipe roughness – Manning's n-value

Outfall

An outfall node is a terminal node of a pipeline with potential boundary conditions. A free outfall can discharge from the storm drain system to a FLO-2D floodplain element, channel or street cell. An outfall discharging to a channel element has to be connected to the channel left bank element. Any other outfall (that is not free) will simply discharge out of the drainage system and off the computational domain. Only one conduit can be connected to an outfall node and there must be at least one outfall node in the network. The required input data is:

- Name
- X and Y Coordinates
- Invert elevation
- Flapgates or Tide Gate (optional) can be assigned to prevent backflow into the pipes.
- Boundary Condition Types:
    - Allow Discharge is 'off' - Free Outfalls can discharge the flow from the storm drain system. Flow will not be added to the surface.
    - Allow Discharge is 'on' - The FLO-2D water surface elevation is imposed on the outfall node. Storm drain water will return to the surface model. This is the only outfall type that allows flow exchange with the surface water.
    - Normal, Fixed, Tidal and Time Series Outfalls discharges flow off the storm drain system with a boundary condition.

Links

Links are defined as those features that connect the junctions and outfalls in the storm drain system. The following components are defined as links in the storm drain system:

- Conduits
- Pumps
- Orifices
- Weirs
- Outlets

Pumps

Pumps are links used to lift water to higher elevations. A pair of nodes can be connected using links as pumps. The flow through a pump is computed as a function of the heads at their end nodes. Pumps can be simulated in FLO-2D as part of the storm drain system or as a hydraulic structure in the surface model. They have to be set up based on the following considerations:

- The pump curve can specify flow as a function of inlet node volume, inlet node depth, or the head difference between the inlet and outlet nodes.
- The pump discharge is limited to the inlet inflow during a given timestep. This will eliminate the possibility of the pump curve being sufficient to drain the inlet node during the time step.
- An ideal transfer pump can be specified where the flow rate equals the inflow rate at its inlet node and no curve is required. In this case, the pump must be the only outflow link from its inlet node.

The parameters for a pump in the storm drain system are:

- Names of the inlet and outlet nodes
- Pump curve name
- Initial status 'on' or 'off' status
- Startup and shutoff depths

Flow Regulators

Flow regulators are devices used to divert flow and can be applied to control releases from storage facilities, prevent surcharging or convey flow to interceptors. They are represented as a link connecting two nodes. The flow regulator discharge is computed as a function of the head at the end nodes. Most of the flow regulators devices control the surface flow, as a consequence they have be simulated using the surface features, example: a ponded area with a weir structure that drains to the storm drain system. The storm drain model can simulate a regulator as a storage unit with a weir. The ponded area belongs to the surface layer; therefore the correct method is to simulate this using a depressed storage area in the surface grid with a Type 4 inlet connecting the storage facility with the storm drain system.

There are some specific configurations where the flow regulators control the storm drain flow. For these cases, the flow regulator feature has to be simulated in the storm drain layer. An example is a large catch basin with an opening in the inlet wall (orifice). This component belongs to the storm drain layer and it needs to be modeled as a storage unit with an orifice.

Orifices

Orifices are used to model outlet and diversion structures. These outlet orifices should be distinguished from the inlet orifice flow and are typically openings in the wall of a manhole, storage facility, or control gate. They can be either circular or rectangular in shape and can be located either at the bottom or along the side of the upstream node. They can have a flapgate to prevent backflow. Orifice flow is based on the following criteria:

- When fully submerged the classical orifice equation is used: $Q_w = C_d A\sqrt{2gh}$.
- A partially submerged orifice applies the modified weir equation:
  $Q_w = C_d A\sqrt{2gDh} f^{1.5}$.
- An orifice surface area contribution to the outlet is based on the equivalent pipe length and the depth of water in the orifice.

where:

A = orifice open area (may be an irregular shape)

D = height of the full orifice opening

h = hydraulic head on the orifice

C_d = discharge coefficient hardcoded to 0.67

g = gravitational acceleration

f = fraction of the orifice that is submerged

Weirs

A weir is an unrestricted overflow opening oriented either transversely or parallel to the flow direction. Weirs can be a link connecting two nodes where the weir itself is placed at the upstream node. A flapgate can be included to prevent backflow. The weir calculations are based on the following criteria:

- When the weir becomes completely submerged, the model switches to the orifice equation to predict flow as a function of the head.
- Weirs do not contribute any surface area to their end nodes.
- The general weir equation $Q = C\ L\ h^m$ is used to compute the discharge as a function of head *h* across the weir when the weir is not fully submerged.

where:

C = the weir coefficient

L = the crest length

m = an exponent that depends on the type of weir being modeled: lateral, transverse, side-flow, V-notch, or trapezoidal. Typically m = 1.5 for a lateral weir. This exponent is hardcoded in the FLO-2D storm drain model.

Outlets

Outlets are used to control discharge from storage units or to simulate special stage-discharge relationships that cannot be characterized by pumps, orifices and weirs. They can have a flap-gate that restricts the flow to only one direction. This option does not discharge to the FLO-2D surface water system.

Chapter 2

FLO-2D Storm Drain Features

Overview of Inlets and Outfalls

FLO-2D storm drain component exchanges discharge between the surface water and the closed conduit system through a series of objects that are representations of the physical components in a storm drain system; primarily inlets and outfalls. The flow exchange can occur bi-directionally capturing surface water runoff when the conduits have capacity and returning flow to the surface water when the pipe system capacity is exceeded. The storm drain system is represented by links and nodes where the links are the closed conduits and nodes are junctions and outfalls as shown in Figure 23.

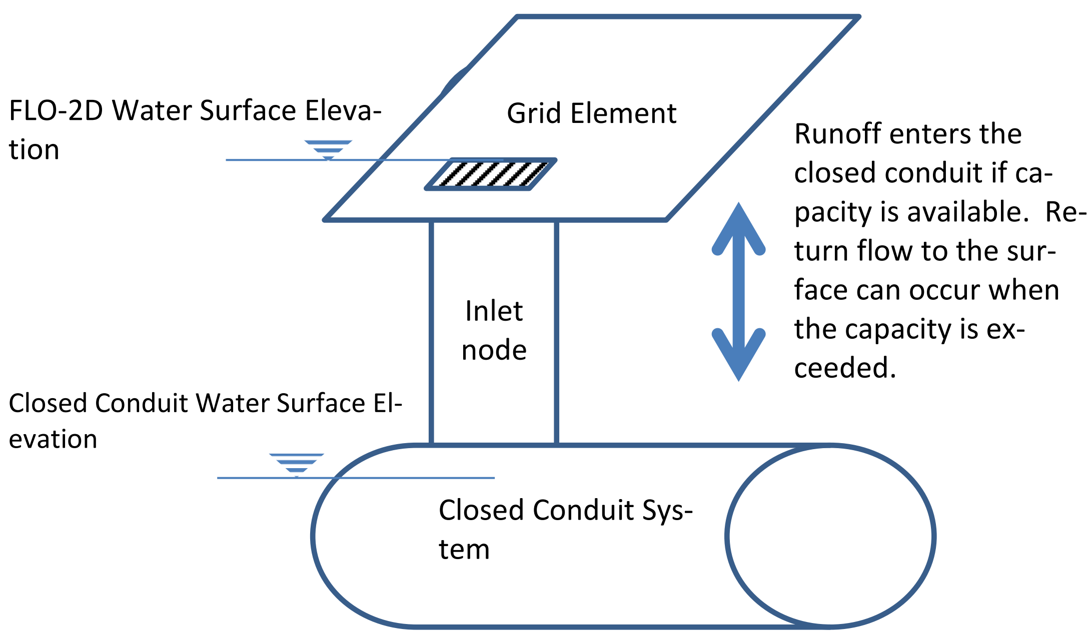


Figure 23. Surface and Closed Conduit Flow Exchange

Junctions represent several features of a storm drain system. One type of junction is the inlet which constitutes an inflow point to the pipe system from the surface water. A junction can represent the connection between two pipes with a change of slope or geometry, in this case the junction is not an inlet because it is not connected to the surface water. The water surface elevation is determined at each junction for each computational timestep based on the junction geometry and the flow hydraulics. Figure 24 is a profile of closed conduit system with three inlets and one outfall. The figure displays a peak water surface elevation for the 10 year return period storm.

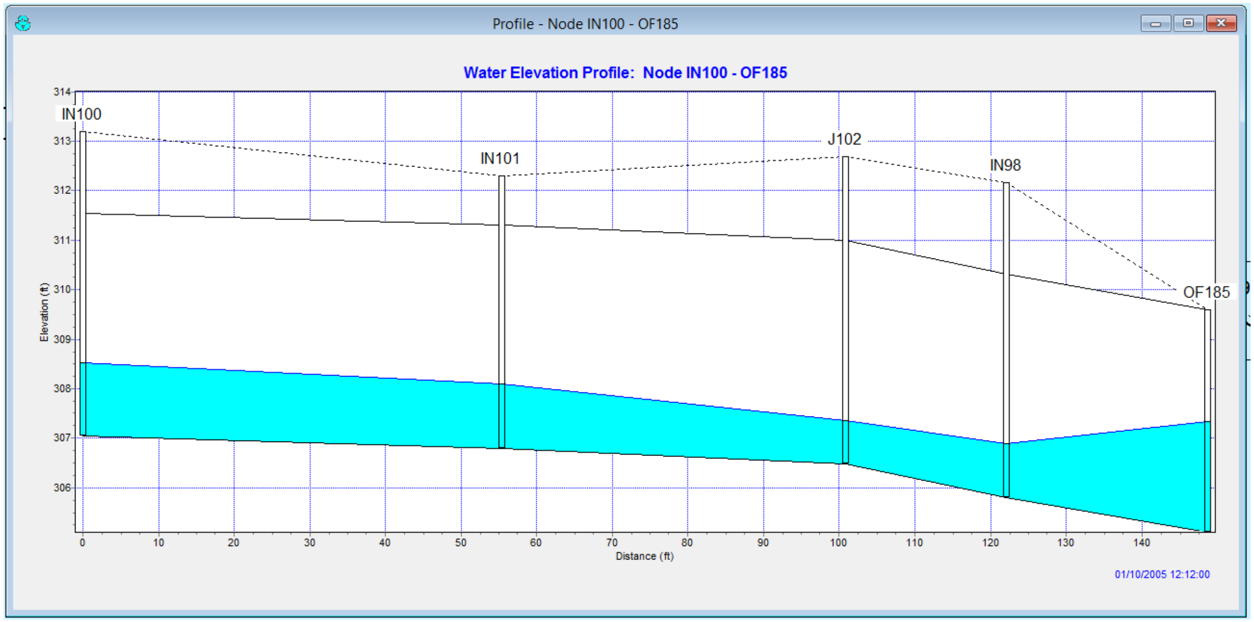


Figure 24. Profile of Typical Closed Conduit System

Inlets are junctions that collect surface water from the FLO-2D grid element. Only one inlet can be assigned to one FLO-2D grid cell. Storm drain inlets are depicted by the GDS as blue in Figure 25 and green nodes are junctions.

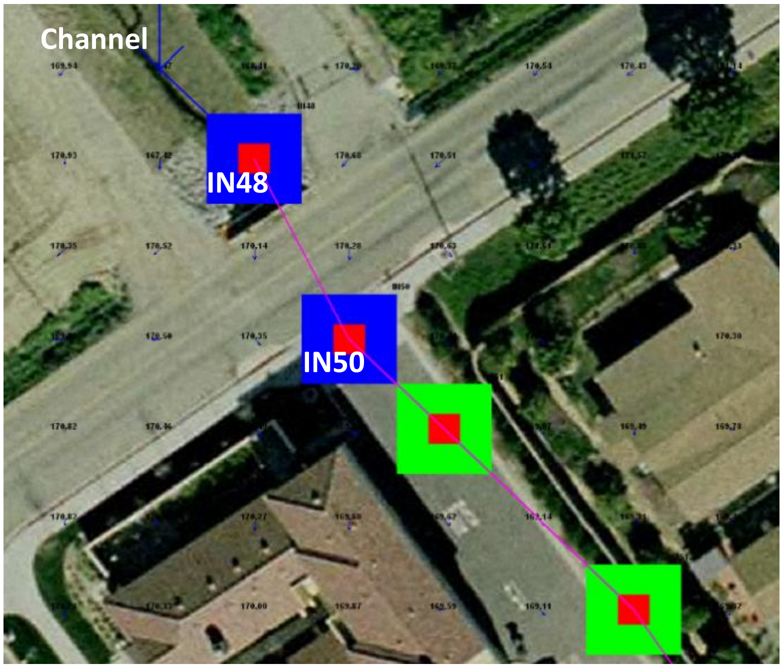


Figure 25. Storm Drain System Represented in the GDS Graphic Display

Outfalls are nodes located at the terminus of a pipe network and discharge flow out the storm drain system. A variety of outfall boundary conditions can be assigned including a fixed stage elevation, critical or normal flow depth, tidal stages, flapgates, timed controlled gates and time varying stage. The outfall can have return discharge to the FLO-2D surface water in channel, street or floodplains elements. If the outfall discharges to a channel element, the downstream boundary is the channel water depth. If the outfall discharge is returned to the surface water, the water surface is the downstream boundary condition with an assigned "free" condition. An outfall to a retention basin is shown in Figure 26. The boundary condition for this outfall would vary with the basin water surface. In Figure 27 the pipe outfall discharges into a channel and may have a potential backwater condition.

Figure 26. Outfall Discharge Retention Basin

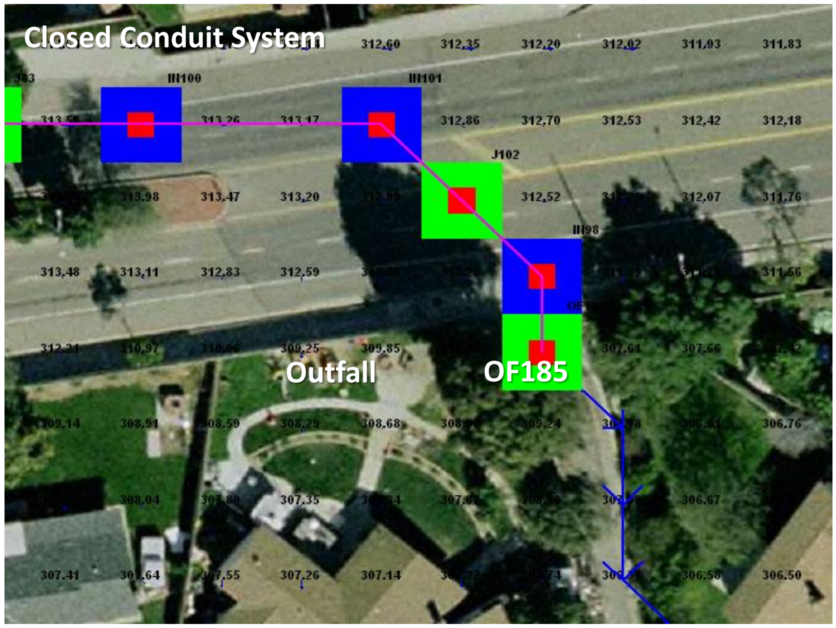


Figure 27. GDS Image of Storm Drain System with Outfall Discharging to a Channel

Storm Drain Inlets

FLO-2D can simulate 5 types of inlet geometries. Three of them are based on the Hydraulic Design Series No. 4 (HDS4) (Johnson and Fred, 1984). Inlets are designated as one of four types: (1) curb-opening inlets, (2) grate inlets, (3) slotted drains and (4) combination inlets. For curb and grate inlets, the inlet geometry is assigned and the discharge is computed using either weir or orifice equations when there is available storm drain capacity. For slotted drains, combination inlets and other non-typical inlets, a rating table must be generated with discharge as a function of headwater depth (Figure 28 and Figure 29).

Figure 28. Combination Curb Opening Inlet with Sag and a Grate

Figure 29. Combination of a Curb Opening at Grade with a Grate

The manhole is a special inlet type that represents a service access point to the storm drain system. Manholes are modeled as junctions with an additional surcharge depth that represents the equivalent pressure required to displace the cover. When the surcharge depth is exceeded, the manhole is popped, and the junction behaves as an inlet type 3.

Inlet Types

Type 1 - Curb Opening Inlet at Grade

The following are the input parameters for a Type 1 inlet (Figure 30):

- Weir coefficient: 2.85 - 3.30 (suggested 3.00 English, 1.6 metric)
- Curb opening length (L)
- Curb opening height (h)

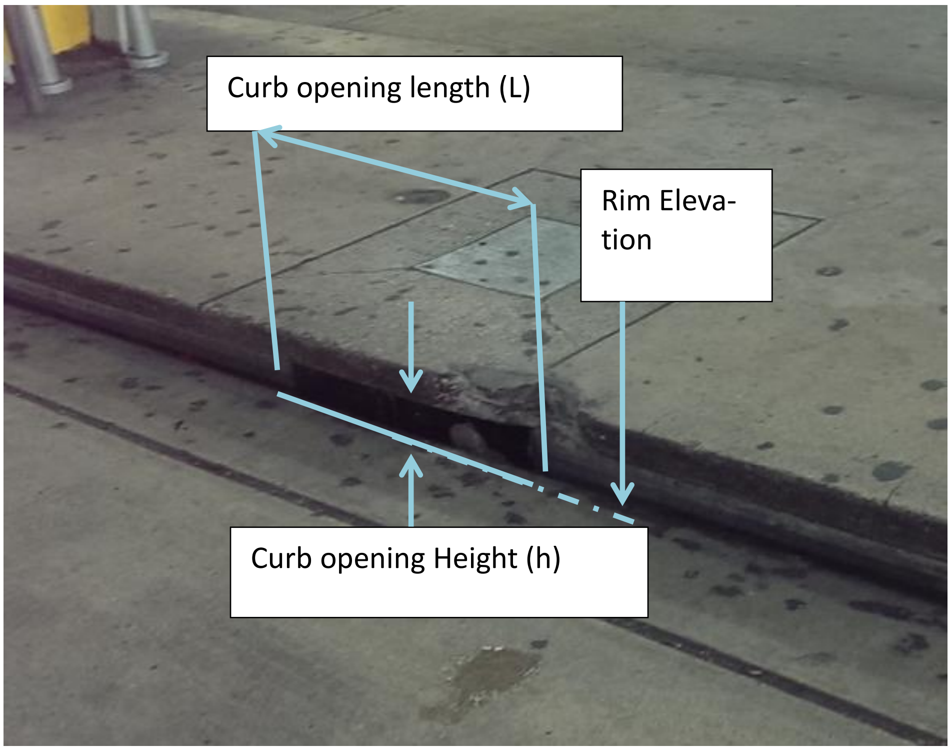


Figure 30. Curb Opening at Grade (Type 1 inlet)

The discharge is based on the flow depth in the FLO-2D grid cell and the inlet geometry:

- The inlet opening height (h), referred to as "Height" column 7 of the SWMMFLO.DAT.
- The inlet opening length (L) specified as "Length", column 5 of the SWMMFLO.DAT.

The discharge is calculated using the weir and orifice equations as follows:

1. If $h \leq H < 1.4h$, then:

    If $Q_w \leq Q_o$, discharge= Q_w Weir Equation Controls

If $Q_w > Q_o$, discharge= Q_o Orifice Equation Controls

2. If $H < h$, discharge= Q_w
3. If $H \geq 1.4h$, discharge= Q_o

Type 2 - Curb Opening Inlet with Sag

The following are the input parameters for a Type 2 inlet (Figure 31):

- Weir coefficient: 2.30 (1.25 metric)
- Curb opening length (L)
- Curb opening height (h)
- Curb opening sag width (W)

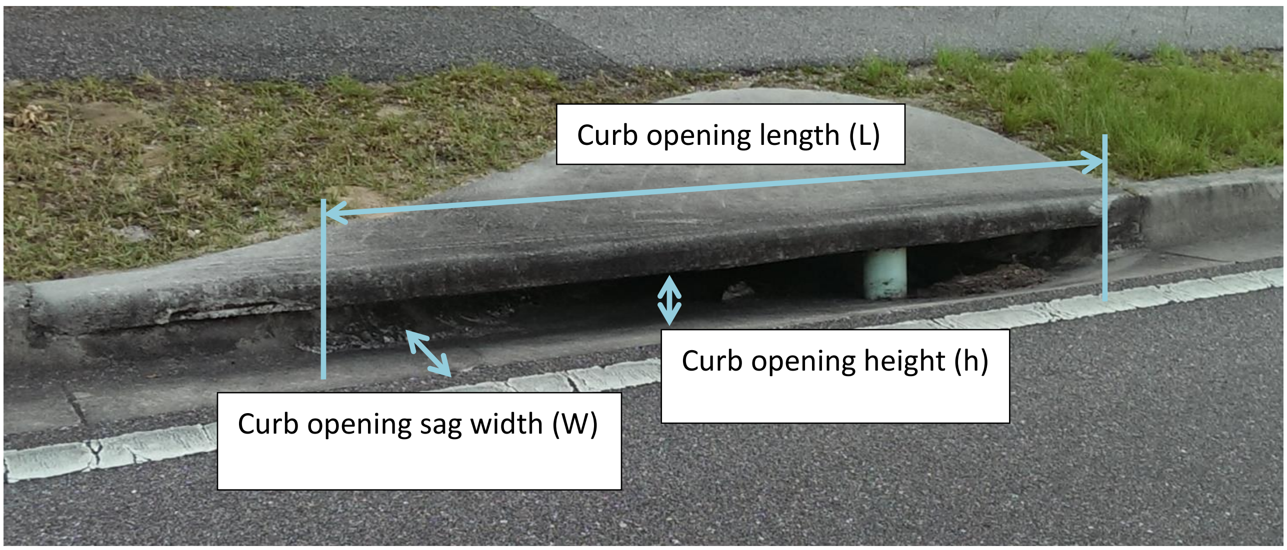


Figure 31. Curb Opening with Sag (Type 2 Inlet, Johnson and Fred, 1984)

Conservatively, the weir or orifice discharge, whichever is smaller, is used for the curb opening with sag. The inlet elevation is assumed to be equivalent to the grid element elevation. For weir flow (Johnson and Fred, 1984):

$$Q_w = C(L + 1.8W)H^m$$

where:

Q_w = weir flow rate at depth H

C = weir coefficient, 'Inlet Weir Coefficient.' field in SWMMFLO.DAT

L = curb opening length, 'Length' field in SWMMFLO.DAT.

W = curb opening sag width, 'Width' field in SWMMFLO.DAT.

H = FLO-2D inlet element flow depth

m = 1.5 exponent for an horizontal weir (hardcoded)

Orifice flow can have two potential sag inlet conditions (Johnson and Fred, 1984):

$$\text{If } h \geq H \text{ then } Q_o = C_d A\sqrt{2gH}$$

$$\text{If } h < H \text{ then } Q_o = C_d A\sqrt{2g\left(H - \frac{h}{2}\right)}$$

where:

Q_o = orifice flow rate at depth H

C_d = discharge coefficient hardcoded to 0.67

A = orifice area Area= L x h

g = gravitational acceleration

H = FLO-2D grid element water depth that contains the inlet structure.

h = curb opening height of the orifice.

Type 2 inlet discharge is determined from the following criteria:

If $h \leq H < 1.4h$, then:

If $Q_w \leq Q_o$, discharge= Q_w

If $Q_w > Q_o$, discharge= Q_o

If $H < h$, discharge= Q_w

If $H \geq 1.4h$, discharge= Q_o

Type 3 - Grate (Gutter) Inlet with/without Sag

The following are the input parameters for a Type 3 inlet (Figure 32):

- Weir coefficient: 2.85 - 3.30 (suggested 3.00 English, 1.6 metric)
- Grate perimeter (not including curb side)
- Grate open area
- Grate sag height (zero for at grade)

Note: Orifice flow coefficient = 0.67 (hardcoded) for all cases.

Figure 32. Grate Inlet in a Street (Type 3 Inlet, Johnson and Fred, 1984)

The smaller of two discharges (weir or orifice) calculated for a grate (gutter) inlet with/without sag is applied for the inlet discharge computation (Johnson and Fred, 1984):

Weir Flow:

If $H \leq h$ then: $Q_w = CPH^m$

If $H > h$ then: $Q_w = CP\left(H + \frac{h}{2}\right)^m$

where:

Q_w = weir flow rate at depth H

C = weir coefficient, 'Inlet Weir Coeff.' field in SWMMFLO.DAT

P = Grate perimeter 'Perimeter' field in SWMMFLO.DAT

H = FLO-2D inlet element flow depth

m = 1.5 horizontal weir exponent (hardcoded).

h = sag height from the 'Sag Height' field of SWMMFLO.DAT.

Orifice Flow:

If $H \leq h$ then: $Q_o = C_d A\sqrt{2gH}$

If $H > h$ then: $Q_o = C_d A\sqrt{2g\left(H + \frac{h}{2}\right)}$

where:

Q_o = orifice discharge at depth H

C_d = discharge coefficient (hardcoded to 0.67)

A = orifice area 'Area' field in SWMMFLO.DAT

g = gravitational acceleration

H = FLO-2D inlet element flow depth

h = sag height from the 'Sag Height' field of SWMMFLO.DAT.

The discharge is determined using the following criteria for Type 3:

If $0.75 \leq \mathrm{H} < 1.8$, then:

If $Q_w \leq Q_o$, discharge= Q_w

If $Q_w > Q_o$, discharge= Q_o

If $\mathrm{H} < 0.75$, discharge= Q_w

If $\mathrm{H} \geq 1.8$, discharge= Q_o

Type 4 – Unique Inlet with Stage-Discharge Rating Table

Inlets that cannot be represented by Types 1, 2 or 3 can have an inflow discharge defined by a rating table. Some storm drain inlets may include entrance types similar to that of a culvert with a vertical opening (Figure 33.). In this case the rim elevation would be ignored as the flows are exchanged based on the invert elevation.

Figure 33. Storm Drain Vertical Inlet with a Culvert Entrance

The following are the input parameters for a Type 4 inlet that are entered in the SWMMFLORT.DAT file:

Stage (depth) above inlet (ft or m)

Discharge (cfs or cms)

Note: The stage-discharge data is assigned in pairs with the first pair being: 0. 0.

Type 5 – Manhole

Manholes are a special case of inlets. Storm drains under high pressure during flooding can result in the manhole covers being popped off (Figure 34. and Figure 35). The FLO-2D storm drain component can simulate covers popping through the application of a surcharge depth. Once the manhole cover has been popped, it remains off and the manhole becomes a Type 3 inlet.

Figure 34. Popped Manhole Cover (source: istock)

The required manhole input parameters are:

- Weir coefficient: 2.85 - 3.30 (suggested 3.00 English, 1.6 metric)
- Manhole perimeter (manhole cover shapes can vary)
- Manhole flow area (ft^2 or m^2)
- Surcharge depth (ft or m).

A manhole is assumed to be level without sag and column 7 in the SWMMFLO.DAT is used to define the surcharge depth (ft or m). The surcharge depth can be estimated by the user as the equivalent depth that the pressure has to overcome to pop the cover. The pressure force must exceed the manhole cover weight represented as the equivalent weight of the volume of water that is above the manhole rim elevation. Manhole covers are typically circular but can be found in other shapes.

The water depth (head) that represents the manhole cover weight can be estimated by:

$$d_s = \frac{w_m}{A_m \gamma_w}$$

where:

d_s = surcharge depth that pops the manhole cover

w_m = weight of the manhole cover

A_m = area of the circular manhole opening ($\pi D^2/4$)

γ_w = clear water density 62.4 lb/ft^3 (1000 kg/m^3)

Figure 35. Popped Manhole Cover with Pressure Head (Source: istock)

To calculate the surcharge depth for manhole cover shown in Figure 34. , the observed head of water is approximately 1.0 feet (0.3 meter). The manhole cover weight and diameter must be known. Assuming a diameter of 2 ft (0.61 m) and a weight of 100 pounds, what surcharge depth d_s had to be exceeded to pop the cover?

Solution:

w_m = 100 lb (45.4 kg)

A_m = $(\frac{\pi(2\text{ft})^2}{4})$ = 3.14 ft^2 (0.29 m^2)

$d_s = \frac{w_m}{A_m \gamma_w} = \frac{100\text{lb}}{3.14\ \text{ft}^2\ 62.4\ \text{lb/ft3}} \approx \frac{45.4\ \text{kg}}{0.29\ \text{m}^2\ 1000\ \text{kg/m3}}$ ≈ 0.5 ft (0.16m)

Table 3 presents some suggested surcharge depths.

Table 3. Manhole Cover Surcharge Depths

Manhole Cover Representative Weight	Surcharge Depth (ft) (assuming 2 ft diameter)
85 to 300 lb manhole cover unbolted	0.4 to 1.5
100 lb to 150 lb rubber cover with plastic bolts	0.5 to 0.75
2,400 lb parked car with one wheel on cover unbolted (~600 lb + 300 lb)	4.6 or greater
5,000 lb steel bolted manhole cover (est.)	25.5 or greater

Storm Drain Outfalls

A variety of outfall conditions are available. For the outfall to discharge back to the surface water, the boundary condition must be set to 'free' in the the SWMM.inp file and the 'allow discharge' switch set ON in the SWMMOUTF.DAT file. Flow into the outfall can occur based on the water surface elevation but may be restricted by a flapgate assignment. Normal, fixed, tidal or time series type of SWMM outfalls will discharge the flow off the storm drain system. In Figure 36, the outfall has a free flow condition but when the river is high, the outfall may be submerged. The location and elevation of the outfall can be assigned in the SWMM GUI.

Figure 36. Storm Drain Free Outfall Condition (source: istock)

During the simulation FLO-2D compares the downstream water surface elevations with the storm drain pressure head to control the flow in or out of the outlet. This may result in back-water pressure on the pipe network. In the case of ponded flow as in detention or retention basins, the outfall might be submerged but still have sufficient pressure to discharge out of the storm drain network (Figure 37). When the retention basin water surface is high enough, water can enter the outfall and flow upstream in the pipe.

Figure 37. Outfall Discharging into a Retention Basin

The discharge at the outfall is controlled as follows:

If storm drain pressure head > WSE, then outfall discharges to the surface water cell

If storm drain pressure head < WSE, then there is no outfall discharge and the outfall node depth = WSE – outfall invert elevation.

Flapgates can be assigned in the SWMM.inp file to stop the flow from going into the storm drain system through the outfall. Flow could leave the storm drain system but not enter it.

Storm drain outfalls should be assigned to the channel left bank element to discharge to the channel. If the outfall coordinates in the SWMM.inp file are the left bank channel element co-ordinates then the GDS will automatically assign the outfall node to the left bank element. The SWMMOUTF.OUT file can be reviewed to ensure that the outfall is correctly paired to the left bank element. If the coordinates in the SWMM.inp are not the left channel element coordinates, then the grid element number in the SWMMOUTF.DAT can be replaced with channel left bank element number (Figure 38). The outfall coordinates in the SWMM.inp file do not need to be modified since they will not affect the storm drain results calculations.

Typically, the outfall invert elevation would be assigned to the channel element thalweg elevation. Assignment of the outfall to the right bank elevation, or channel interior elements would generate an error message. If the outfall invert elevation is lower than the thalweg elevation (underground), then the storm drain would be assumed to be underwater with an initial tailwater depth.

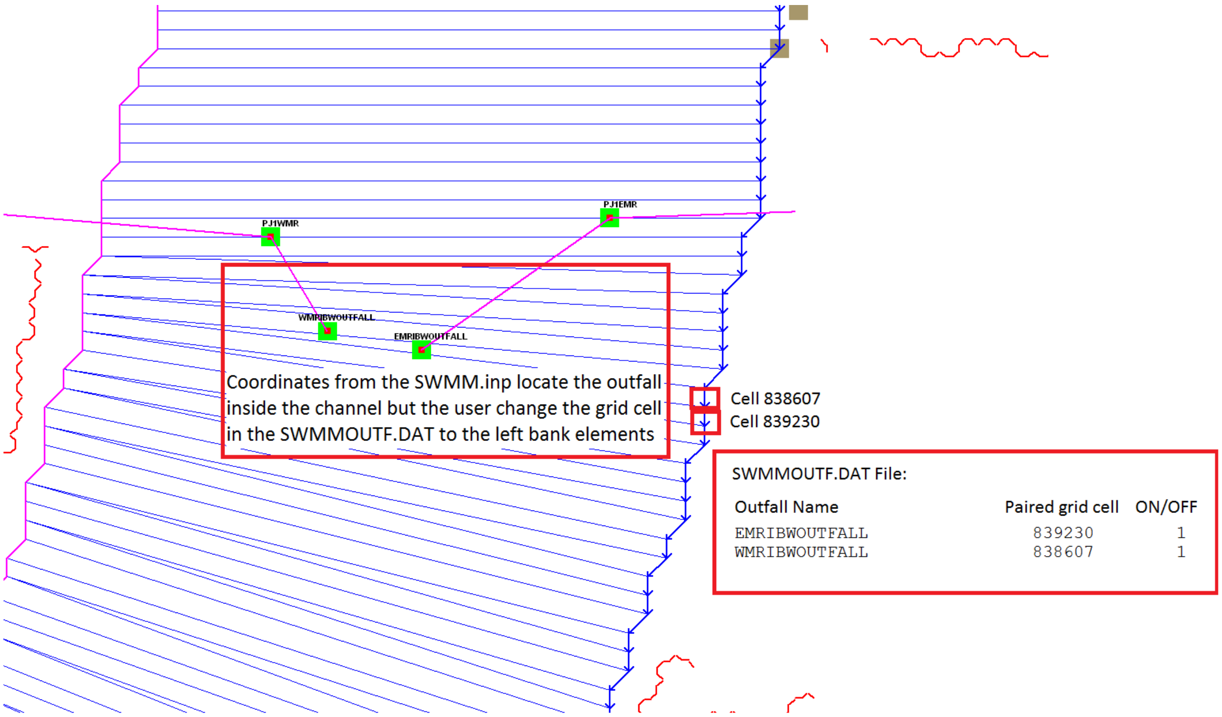


Figure 38. Outfall Location with Respect the Channel Left Bank Elements.

Elevations and Datum

The elevation and location of the inlets and outfalls are required to exchange flow with the surface water. The floodplain water surface elevation is compared to the storm drain pressure head based on a common reference such as the inlet rim elevation, inlet invert elevation (Vertical Type 4 Inlets) or the outfall invert elevation. It is possible that the geometry of the closed conduit system may have a different datum than that applied to the FLO-2D surface topography. The elevation of each inlet rim and outfall invert should be reviewed especially for unique conditions such as combination inlets (Figure 39).

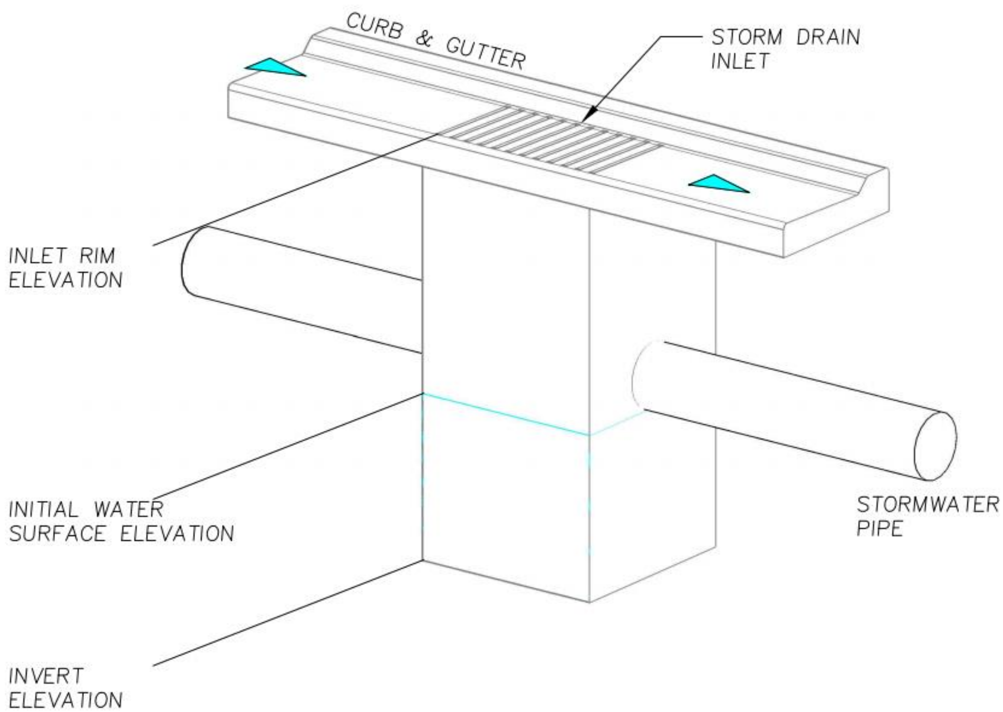


Figure 39. Diagram of a Combination Inlet Rim Elevation.

For curb opening inlets at grade (Type 1), curb opening inlets with sag (Type 2), grates with or without sag (Type 3) and manholes (Type 5), the FLO-2D model computes the rim elevation at the node by adding the maximum depth in the SWMM.inp file to the invert elevation. The model compares the rim elevation with the grid element elevation and reports the differences in the FPRIMELEV.OUT. If the rim elevation and floodplain elevations are different, the model reassigns the floodplain elevation to the rim elevation based on the assumption that the rim elevation was surveyed and the grid element elevation was interpolated. The model uses the rim elevation as the reference to determine the water surface elevation and inlet flow depth.

A Type 4 inlet with a defined rating table can represent any inlet not defined by Types 1 thru 3. The rating table reference elevation must correlate with the inlet rim or invert elevation. If the Type 4 inlet is a horizontal inlet (feature = 0 in the SWMMFLO.DAT file), then the floodplain elevation is automatically reset to the rim elevation. Typically, inlets have horizontal inlets, but some inlets such as culverts have vertical openings. For vertical inlets, the water surface elevations are compared at the node, but the rim elevation is ignored. Vertical inlets have some unique constraints:

1. For an inlet on a channel (channel flow is discharging to the storm drain pipe), the invert elevation should be set to the channel bed elevation. If the Type 4 inlet is a

vertical inlet and it is located in a channel cell, the 'Feature' column has to be equal to 1 in the SWMMFLO.DAT file and the floodplain elevation is not modified.

2. For a floodplain swale where the flow is discharging to a storm drain pipe or culvert, the floodplain grid element elevation should match the pipe invert elevation. If the Type 4 inlet is a vertical inlet and it is located in a floodplain cell, the 'Feature' column has to be equal to 1 in the SWMMFLO.DAT file and the floodplain elevation equal to the pipe invert elevation otherwise the floodplain elevation is modified at runtime.
3. The 'Feature' column in the GDS inlet dialog window has three options:

    0 - default, no flapgate, no vertical inlet opening

    1 - vertical inlet opening

    a. Channel pipe inlet invert elevation = channel bed elevation
    b. Floodplain grid element elevation = pipe invert elevation

    2 - flapgate (outfall)
4. Revised floodplain elevations are not changed in the FPLAIN.DAT file. These modifications are only implemented at runtime. For permanent floodplain revisions, the user must adjust the elevations in FPLAIN.DAT to match the modifications in FPRIMELEV.OUT. Rim elevations for the inlets located in channel or street cells are not checked and must be verified by the user.
5. Unique inlet conditions such as those with unusual shape openings are simulated with a rating table.

All runtime changes in the floodplain elevation are reported in FPRIMELEV.OUT and Stormdrain_error.chk files. They are reported to the FPLAIN_SDElev.RGH and TOPO_SDElev.RGH files and the FPLAIN.DAT AND TOPO.DAT files can be replaced by renaming the files to apply the storm drain elevation adjustments to the next simulation.

Chapter 3

FLO-2D Storm Drain Data Files

Input and Output File - General

This chapter describes the input and output data and their format for the FLO-2D storm drain component. The storm drain input data files are divided into two categories:

- Files that have to be created using a storm water user interface GUI. Some software GUI options are: EPA SWMM 5, inpPINS, Autodesk Storm and Sanitary Analysis, Civil 3D Storm and Sanitary Analysis, and the QGIS plug-in. The GUI for inpPINS and Civil 3D Storm and Sanitary Analysis integrate storm water objects with Geographical Information Systems (GIS).
- Files created by the FLO-2D GDS and QGIS plug-in.

The following folders will contain the complete storm drain model system:

- FLO-2D PRO folder in C:\Program Files (x86). All executable program files for the FLO-2D model including pre- and post-processing and the FLOPRO.exe are located in this folder.
- FLO-2D PRO documentation folder in C:\Users\Public\Documents\FLO-2D PRO Documentation (My Documents). Manuals, Handout Documents, PowerPoint Presentations and tutorials are located in this directory.

Data Input Files

The FLO-2D data input files are generated by the GDS and the QGIS plug-in. These tools read the SWMM.inp file which associates the designated inlets with grid elements. Typically, the FLO-2D grid system is prepared prior to developing the storm drain model. The GDS and QGIS plug-in will generate the files needed to run the FLO-2D Storm drain model. The following data files have to be created to run a FLO-2D storm drain simulation.

CONT.DAT

This file contains the simulation control parameters:

- Set SWMM = 1 to initiate the storm drain component;
- For no storm drain simulation, SWMM = 0 (default).

SWMM.inp

SWMM.inp is the input file that contains most of the storm drain project data. It includes pipe network geometry, inlet/outlet and junction locations, simulation control settings and hydraulic routing properties. The pipe network components in SWMM.inp have to be entered in the SWMM GUI, QGIS plug-in or other storm drain software. Table 4 lists the section data in the SWMM.inp file. Each section begins with a keyword.

Table 4. Sections that Enclose Data in the SWMM.inp (Rossman, 2005)

[TITLE]	Project title
[OPTIONS]	Analysis options
[REPORT]	Output reporting instructions
[FILES]	Interface file options
[JUNCTIONS]	Junction node information
[OUTFALLS]	Outfall node information
[DIVIDERS]	Flow divider node information
[STORAGE]	Storage node information
[CONDUITS]	Conduit link information
[PUMPS]	Pump link information
[ORIFICES]	Orifice link information
[WEIRS]	Weir link information
[OUTLETS]	Outlet link information
[XSECTIONS]	Conduit, orifice, and weir cross-section geometry
[TRANSECTS]	Transect geometry for conduits with irregular cross-sections
[LOSSES]	Conduit entrance/exit losses and flap valves
[CONTROLS]	Rules that control pump and regulator operation

The structure of the SWMM.inp file follows:

- Sections appear in any arbitrary order in the input (*.inp) file. Section keywords can appear in mixed lower and upper case letters. Only the first four characters (plus the open bracket) are used to distinguish one keyword from another (e.g., [DIVIDERS] and [Divi] are equivalent).
- Not all sections must be present on each project.
- Each section can contain one or more lines of data.
- Blank lines may appear anywhere in the file.
- A semicolon (;) can be used to indicate that comment follows, not data.
- Data items can appear in any column of a line.
- The data is ordered creating a tabular appearance complete with column headings.
- When listing the format of a line of data, mandatory keywords are shown in boldface while optional items appear in parentheses.
- A list of keywords separated by a slash (YES/NO) means that only one of the words should appear in the data line.
- In the [OPTIONS] section, flow units can be selected as either cubic feet per second (CFS), gallons per minute (GPM), million gallons per day (MGD), cubic meters per second (CMS), liters per second, (LPS), or million liters per day (MLD). If cubic feet or gallons are

chosen for flow units then US units are used for all other quantities. If cubic meters or liters are chosen then metric units apply to all other quantities. The default flow units are CFS.

SWMM.inp Example

The typical structure of a SWMM.inp file can be shown as follows:

```
[TITLE]
INP file created on 10/17/2014 9:30:50 AM by FLO-2D Team
Title/Notes:

[OPTIONS]
FLOW_UNITS          CFS
INFILTRATION        HORTON
FLOW_ROUTING        DYNWAVE
START_DATE          03/25/2013
START_TIME          00:00:00
REPORT_START_DATE   03/25/2013
REPORT_START_TIME   00:00:00
END_DATE            03/25/2013
END_TIME            00:24:00
SWEEP_START         01/01
SWEEP_END           12/31
DRY_DAYS            0
REPORT_STEP         00:01:30
WET_STEP            00:01:00
DRY_STEP            00:01:00
ROUTING_STEP        0.05
ALLOW_PONDING       YES
INERTIAL_DAMPING    PARTIAL
VARIABLE_STEP       0.00
LENGTHENING_STEP    0
MIN_SURFAREA        0
NORMAL_FLOW_LIMITED BOTH
SKIP_STEADY_STATE   NO
FORCE_MAIN_EQUATION D-W
LINK_OFFSETS        DEPTH
MIN_SLOPE           0

[EVAPORATION]
;;Type       Parameters
;;---------- ----------
CONSTANT     0.0
DRY_ONLY     NO

[JUNCTIONS]
;;               Invert     Max.       Init.      Surcharge  Ponded
;;Name           Elev.      Depth      Depth      Depth      Area
;;-------------- ---------- ---------- ---------- ---------- ----------
```

```
I1CP1C14   1264.35    6.34            0         0         0

[OUTFALLS]
;;               Invert     Outfall    Stage/Table      Tide
;;Name           Elev.      Type       Time Series      Gate
;;-------------- ---------- ---------- ----------------      ----
OUTFALL   1254.20    FREE                          NO

[CONDUITS]
;;               Inlet           Outlet                       Manning    Inlet      Outlet     Init.      Max.
;;Name           Node            Node            Length     N          Offset     Offset     Flow       Flow
;;-------------- ---------------- ---------------- ---------- ---------- ---------- ---------- ---------- ----------
C1             PJ1C1          FOUTFALL        176        0.013      0         0         0         0

[XSECTIONS]
;;Link           Shape        Geom1            Geom2      Geom3      Geom4      Barrels
;;-------------- ------------ ---------------- ---------- ---------- ---------- ----------
C1             CIRCULAR     10              0         0         0         1

[LOSSES]
;;Link           Inlet      Outlet     Average    Flap Gate
;;-------------- ---------- ---------- ---------- ----------

[REPORT]
INPUT      YES
CONTROLS   YES
NODES ALL
LINKS ALL

[TAGS]

[MAP]
DIMENSIONS 696350.607 908521.547 697144.791 909040.891
Units      None

[COORDINATES]
;;Node           X-Coord            Y-Coord
;;-------------- ------------------ ------------------
I1CP1C14         696446.346         908983.021

[VERTICES]
;;Link           X-Coord            Y-Coord
;;-------------- ------------------ ------------------
```

SWMMFLO.DAT

A junction must have an ID starting with an 'I' to differentiate the inlet from a pipe junction. Surface water discharge is only shared between those junctions that have the correct ID assigned in the *.inp file. Inlets are automatically paired by the GDS to a corresponding FLO-2D grid element. The SWMMFLO.DAT file contains the inlet geometry as well as the names or numbers that identify the inlet that is paired with the grid cell (Table 5).

Table 5. SWMMFLO.DAT Variables

SWMMFLO.DAT File Variables
Line 1 to Number of Inlets (JT): SWMMCHAR='D' SWMM_JT(JT) SWMM_IDEN(JT) INTYPE(JT) SWMMlength(JT) SWMMwidth(JT) SWMMheight(JT) SWMMcoeff(JT) FLAPGATE(JT) CURBHEIGHT(JT) Notes: Multiple Inlets (SWMM_IDEN(JT)) assigned to the same grid cell (SWMM_JT(JT)) constitutes a data error. An Inlet (SWMM_IDEN(JT)) assigned to more than one grid cell (SWMM_JT(JT)) constitutes a data error.

The Table 6 outlines the required data in the SWMMFLO.DAT.

Table 6. SWMMFLO.DAT Input File Example

SWMMFLO.DAT
D 452161 I19CP1SEMDRHRL 1 12 0 0.417 3 0 0 D 451297 I20CP1SEMDRHRL 1 4.8 0 0.417 3 0 0 D 451302 I21CP1SEMDRHRL 1 12 0 0.417 3 0 0 D 450875 I21CP2SEMDRHRL 3 6 4.4 0 3 0 0 D 441072 I22CP1SEMDRHRL 1 12 0 0.417 3 0 0 D 440649 I22CP2SEMDRHRL 3 5 3.0 0 3 0 0

Table 7 lists the variable descriptions for the SWMMFLO.DAT file:

Table 7. SWMMFLO.DAT Input Variable Descriptions

Variable	Format	Range	Description
SWMMCHAR	c	--	Character Line Identifier
SWMM_JT(JT)	i	--	Cell Grid paired with the Inlet
SWMM_IDEN(JT)	c	--	Inlet Name
INTYPE(JT)	i	1, 2, 3, 4 or 5	Type of inlet: 1: Curb opening inlet at grade 2: Curb opening inlet with sag 3: Grate (gutter) inlet with/without sag

			4: Unique inlet with stage-discharge rating table 5: Manhole
SWMMlength(JT)	r	0.01- ∞	Curb opening length for INTYPE= 1 or 2 Grate perimeter (not including curb side) for INTYPE= 3 For INTYPE = 4 (set to 0 – not needed) Manhole perimeter for INTYPE = 5
SWMMwidth(JT)	r	0.01- ∞	For INTYPE = 1 (set to 0 – not needed) Curb opening sag width for INTYPE=2 Grate open area for INTYPE=3 For INTYPE = 4 (set to 0 – not needed) Manhole flow area for INTYPE = 5
SWMMheight(JT)	r	0.01- ∞	Curb opening height for INTYPE=1 Curb opening height for INTYPE=2 Grate sag height for INTYPE = 3 For INTYPE = 4 (set to 0 – not needed) Surcharge depth for INTYPE=5
SWMMcoeff(JT)	r	2.8-3.3	Recommended weir coefficients are: For INTYPE= 1,3 and 5: Range 2.8 to 3.2 For INTYPE=2: 2.3 For INTYPE = 4 (set to 0 – not needed)
FLAPGATE(JT)	i	0, 1, or 2	For INTYPE = 4: 0 = default, no flap gate, no vertical inlet opening 1 = vertical inlet opening 2 = flapgate, controls outlet node discharge For a fake outfall INTYPE = 1, 2, 3 and 5 can be 0 or 2
CURBHEIGHT(JT)	r	0.01- ∞	Curb height used to calculate discharge on inlets for all INTYPE

(i) = Integer variable (r) = Real variable (c) = Character

To create the SWMMFLO.DAT, open the project in GDS and locate Tools | Storm Drain | View Storm Drain Inlets Dialog command. Browse for the SWMM.inp file using the GDS Window “Select a Storm Drain *.inp file” shown in Figure 40.

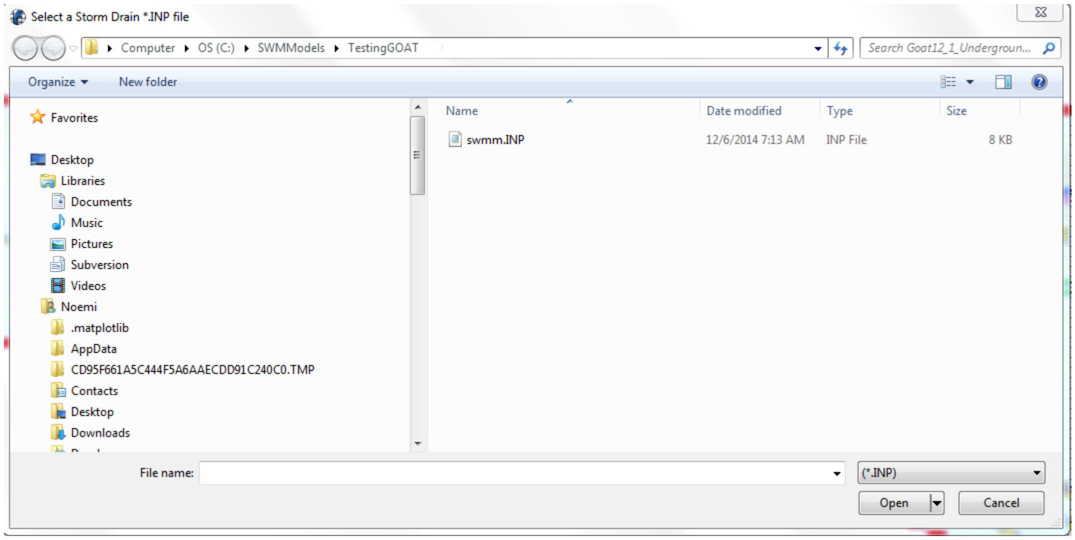


Figure 40. GDS Open *.inp File

The GDS will read the *.inp data file and those junctions which names start with an 'I' is identified by the GDS. The data from the storm drain file will then be paired with the data in the SWMMFLO.DAT file. Enter the inlet geometry and save the SWMMFLO.DAT file. The SWMMFLO.DAT file dialog box is shown in Figure 41.

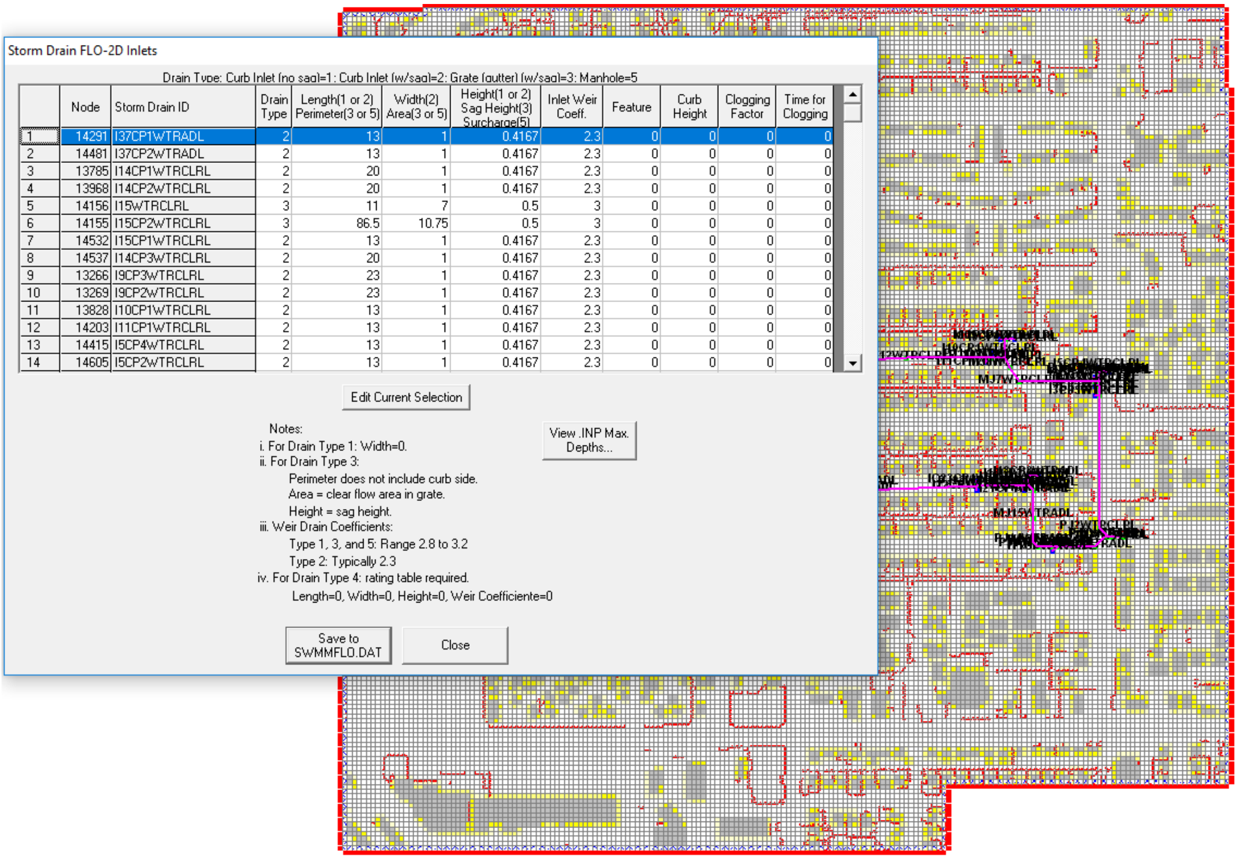


Figure 41. GDS Enter Storm Drain Inlet Geometry Data

The typical structure of a SWMMFLO.DAT file is shown in Figure 42.

```
SWMMFLO.DAT - Notepad
File   Edit   Format   View   Help
D        392753  I18CP3WTRADL     1    16     0   0.42     3     0     0
D        392752  I18CP2WTRADL     2    13     1  0.4167    2.3    0     0
D        394086  I11CP1WTRADL     3   2.13   1.05     0     3     0     0
D        404201  I12CP1WTRADL     4     0     0     0     0     0     0
D        403476  I26CP1WTRADL     4     0     0     0     0     0     0
D        394782  I25CP1WTRADL     5   4.00   4.00   3.00   3.00    0     0
D        394076  I25CP2WTRADL     1    4.8    0   0.42     3     0     0
D        395498  I24CP1WTRADL     1    7.2    0   0.42     3     0     0
D        394822  I22CP1WTRADL     1   10.4    0   0.42     3     0     0
D        394112  I23CP1WTRADL     1   10.4    0   0.42     3     0     0
D        394832  I21CP1WTRADL     1   10.4    0   0.42     3     0     0
D        394122  I20CP1WTRADL     1   10.4    0   0.42     3     0     0
D        381176  I3CP1WTRCLRL     1    7.2    0   0.42     3     0     0
D        377649  I37CP2WTRADL     1   10.4    0   0.42     3     0     0
D        377648  I37CP1WTRADL     1   10.4    0   0.42     3     0     0
D        400511  I29CP2WTRADL     1    16     0   0.42     3     0     0
D        377771  I5CP4WTRCLRL     1   10.4    0   0.42     3     0     0
D        377773  I5CP2WTRCLRL     1   10.4    0   0.42     3     0     0
D        380495  I4CP1WTRCLRL     4     0     0     0     0     0     0
D        379132  I6CP1WTRCLRL     1   10.4    0   0.42     3     0     0
D        373666  I9CP2WTRCLRL     1   18.4    0   0.42     3     0     0
```

Figure 42. Typical SWMMFLO.DAT Input Data File

SWMMOUTF.DAT

This file lists the outfall data found in the SWMM.inp file. The option to allow the outfall discharge to be returned to the surface water can be selected by checking the 'Allow Discharge' box in Figure 43. . This is the 'Free' outfall condition. Unchecking the 'Allow Discharge' box ('off') forces outfall discharge to be removed from the complete model system.

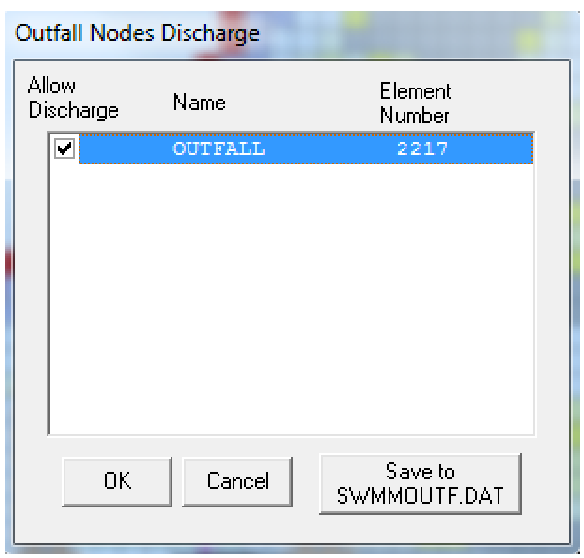


Figure 43. GDS Dialog Box for Storm Drain Outfall Switch

The required data in the SWMMOUTF.DAT is shown in **Error! Not a valid bookmark self-reference.** and Table 9:

Table 8. SWMMOUTF.DAT Variables

SWMMOUTF.DAT File Variables
Line 1 to Number of Outfalls (JT): *OUTF_NAME(JT) OUTF_GRID(JT) OUTF_FLO2DVOL(JT)*

Table 9. SWMMOUTF.DAT Input File Example

SWMMOUTF.DAT File Example		
OF1	72565	1
OF2	98863	0
OF3	97479	1
OF4	51882	1
OF5	216195	0
OF7	382089	1
OF1-79S	287432	0
OF1-51S	320048	0
OF185	232611	0

Table 10 lists the variable descriptions for the SWMMOUTF.DAT file:

Table 10. SWMMOUTF.DAT Input Variable Descriptions

Variable	For-mat	Range	Descriptions
OUTF_NAME(JT)	c	--	Outfall name
OUTF_GRID(JT)	i	--	Cell grid paired with the outfall
OUTF_FLO2DVOL(JT)	i	0: off or 1: on	Allow discharge from the storm drain to FLO-2D

(i) = integer variable (r) = real variable (c) = character

SWMMOUTF.DAT file should contain the list of outfalls in the same order as it appears on the SWMM.inp. When the outfall order is modified in the SWMM.inp file because an outfall node was added or deleted, the list of outfall nodes in the GDS should be edited and the SWMMOUTF.DAT file saved. The functionality of the outfall nodes is as follows:

- If the outfall discharge is 'off' the outfall will discharge off the complete model system. No discharge is returned from the storm drain to the surface water.
- If the outfall discharge switch is 'on' the surface water elevation and storm drain pressure head are compared and the outfall will discharge until WSE is equal or greater than the storm drain head. The outfall flow drain back to the surface water.
- Potential backflow into the outfall pipe will depend on the comparison of the WSEL, the storm drain pressure head and the flapgate assignment.

SWMMFLORT.DAT

The SWMMFLORT.DAT file contains a list of the rating table data. This option (INTYPE = 4 in the SWMMFLO.DAT) is assigned by the GDS in the inlet geometry dialog box (Figure 44.). The rating table is used throughout the simulation without adjustment.

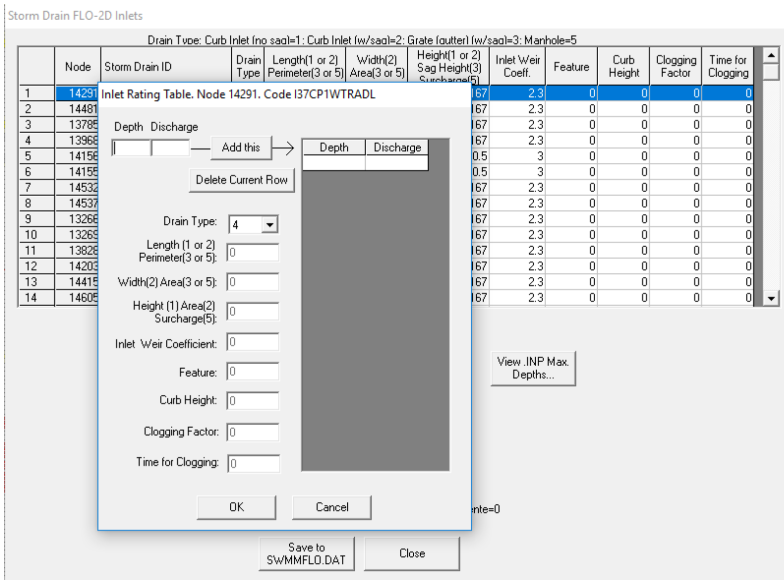
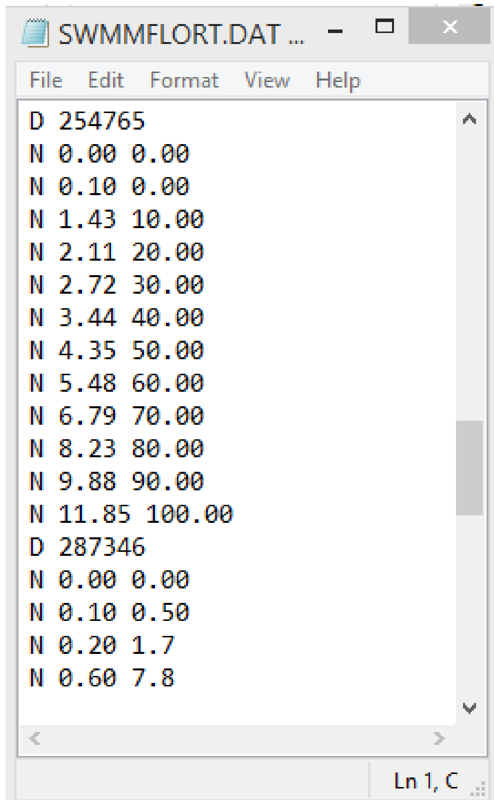


Figure 44. GDS Dialog Box for Entering Rating Table data for INTYPE 4

The structure of a SWMMFLORT.DAT file is:

ID Grid Cell
ID Depth Discharge
ID Depth Discharge
ID Depth Discharge
…

The first pair of numbers should be zero depth and zero discharge. This is repeated from 1 to number of storm drain inlets with INTYPE= 4 (Figure 45 and Table 11).



```
D 254765
N 0.00 0.00
N 0.10 0.00
N 1.43 10.00
N 2.11 20.00
N 2.72 30.00
N 3.44 40.00
N 4.35 50.00
N 5.48 60.00
N 6.79 70.00
N 8.23 80.00
N 9.88 90.00
N 11.85 100.00
D 287346
N 0.00 0.00
N 0.10 0.50
N 0.20 1.7
N 0.60 7.8
```

Figure 45. Typical SWMMFLORT.DAT input file

Table 11. SWMMFLORT.DAT Variables

SWMMFLORT.DAT File Variables
Line 1 to Number of INTYPE 4 (JTSWMM(JT)): SWMMCHARRT GRIDCELL SWMMCHARRT DEPTHSWMMRT(JTSWMM(JT),K) QSWMMRT(JTSWMM(JT),K)

The required data in the SWMMFLORT.DAT is listed in Table 10.

Table 12. SWMMFLORT.DAT Input File Example

SWMMFLORT.DAT File Example
D 153076
N 0.00 0.00
N 0.70 10.00
N 1.12 20.00
N 1.46 30.00
N 1.77 40.00
N 2.06 50.00
N 2.32 60.00
N 2.57 70.00
N 2.81 80.00
D 199236
N 0.00 0.00
N 0.79 10.00
N 1.19 20.00
N 1.56 30.00
N 1.89 40.00
N 2.19 50.00
N 2.47 60.00
N 2.74 70.00

Table 13 lists the variable descriptions for the SWMMFLORT.DAT file.

Table 13. SWMMFLORT.DAT Input Variable Descriptions

Variable	Format	Range	Description
SWMMCHARRT	c	D or N	D: line with the grid cell paired with the INTYPE 4 N: line with the rating table data
GRIDCELL	i	--	Cell Grid paired with the INTYPE 4
DEPTHSWMMRT(JTSWMM(JT),K)	r	0.00- ∞	Depth (ft or m) for the rating table
QSWMMRT(JTSWMM(JT),K)	r	0.00- ∞	Discharge (cfs or cms) for the rating table

(i) = Integer variable (r) = real variable (c) = character

SDCLOGGING.DAT

A clogging factor was created to simulate the debris reduction of the inlet capacity. This option (INTYPE = 1,2,3,4 or 5 in the SWMMFLO.DAT) is assigned by the GDS in the inlet geometry dialog box (Figure 46.). The SDCLOGGING.DAT file contains the data with the following format:

ID	Grid Cell	Inlet ID	Clogging Factor - C_f (%)	Time for clogging -T_c (hr)
D	2694	I1	25	0.50
D	2409	I2	50	3.25

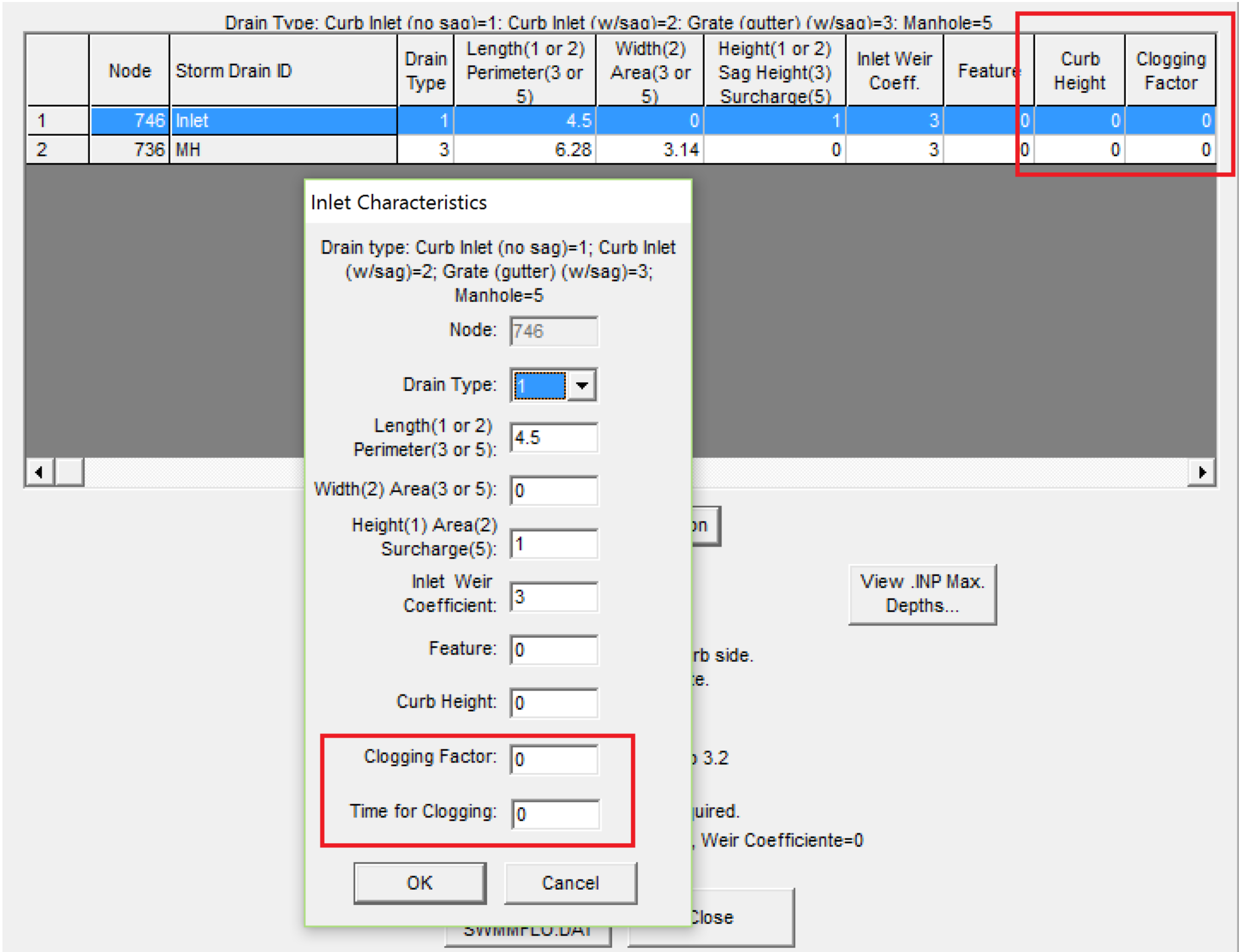


Figure 46. GDS Dialog Box for Entering clogging factor data.

SWMM.ini

After a project is saved in the storm drain GUI, the control settings file SWMM.ini will automatically be generated. This file has the same name as the project file and the extension *.ini. It contains global settings and model output options such as map display, legend colors and intervals, object default values, etc. To enable the display of results in the storm drain GUI the user must set the last two lines in Figure 47 as shown.

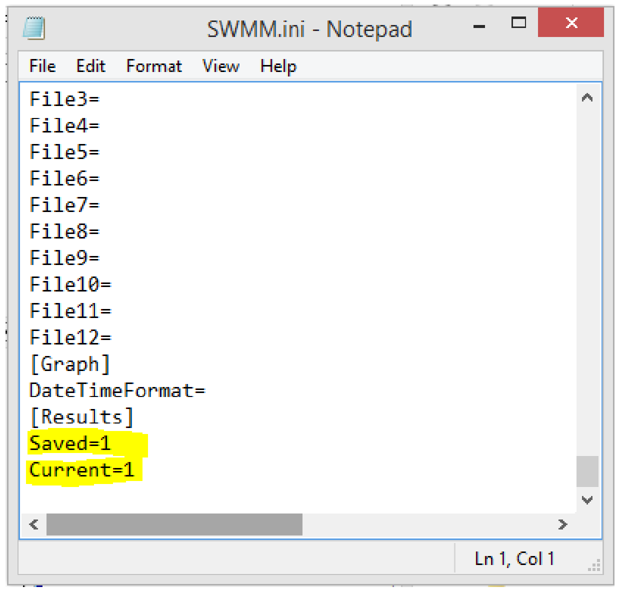


Figure 47. SWMM.ini file.

Output files

With the successful completion of a FLO-2D storm drain simulation, the storm drain output files listed in Table 14. List of the Storm Drain Data Output Files and Description .

Table 14. List of the Storm Drain Data Output Files and Description

Output File	Description
SWMM.RPT	Output report file generated by the storm drain model at the end of a simulation containing the pipe routing results for each output interval.
SWMM.OUT	Output file in a binary format that contains the storm drain results. This file is similar to the SWMM.RPT file and it is used by the SWMM GUI to interactively create time series plots and tables, profile plots, and statistical analyses of the results.
SWMMQIN.OUT	Reports the inflow and return flow discharge for each storm drain inlet. The reported discharges may be different from those in the SWMM.RPT file since the storm drain results include lateral pipe inflow and outflow.
SWMMOUTFIN.OUT	This file reports the outfall hydrographs for return flow to the surface water system. This file lists the grid or channel element in the first line followed by the time and discharge pairs.
FPRIMELEV.OUT	This output file reports the differences in elevation between the rim/invert elevation in the SWMM.inp file and the FLO-2D grid element elevation. This file should be reviewed to evaluate the elevations representing the inlet reference.
StormDrain_ERROR.CHK	This output file reports the storm drain error and warning messages. They were removed from the ERROR.CHK file.
SDManholePopUp.OUT	This output file reports the information for the manhole popping in the storm drain system. The information reported is: manhole name, time of occurrence, elevation comparison for the popping.
ManholePop.OUT	This output file reports the information for the manhole popping in the storm drain system in a table. The information reported is: X coordinate, Y coordinate, grid cell, manhole name, time of occurrence, pressure head, rim + surcharge depth, WSE.
FPLAIN_SDElev.RGH	Changes in elevation (FPRIMELEV.OUT) are reported to the FPLAIN_SDElev.RGH file. Replace the FPLAIN.DAT file to apply the changes to the next simulation
TOPO_SDElev.RGH	Changes in elevation (FPRIMELEV.OUT) are reported in the TOPO_SDElev.RGH. Replace the TOPO.DAT file to apply the changes to the next simulation.

SUMMARY.OUT	Volume conservation, run time, and final disposition of the volumes between the model components are reported in this file. The storm drain volume is reported as: 1. Storm Drain Inflow in the inflow volume that flows from surface water to storm drain system through inlets and outfalls (compare total w/SWMM.rpt Wet Weather Inflow): • SURFACE TO STORM DRAIN SYSTEM THROUGH INLETS: Volume that flows from surface water to storm drain network through the inlets. • SURFACE TO STORM DRAIN THROUGH OUTFALLS: Volume that flows from surface water to storm drain network through the outfalls as backwater. 2. Storm Drain Outflow from Outfalls is the volume that leaves the storm drain system through outfalls (compare total w/SWMM.rpt External Outflow): • STORM DRAIN TO SURFACE THROUGH OUTFALLS: Volume that flows from the storm drain to the surface water through the outfalls. Only those outfalls that were turned 'on' in the SWMMOUTF.DAT will discharge the volume back to the surface layer. • STORM DRAIN OUTFALL (OFF SYSTEM): Volume that flows off the storm drain through the outfalls. Only those outfalls that were turn 'off' in the SWMMOUTF.DAT will discharge the volume off the storm drain system. 3. Storm Drain Return Flow to Surface • TOTAL STORM DRAIN RETURN FLOW: Volume that returns to the surface water from the storm drain network. Return volume is possible when the storm drain pressure head is greater than both the RIM elevation and the FLO-2D water surface elevation in the inlet grid element. 4. STORM DRAIN SYSTEM STORAGE**:** volume in the pipe system at the end of the simulation. This volume is extracted from the swmm.RPT file. 5. STORM DRAIN MASS BALANCE: mass balance volume conservation calculated as: Total Inflow - Total Outflow - Total Storm Drain Return Flow - Storm Drain System Storage
SUMMARY.OUT	6. Extracted from Storm Drain File (swmm.rpt): • WET WEATHER INFLOW: Compare with Storm Drain Inflow. • EXTERNAL INFLOW: Other node inflows in addition to any surface runoff such as direct inflows as user-defined time series, dry weather inflows that are continuous inflows as base flows, and rainfall dependent infiltration/inflow (RDII). • EXTERNAL OUTFLOW: Compare with Storm Drain Outflow from Outfalls. • Total Storm Drain Storage (nodes + links): Sum of the storage. • Continuity Error (%)
CHVOLUME. OUT	This file reports the channel flow distribution including inflow, outflow, overbank flow, infiltration losses and volume conservation. This includes the volumes for channel outflow to the storm drain and channel inflow from the storm drain.
ERROR.CHK	This FLO-2D file contains input data error warnings and messages for the surface model. This file should be reviewed for messages after each simulation.

Chapter 4

Introduction

These guidelines will assist in understanding the FLO-2D surface water-storm drain interface.

- Surface water hydrology and hydraulics including the rainfall runoff, infiltration, and flood routing in channels, streets or unconfined overland flow are simulated by the FLO-2D surface water model.
- The storm drain engine only solves the pipe flow routing and hydraulics.
- A storm drain inlet must have a name starting with an 'I' to distinguish it from a junction and to collect water from the surface.
- The inlet locations are assigned in the SWMM.INP file and are read by the GDS to establish the connectivity between the inlet and the surface water grid system.
- Storm drain outfall nodes can be assigned to discharge pipe flow to the surface water system or off the models completely. Only the "Free" outfall boundary condition can discharged back to the surface water model.
- Return flow from inlets to the water surface depends on the relationship between the surface water elevation and the pipe pressure.

To get started on a new project, first develop the FLO-2D model by importing the digital terrain model (DTM) points, selecting a grid size, outlining the computational domain, assigning the grid element elevations and importing an aerial image. Other features of surface water flooding such as inflow-outflow nodes, buildings, levees and others can then be added either before or after the storm drain system model is created. The SWMM GUI (activated from the GDS command: Tools| Storm Drain| Run Storm Drain GUI…) or other storm drain GUI's can be used to assign the storm drain data base. The GDS or QGIS has several functions that integrate the FLO-2D surface water model with the storm drain model:

- Activates the SWMM storm drain GUI;
- Reads the data from the SWMM.inp file and associates inlets/outlets with FLO-2D cells.
- Displays a dialog box to enter the storm drain inlet geometry data.
- Saves and edits the SWMMFLO.DAT file containing the inlet geometry data.
- Displays the inlets and outlets and the piping network connections:
  - ✓ Inlets are identified as blue cells.
  - ✓ Junctions are green cells
  - ✓ The pipe system is represented as a magenta polyline.
- Displays a dialog box containing the outfall list from the SWMM.inp.
- Edits the switch that turns on the discharge from an outfall to the FLO-2D grid elements.
- Saves the SWMMOUTF.DAT file.

- Displays warning messages.
- Graphically displays results including inflow and return flow hydrographs and the pipeline energy grade line.

When the SWMM GUI is called from the GDS, the user can open an existing storm drain model. When the SWMM GUI is closed, the SWMM.inp data file is automatically saved. The SWMMFLO.DAT file is then automatically generated by GDS using the SWMM.inp data to locate the inlet positions and associate them with a grid element.

To summarize, a FLO-2D storm drain system model is generated through the following steps:

1. Create or open an existing FLO-2D model project;
2. Open the SWMM GUI from the GDS and develop the SWMM.inp data file;
3. Close the SWMM GUI and view the storm drain network in the GDS;
4. Generate the SWMMFLO.DAT file with the storm drain inlet data;
5. Create the SWMMOUTF.DAT using the Outfall Node Discharge Option;
6. Run the integrated FLO-2D Storm Drain System model.

After the FLO-2D grid system is prepared, the storm drain model switch (CONT.DAT file) must be activated to run the model in the GDS. For further guidance in building a storm drain model, the SWMM model GUI options and tools, and the EPA SWMM user manual can be reviewed. The specific storm drain instructions, guidelines and comments are followed by suggested initial parameters and a checklist for entering and reviewing the data.

Detailed Storm Drain Instructions and Guidelines

Virtually all storm drain models are developed for urban areas. Specific instructions for creating a storm drain in a FLO-2D urban environment are:

- A storm drain inlet/outfall should not be assigned to a completely blocked cell (ARF = 1); A reasonable amount of surface area should be available for the storm drain feature to interact with the surface water.
- Higher roughness values can be assigned to floodplain elements with storm drain features (inlet or outfalls) to represent the storm drain flow disturbance around the inlet or outfall. This would help surface routing numerical stability.
- Inlet rim elevations should match the floodplain elevation. It is possible to assign inlet rim elevations higher than the floodplain cell elevation, but this may lead to oscillating or poor results. For an inlet rating table, the inlet discharge is zero if the floodplain water surface elevation is less than the inlet rim elevation.
- Levees and walls may isolate a storm drain feature in the model resulting in oscillating storm drain discharge. A review of the inlet/outfall location may be required.

- Storm drain features should not be assigned to surface water inflow or outflow elements. There is an error message generated for this conflict.

Other specific urban guidelines are listed in Table 15.

Table 15. FLO-2D Storm Drain Instructional Comments.

<table>
<tr><th></th><th>Guidelines and Instructional Comments</th></tr>
<tr><td rowspan="2">WSE is greater than pipe pressure head</td><td>Inlet discharge is computed by FLO-2D and exchanged with the storm drain system. FLO-2D will use the grid element water depth (floodplain, channel or street) and the inlet geometry to calculate the inlet discharge.
Horizontal inlets refer to gutter inlets and vertical inlets are culvert or pipe openings to the surface. Floodplain grid element elevation (FPE) is automatically set to the inlet rim elevation. For horizontal inlets changes to FPE are reported to a file named as FPRIMELEV.OUT.
For vertical inlets:
An inlet on a 1-D channel (end of segment) where the channel flow is discharging to the storm drain pipe, the invert elevation should be equal to the channel bed elevation.
If the flow is discharging to a storm drain pipe or culvert in a floodplain swale, the invert elevation should be equal to the cell floodplain elevation.
The last column parameter ‘Feature’ in the GDS inlet dialog has three options:
0 - default, no flapgate, no vertical inlet opening
1 - vertical, inlet opening
2 - flapgate (outlet)
If option 1 is assigned for a vertical inlet opening then there are two cases:
The channel pipe inlet invert elevation has to be reset to the channel bed elevation. Automated runtime changes for this case do not occur. The user has to manually implement the elevation revision.
Grid element elevation is reset to the pipe invert elevation at runtime.
The corrected FPE is not revised in the FPLAIN.DAT file. The user must review the FPRIMELEV.OUT modifications and rename FPLAIN_SDElev.RGH and the TOPO_SDElev.RGH files to FPLAIN.DAT AND TOPO.DAT files respectively to make the elevation changes permanent. Rim elevations for the inlets located in channel/street cells must be verified and manually revised by the user.</td></tr>
<tr><td>The discharge and volume that enters the storm drain is based on the inlet geometry and on the relationship between the water surface elevation and the storm drain pressure head. Storm drain inflow discharge is inlet controlled until the system capacity is reached.</td></tr>
</table>

	Inlet conditions: • Curb opening inlet at grade (no sag) INTYPE=1; • Curb opening inlet with sag INTYPE= 2; • Grate (gutter) inlet with/without sag INTYPE=3; • Unique inlet with stage - discharge rating table INTYPE=4; • Manhole INTYPE=5. Weir/orifice equations are used to calculate the discharge for inlets 1 thru 3 and 5. For the rating table option (INTYPE = 4), a relationship between cell flow depth and discharge is assigned in the GDS. An additional file (SWMMFLORT.DAT) is created by the GDS for these type of inlets.
Storm drain pressure is greater than the WSEL	Surcharging or return flow is computed from the storm drain to the surface water.
	Not all return flow or flooding reported in the SWMM.RPT file passes from the storm drain to the surface water since the pressure head has to be greater than the WSEL.
	Return flow volume is distributed over the grid element surface area as an increased incremental flow depth added to the existing cell depth.
	Inflow to storm drain from the surface water is not allowed in this case.
WSE is greater than the pressure head and the rim elevation	No return flow from the storm drain to the surface water is computed.
	Volume in the inlet node stays in the pipe and the overflow volume is set to 0.
	Inflow to the storm drain from surface water is not computed.
Volume conservation	Inflow to the storm drain and return volumes (flooding and outfall volumes) to the surface water are compiled and reported by the FLO-2D model.
Reporting results	The FLO-2D file SWMMQIN.OUT reports the inflow and return flow discharge for each inlet. This is different from the discharge values reported in the SWMM.RPT file which includes lateral pipe inflow and outflow. SWMMOUTFIN.OUT file lists time and discharge pairs for storm drain outfall hydrographs when they discharge back to the surface water.
Detention basin outlet	Flapgates can be used to stop flow from going into the storm drain system. Flow only goes out of the outlet. The SWMMFLO.DAT file assigns a switch (FEATURE) that can have one of the following values: 0 = default, no flap gate, no vertical inlet opening 1 = vertical inlet opening 2 = flapgate, controls outlet node discharge
Free Outfalls	Any type of SWMM outfall can be assigned. To discharge back to the FLO-2D surface water, the user has to set the outfall to type ‘free’. Discharge is based on the surface water elevation and storm drain pressure head. Outfall discharge will occur if WSE is greater than the pressure head. If the pressure head > WSEL: Outfall discharges to the FLO-2D grid cell. If the pressure head < WSEL: There is no outfall discharge but depth is equal to WSEL. Flow into the outfall depends on flap gate assignment and the WSEL. This is available only for the ‘free’ type of outfalls. It does not apply to normal, fixed, tidal or time series type of outfalls.

<table>
<tr><td></td><td>When the outfall does not discharge to the surface water, the outfall head is assigned based on the type of outfall node. The following types can be set up:
a. FREE: minimum between normal and critical depth.
b. NORMAL: normal depth.
c. FIXED: fixed stage entered in the data.
d. TIDAL: head computed from tide stage curve.
e. TIME SERIES: head computed from the time series.</td></tr>
<tr><td>Manholes</td><td>Popping a manhole cover can be simulated. The surcharge depth is entered in the SWMMFLO.DAT file. The user can define the surcharge depth in the junction properties (SWMM.inp file). When the surcharge depth is set to different values in SWMMFLO.DAT and in the SWMM.inp file, the model uses the surcharge depth from the SWMMFLO.DAT.
• If Pressure Head + Surcharge Depth < WSEL:
✓ Cover remains in place.
✓ Inflow to the manhole is not allowed.
✓ Return flow will not occur.
• If Pressure Head + Surcharge Depth > WSEL:
✓ Cover is popped.
✓ Surcharge depth is reset to 0.
✓ Inflow to the storm drain is permitted.
✓ Return flow can occur.</td></tr>
</table>

Default Parameters

The following storm drain parameters are automatically assigned in the SWMM.inp file:

- Runoff Wet Weather (WET_STEP) and Runoff Dry Weather (DRY_STEP) timesteps equal 00:01:00 min. These watershed routing parameters are not directly used by the storm drain model but can affect the storm drain routing if they are less than the ROUTING_STEP. This avoids an inappropriate assignment of these parameters.
- Lengthening Step (LENGTHENING_STEP) is set as the routing timestep (ROUTING_STEP) divided by 6. This option increases the length of the shorter conduits based on the Courant-Friederick-Levy (CFL) stability criteria. The storm drain model bases the new equivalent pipe length on an estimate of the full flow velocity in the conduit as well as on the wave celerity. The full area, width and hydraulic radius are unchanged in the modified link but the length, slope and roughness are altered.

Length Factor = (Wave Celerity + Full Depth Velocity) * Timestep/ Pipe Length

For a Length Factor > 1:

New Roughness= Old Roughness /Length Factor**0.5

New Slope = Old Slope Slope/Length Factor**0.5

- The routing model (FLOW_ROUTING) for the FLO-2D storm drain component is set as dynamic wave (DYNWAVE) to account for backwater effects, entrance/exit losses, flow reversal or pressurized flow.
- The inertial term (INERTIAL_DAMPING) is set to PARTIAL or dampen (Partial=1). The inertial terms in the Saint Venant Equation are reduced for critical flow and ignored for supercritical flow.
- Report Control Actions and Report Input Summary ([Report] CONTROLS and INPUT) are automatically turned on.
- "Start Reporting on" variables (START_DATE and START_TIME) in the SWMM GUI is automatically set as Start Analysis on. Both the FLO-2D and the storm drain model will start reporting results at the beginning of the simulation.
- "End Analysis on" variables (END_DATE and END_TIME) in the SWMM GUI are automatically set based on the simulation time entered in FLO-2D. The Storm Drain results (*.RPT,*.OUT) are automatically saved when the model simulation runs until completion. If the model is not completed, storm drain results are not written.

Initial Parameters

When building the storm drain model, the following data assignments in the SWMM.inp file are recommended:

1. It is recommended to keep the names of the various storm drain components simple, short and uniform such as I1, I2, I3… for inlets. The number of characters should be less than 25. Use O = outlets and C = pipe conduits. This will simplify the GDS graphics display of the storm drain components and make it easier to differentiate between inlets, manholes and outlets. ID Prefixes can be assigned in the SWMM GUI from Project>>Defaults on the ID Labels page of the dialog. The GUI will automatically label new objects with consecutive numbers following the specified prefixes.
2. The END_TIME for the model duration is automatically assigned so that the END_TIME minus the START_TIME is equal to the simulation time SIMUL in the FLO-2D model CONT.DAT file.
3. The ROUTING_STEP is automatically assigned as equal to the FLO-2D Timestep. The FLO-2D Timestep is used as the Routing_STEP for all conditions including a VARIABLE_STEP equal or different to zero.

Note: The WET_STEP and DRY_STEP values are hardcoded to 1 minute.

4. The REPORT_START_DATE and REPORT_START_TIME are automatically assigned as the start date and time. The storm drain component reporting time (REPORT_STEP) in SWMM.inp file is recommended to be set up as equal to the FLO-2D output interval (TOUT) in the CONT.DAT file. REPORT_STEP is a mixture of hours, minutes and seconds with the format 00:00:00. The unit for TOUT is hours.

Integration of a Storm Drain Network into a Complex Urban Model

Integrating a storm drain network into an urban model requires an understanding of how the storm drains will interact with the surface flow. In the FLO-2D model, the most complex interaction occurs between the storm drain features and the 1-D channel component. The storm drain system interfaces with channels through inlets and outfalls. An early identification of storm drain database deficiencies can save time and effort on a project.

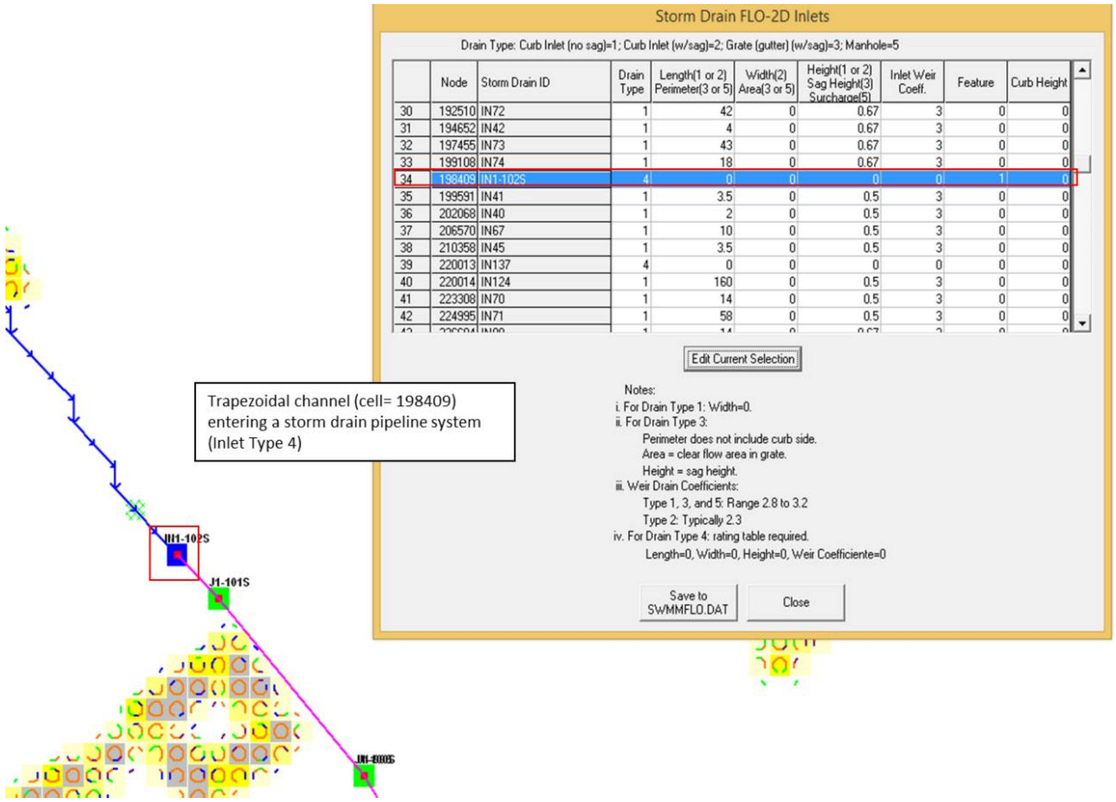
Storm Drain Inlet – Channel Considerations

After the urban project is prepared, the following questions related to the storm drain inlet to channel system can be addressed:

- Inlet locations:
    - ✓ Are all inlets correctly assigned inside the FLO-2D computational domain?
    - ✓ Are there inlets incorrectly assigned to the interior channel elements?
    - ✓ Are there inlets incorrectly assigned to the right channel bank elements?
- Inlet elevations:
    - ✓ Are there channels discharging to a storm drain inlet?
    - ✓ Does the channel thalweg elevation match the inlet invert/RIM elevation?
    - ✓ Is the inlet set up as a vertical inlet opening in the SWMMFLO.DAT?

Storm drain inlets should not be set up in an interior channel element. If the channel discharges directly to a storm drain pipe via a culvert, assign the inlet to the channel left bank element. For this configuration a vertical Type 4 inlet can be assign (See the Storm Drain Manual for details). Figure 48 and Figure 49. Elevation of a Trapezoidal 1-D Channel Discharging to a Storm Drain Inlet provide some additional details about setting up the inlet/channel interface. This system shows that the bed elevation of the channel is equal to the invert elevation of the inlet.

- Channel Bed Elevation = Bank Elevation – Depth = (299.8 – 3.5) = 296.3 ft
- Inlet Elevation = 296.3 ft



Storm Drain FLO-2D Inlets

Drain Type: Curb Inlet (no sag)=1; Curb Inlet (w/sag)=2; Grate (gutter) (w/sag)=3; Manhole=5

	Node	Storm Drain ID	Drain Type	Length(1 or 2) Perimeter(3 or 5)	Width(2) Area(3 or 5)	Height(1 or 2) Sag Height(3) Surcharge(5)	Inlet Weir Coeff.	Feature	Curb Height
30	192510	IN72	1	42	0	0.67	3	0	0
31	194652	IN42	1	4	0	0.67	3	0	0
32	197455	IN73	1	43	0	0.67	3	0	0
33	199108	IN74	1	18	0	0.67	3	0	0
34	198409	IN1-102S	4	0	0	0	0	1	0
35	199591	IN41	1	3.5	0	0.5	3	0	0
36	202068	IN40	1	2	0	0.5	3	0	0
37	206570	IN67	1	10	0	0.5	3	0	0
38	210358	IN45	1	3.5	0	0.5	3	0	0
39	220013	IN137	4	0	0	0	0	0	0
40	220014	IN124	1	160	0	0.5	3	0	0
41	223308	IN70	1	14	0	0.5	3	0	0
42	224995	IN71	1	58	0	0.5	3	0	0

Figure 48. Trapezoidal 1-D Channel Discharging to a Storm Drain Inlet

CHAN.DAT FILE

T	191815	304.22	304.22	0.021	5.00	3.50	30.40	1.50	1.50
T	192637	303.67	303.67	0.022	5.00	3.50	30.40	1.50	1.50
T	193459	303.12	303.12	0.020	5.00	3.50	30.40	1.50	1.50
T	194283	302.56	302.56	0.023	5.00	3.50	30.40	1.50	1.50
T	195107	302.01	302.01	0.020	5.00	3.50	30.40	1.50	1.50
T	195931	301.46	301.46	0.020	5.00	3.50	30.40	1.50	1.50
T	196756	300.91	300.91	0.020	5.00	3.50	30.40	1.50	1.50
T	197582	300.35	300.35	0.020	5.00	3.50	30.40	1.50	1.50
T	198409	299.80	299.80	0.020	5.00	3.50	30.40	1.50	1.50

Thalweg Channel Depth= 3.50 ft

Grid Elevation (198409)= 299.80 ft

SWMM.INP FILE

[JUNCTIONS]

;;Name	Invert Elev.	Max. Depth	Init. Depth	Surcharge Depth	Ponded Area
IN1	287.82	11.60	0	0.5	0
J10	275.92	7.49	0	0.5	0
IN100	307.04	6.13	0	0	0
IN101	306.8	6.27	0	0	0
IN10-1S	188.81	2.63	0	0	0
J102	306.49	6.19	0	0	0
IN103	274.06	4.46	0	0	0
J104	279.23	4.58	0	0	0
J105	295.68	6.59	0	0	0
J106	270.83	4.25	0	0	0
J107	238.96	8.72	0	0	0
J108	237.12	9.49	0	0	0
IN11	273.22	6.62	0	0	0
J110	272.93	4.56	0	0	0
J1-100S	285.46	9.28	0	0.5	0
J1-101S	291.78	11.92	0	0	0
IN1-102S	296.3	0	0	0	0
J111	263.47	4.87	0	0	0

Figure 49. Elevation of a Trapezoidal 1-D Channel Discharging to a Storm Drain Inlet

Figure 50 shows a storm drain system interfacing with a channel system. The surface water and the storm drain exchange flow is based on the water surface elevation and the pressure head which are a function of the following:

- Channel bed elevation = inlet invert elevation
- Channel bank elevation = inlet rim elevation

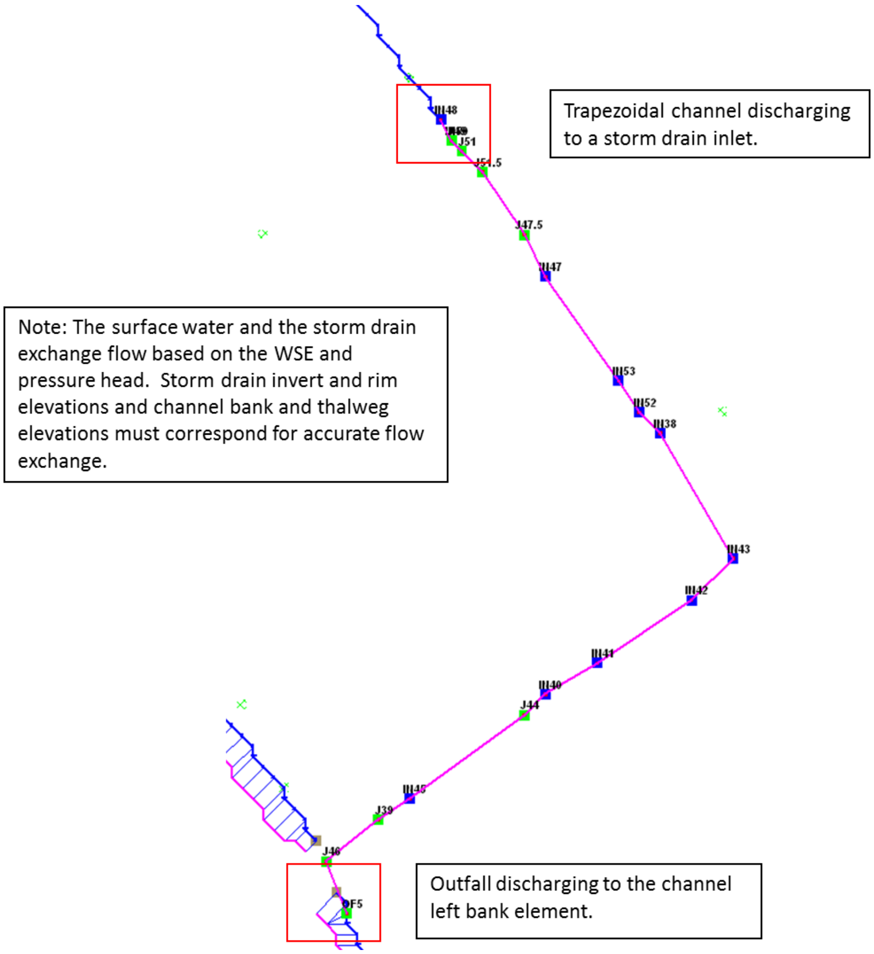


Figure 50. Complex Interaction between a Storm Drain Pipe and 1-D Channel

To connect a storm drain outfall to a channel element the following issues should be addressed:

- ✓ Are outfalls set up as a 'FREE' condition type?
- ✓ Is the switch to discharge flow back to the surface 'ON' in the SWMMOUTF.DAT file?

Figure 51 shows a complex storm drain – channel system where a channel feeds the storm drain as an inlet and flow returns to surface channel downstream.

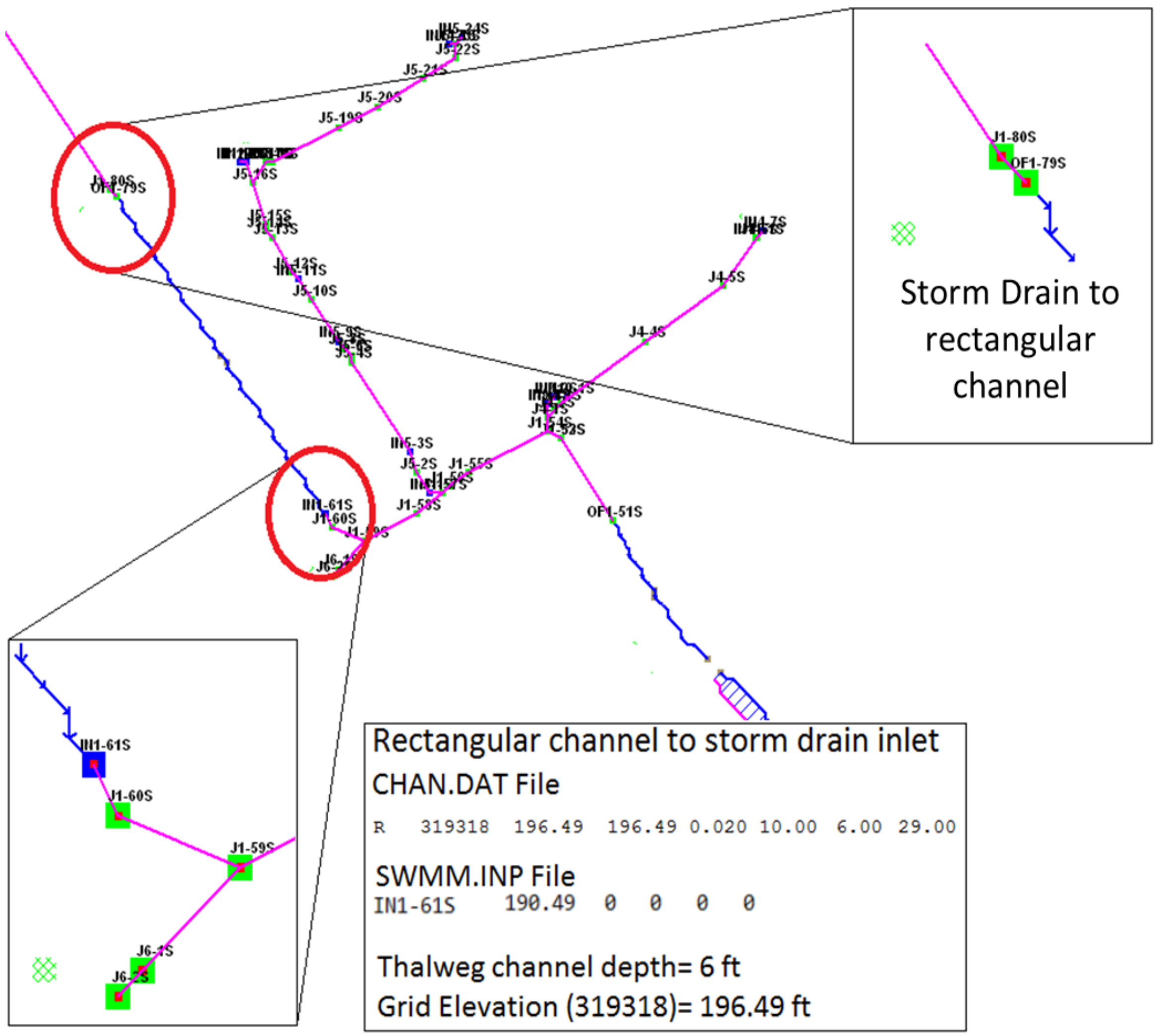


Figure 51. Complex Flow Exchange between a Storm Drain System and 1-D Channel

Storm drain outfalls are assigned to the channel left bank element. For most cases, the outfall invert elevation would be assigned to the channel element thalweg elevation. If the coordinates in the SWMM.inp file are the left bank element channel coordinates then the GDS will automatically assign the outfall node to the left bank element. The outfall should be correctly paired to the left bank element in the SWMMOUTF.DAT (Figure 52).

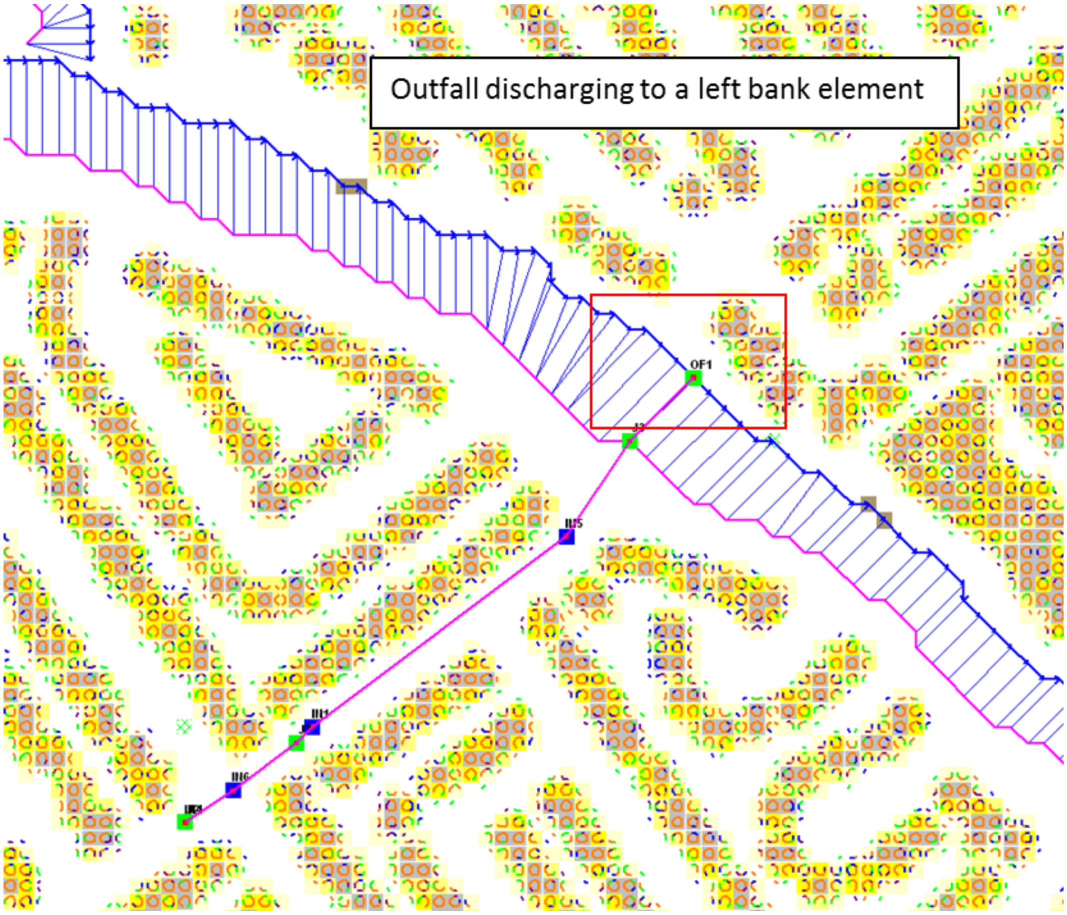


Figure 52. Typical Configuration of a Storm Drain Outfall Discharging to a Natural Channel

The GDS pairs the coordinates of the different storm drain components from the SWMM.inp file to the grid elements in the surface layer. It is not necessary that the outfall coordinates in the SWMM.inp match the left bank channel element coordinate. The position is correlated in the SWMMOUTF.DAT updating the grid element number to the left bank element number (Figure 53.). The outfall coordinates in the SWMM.inp file do not have to be replaced since the storm drain discharge calculations will not be affected.

The bank elements in FLO-2D act as both floodplain and channel elements in order to facilitate the channel to floodplain exchange. The outfall should not be assigned to the left bank floodplain element. If the outfall physically discharges to the floodplain elevation instead of the channel bed elevation, assign the outfall position to a contiguous element that is not a channel bank element. Assignment of the outfall to a right bank element, or a channel interior element will generate an error message.

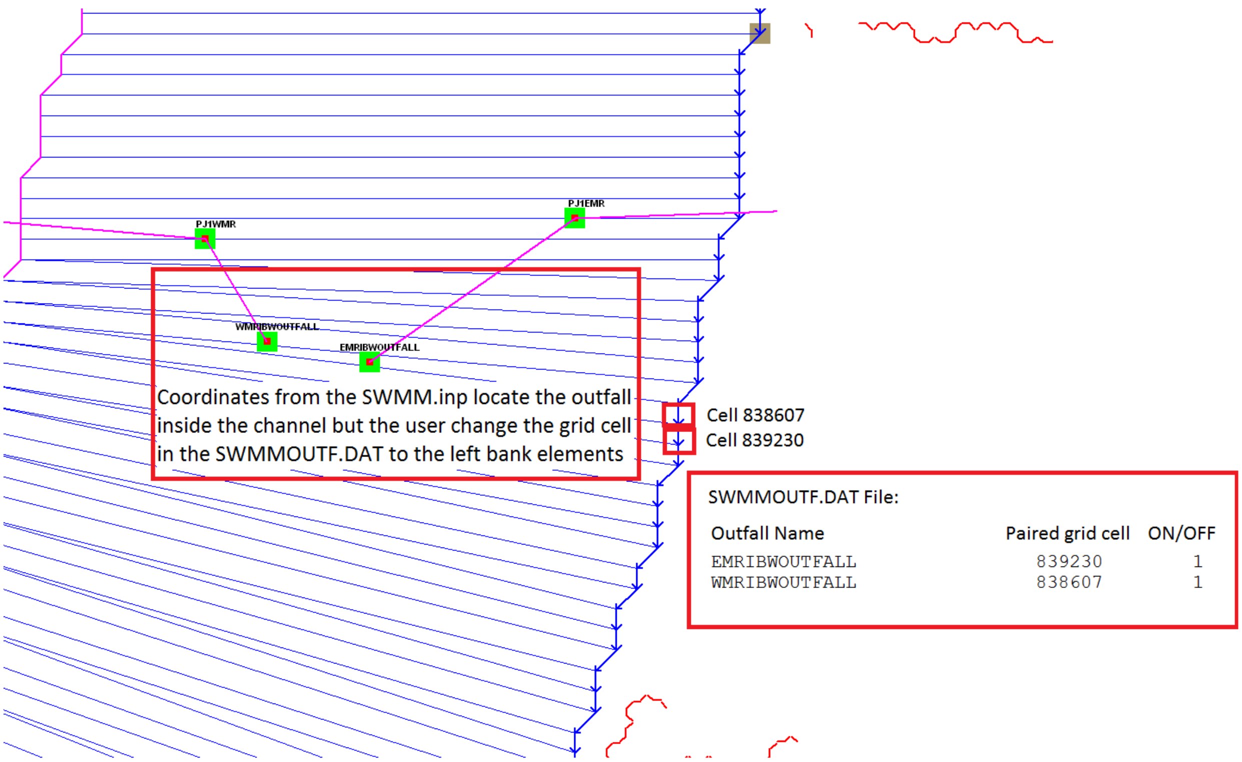


Figure 53. Outfall Nodes Paired to Interior Channel Elements by GDS

The outfall invert elevation can be less than the channel thalweg elevations (underground), and the storm drain would be assumed to be underwater with an initial tailwater depth. The pipe conduit should have a positive slope to the outfall. This configuration may represent the case for a ponded surface water condition that is assigned as a ground elevation because the ponded water will not contribute to downstream flooding. If the outfall invert is underground (underwater), an artificial head equal to the ground elevation is assigned to the outfall node for the entire simulation. This artificial head may fill the pipe, but the volume that goes into the pipe is not considered in the FLO-2D volume conservation accounting because the grid element is dry. The artificial volume is accounted for in the storm drain model. When the model runs, inflow may be added to either the outfall grid element or the upstream storm drain pipe network and the flow can go either in or out of the outfall pipe based on the pressure head (Figure 54). To account for volume conservation, the storm drain outflow that represents inflow volume to a FLO-2D channel is reported in the CHVOLUME.OUT file.

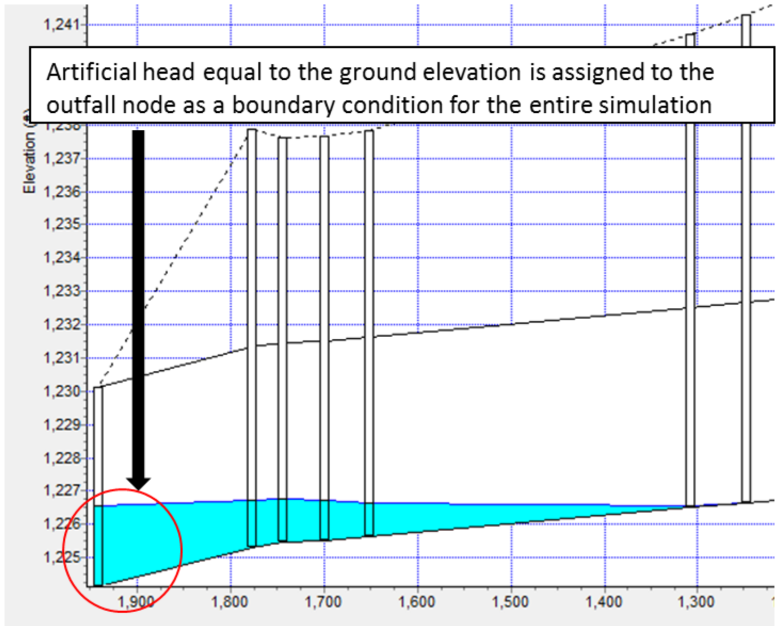


Figure 54. Underground Outfall Condition

Water will flow in or out of the outfall pipe based on the relationship between the water surface elevation and pipe pressure head. Water can enter the storm drain when the water surface elevation is greater than the invert, but it can evacuate from the storm drain if the invert is above the water surface elevation. This behavior can introduce oscillations in the system that can be explained as a respond to the surface water and storm drain pressure interaction (Figure 55).

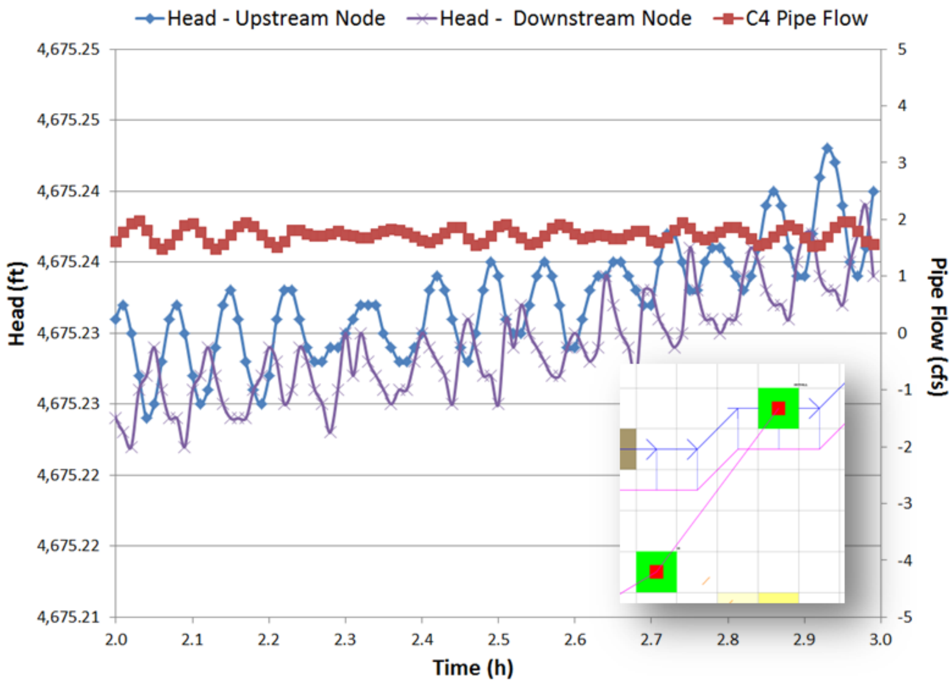


Figure 55. Inlet and Outfall Pressure Head Variation Cause Pipe Discharge Oscillations

Model Checklist

The following checklist was prepared to review the storm drain data files:

- ☐ Inlet location. All inlets must be inside the FLO-2D computation domain.
- ☐ SWMM hydrology components for surface water modeling are eliminated. If an existing SWMM model was ported to the FLO-2D model, all of the hydrology features have to be removed including subcatchments and rain gage features.
- ☐ Pipe lengths. If pipe lengths are less than 20ft or less than the FLO-2D grid element side length, adjust short pipe length or delete that section of pipe.
- ☐ Adverse pipe slope. Check adverse pipe slope for accuracy.
- ☐ Inlet elevations. Grid elevations or channel thalweg elevations should match the inlet rim elevations or invert elevations.
- ☐ Type 4 - Defined by a Stage-Discharge Rating Table. Rating tables must be assigned in the SWMMFLORT.DAT file.
- ☐ Inlet geometry. Check the size and location.
- ☐ Outfall discharge. For outfall discharge to the surface water, the outfall must have a ‘free’ condition and must be turned ‘on’ in the SWMMOUTF.DAT file.

- Outfall location. Check the outfall location with respect to the grid element and check the invert elevation with respect to floodplain elevation.
- Outfall to the channel. The outfall has to be connected to the left bank grid element.
- Flapgates for outfall nodes. They have to be assigned in the SWMM.inp file.
- Manholes. Check the assigned surcharge depth.
- Storm drain model simulation completion. Check the SUMMARY.OUT file last line for the time stamp indicating that the model properly terminated.
- SWMM.ini file. The SWMM.ini file should be modified to review the results.

Chapter 5

Troubleshooting

Introduction

When a FLO-2D storm drain model ends prematurely, it is probable that an error statement is written to the ERROR.CHK file as well as to the report file (SWMM.RPT). Even for successful simulations, the ERROR.CHK should be reviewed for warning messages. All the potential errors that may be encountered cannot be anticipated, but some suggestions to reduce the conflicts and have a better understanding of how to improve the storm drain model are presented:

- Number of inlets (SWMMFLO.DAT) should be equal to the number of junctions with an ID that starts with "I" (SWMM.inp).
- The SWMMOUTF.DAT file should be created through the GDS if the storm drain system has outfalls defined. If the number of outfalls is modified, the user must recreate the SWMMOUTF.DAT from the GDS.
- Outfall to a channel must be assigned to the left bank channel element.
- The SWMMFLORT.DAT file needs to be created if Type 4 inlet rating tables were assigned.
- The inlet rim elevation has to be equal to the FLO-2D grid element floodplain elevation. Cell elevation is adjusted to the rim elevation at runtime and a warning message is generated. The user has to make the adjustment permanent in FPLAIN.DAT and TOPO.DAT.
- The path names of FLO-2D storm drain files are recommended to be less than 254 characters. Paths names approaching that number of characters may cause the storm drain model to crash. The filenames and paths are defined as character pointers in the storm drain model so there is no text length issue but there are several opening and write statements in the output files that include format specifiers that could trigger this problem. Run the model in a folder with a short path and a simplified name.

FLO-2D Error Messages

Error/warning messages are listed in the ERROR.CHK file. The most important messages are listed in Table 16.

Table 16. FLO-2D Error Messages

Issue	Message in ERROR.CHK
Channel bed elevation.	THERE ARE STORM DRAIN INLETS ON CHANNEL GRID ELEMENTS. THE CHANNEL BED ELEVATION MIGHT BE DIFFERENT THAN THE INVERT ELEVATION. NO ACTION WAS TAKEN DURING THE SIMULATION. PLEASE REVIEW AND REVISE IF NECESSARY.
Elevations for outfall nodes.	THE STORM DRAIN OUTFALL INVERT ELEVATION SHOULD BE EQUAL TO OR GREATER THAN THE FLOODPLAIN, CHANNEL, STREET ELEVATION. NO ACTION IS TAKEN. PLEASE REVIEW AND REVISE IF NECESSARY.
Outfall node in channel interior element.	THE FOLLOWING STORM DRAIN OUTFALL NODES ARE IN CHANNEL INTERIOR ELEMENTS, RE-ASSIGN TO THE CHANNEL ELEMENTS IN CHAN.DAT.
Outfall node assigned to a FLO-2D outflow element.	THERE IS AN OUTFLOW NODE AND A STORM DRAIN OUTFALL ASSIGNED TO THE SAME GRID CELL.
Grid element floodplain has to be revised.	THE GRID ELEMENT FLOODPLAIN WAS REVISED DURING THE SIMULATION TO THE STORM DRAIN INLET RIM ELEVATIONS FOR THE FOLLOWING GRID ELEMENTS (PLEASE REVIEW AND REVISE FPRIMELEV.OUT FILE IF NECESSARY).
Horizontal type 4 storm drain inlet elevations.	THE HORIZONTAL TYPE 4 STORM DRAIN INLET ELEVATIONS WERE REVISED DURING THE SIMULATION TO THE STORM DRAIN INLET RIM ELEVATIONS FOR THE FOLLOWING GRID ELEMENTS (PLEASE REVIEW AND REVISE FPRIMELEV.OUT FILE IF NECESSARY).
Type 4 inlet is a vertical inlet and it is in a floodplain cell.	THE GRID ELEMENT ELEVATIONS IN "FLOODPLAIN SWALES" WERE REVISED DURING THE SIMULATION TO THE TYPE 4 VERTICAL INLET INVERT ELEVATIONS. PLEASE REVIEW FPRIMELEV.OUT FILE. *NOTE: If a floodplain swale is discharging into a storm drain pipe or culvert, the invert elevation should be equal to the swale bed elevation.*

Type 4 inlet is a vertical inlet and it is in a channel cell.	THERE ARE VERTICAL TYPE 4 INLETS ASSIGNED TO CHANNEL ELEMENTS AND THE CHANNEL BED ELEVATION IS DIFFERENT THAN THE INVERT ELEVATION. NO ACTION WAS TAKEN DURING THE SIMULATION. PLEASE REVIEW AND REVISE IF NECESSARY. *NOTE: If an inlet is assigned to the end of a 1-D channel segment and the channel flow discharges into the storm drain, the invert elevation should equal to the channel bed elevation.*
No elevation differences between surface and storm drain layers.	NOTE: THERE ARE NO DIFFERENCES BETWEEN FLOODPLAIN GRID AND STORM DRAIN RIM ELEVATIONS. *NOTE: THERE ARE NO DIFFERENCES BETWEEN FLOODPLAIN GRID AND TYPE 4 INVERT INLET ELEVATIONS. FPRIMELEV.OUT FILE WAS NOT CREATED.*
Elevations are revised.	THE GRID ELEMENT FLOODPLAIN OR STREET ELEVATIONS WERE REVISED DURING THE SIMULATION TO THE STORM DRAIN INLET RIM ELEVATIONS FOR THE FOLLOWING GRID ELEMENTS (PLEASE REVIEW THE FPRIMELEV.OUT FILE)
More than one storm drain inlet is assigned to one grid element. Simulation does not start.	THERE ARE POTENTIAL DATA ERROR(S) IN FILE SWMM.inp AND SWMMFLO.DAT. MULTIPLE INLETS ASSIGNED TO ONE GRID CELL
Multiple cells are assigned to one inlet. Simulation does not start.	MULTIPLE CELLS ASSIGNED TO ONE INLET
Missing storm drain inlet geometry or inappropriate geometry, simulation does not start.	THERE ARE A MISSING OR INAPPROPIATE STORM DRAIN INLET GEOMETRY IN FILE: SWMMFLO.DAT. REVIEW STORM DRAIN INLET: XX, DRAIN TYPE: IX ON GRID CELL: XXX
Missed or inappropriate geometry.	'LENGTH MUST BE GREATER THAN ZERO' 'WIDTH OR HEIGHT MUST BE GREATER THAN ZERO' 'PERIMETER MUST BE GREATER THAN ZERO' 'TYPICAL WEIR COEFFICIENT: 2.3' 'TYPICAL WEIR COEFFICIENT RANGE: 2.8 - 3.2'
General Error in SWMM Model.	COMPUTATIONAL ERROR IN SWMM MODEL: REVIEW *.RPT FILE FOR THE ERROR DESCRIPTION

General Error in SWMM Model reported to the ERROR.CHK file.	THERE IS AN ERROR IN THE SWMM MODEL. PLEASE REVIEW THE FOLLOWING ERROR CODE IN THE SWMM ERROR LIST OR CONTACT THE FLO-2D TEAM FOR SUPPORT Error Code: XXX. (See Table 17 below.)

SWMM error messages

Table 17 lists the errors reported by SWMM (Rossman, 2005) to the SWMM.RPT file as well as on the ERROR.CHK file. Some of these errors are not relevant to a FLO-2D storm drain model.

Table 17. SWMM Error Numbers.

Storm Drain Engine Error Number	Description of the error
101	Memory allocation error
103	Cannot solve KW equations for Link
105	Cannot open ODE solver
107	Cannot compute a valid time step
108	Ambiguous outlet ID name for subcatchment
109	Invalid parameter values for aquifer
110	Ground elevation is below water table for Subcatchment
111	Invalid length for conduit
112	Elevation drop exceeds length for conduit
113	Invalid roughness for conduit
114	Invalid number of barrels for conduit
115	Adverse slope for conduit
117	No cross section defined for link

119	Invalid cross section for link
121	Missing or invalid pump curve assigned to pump
131	The following links form cyclic loops in the drainage system
133	Node xxx has more than one outlet link
134	Node has illegal DUMMY link connections
135	Divider xxx does not have two outlet links
136	Divider xxx has invalid diversion link
137	Weir Divider xxx has invalid parameters
138	Node xxx has initial depth greater than maximum depth
139	Regulator xxx is the outlet of a non-storage node
141	Outfall xxx has more than 1 inlet link or an outlet link
143	Regulator xxx has invalid cross-section shape
145	Drainage system has no acceptable outlet nodes
151	A Unit Hydrograph in set xxx has invalid time base
153	A Unit Hydrograph in set xxx has invalid response ratios
155	Invalid sewer area for RDII at node
156	Inconsistent data file name for rain gage
157	Inconsistent rainfall format for rain gage

158	Time series for rain gage xxx is used by another object
159	Recording interval > time series interval for Rain Gage
161	Cyclic dependency in treatment functions at node
171	Curve xxx has invalid or out of sequence data
173	Time series xxx has its data out of sequence
181	Invalid snow melt climatology parameters
182	Invalid parameters for snow pack
183	No type specified for LID
184	Missing layer for LID
185	Invalid parameter value for LID
186	Invalid parameter value for LID placed in subcatchment
187	LID area exceeds total area for subcatchment
188	LID capture area exceeds impervious area for subcatchment
191	Simulation start date comes after ending date
193	Report start date comes after ending date
195	Reporting time step or duration is less than routing time step
200	One or more errors in input file
201	Too many characters in input line

203	Too few items
205	Invalid keyword xxx
207	Duplicate ID name xxx
209	Undefined object xxx
211	Invalid number xxx
213	Invalid date/time xxx
217	Control rule clause out of sequence
219	Data provided for unidentified transect
221	Transect station out of sequence
223	Transect xxx has too few stations
225	Transect xxx has too many stations
227	Transect xxx has no Manning's n
229	Transect xxx has invalid overbank locations
231	Transect xxx has no depth
233	Invalid treatment function expression
301	Files share same names
303	Cannot open input file
305	Cannot open report file

307	Cannot open binary results file
309	Error writing to binary results file
311	Error reading from binary results file
313	Cannot open scratch rainfall interface file
315	Cannot open rainfall interface file
317	Cannot open rainfall data file
318	Date out of sequence in rainfall data file
319	Invalid format for rainfall interface file
321	No data in rainfall interface file for gage
323	Cannot open runoff interface file
325	Incompatible data found in runoff interface file
327	Attempting to read beyond end of runoff interface file
329	Error in reading from runoff interface file
330	Hotstart interface files have same names
331	Cannot open hotstart interface file
333	Incompatible data found in hotstart interface file
335	Error in reading from hotstart interface file
336	No climate file specified for evaporation and/or wind speed

337	Cannot open climate file
338	Error in reading from climate file
339	Attempt to read beyond end of climate file
341	Cannot open scratch RDII interface file
343	Cannot open RDII interface file
345	Invalid format for RDII interface file
351	Cannot open routing interface file
353	Invalid format for routing interface file
355	Mismatched names in routing interface file
357	Inflows and outflows interface files have same name
361	Could not open external file used for time series
363	Invalid data in external file used for time series
401	General system error
402	Cannot open new project while current project still open
403	Project not open or last run not ended
405	Amount of output produced will exceed maximum file size; either reduce Ending Date or increase

Updating FLOPro.exe

To run a new FLO-2D model update (FLOPro.exe) but keep the previous version of the storm drain engine VC2005-CON.DLL for other projects, do the following:

1. Rename the previous VC2005-CON.dll file as VC2005-CON_OLD.dll from all the specified locations (C:\Program Files (x86)\FLO-2D PRO, C:\Windows\System32 or C:\Windows\SysWOW64).
2. Copy the new VC2005-CON.dll into the project folder along with the new FLOPro.exe.

Dynamic Link Library VC2005-CON.DLL

VC2005-CON.dll is the dynamic link library (engine) that runs the storm drain system in the FLO-2D mode. When updating FLOPro.exe, it is necessary to update VC2005-CON.dll. An out of date DLL library may cause a crash error or inconsistent results. The VC2005-CON.dll file (based on the file date) should be current in the following directories:

- C:\Program Files (x86)\FLO-2D PRO
- C:\Windows\System32[1]
- C:\Windows\SysWOW64[1]

Volume Conservation and Numerical Instability

When a model run is completed, volume conservation is reported in the SWMM.RPT file under Flow Routing Continuity. The FLO-2D model volume conservation results are written to the SUMMARY.OUT file. Volume conservation represents the difference between the inflow volume and the outflow plus storage volumes for the system. The storm drain system should be reviewed if the mass continuity error in SWMM.RPT exceeds some reasonable level, such as plus or minus 1 percent. Ideally, this error should be less than 1 percent. The original SWMM model might have volume conservation errors as high as 10 percent and this should not occur with the FLO-2D storm drain model. Excessive volume conservation error is primarily the result of short conduit lengths. It is recommended that the minimum pipe length be at least the size of the FLO-2D grid element side. Generally the volume conservation errors and numerical instabilities may be reduced or eliminated by:

- Increasing pipe roughness n-values;
- Reviewing the selection of reporting timestep (REPORT_STEP) in the SWMM.inp file and the simulation time (TOUT) in the CONT.DAT file;
- Assessing the reporting and plot control variables ([REPORT] INPUT and CONTROLS) in the SWMM.inp file;
- Reviewing the system connectivity for adverse slopes and incorrect inlet geometry.

The most common sources of numerical instability are:

[1] It is necessary to put the VC2005-CON.dll in the system folders if the FLOPro.exe is run from a project folder.

- Short conduits;
- Conflicting or poor system connectivity in the pipe network.

Discharge oscillations at inlets primarily occur when the pipe system is under pressure (full capacity). In the original SWMM model, when the pipe pressure exceeded the rim elevation, the inlet inflow discharge ceased and the water in the pipe could either be lost to 'flooding' or stored in artificial 'ponding' to be returned to the pipe when capacity became available. The artificial ponding routine was removed from the FLO-2D storm drain component.

For a given water surface elevation condition, the storm drain can receive inflow or discharge overflow to the surface water based on the pipe capacity. If the pipe is full, there is no inflow discharge for that computational timestep. As the water flows through or out of the storm drain and pipe capacity becomes available, inlet inflow can occur. The model will only allow inlet inflow if there is available capacity. There is zero inflow if the storm drain system is full regardless of the surface water elevation above the rim.

High Flow Instability Indices (HFII)

The SWMM.RPT file lists those nodes of the drainage network that have the largest flow continuity errors. The following is an example from a SWMM.RPT file:

```
High Flow Instability Indexes
*******************************
Link C9 (9)
Link C8 (8)
Link C10 (7)
Link C7 (6)
Link C6 (6)
```

To improve the volume conservation error, the links listed on the HFII table have to be reviewed. Fixing the top 5 listed HFII links usually result in a more stable model. The index is the number of flow turns that exists in a link during the simulation. A flow turn occurs when the difference between a new and old flow results in a perturbation which is defined as a significant flow difference or a change in flow direction.

Timestep and Conduit Length

Similar to the Courant criteria used on the FLO-2D surface water model, stability issues can arise if the timestep is greater than about two times the travel time through a pipe. This would be comparable to the wave celerity being equal to about 1.5 times the average flow velocity V in the pipe. To improve a model with numerical stability issues:

- A minimum conduit length of 20 ft or the FLO-2D grid element side length is recommended. Pipes shorter than 20 ft are reported as a warning message in the ERROR.CHK file and by the GDS.
- Dynamic wave routing numerical stability requires that the timestep be less than the time it takes for a dynamic wave (flow velocity plus wave celerity) to travel through the shortest conduit in the storm drain system. A maximum timestep of 1 second is sufficient for most storm drain simulations. The FLO-2D timesteps are used for both the surface water and the storm drain model and they are small enough for the storm drain solution to converge. A timestep calculation of a short pipe is:

  Conduit length Δx = 20 ft
  Average conduit velocity V = 7.30 fps
  COURANTFP from CONT.DAT C = 0.6
  Wave celerity c = 1.5 x V
  Applying the Courant equation:
  $\Delta t = C\,\Delta x/(V + c) = 0.6\ (20\text{ ft}) / (7.3\text{ fps} + 1.5 \times 7.3\text{ fps}) = 0.66\text{ seconds}$

Unstable Results

Oscillations that grow in time are a form of numerical instability. The solution is not converging and the following issues should be reviewed:

- A pipe is short relative to other adjacent pipes. A longer pipe length is recommended. A careful check of the storm drain connections in all contiguous connections of the unstable pipe should be completed prior to pipe length adjustments.
- Excessive discharges in adjacent downstream pipe elements generate an excessive decrease in the upstream water surface.
- A node dries on each timestep despite an increasing inflow. This is the result of an excessive discharges in adjacent downstream pipe elements.
- Excessive velocities (over 20 ft/sec) and discharges grow without limit. Increase the pipe length or the pipe roughness.
- There is a large continuity error. If the continuity error exceeds ± 10%, the user should check the pipe results for zero flow or oscillating flow. This may indicate an improperly connected system.

In general, excessive discharge or pressure head oscillations should be eliminated. There are physical system configurations that in reality might generate some instability oscillations, but these will usually decay over time. Other possible modifications to reduce storm drain numerical instability include:

- Increasing pipe roughness;

- Decreasing pipe slope;
- Increasing or adjusting pipe geometry;
- Eliminating a junction between two short pipe sections;
- Reducing or eliminating connections to isolate the unstable portion of the pipe network.

Conservatively high n-values (0.1 and higher) have been used to reduce pipe network instability. Uncertainty associated with pipe material, obstructions, debris, pipe bends, junction entrance and exit losses, and unsteady flow may warrant the application of conservative n-values. The numerical stability will improve with n-values higher than those typically assigned to straight pipes with uniform geometry in a steady flow condition. To save time, there are two checks that can be made prior starting a complete simulation:

1. Perform a short test run to confirm that the different links, nodes, subcatchments, etc. are properly connected and represent the actual pipe network.
2. Review the junction inverts to make sure that they are at same elevation as the invert of the lowest pipe entering or leaving the junction otherwise errors in the pipe hydraulics can occur.

For complex models, it is sometimes difficult to differentiate between oscillations that are produced by a numerical instability and those real oscillations that represent rapid discharge flux linked to inlets, junctions, conduits and outfall. Resolving potential error sources requires an understanding of the project and model application. Troubleshooting storm drain instability during the initial phase of a project can be accomplished with test runs. To summarize, some of methods for reducing storm drain pipe routing instability are:

1. Conservatively high n-values (up to 0.100) can be used to reduce pipe network dynamic instability. For large complex project, local conservative pipe n-values can reduce oscillations in return flows.
2. Eliminate short conduits in the simulation. Conduits should be longer than 20 ft (6 m) or at least the length of a FLO-2D grid element. A careful check of the storm drain connections in all contiguous connections of the unstable pipe should be completed prior to pipe length adjustments.
3. Investigate flooded or surcharged inlets. Storm drain systems may have local conditions that may be explained by analyzing the actual physical behavior of the system. For flooded or surcharge inlets or junctions displaying oscillations, upstream inlets should be examined to determine where the oscillations originate.
4. Review the system connectivity. Search for adverse slopes and incorrect inlet geometry.
5. Review the SWMM.rpt file for critical timestep elements and check the highest flow instability indexes (FII). This index is normalized with respect to the expected number of flow reversals (turns) that would occur for a purely random series of values and can range from 0 to 150. Inflow and flooding hydrographs for the FII elements should be

checked for oscillations. Check upstream and downstream plots (flow, depth, velocity, Froude No., and capacity) of the links having the highest FII's numbers.

6. Check for oscillations or instabilities associated with pumps.
7. If an instability or oscillation cannot be explained as a physical response of the system then try to isolate the problem by changing roughness in contiguous links or by removing sections of the storm drain system.
8. Reduce the reporting timestep to 30 s when oscillations are identified to have a more complete picture of the dynamic behavior of the system.

The storm drain dynamic wave routing uses an explicit scheme numerical solution that may fluctuate or oscillate. In the original SWWM model, most volume conservation errors were associated with numerical surging and typically volume conservation errors of 10% or more were acceptable. If the volume conservation error exceeds 1 % in the FLO-2D storm drain system, the model can be improved.

Chapter 6

Reviewing Storm Drain Model Results

Introduction

After a FLO-2D storm drain model is complete, the results can be reviewed with both the FLO-2D GDS program and the SWMM GUI. The GDS displays the storm drain inflow and return flow hydrographs, the water surface head on the storm drain inlet and outlet, and the hydraulic and energy grade lines of the storm drain system. The GDS is used to import the shapefiles for surface water velocity results, surface water flow depths and channel results from a given simulation generated in Mapper Pro or Mapper ++. With the SWMM GUI the user can access the results from the various storm drain objects. A FLO-2D storm drain simulation will generate the following files in Table 18.

Table 18. FLO-2D Storm Drain Simulation Files

Input Files	Output Files
SWMM.ini	SWMM.RPT
SWMM.rain	SWMM.OUT
SWMM.inp	SWMMOUTFIN.OUT
SWMMFLO.DAT	SWMMQIN.OUT
SWMMFLORT.DAT	FPRIMELEV.OUT
SWMMOUTF.DAT	

Viewing Results in the GDS

FLO-2D provides a variety of post-processing tools that include Mapper, PROFILES, HYDROG, MAXPLOT, and QGIS plug-in programs. The Mapper Pro and Mapper ++ read FLO-2D output and generate grid element plots, contours, shaded contours and shapefiles. The shapefiles can be imported to a GIS environment. These GIS tools complement the GDS capabilities of displaying FLO-2D output. The GDS has the ability to customize different GIS layer views. A possible approach to reviewing a storm drain simulation is outlined.

Step 1. Review the SUMMARY.OUT File

After a FLO-2D simulation is completed, the SUMMARY.OUT file can be reviewed with an ASCII text editor program. A FLO-2D model has been successfully completed if the termination time and date is reported at the end of the SUMMARY.OUT:

```
COMPUTER RUN TIME IS :  1.39575 HRS

THIS OUTPUT FILE WAS TERMINATED ON:   6/17/2015  AT:   0:10:39
```

Volume exchange and conservation between the surface water and storm drain is reported in the SUMMARY.OUT file as shown below.

```
==============================================================================
          *** FLO-2D STORM DRAIN EXCHANGE VOLUME (ACRE-FT) ***

    Storm Drain Inflow

SURFACE TO STORM DRAIN SYSTEM THROUGH INLETS                    23.211

SURFACE TO STORM DRAIN THROUGH OUTFALLS                          0.000
                                                             ---------

TOTAL (compare w/SWMM.rpt Wet Weather Inflow)                   23.211

    Storm Drain Outflow from Outfalls

STORM DRAIN TO SURFACE THROUGH OUTFALLS                          0.000

STORM DRAIN OUTFALL (OFF SYSTEM)                                22.836
                                                             ---------

TOTAL (compare w/SWMM.rpt External Outflow)                     22.836

    Storm Drain Return Flow to Surface

STORM DRAIN RETURN FLOW TO SURFACE THROUGH INLETS                0.657

STORM DRAIN STORAGE (PONDED FLOW INLETS)                         0.000

    Extracted from Storm Drain File (swmm.rpt)

WET WEATHER INFLOW                                              23.578

EXTERNAL INFLOW                                                  0.000

EXTERNAL OUTFLOW                                                22.836

Total Storm Drain Storage (nodes+links)                          0.214

Continuity Error (%)                                            -0.555

==============================================================================
```

Storm Drain Inflow: Reports the volume that represents inflow to the storm drain through the inlets and outfalls.

Storm Drain Outflow from Outfalls: Total volume of outfalls that discharge to the FLO-2D surface water and off the system.

Storm Drain Return Flow to Surface: Return flow to the surface water through the inlets as pressure flow.

Step 2. View the Surface Water Data and Results in the GDS

Open the GDS and load the project as shown in Figure 56. The components displayed in this image are the building area reduction factors (ARF-values) and the storm drain features. The storm drain features and parameters can be edited in the GDS while viewing the project (see the GDS manual). Alternatively, the project could be opened in the FLO-2D QGIS plug-in.

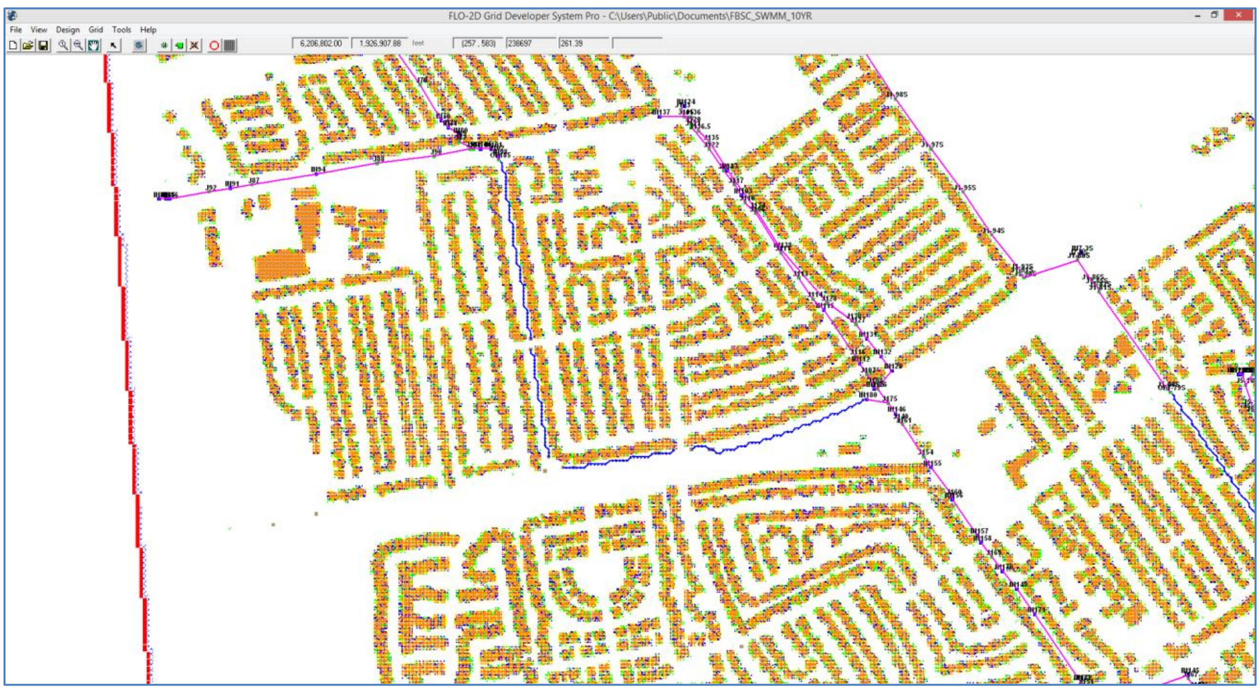


Figure 56. Typical GDS Storm Drain Project

Step 3. Import Shapefiles from Mapper (Optional)

If the output data was plotted in Mapper, the automatically generated shapefiles can be imported into the GDS or QGIS plug-in. The FLO-2D model maximum depths and velocities and other output can be imported as shapefiles to view the flooding with the various model components including the storm drain. The layer properties for each shapefile can be modified in the layers menu. A 10-year return period area of inundation is shown in Figure 57. The flooding is shown in the streets and is being captured by the storm drain system.

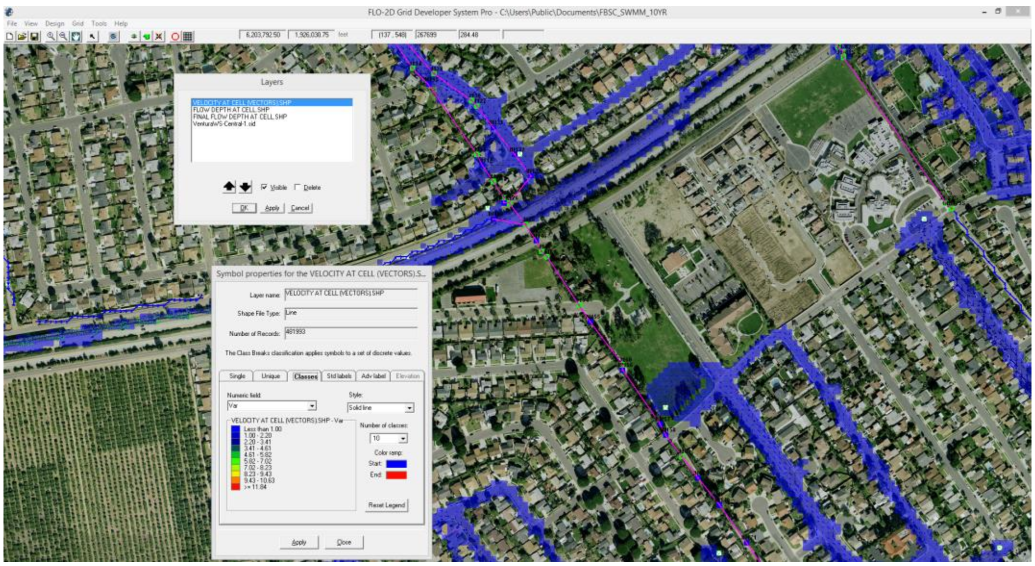


Figure 57. Maximum Flow Depth Shapefile in the GDS with Storm Drain Components

Step 4. Display the Storm Drain Results in the GDS

The GDS can display the storm drain inlet discharge hydrograph including the return flow (pressure flow) to the surface water. GDS will plot the water surface head on the storm drain inlet and outlet and the hydraulic and energy grade lines. Click Tools | Storm Drain | Storm Drain Discharge Display (Figure 58) to view results plots.

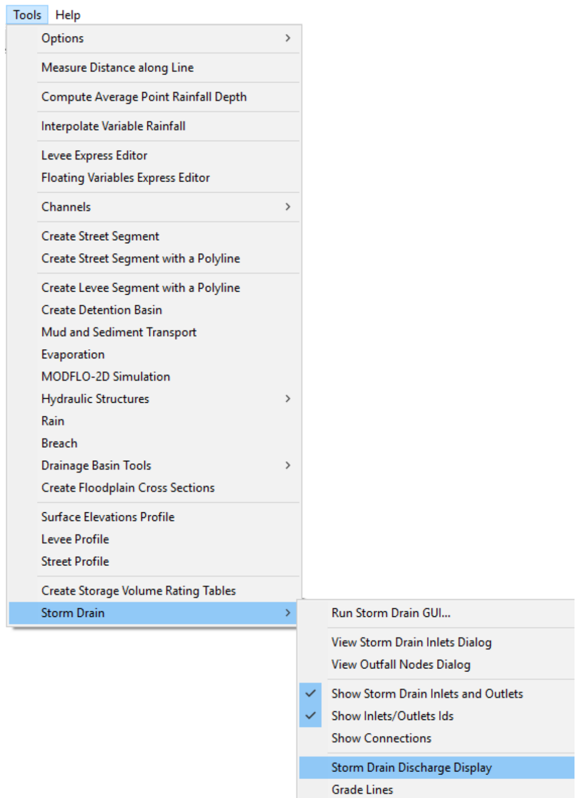


Figure 58. Storm Drain Display Menu Options

Select the SWMM.RPT output file from the project folder to display results (Figure 59), then click on the individual inlet/junction to generate the plots.

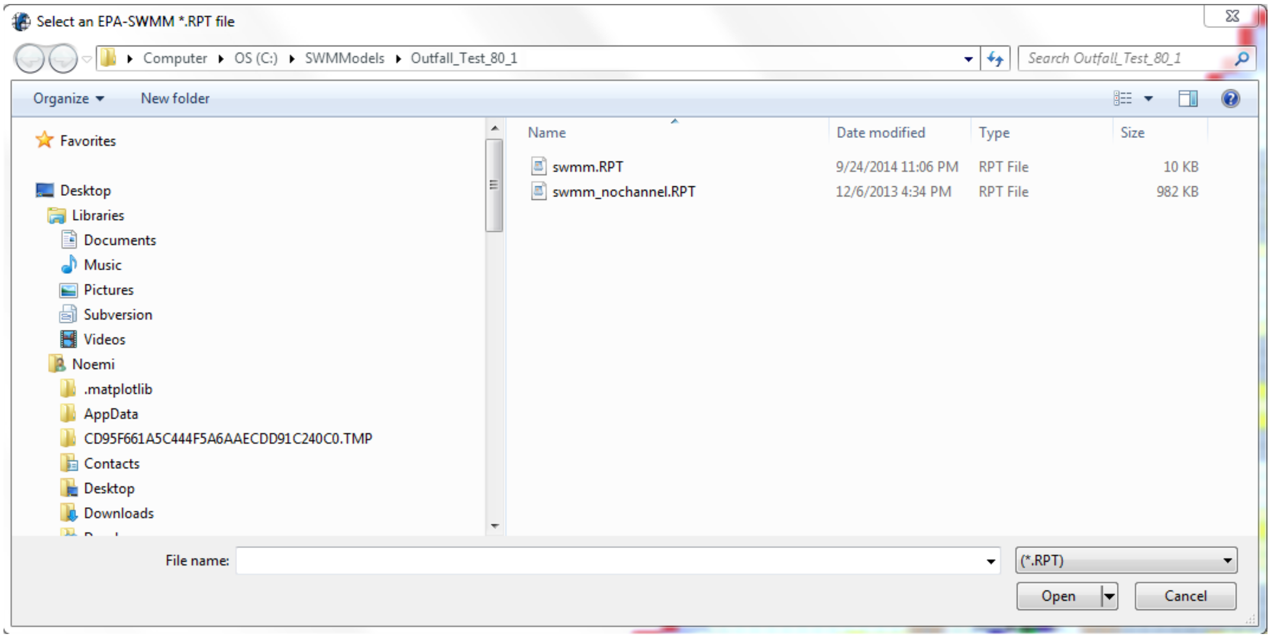


Figure 59. Dialog to select the SWMM.RPT Output File

Figure 60 to Figure 63 displays an inlet hydrograph, return flow, water surface head and energy grade line as displayed by the GDS.

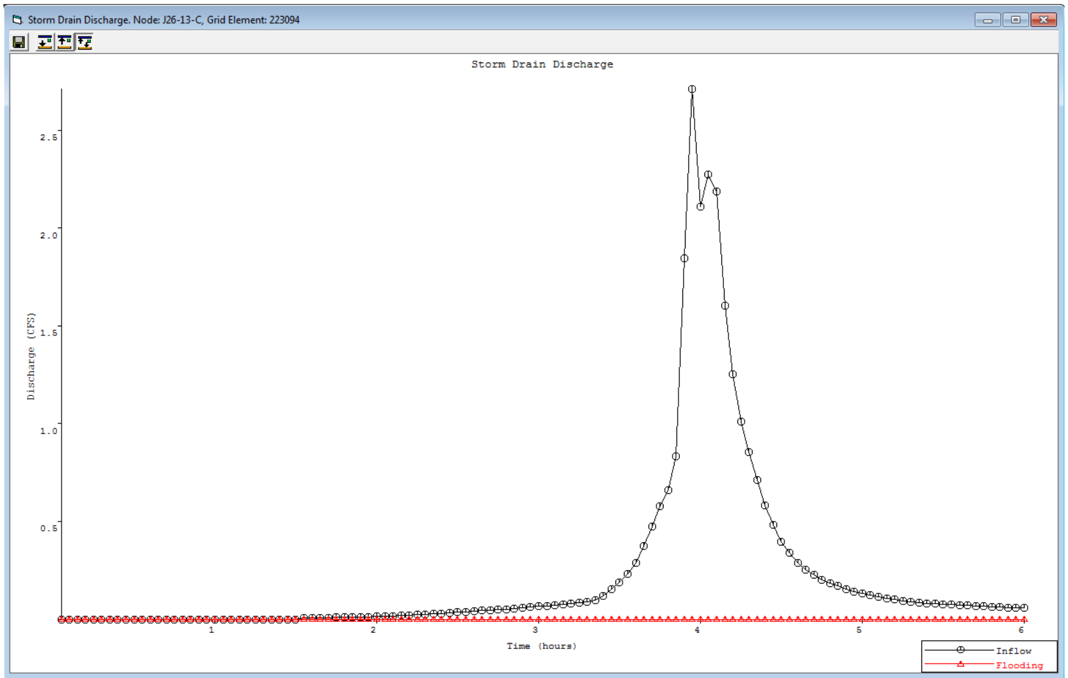


Figure 60. Display of Storm Drain Node Inflow Discharge

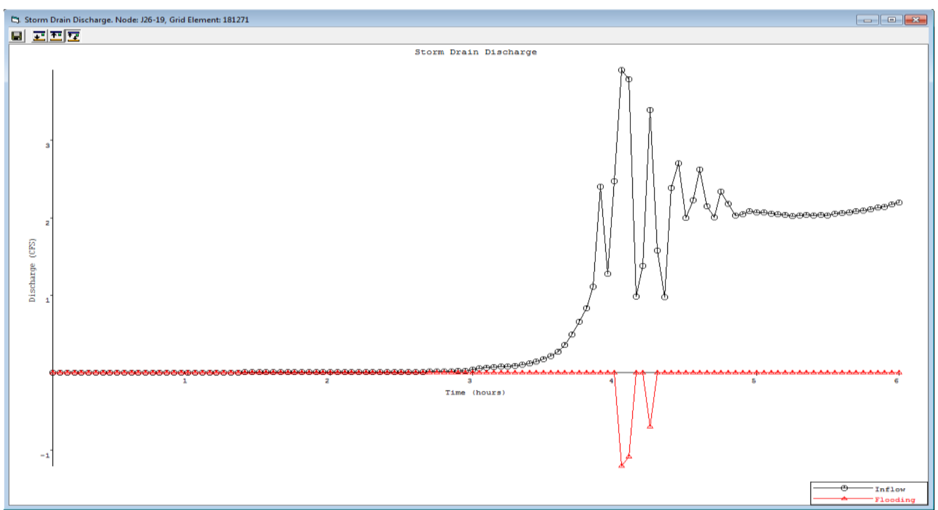


Figure 61. Display of Storm Drain Inflow and Return Flow Hydrograph

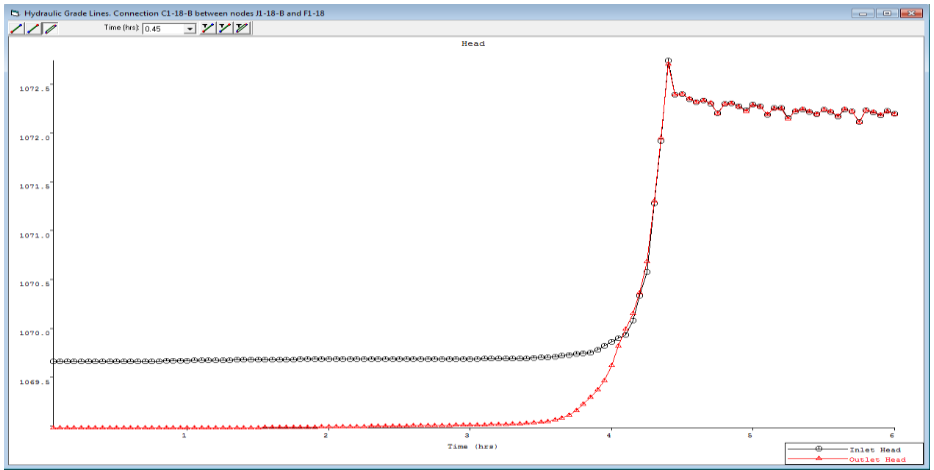


Figure 62. Display of Water Surface Head on the Storm Drain Inlet

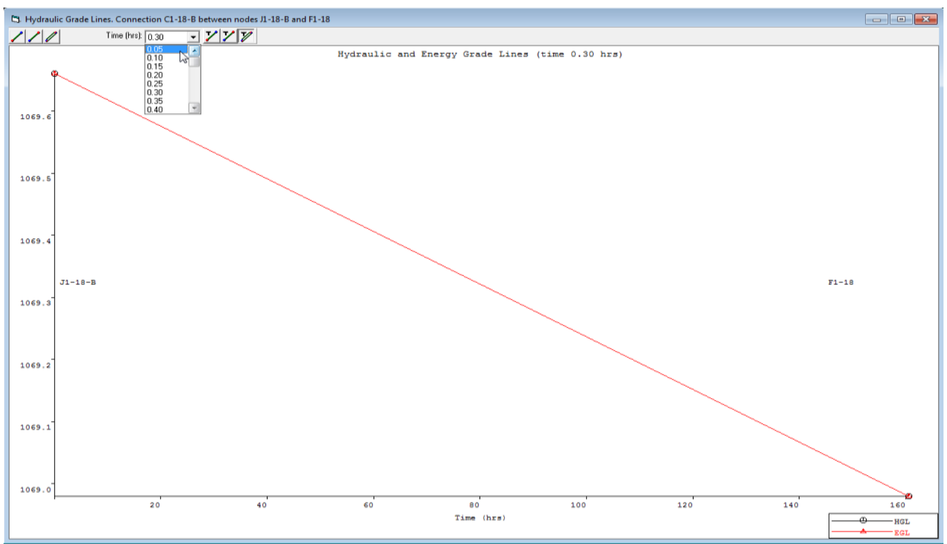


Figure 63. Display of the Energy Grade Line on a Storm Drain Conduit

Viewing the Storm Drain Output in the SWMM GUI

The SWMM GUI creates the storm drain data input files and graphically displays the results. A FLO-2D storm drain simulation will generate output files that are compatible with the SWMM GUI. To view the output, first make sure that the following files are present in the project folder:

- SWMM.ini
- SWMM.inp
- SWMM.rain
- SWMM.RPT
- SWMM.OUT

The SWMM.RPT file reports the storm drain output data in ASCII format and can be read with any ASCII editor. The SWMM.OUT file is a binary output file with results reported temporally and spatially that can be read by the SWMM GUI.

SWMM.ini

The storm drain results are not automatically loaded into the SWMM GUI. To load the results, navigate to the project folder directory and open the SWMM.ini file using an ASCII text editor. The user needs to set 'Saved = 1" and "Current = 1" as highlighted in Figure 64.

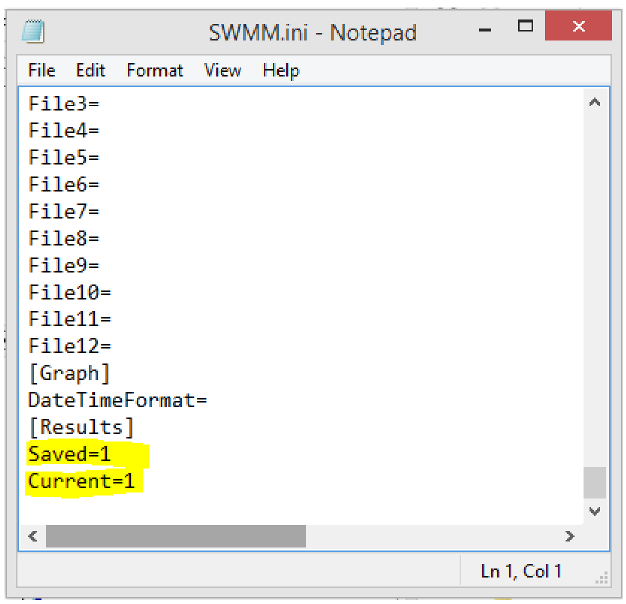


Figure 64. SWMM.ini File with Required Data to View the Results in the SWMM GUI

View the Storm Drain Results

Open the SWMM GUI and navigate to the project folder. Open the SWMM.inp file from the File|Open menu. This will access the model output and enable the GUI to display the results as shown in Figure 65. The SWMM GUI has controls to label and color code the results in a variety of configurations.

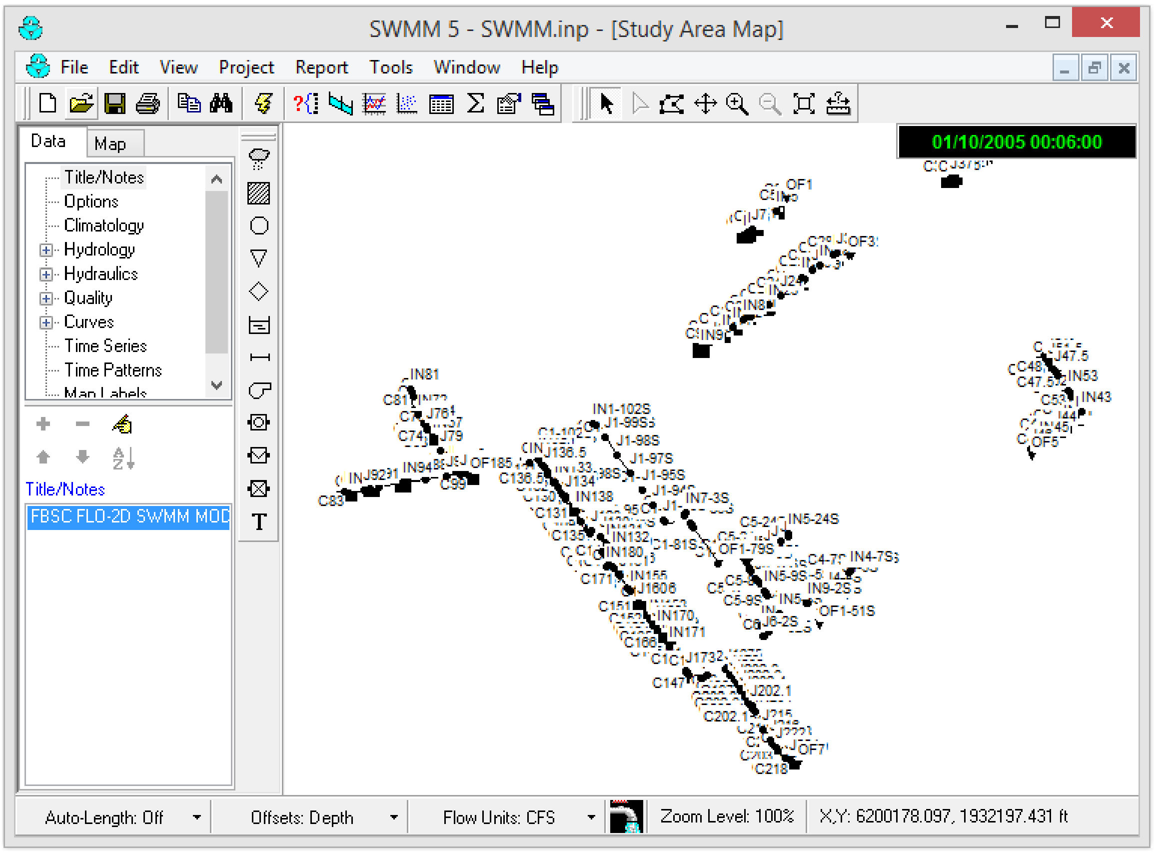


Figure 65. Example Storm Drain Model in the SWMM GUI

The SWMM GUI has some tools that can assist in the review of the storm drain output. The SWMM manual should be used referenced when using the SWMM GUI. The GUI environment is shown in Figure 66 consisting of a main menu, tool bars, study area, and working area. These attributes and some of the options are explained briefly.

File Menu

Open: Opens an existing project

Reopen: Opens recently used projects

Save: Saves the current project

Save As: Saves the project under a different name

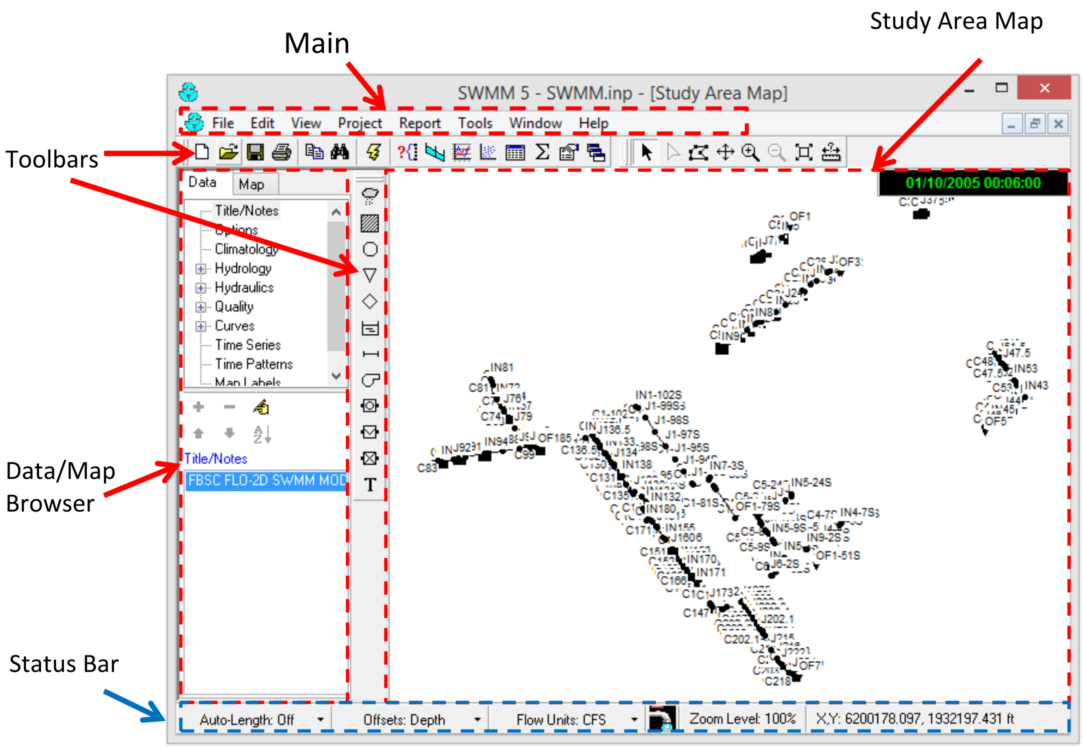


Figure 66. SWMM GUI Environment

Edit Menu

Select Object: Select an object on the map

Find Object: Locates a specific object by name in the map

Find Text: Locates specific text in a status report

Group Edit: Edits a property for the group of objects within selected region

View Menu

Dimensions: Sets reference coordinates and distance units for a study area

Backdrop: Allows a background image to be added, positioned and viewed.

Query: Highlights objects that meet specific criteria

Objects: Toggles display of classes of objects

Legends: Controls display of the map legends

Toolbars: Toggles display of tool bars

Project Menu

Defaults: Edits a project's default properties

Report Menu

Status: Displays a status report for the last simulation. The Bookmark feature enables quick review of the SWMM.RPT file sections (Figure 67).

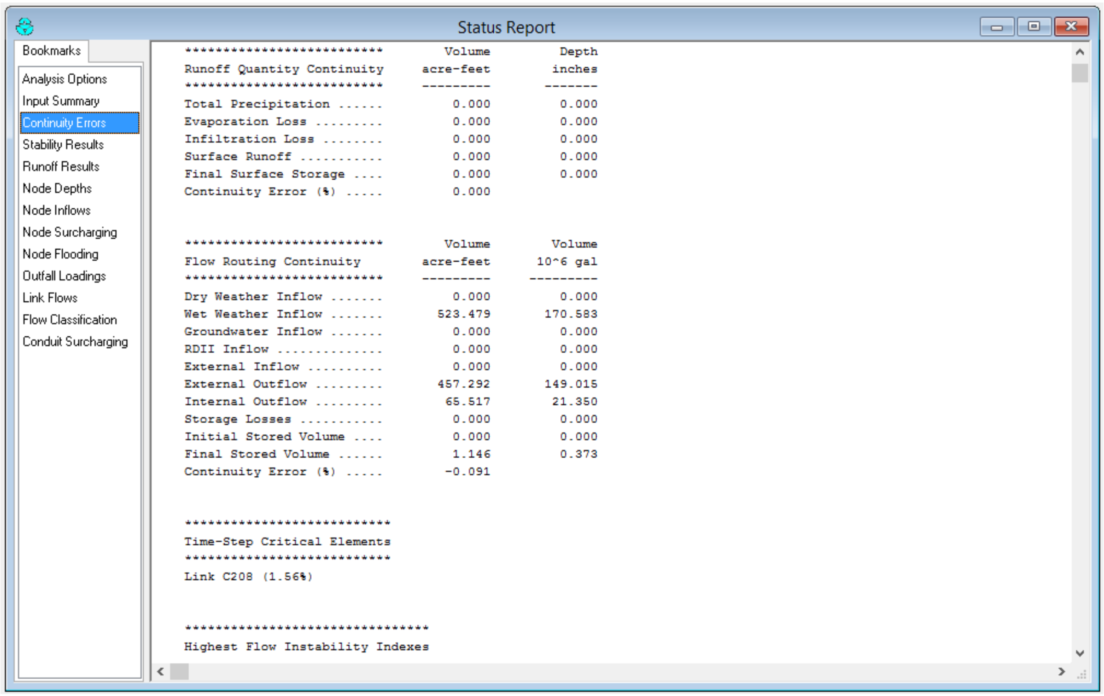


Figure 67. Bookmark Interface that Facilitates a Review the SWMM.RPT File

Graph: Command that displays simulation results in graphical form (Figure 68. Graph: Time Series).

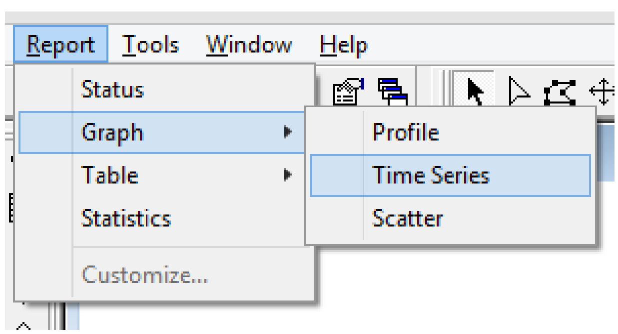


Figure 68. Graph: Time Series

Table: Displays simulation results in tabular form**.**

Tools Menu – Commands that Provide Control Detail

Program Preferences: Sets program preferences, such as font sizes, deletions, number of decimal places displayed, etc.

Map Control Display Options: Sets appearance options for the Map, such as object size, annotation, flow direction arrows, and background colors (Figure 69).

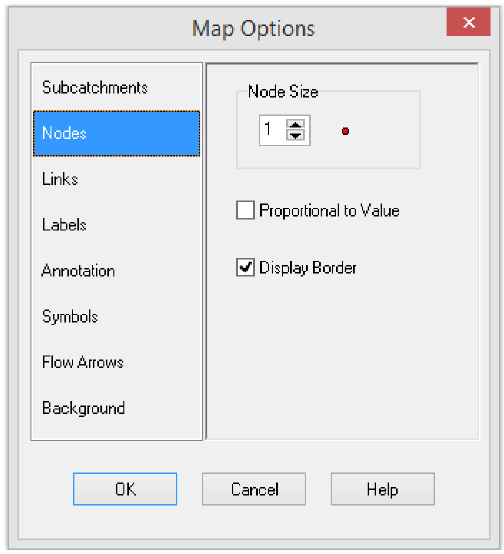


Figure 69. Map Options

Help Menu

Help Topics: Displays the Help system's Table of Contents

How Do I: Displays a list of the most common operations

Tutorial: Presents a short tutorial introducing the user to the EPA SWMM

Study Area Map

This map is a working environment with of storm drain system.

Toolbars

Provide shortcuts to common operations and includes the standard toolbar.

Map Toolbar and Object Toolbar

Data/map browser provides access to all the data objects in a project and controls the mapping themes and time periods viewed on the study area map.

Status Bar

Auto-length: Indicates whether the automatic computation of conduit lengths. It is recommended that the auto-length be turn 'off" if the data is imported from a GIS database.

Offsets: Indicate whether the positions of links above the invert of their connecting nodes are expressed as depth above the invert or as an offset elevation. The FLO-2D component is configured for the depth.

Flow units: English or metric. The previously entered data is not automatically adjusted for the unit system if the system is changed.

Zoom level: Zoom level percentage.

XY location: Mouse cursor coordinates.

Data Browser

The data browser has three sections (Figure 70).

- A tree box that lists all the storm drain categories of data objects;
- Edit buttons;
- A list of individual objects in the selected category.

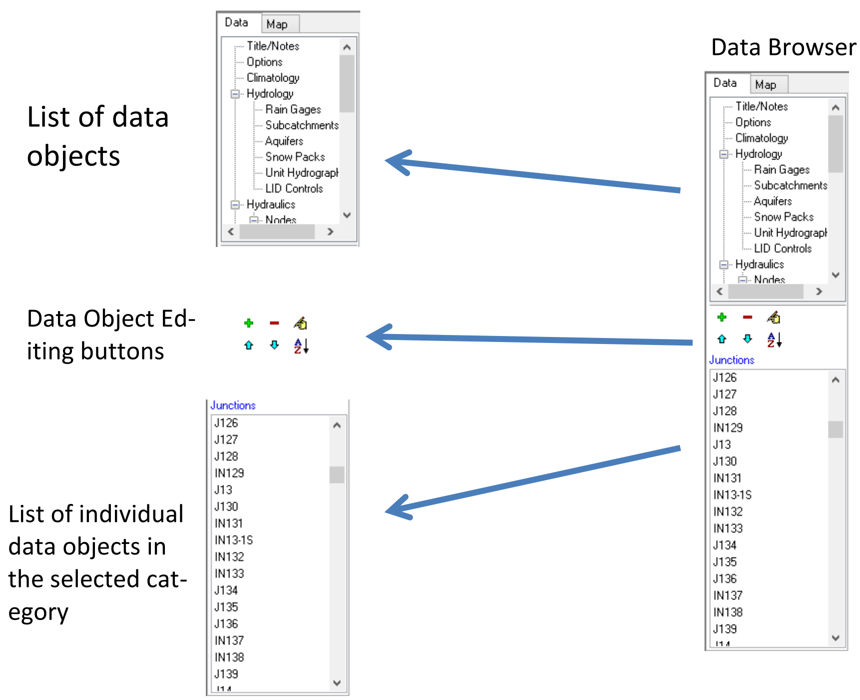


Figure 70. SWMM GUI Data Browser Panel

Map Browser

The map browser consists of three panels that control the results display (Figure 71).

- Themes Panel selects variables according to color on the map.
- Time Period Panel indicates a specific output interval to display results on the map.
- Animator Panel controls the animation of the temporal output and profile plots.

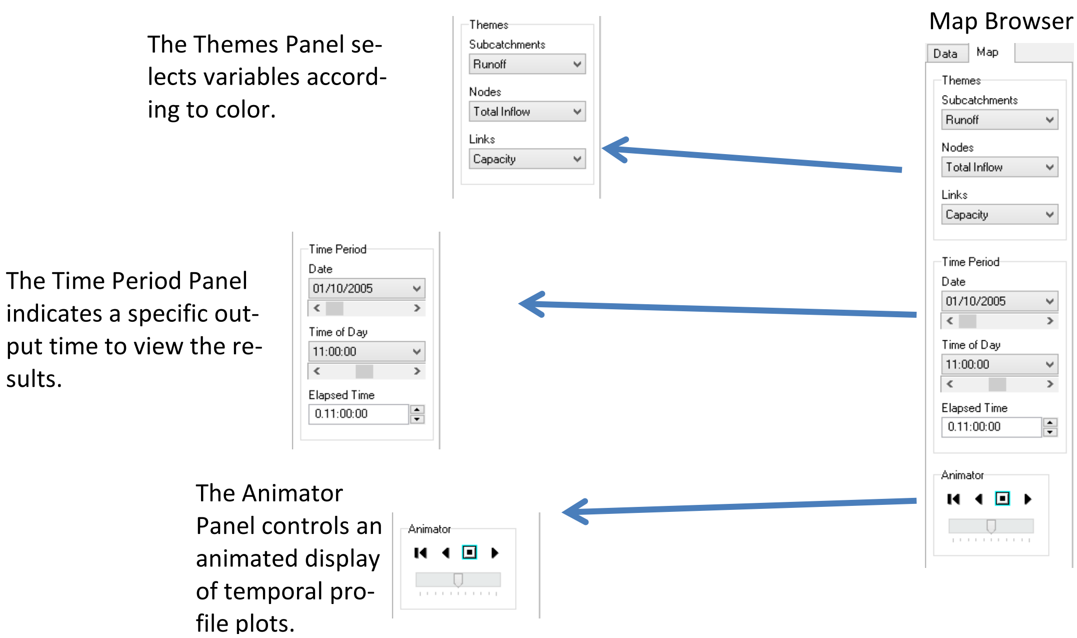


Figure 71. Map Browser, Time Output Interval to View Results and Animator Controls

Output Display

There are a number of options to analyze the output files with the GDS and the SWMM GUI. The following figures show a few of the options to display the results. Figure 72 shows the inlet discharge computed by FLO-2D. This figure is plotted in the GDS and shows the exchanged flow between the surface water and the storm drain system. The plot represents the volume that is returned from the storm drain system to the surface water under pressure. This occurs when the there is no more capacity in the system.

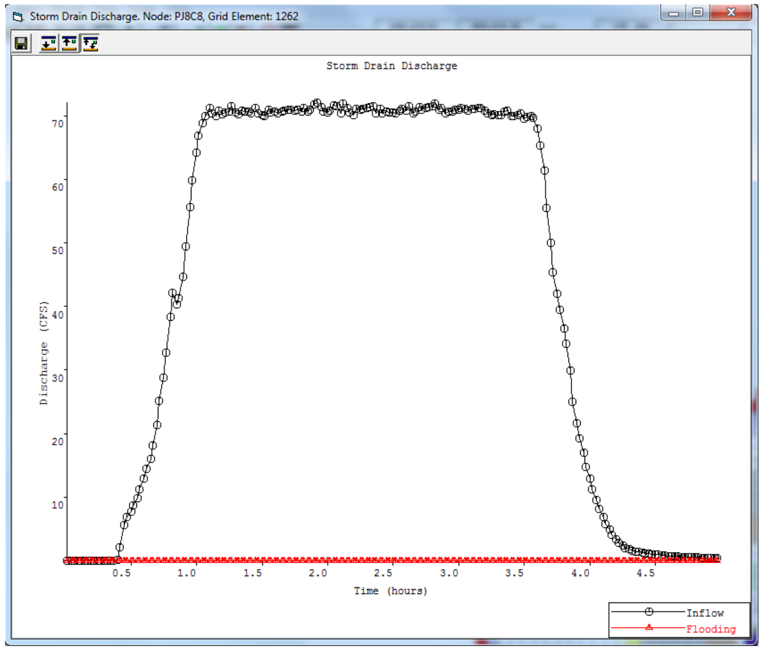


Figure 72. Storm Drain Discharge Reported in SWMMQIN.OUT (Plotted in the GDS)

Figure 73 shows the time series of an outfall depth. Other variables are plotted as a time series by the SWMM GUI. Outfall discharge may be computer as flow off the storm drain system or as return flow to the FLO-2D surface water.

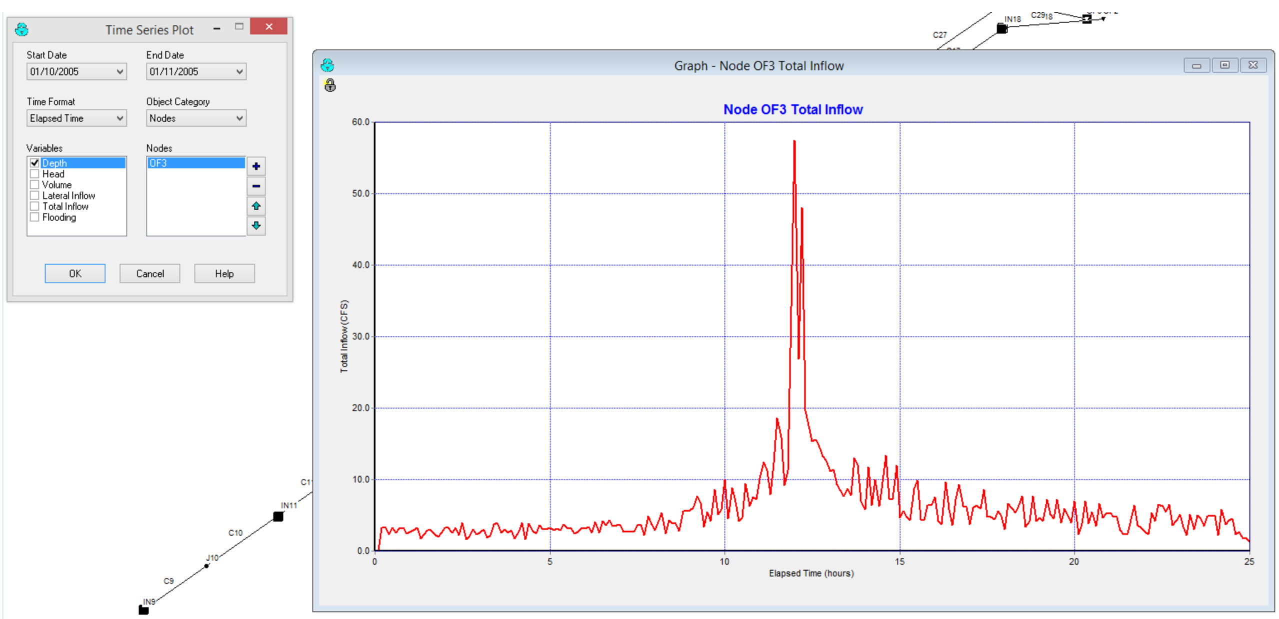


Figure 73. Time Series Plots of Depth, Head, Inflow and Return Flow in the SWMM GUI.

Figure 74. displays the flow in a series of connected pipes. A number of hydrographs from various locations in the storm drain can be plotted on a single graph. Combined hydrograph plots are helpful to determine the volume distribution and to review the timing of the flood wave progression throughout the storm drain system.

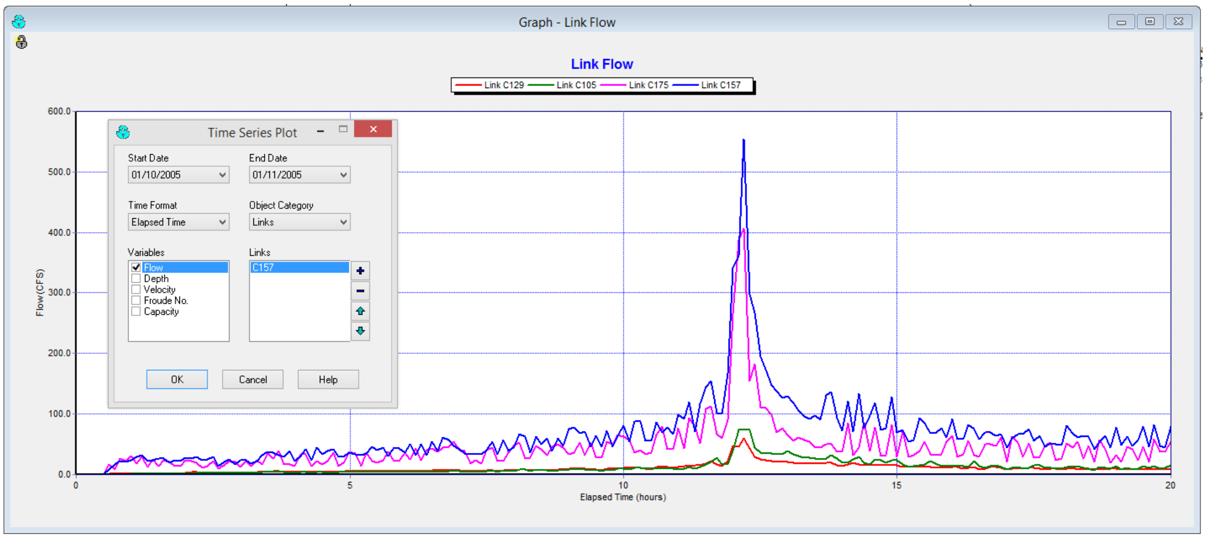


Figure 74. Hydrographs for Each Link in the Storm Drain System

The conduit flow depth profile at the peak discharge is shown in Figure 75. This profile can be animated using the map panel options. This tool will display the flow progression through the pipe system.

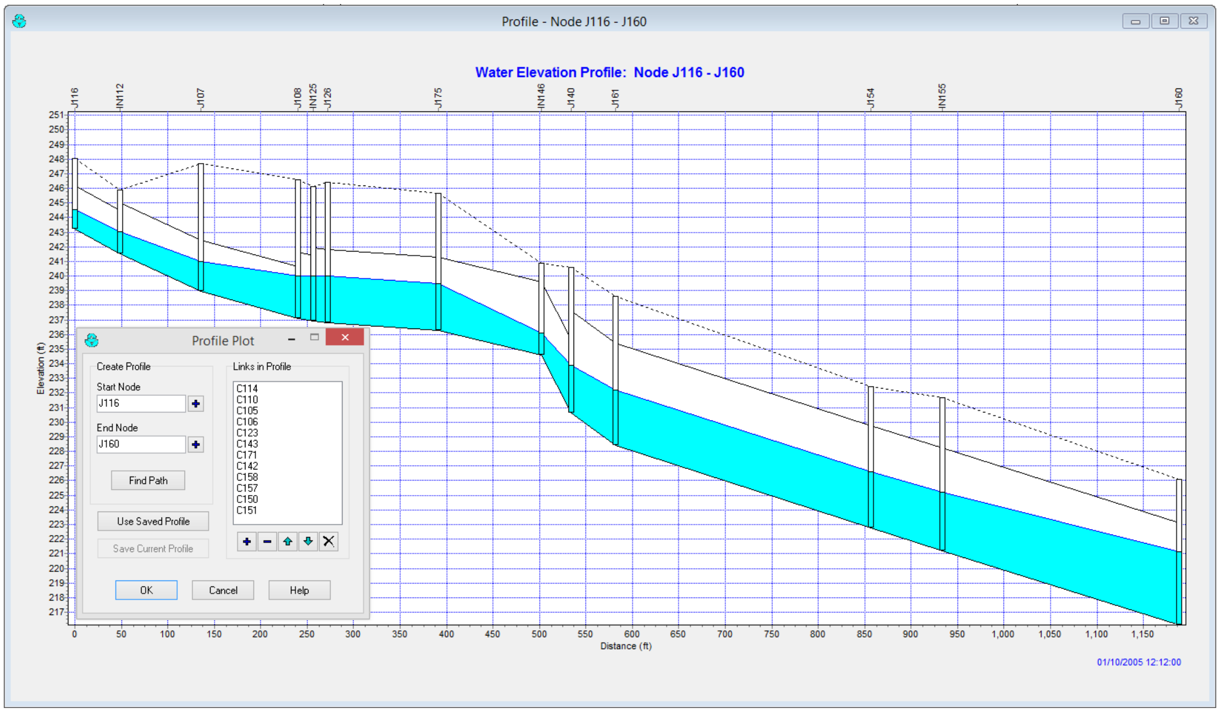


Figure 75. A Conduit Depth Profile

The storm drain system results can be colored and animated with the mapper tools. The SWMM.RPT file can be reviewed using the status report command in the report menu. The use of colors and labels enables a dynamic visual display of the flow distribution (Figure 76).

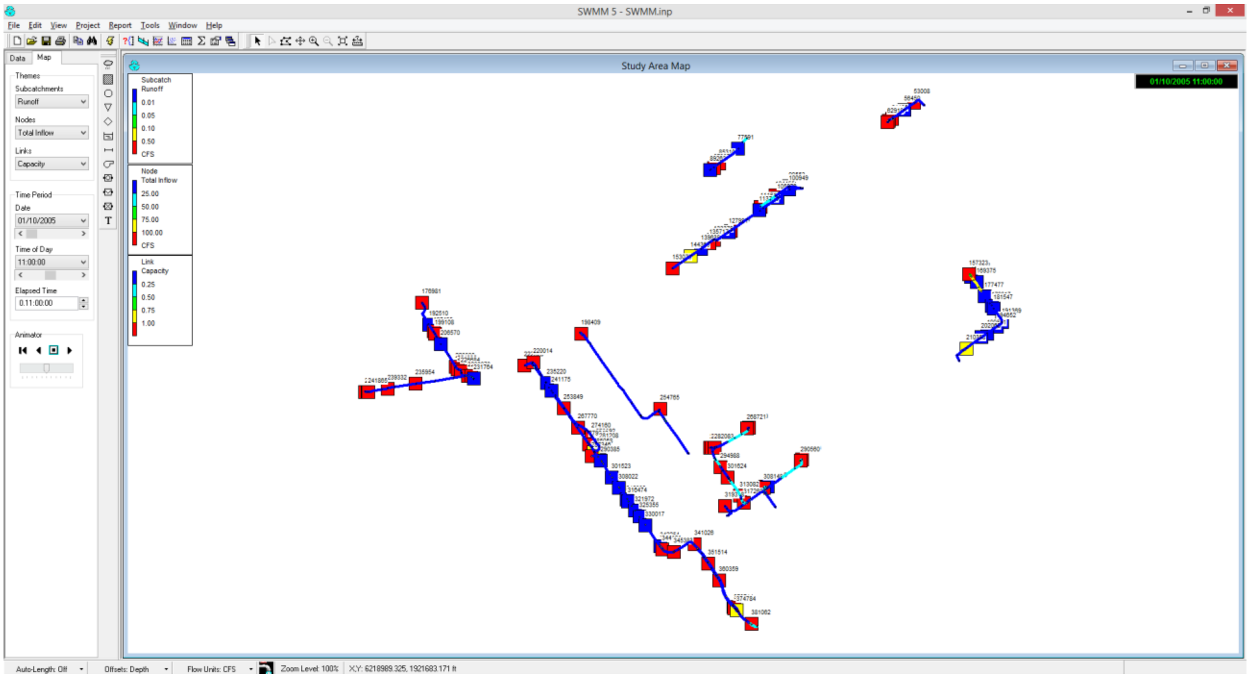


Figure 76. The Display of the Storm Drain System with the Map Settings

Chapter 7

Porting a SWMM Model to the FLO-2D Storm Drain System

Introduction

This chapter describes the porting procedure to modify and adapt the input data for a 'stand-alone' EPA SWMM Version 5.0.022 to the FLO-2D Pro storm drain model. The SWMM engine has been modified and integrated into the FLO-2D Pro model. The FLO-2D storm drain model is a distinctly new model that is superior to the original EPA SWMM model. The following data file revisions are required to convert an existing 'stand-alone' SWMM model to a FLO-2D storm drain model.

Porting SWMM Data Files to a FLO-2D Storm Drain Model

The modifications that have to be made to the SWMM.inp file are discussed.

Rain gages

Rain gages are not required in the FLO-2D Storm Drain Model.

Subcatchments

Subcatchments are not required in the FLO-2D Storm Drain Model.

Inlets

For junctions to function as inlets they must have an ID that starts with 'I'. The required data for inlets are (Figure 77):

- Name (field): Starts with the character 'I'.
- X-coordinate and Y-coordinate: It is required to have a spatial reference to pair FLO-2D and the 'stand-alone' SWMM system.
- Invert Elevation and Maximum Depth (fields):
    - The invert elevation and maximum depth can be edited manually in the SWMM GUI or by using other software editors.
    - The maximum depth and the invert elevation are used to determine the rim elevation.
    - The rim elevation is compared with the FLO-2D grid element elevations.
    - These comparisons are listed in the FPRIMELEV.OUT file.
    - Ideally the elevations in the *.inp file should match those of the grid system.

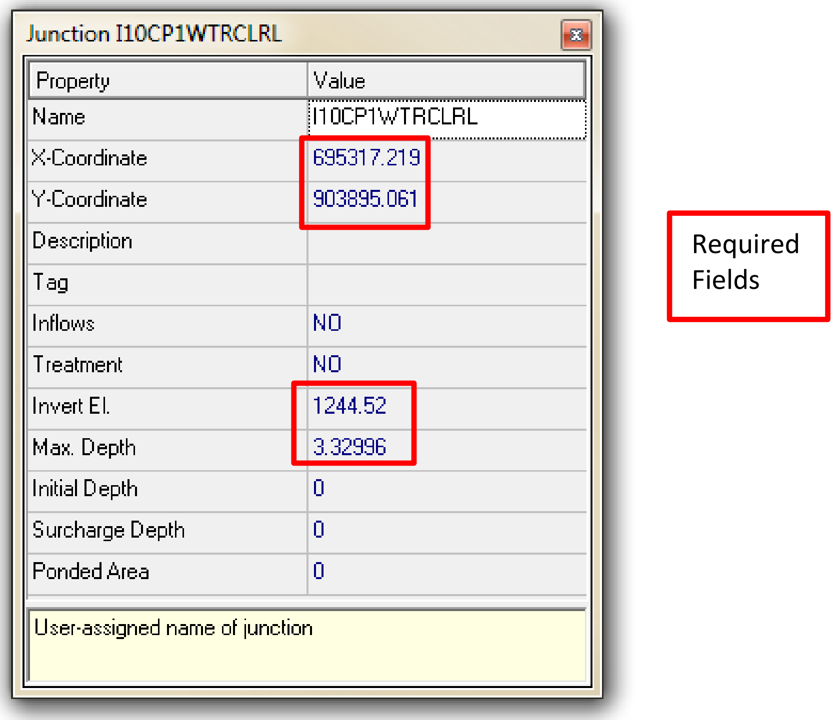


Figure 77. SWMM GUI Junction Dialog Box Showing the Required Data

Outfalls

Outfalls are terminal nodes in the drainage system used to define final downstream boundaries. Only a single link can be connected to an outfall node. The required data includes (Figure 78):

- Name (field)
- X- and Y-coordinate: It is required to have a spatial reference to pair FLO-2D and the 'stand-alone' SWMM system.
- Invert Elevation:
  - An outfall that return flows to a channel has to be assigned to the left bank.
  - An option to have an underground (underwater representing ponded flow) outfall was added in the FLO-2D storm drain.
  - This elevation can be lower than the grid element and is not reported in the FPRIMELEV.OUT file.
  - The outfall can discharge off the model.
- Outfall Type (Figure 78):
  - Set to 'FREE' to exchange flow with the FLO-2D surface water system.
  - Other types such as NORMAL, FIXED, TIDAL or TIME SERIES can be assigned but the flows will not be exchanged with the surface water but will debouch the flow off the storm drain system.
- Flapgate: Set to 'NO' for the outfalls to exchange flows with the surface water.

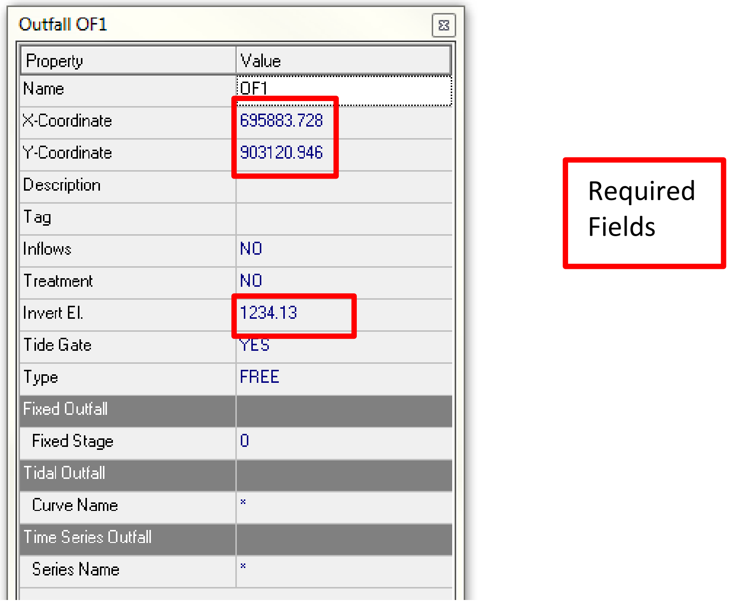


Figure 78. SWMM GUI Outfall Dialog Box.

SWMM.ini

This file contains information about the model global settings and output and is saved by the SWMM GUI. To enable the display of the FLO-2D storm drain results in the SWMM GUI the user must set the following lines as follows:

[Results]

Saved=1

Current=1

SWMMFLO.DAT

This file contains the inlet geometry as well as the names/numbers that identify the paired inlet with the FLO-2D grid cells. It is created from the SWMM.inp file by the GDS. To use the SWMM.inp to create the SWMMFLO.DAT, the following SWMM components have to be removed from file:

- Hydrologic processes such as time-varying rainfall, rainfall interception, evaporation, depression storage, groundwater percolation, snow accumulation and standing water surface.
- Watershed processes such as infiltration trenches, streets, and swales.
- Surface flood simulation features such as canals, culverts and bridges. FLO-2D engine performs all the surface flood routing.

After the SWMM.inp file has been edited, the storm drain command can be called from the FLO-2D GDS (Figure 79.).

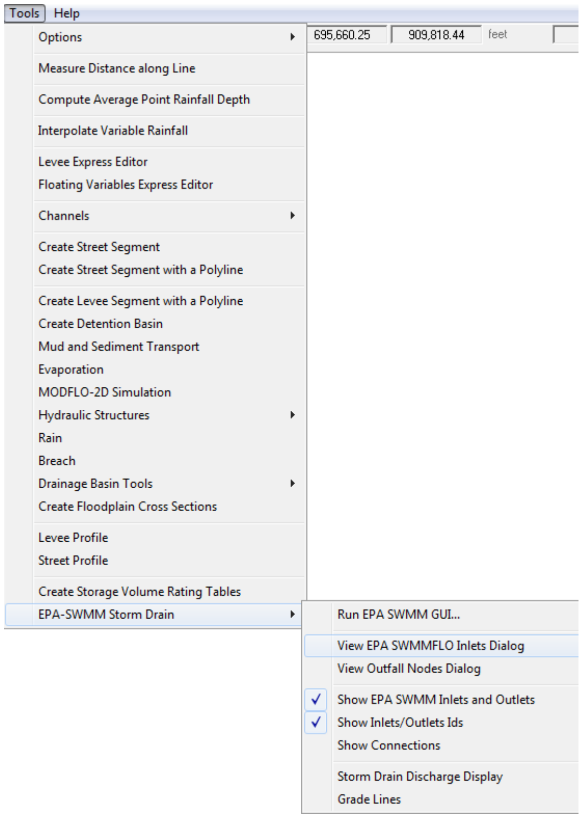


Figure 79. GDS Storm Drain Commands

The GDS then locates and reads the SWMM.inp file (Figure 80.) and loads the SWMM inlet nodes and associates them with a FLO-2D grid element as shown in Figure 81. At this point the storm drain geometry can be entered into the GDS.

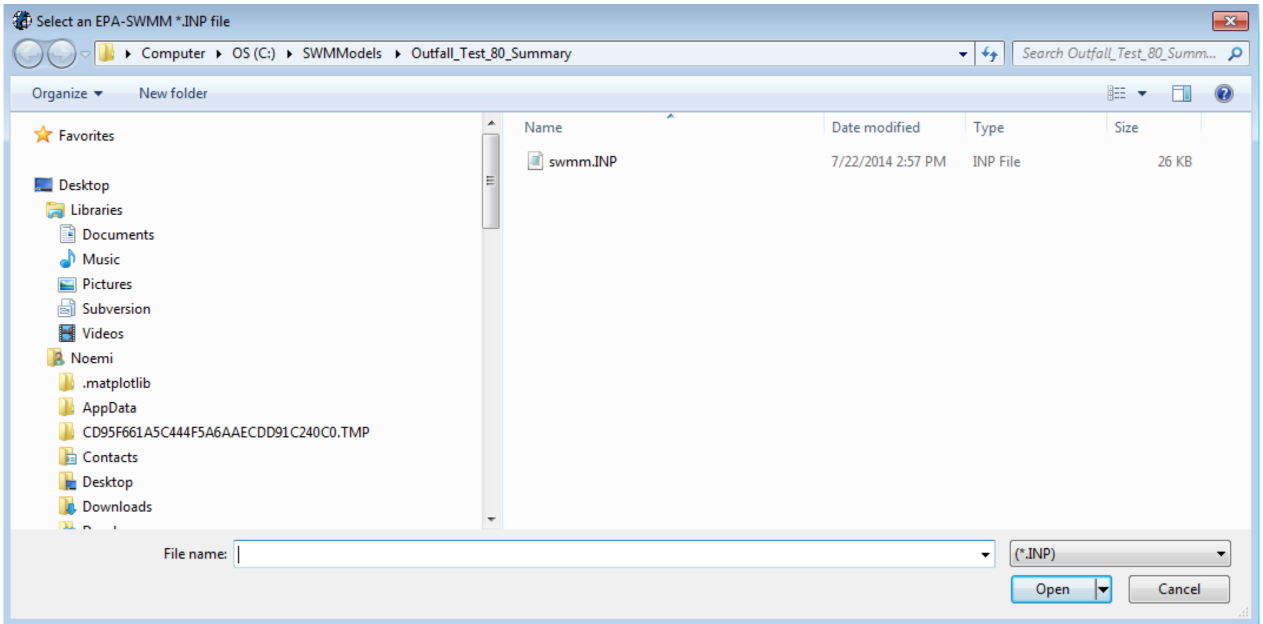
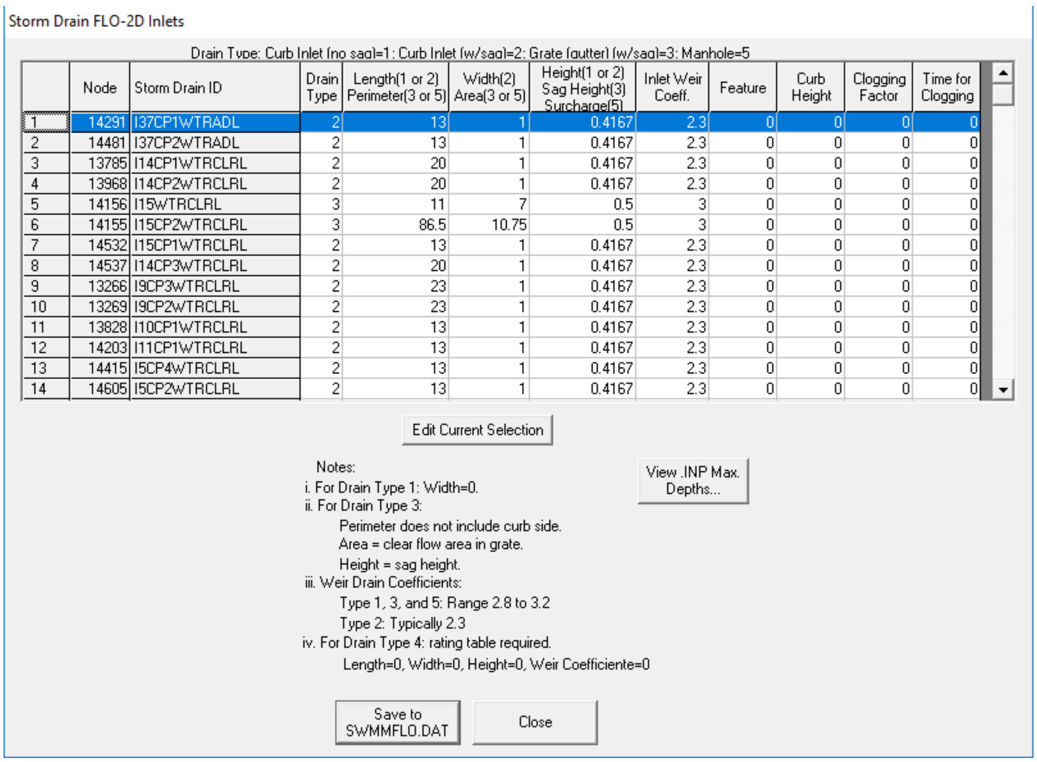


Figure 80. Reading the SWMM.inp File in the GDS.



	Node	Storm Drain ID	Drain Type	Length(1 or 2) Perimeter(3 or 5)	Width(2) Area(3 or 5)	Height(1 or 2) Sag Height(3) Surcharge(5)	Inlet Weir Coeff.	Feature	Curb Height	Clogging Factor	Time for Clogging
1	14291	I37CP1WTRADL	2	13	1	0.4167	2.3	0	0	0	0
2	14481	I37CP2WTRADL	2	13	1	0.4167	2.3	0	0	0	0
3	13785	I14CP1WTRCLRL	2	20	1	0.4167	2.3	0	0	0	0
4	13968	I14CP2WTRCLRL	2	20	1	0.4167	2.3	0	0	0	0
5	14156	I15WTRCLRL	3	11	7	0.5	3	0	0	0	0
6	14155	I15CP2WTRCLRL	3	86.5	10.75	0.5	3	0	0	0	0
7	14532	I15CP1WTRCLRL	2	13	1	0.4167	2.3	0	0	0	0
8	14537	I14CP3WTRCLRL	2	20	1	0.4167	2.3	0	0	0	0
9	13266	I9CP3WTRCLRL	2	23	1	0.4167	2.3	0	0	0	0
10	13269	I9CP2WTRCLRL	2	23	1	0.4167	2.3	0	0	0	0
11	13828	I10CP1WTRCLRL	2	13	1	0.4167	2.3	0	0	0	0
12	14203	I11CP1WTRCLRL	2	13	1	0.4167	2.3	0	0	0	0
13	14415	I5CP4WTRCLRL	2	13	1	0.4167	2.3	0	0	0	0
14	14605	I5CP2WTRCLRL	2	13	1	0.4167	2.3	0	0	0	0

Figure 81. GDS dialog Inlet Geometry Data

SWMMOUTF.DAT

In the SWMMOUTF.DAT file the “Allow Discharge” switch can be selected to enable outfall discharge to the FLO-2D surface water system. The GDS reads the outfall nodes from the SWMM.inp file, then opens the dialog box to set the switch ‘ON’ or ‘OFF’ for each outfall node and saves the data. If the ‘OFF’ option is selected, the outfall node is treated as a regular outfall in SWMM discharging flow off the pipe network system. To create this file, run the GDS and go to Tools | Storm Drain | View Outfall Nodes Dialog command (Figure 82.). The dialog can then be used to turn ‘ON’ or ‘OFF’ the outfall nodes and save the SWMMOUTF.DAT file.

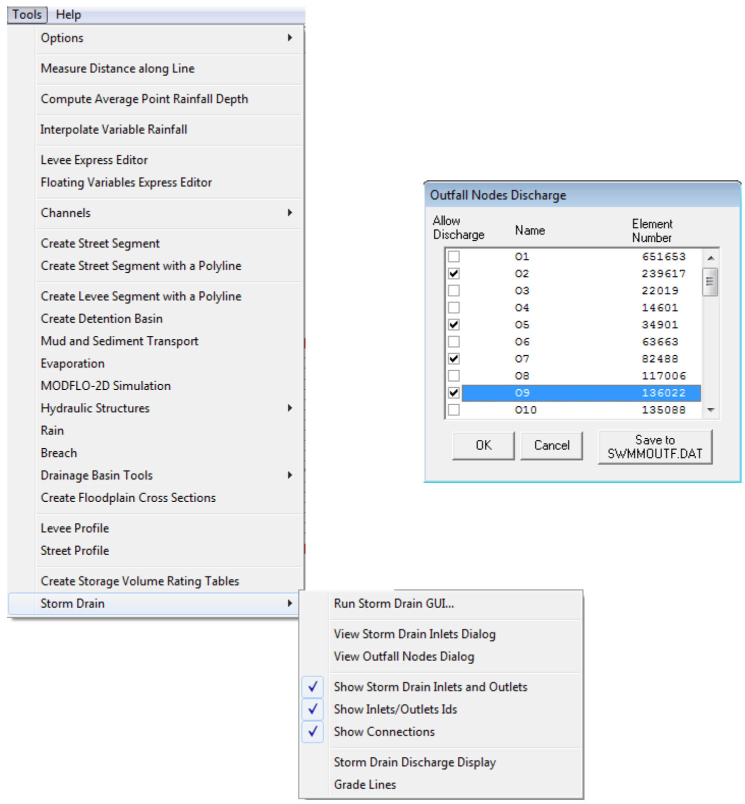


Figure 82. GDS Outfall Dialog Box Command

Simulation Options

The following simulation options have to be selected:

- The “FLOW_UNITS” must match those of the FLO-2D model (English or Metric);
- The remaining storm drain parameters should be selected in accordance with the SWMM User’s Manual (Figure 83).

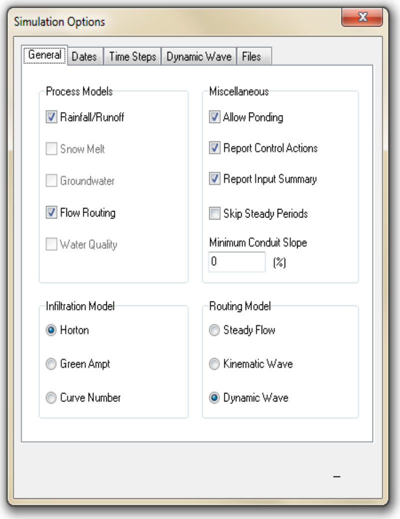


Figure 83. SWMM Dialog with the General Tab Selections

Chapter 8

FLO-2D Storm Drain Example

Two storm drain examples are provided with the FLO-2D installation. Lesson 16 demonstrates the procedure to set up the storm drain features using the SWMM GUI and the GDS. The following tutorial outlines the steps to run the Storm Drain project.

To create a FLO-2D Storm Drain model the following tasks are performed:

1. Create or open an existing FLO-2D model project;
2. Open the SWMM GUI from the GDS and develop the SWMM.inp data file. There are other GUIs that can be used to create the *.inp file.
3. Close the SWMM GUI and view the storm drain network in the GDS;
4. Generate the SWMMFLO.DAT file with the storm drain inlet data;
5. Create the SWMMOUTF.DAT using the Outfall Node Discharge Option;
6. Run the FLO-2D storm drain model.

Data Required

Storm drain data includes inlet geometry, conduit sizes, and inlet/outlet locations. After the FLO-2D surface water model is prepared, the storm drain data can be created. The steps are:

Create or open a FLO-2D model

1. Open the GDS and locate the project folder.
2. Import any aerial images to visualize the location of the storm drain system.

Open the SWMM GUI

3. From the GDS open SWMM GUI (Figure 84). It is possible to open the GUI directly by calling EPA SWMM 5.0 from the Windows search bar.

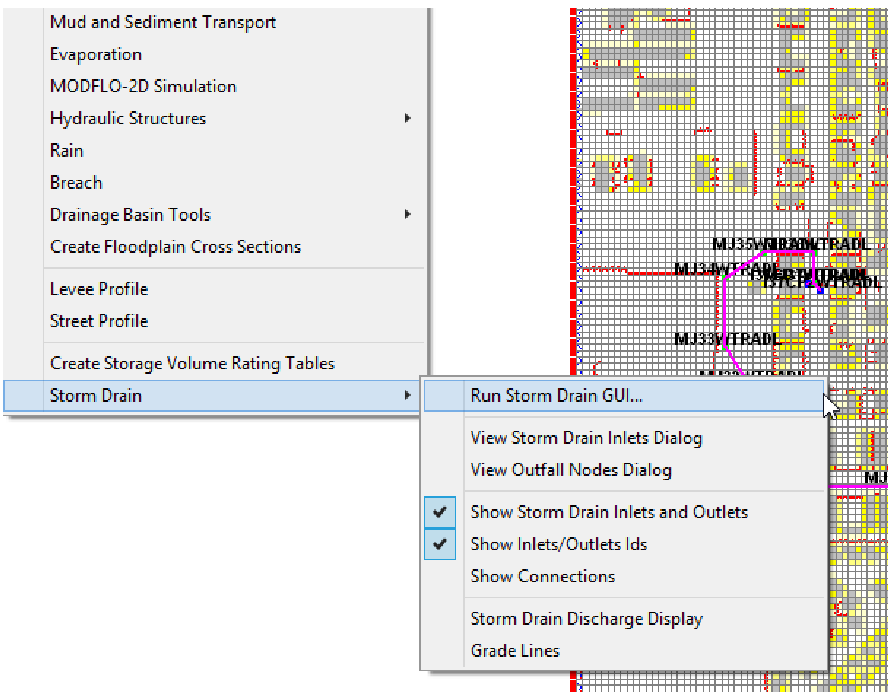


Figure 84. GDS Open Storm Drain GUI

Create SWMM.inp file

4. If there is an existing 'stand-alone' storm drain model for this project, the user can port it to the FLO-2D model using the previous Chapter 7 guidelines. To view the storm drain model in the SWMM GUI, find the SWMM.inp file in the FLO-2D project folder as shown in Figure 85.

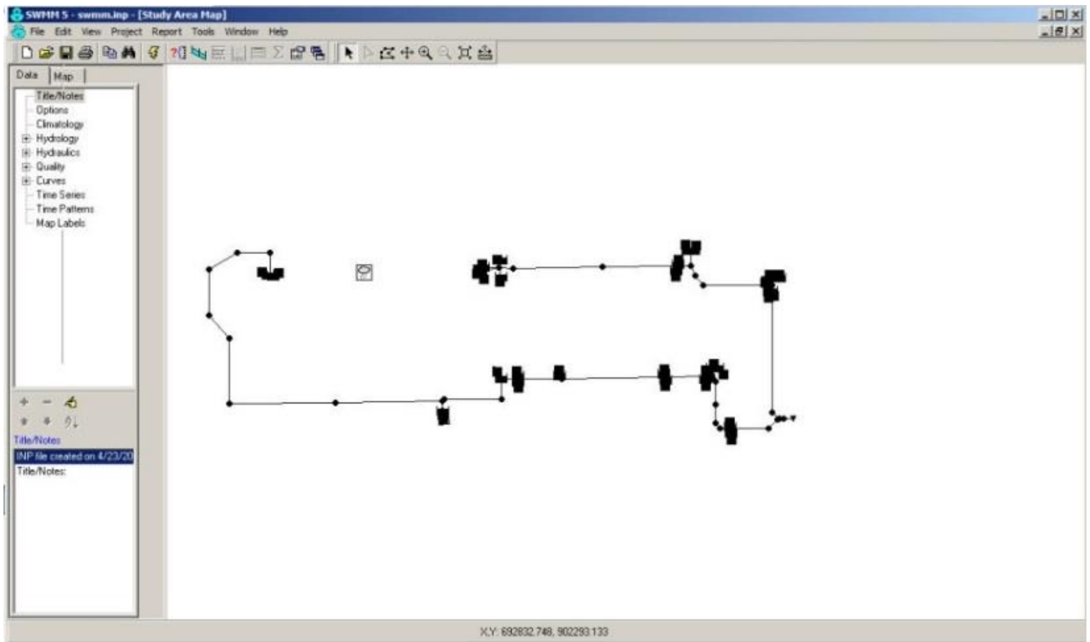


Figure 85. SWMM GUI

5. To create a new storm drain system, go to File|New| (Figure 86). Some of the instructions are discussed below.

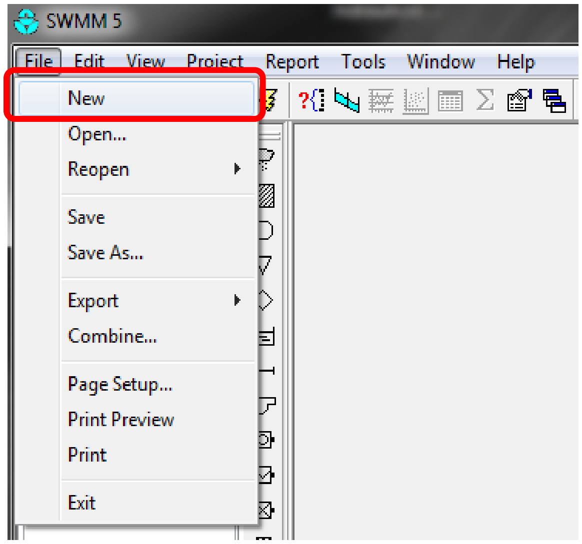


Figure 86. Open a New Project

6. Use the Junction Button to add junctions in the project workspace (Figure 87).

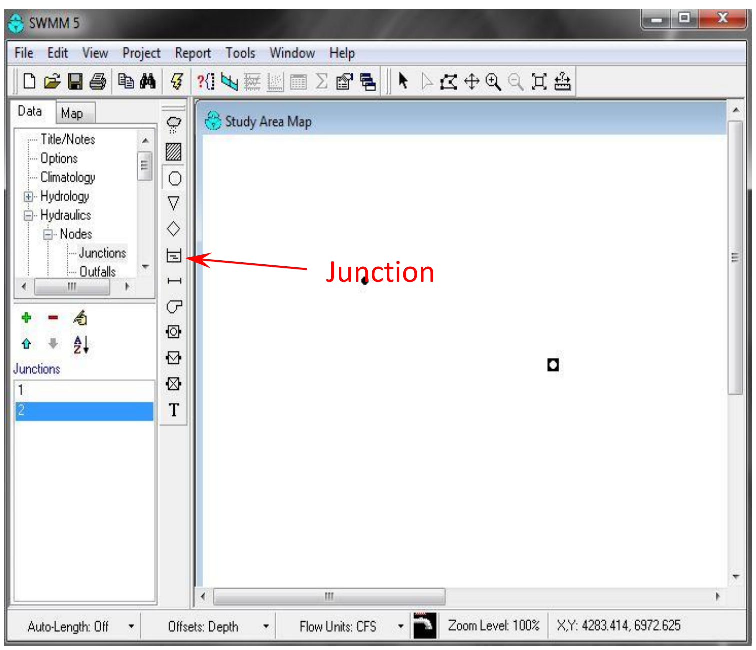


Figure 87. Assign a Junction

7. To add junction data, change the cursor to a pointer and double click on each junction (Figure 88).

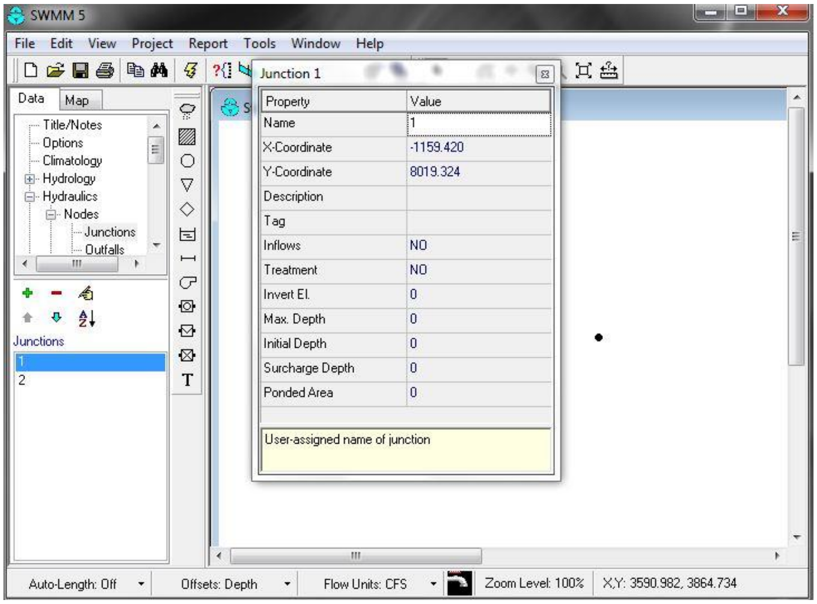


Figure 88. Junction Properties

The coordinate data for each storm drain component must be entered. Using a shapefile with the storm drain feature attributes is the preferred method, but the SWMM GUI does not have the option to import shapefiles. There are several options to enter this data:

- Import an image and click on the approximate location of the storm drain.
- Enter the coordinates manually in the Figure 88 dialog box.
- Use third party software such as Autodesk Storm and Sanitary Analysis, or Arc Map. An open source program (inpPINS) is available from Geospatial Software Lab that will generate the SWMM.inp file based on shapefiles. The links are:

  The open source GIS for software:

  http://www.mapwindow.org/downloads/index.php?show_details=62

  The free domain inp.PINS software:

  https://sites.google.com/site/inppins/

8. Select the Connect Junctions button to add a conduit (Figure 89).

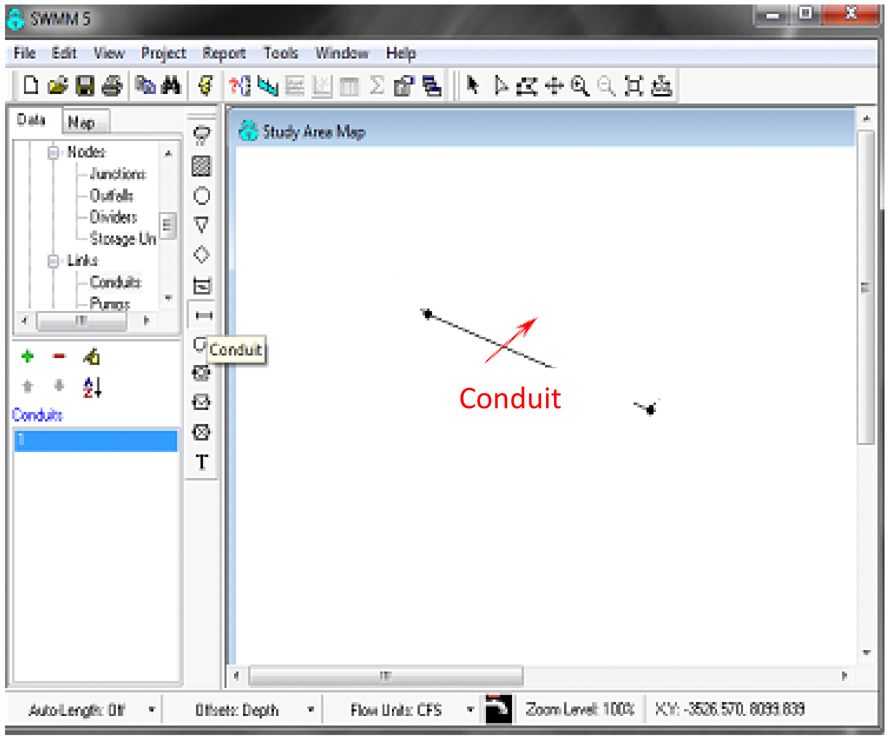


Figure 89. Create a Conduit

9. To add conduit data, change the cursor to pointer and double click on each conduit to open the data dialog box (Figure 90.).

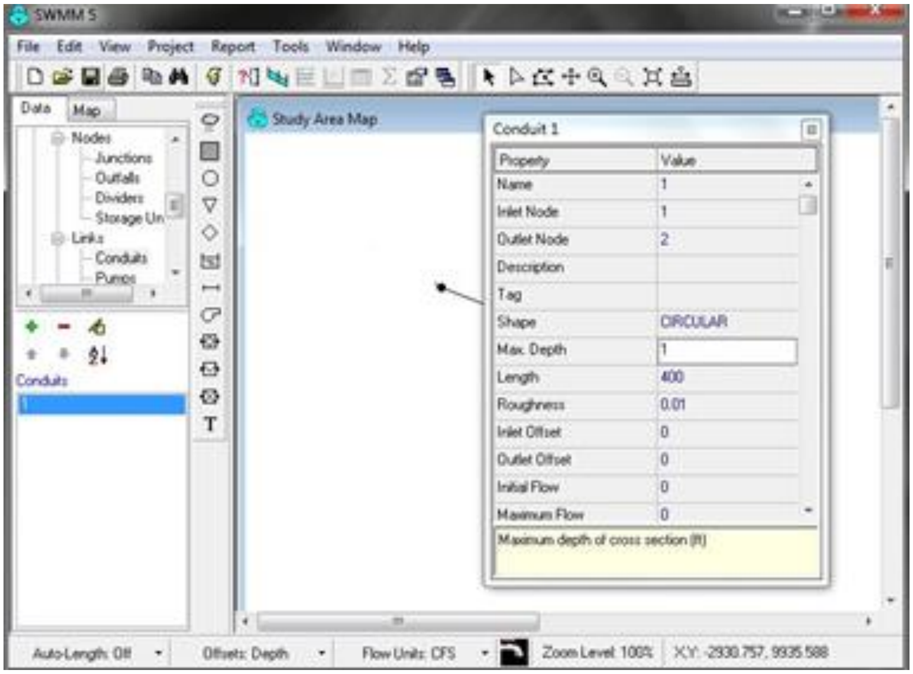


Figure 90. Edit the Conduit Data

It is recommended to keep the names of the various Storm Drain components simple and consistent with names such as I1, I2, I3, ... for inlets, J1, J2, J3, … for junctions. Use O = outlets, C = pipe conduits, etc.

10. The Simulation Options|Dates are setup automatically by the FLO-2D Storm Drain model using the SIMUL time assignment in the CONT.DAT file.

11. Go to Simulation Options|Dynamic Wave and set up the Force Main Equation by selection either Hazen-Williams or Darcy-Weisbach and save the project (Figure 91).

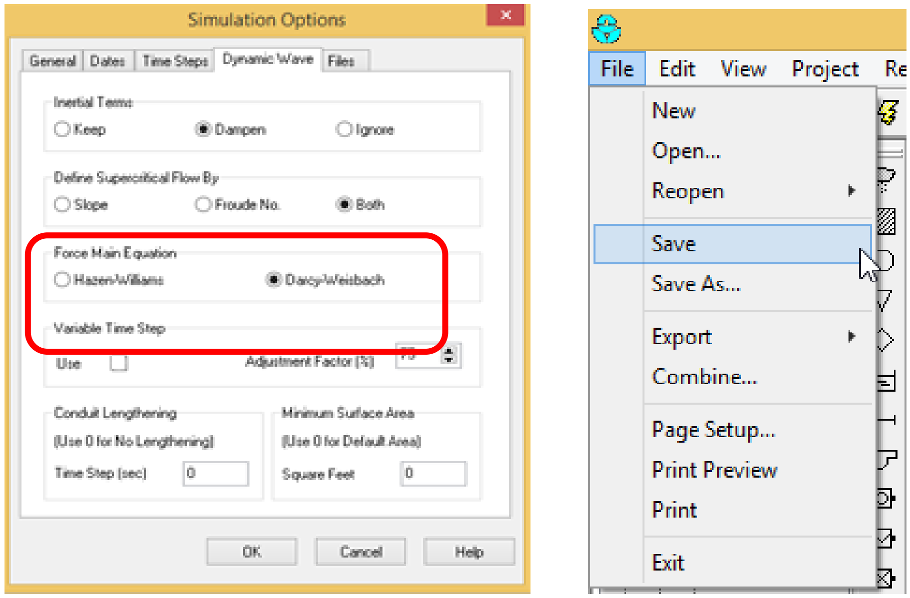


Figure 91. Simulation Options - Dynamic Wave

Create SWMMFLO.DAT file

12. Open the GDS FLO-2D inlet geometry dialog box (Figure 92.).

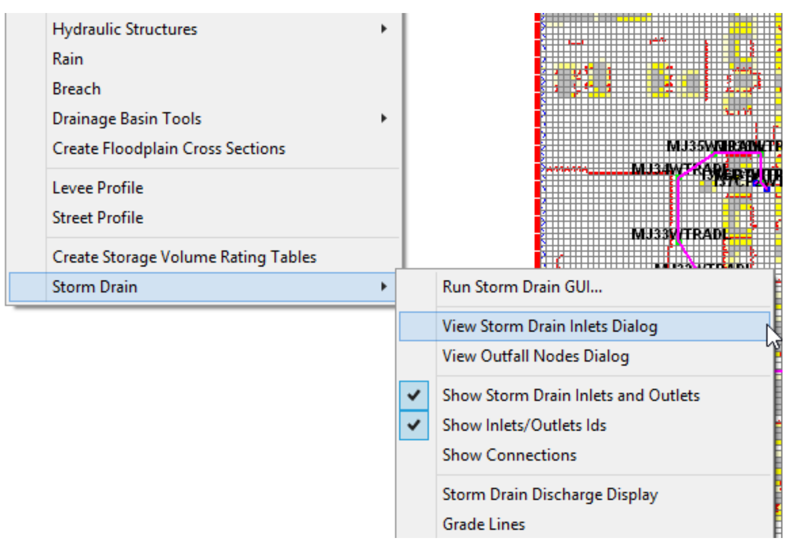


Figure 92. Storm Drain GDS Menus

13. Input the SWMMFLO.DAT file storm drain inlet geometry (Figure 93.).

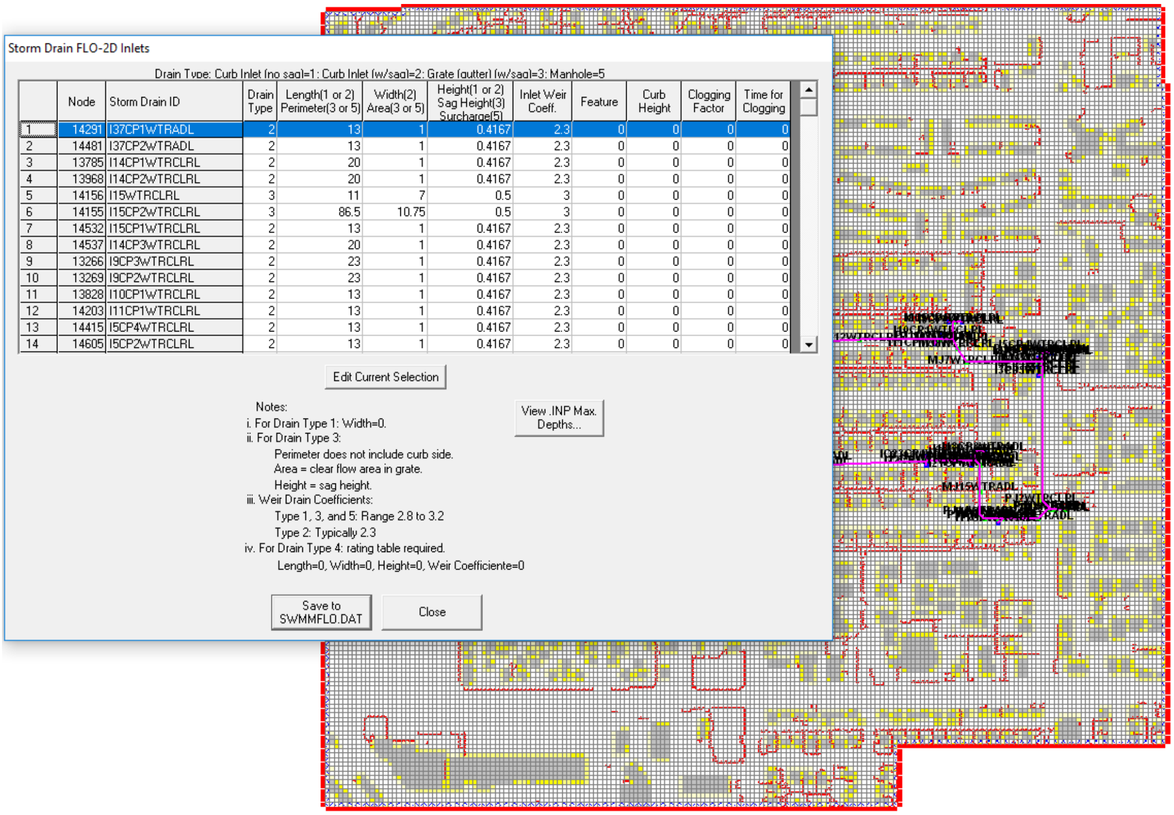


Figure 93. Storm Drain Inlet Dialog GDS

14. Inlets to channel grid elements must be assigned to left bank channel element listed in CHAN.DAT and the SWMMFLO.DAT file must be updated.
15. View the Storm Drain storm drain system in the GDS. Inlets are displayed as blue cells and junctions as green cells. Pipe are displayed as a magenta polyline.
16. The inlet inflow discharge and the flooding are reported in SWMMQIN.OUT.

Create SWMMOUTF.DAT file

17. View the Outfall nodes dialog to create the SWMMOUTF.DAT file (Figure 94.). This dialog reads the outfall nodes from the SWMM.inp file. The user has to select “Allow Discharge” to enable outfall discharge to the FLO-2D grid (Figure 95.). If that option is ‘off’, the outfall node will discharge flow off pipe system.

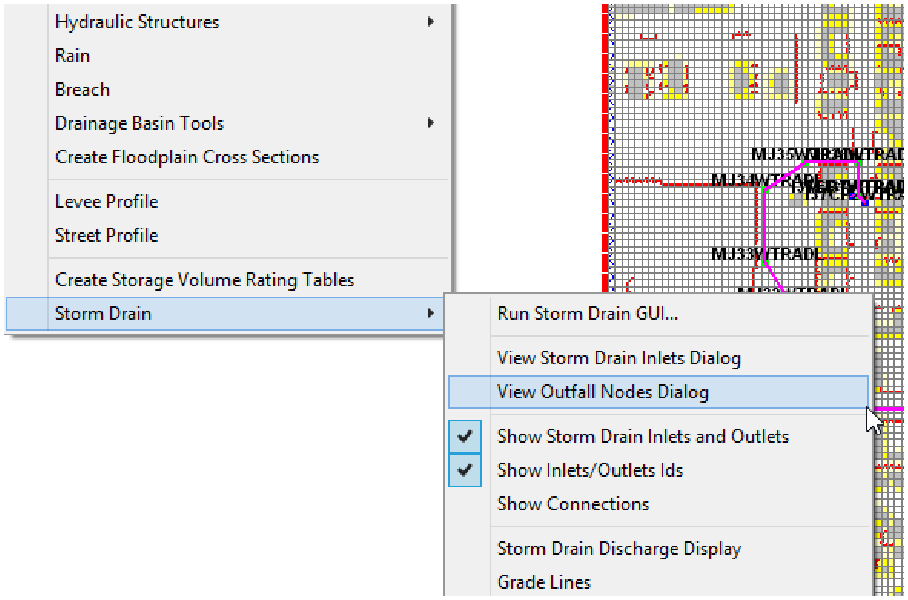


Figure 94. GDS Storm Drain Menu - View Dialog

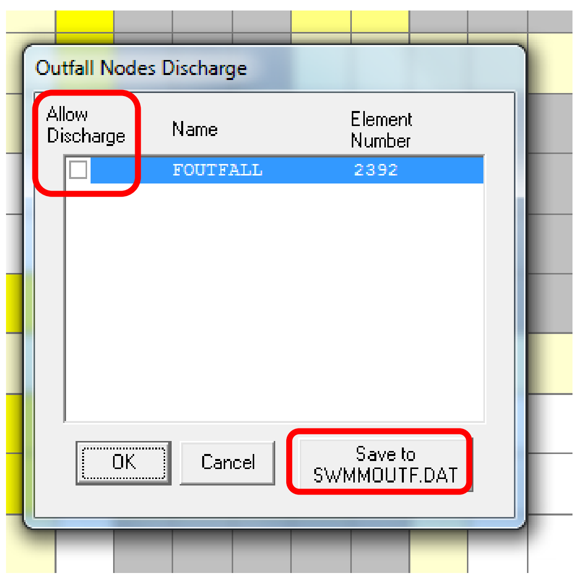


Figure 95. Outfall Node Discharge

18. Outfall Discharge to FLO-2D is reported in the SWMMOUTFIN.OUT file.

Run FLO-2D storm drain model

19. Run the project from GDS (Figure 96).

Figure 96. FLO-2D Control Variables.

Review FLO-2D storm drain model results

20. The SWMM results are reported in the SWMM.RPT file (ASCII) and SWMM.OUT file (Binary). Results can be used to create plots in the GDS and in the SWMM GUI.

Chapter 9

Verification

Verification tests have been prepared to demonstrate that the FLO-2D storm drain component correctly routes the conduit flow. Model verification is an essential process in the evolution of a numerical model. Verification testing is designed to ensure that the governing equations and solution algorithms provide accurate results. One of the products of verification tests is error identification and propagation. Upon implementing corrections, retesting is required to ensure code integrity. This testing program provides a better understanding of the model's capabilities, limitations, and appropriateness for a range of applications. A large number of simplified storm drain tests were prepared and compared with analog results. Six of these tests are discussed.

Case 1 - Weir and Orifice Flow

FLO-2D model computes storm drain inlet discharge using the weir and orifice equations and assigned geometry parameters. The simplified test has two inlets and one outfall. The topography drains to the inlets IN1 (inlet at the left side) and MH1 (inlet in the middle of the grid), which are located at the lowest elevations in the grid system (see Figure 97.).

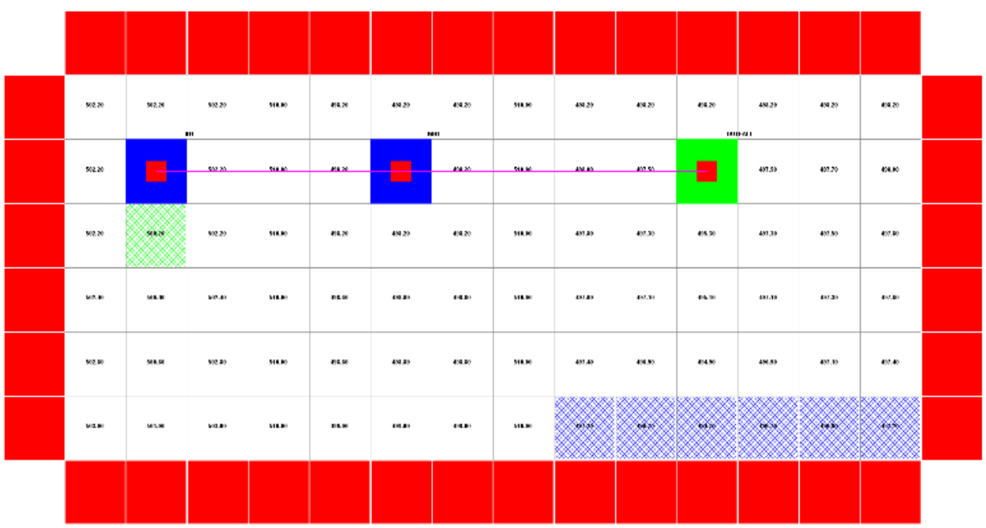


Figure 97. Grid for Verification Case 1.

There are three separate basins divided by a high elevation ridge. The basin containing the outfall has drainage away from the outfall toward the grid element outflow nodes (). The storm drain profile is shown in Figure 99.

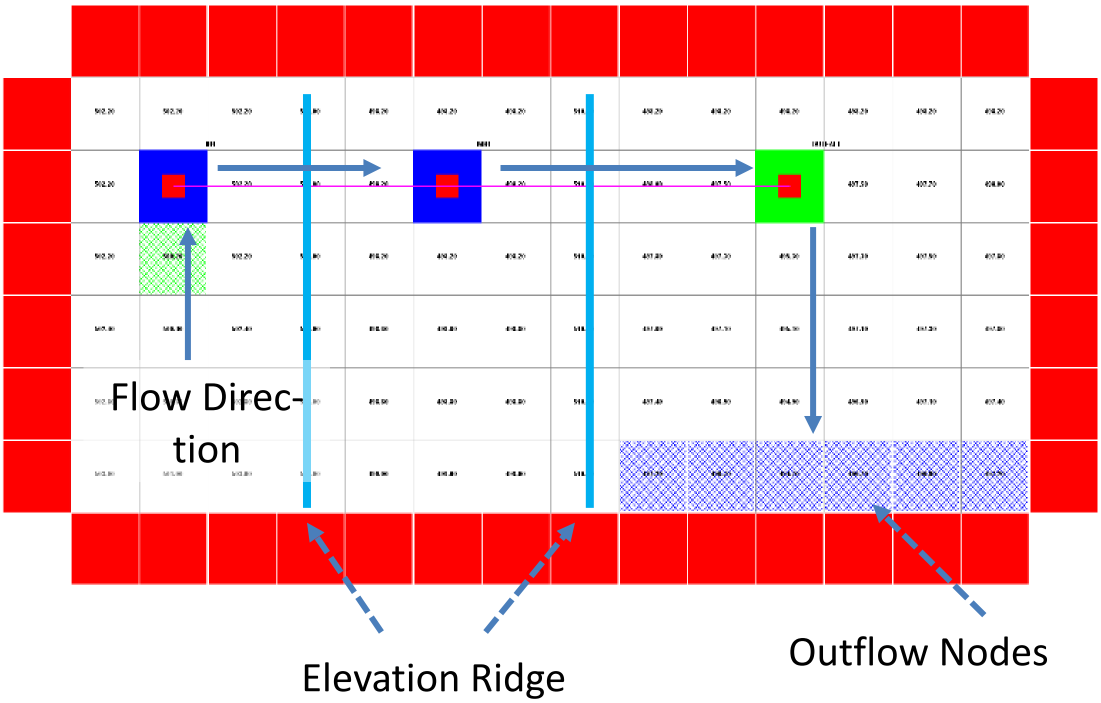


Figure 98. Separate Basins for the Verification Case 1.

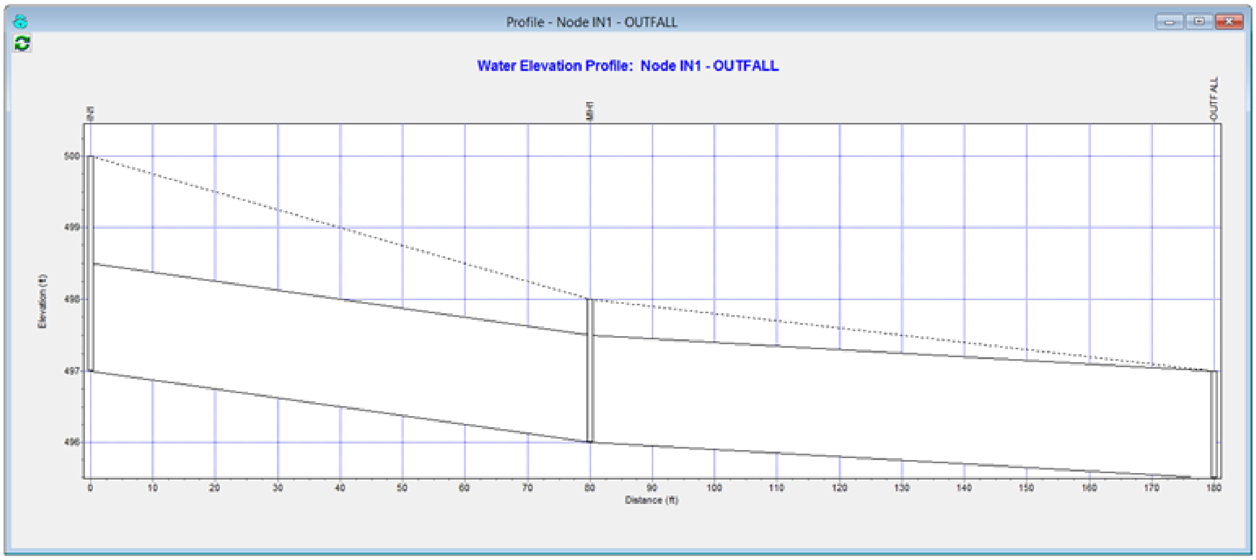


Figure 99. Storm Drain Profile for the Verification Case 1.

For inlet types 1 to 3 and 5, FLO-2D computes the discharge based the flow depth using either the orifice or weir equations. In this verification test a Type 1 inlet was applied as shown in Figure 100. .

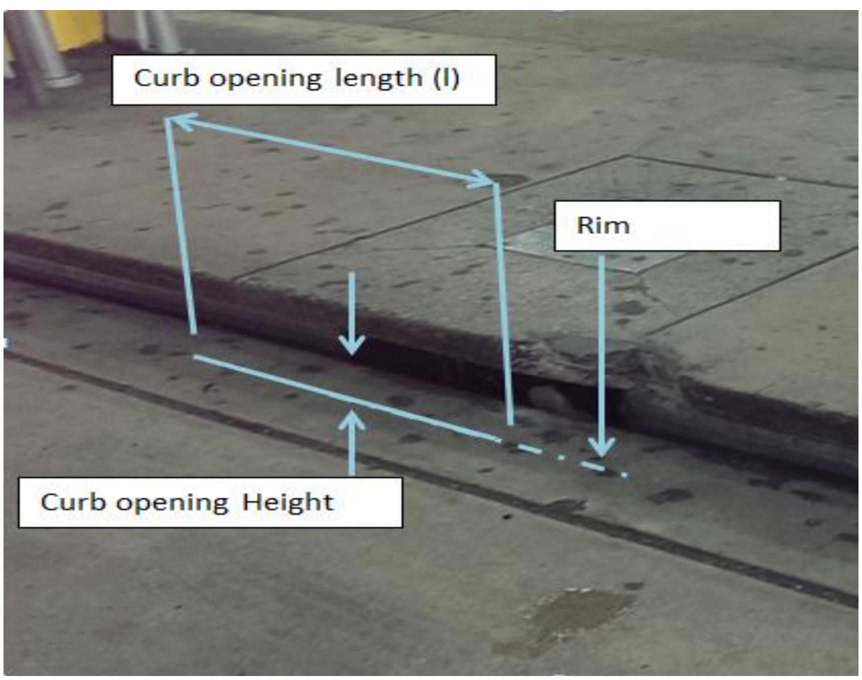


Figure 100. Storm Drain Inlet Configuration.

The input parameters are:

- Weir coefficient: 3.0
- Curb opening length (l): 10ft
- Curb opening height (h): 0.5ft
- Weir Case: 2.0 cfs pulse (0.5hr)
- Orifice Case: 16 cfs pulse (0.5hr)

Table 19 presents the analog discharges calculated from the orifice and weir equations for the given floodplain depth. These are compared with the flows calculated by the model.

Table 19. Comparison of Discharges

Inlet	Depth (ft)	Analog Q (cfs)	Model Q (cfs)
Weir	0.1647	2.005	2.003
Orifice	3.6982	51.70	51.75

Figure 101.   and Figure 102 show the comparison between discharges reported by the SWMM.RPT file and the SWMMQIN.OUT file inlet behaving as a weir and orifice respectively.

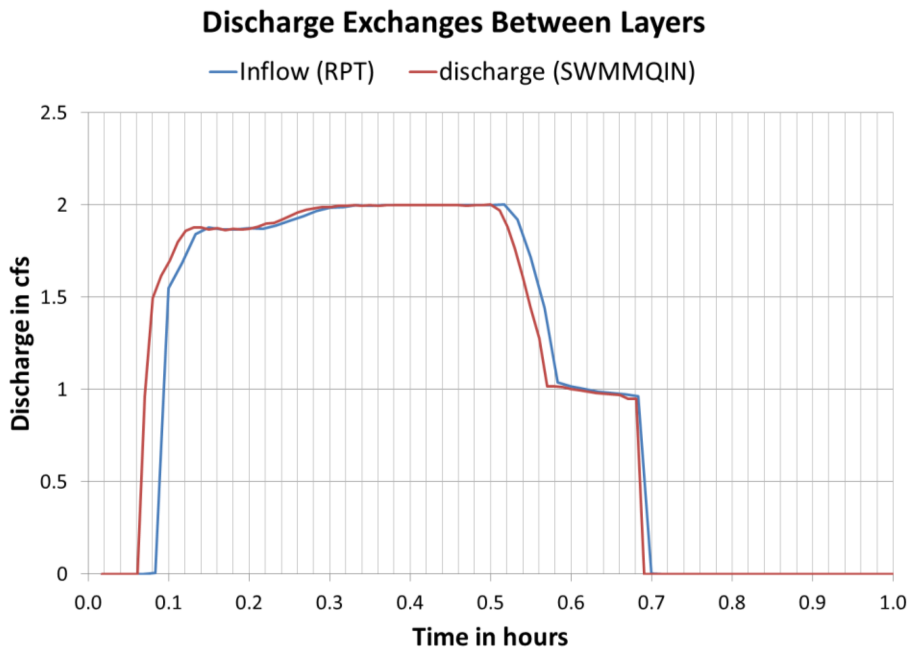


Figure 101. Discharge Exchange between Layers for the Inlet Behaving as a Weir.

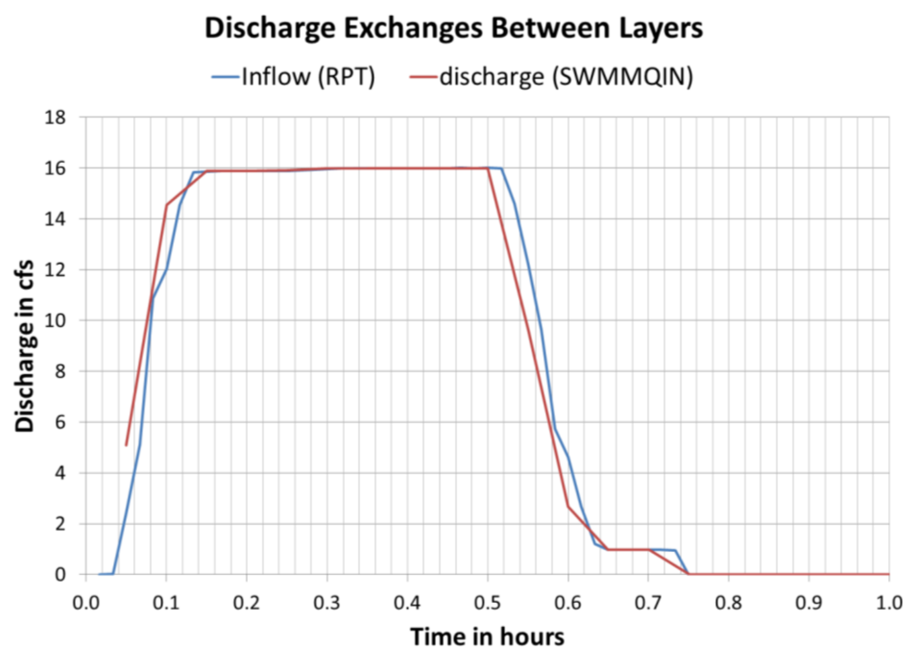


Figure 102. Discharge Exchange between Layers for the Inlet Behaving as an Orifice.

In the above figures, the discharges computed and reported by the FLO-2D model as entering the storm drain system correctly compare with those reported by the SWMM.rpt file. The discharge being transferred between the FLO-2D surface layer and the storm drain system is consistent.

Case 2 - Type 4 Inlet Rating Table

The FLO-2D model uses a rating table for inlet Type 4 as shown in Table 20.

Table 20. Rating Table Imposed for Inlet Type 4.

Depth (ft)	Discharge (cfs)
0	0
0.1	1
0.5	2
1	4
2	8
3	16
4	32
5	33

The horizontal inlet inflow consists of a pulse of 10 cfs for 0.5 hrs. Figure 103. and Figure 104 present the comparison of depths and discharge for the Type 4 inlet.

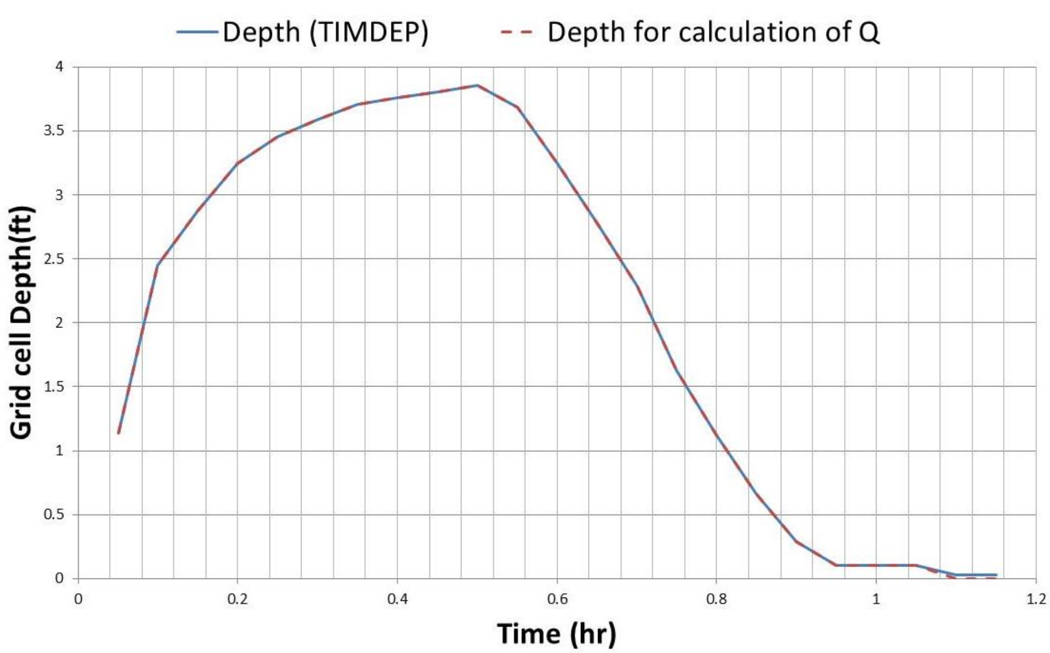
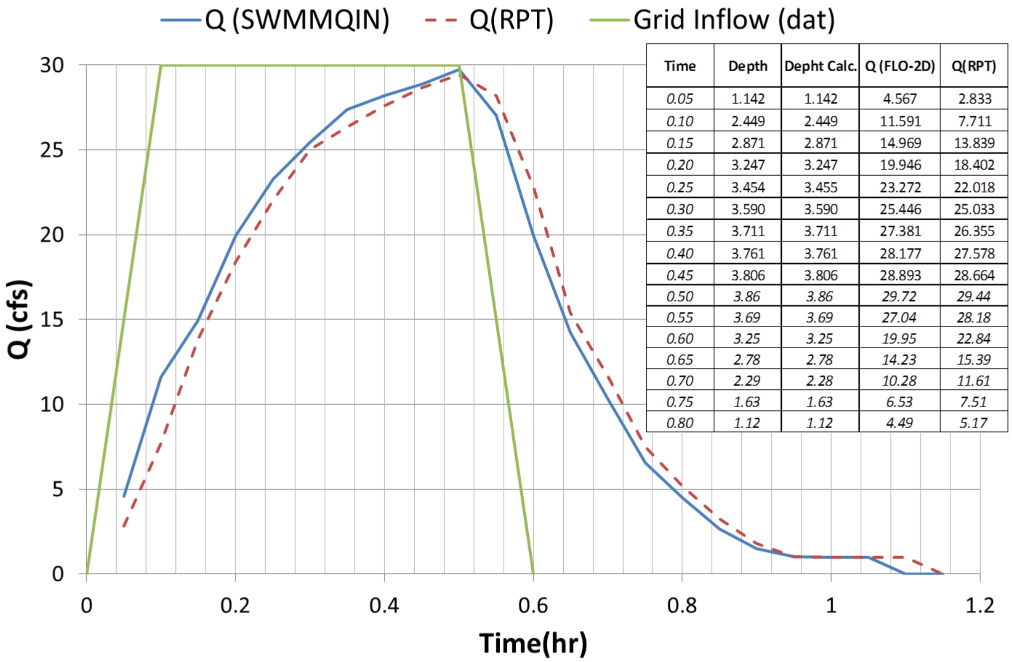


Figure 103. Reported Depth vs Depth used for Calculation of Q at the Inlet.



Time	Depth	Depht Calc.	Q (FLO-2D)	Q(RPT)
0.05	1.142	1.142	4.567	2.833
0.10	2.449	2.449	11.591	7.711
0.15	2.871	2.871	14.969	13.839
0.20	3.247	3.247	19.946	18.402
0.25	3.454	3.455	23.272	22.018
0.30	3.590	3.590	25.446	25.033
0.35	3.711	3.711	27.381	26.355
0.40	3.761	3.761	28.177	27.578
0.45	3.806	3.806	28.893	28.664
0.50	3.86	3.86	29.72	29.44
0.55	3.69	3.69	27.04	28.18
0.60	3.25	3.25	19.95	22.84
0.65	2.78	2.78	14.23	15.39
0.70	2.29	2.28	10.28	11.61
0.75	1.63	1.63	6.53	7.51
0.80	1.12	1.12	4.49	5.17

Figure 104. Comparison of Discharges at the Type 4 Inlet.

Figure 104. displays a verification that the rating table for the Type 4 inlet is being correctly read and assigned as the inlet inflow.

Case 3 - Return Flows from the Storm Drain to the Surface Water

Storm drain inlets under pressure may return flow to the surface water (Figure 105). In this case, the Type 1 inlet is the only connection between the layers, so the return flow can be isolated. There is a storm drain inflow for a short period of time as a time series in the *.inp file.

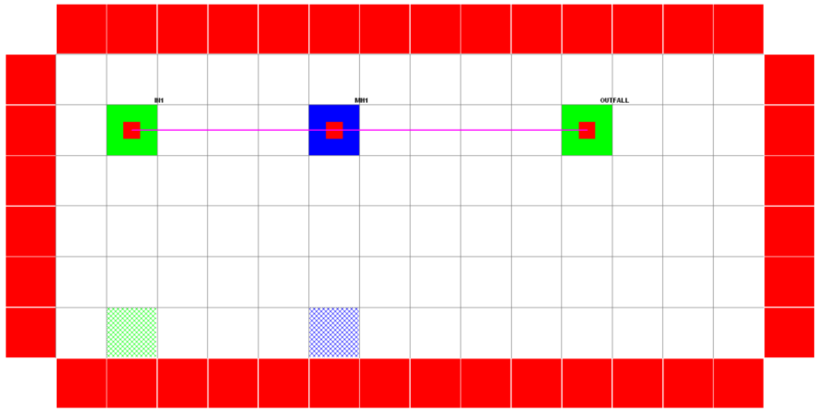


Figure 105. Grid for the Verification Case 3.

Figure 106 The comparison of return flow at the inlet reported by the SWMM.rpt file and the SWMMQIN.OUT file is shown in Figure 101. The outfall discharge extracted from the SWMM.rpt file is plotted.

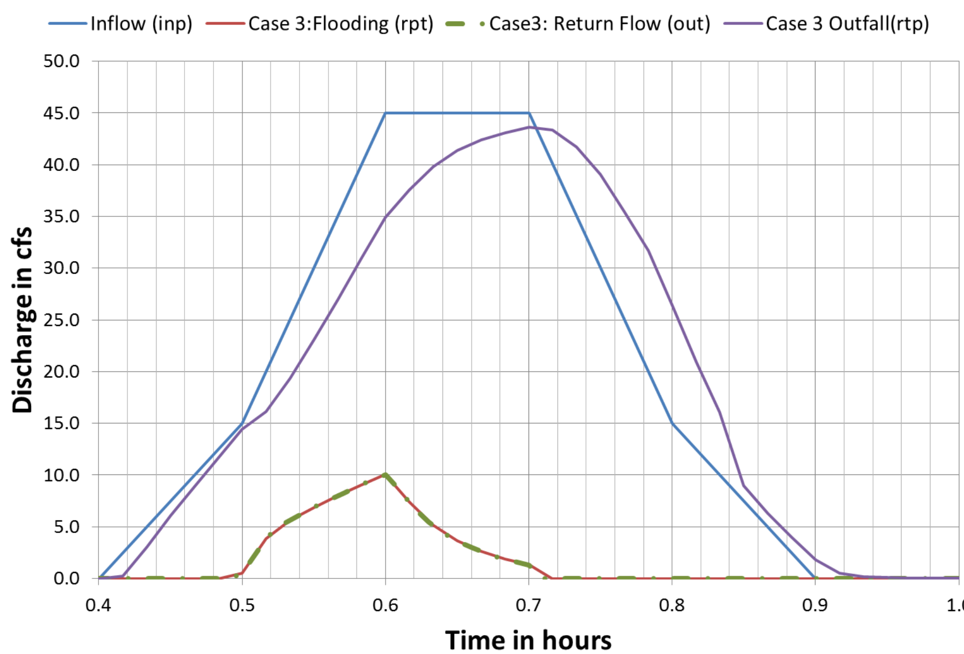


Figure 106. Case 3 Discharge Passes Through the Inlet as Return Inflow.

This verification test supports the conclusion that when the capacity is exceeded in the storm drain, the volume that leaves the system is correctly being transferred to the surface layer.

Case 4 - Manhole Cover Popping and Type 3 Inlet

In this verification test a Type 5 inlet manhole cover pop under pressure and then acts as a Type 3 inlet (Figure 102). The manhole is set up to have a 5 ft surcharge (980lb manhole cover.) The manhole inlet dimensions are:

Perimeter: 6.28ft

Area: 3.14 ft^2

Surcharge: 5ft

Weir Coefficient: 3.0

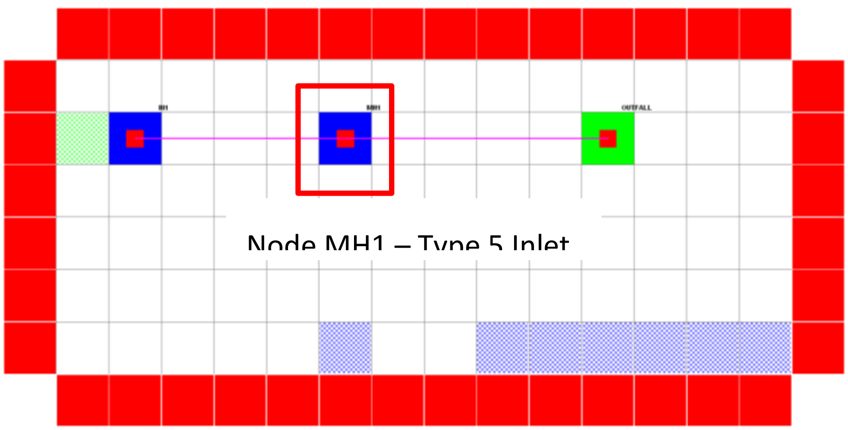


Figure 107. Grid for the Verification Case 4.

The results are shown in Figure 108 and Figure 109. Figure 108 illustrates that the manhole cover pops off and Figure 109 demonstrates that the manhole functions as a Type 3 inlet after the manhole cover is removed.

Pressure Head > Inv Elev + Max Depth + Surcharge Depth: Manhole cover pops off

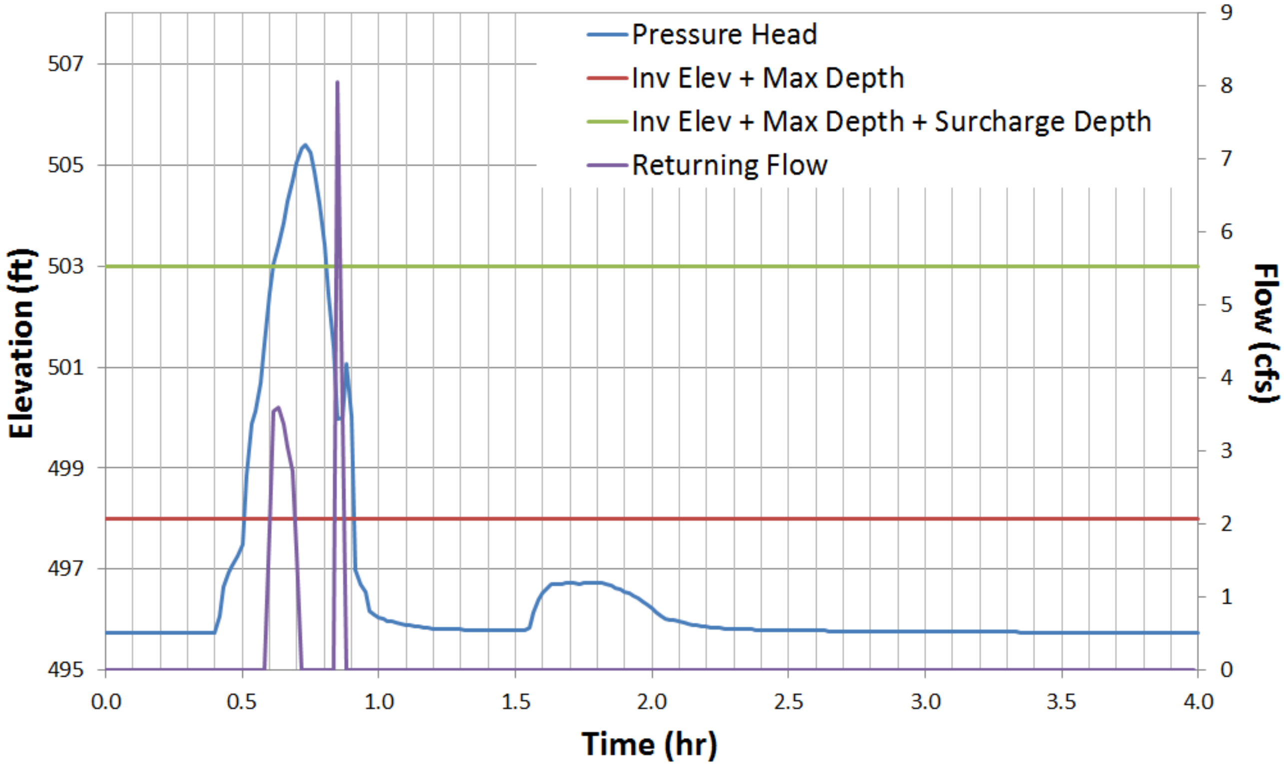


Figure 108. Verification Case 4 shows Manhole Popping.

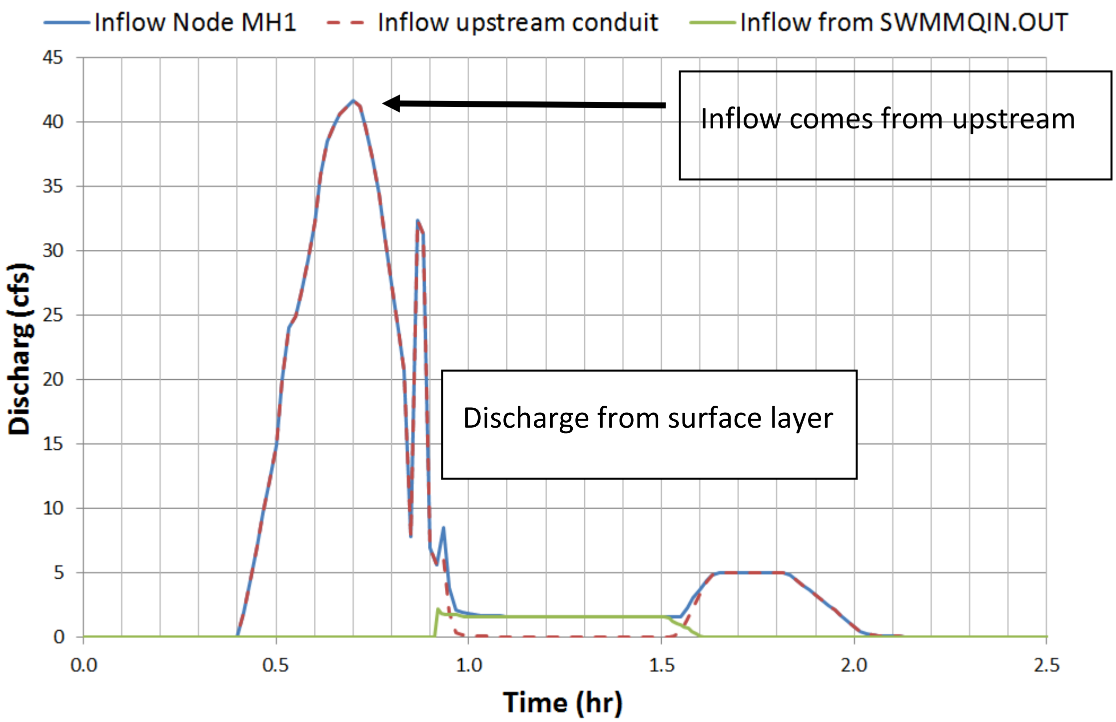


Figure 109. Manhole Behaves as a Type 3 Inlet Allowing Inflow Flow from the Surface Layer.

Case 5 - Outfall Discharge

In this test the storm drain outfall discharges to the surface water. The data base is the same as that used for the Case 1 test. The inflow is a pulse of 12 cfs for 0.5 hr. The following figures show that the discharge is being transferred to the grid element containing the outfall. Figure 110 compares the outfall water surface elevation in the SWMM.rpt file with the water surface reported by the FLO-2D model. Figure 111 compares the reported discharges by the two model output files showing an exact match.

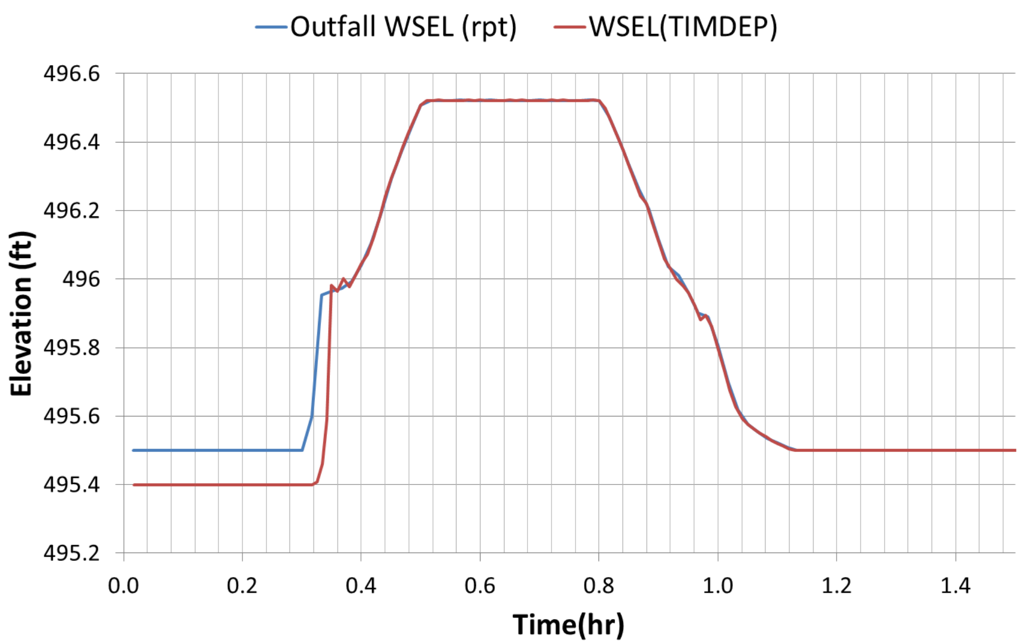


Figure 110. Comparison of WSE at the Outlet Node and at the Grid Element.

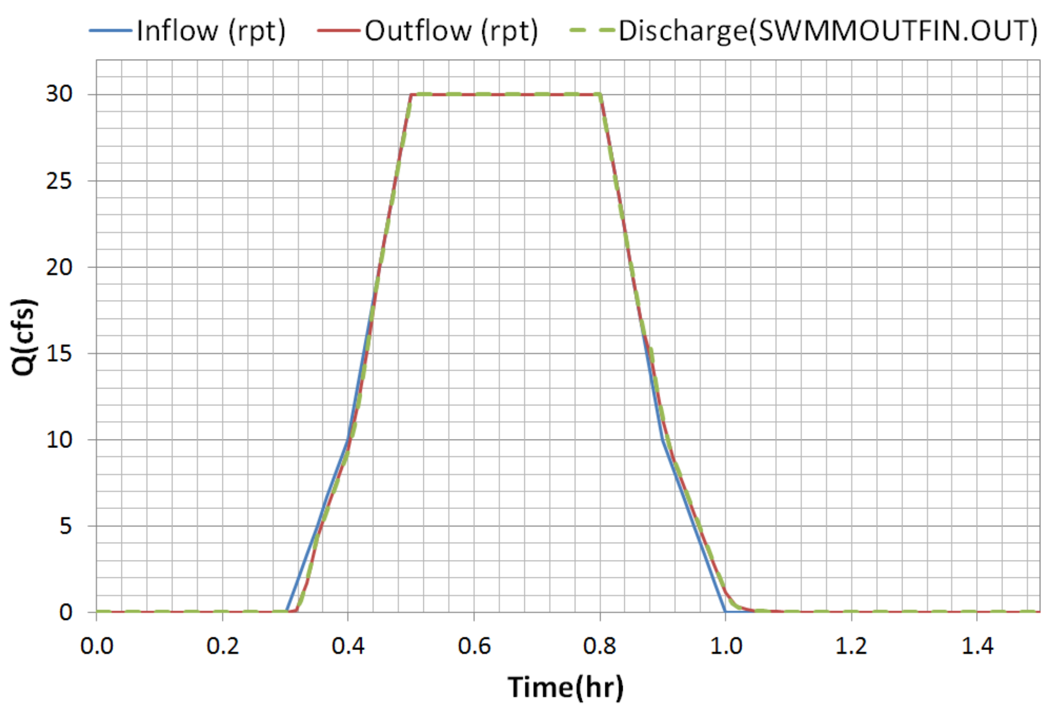


Figure 111. Discharge Outfall Node (SWMM.rpt) and SWMMOUTFIN.OUT.

This test verifies that a storm drain outfall can discharge volume to the surface layer and that the WSE is being correctly calculated for an outfall node.

Case 6 - Outfall to a Natural Channel

This verification test represents a complex storm drain configuration with multiple inlets, junctions and outfalls all interacting (Figure 112). The outfalls discharge to a natural channel. One of the outfalls is setup to be underground. The invert is below the channel bed elevation and the channel water surface constitutes a boundary condition (Figure 113).

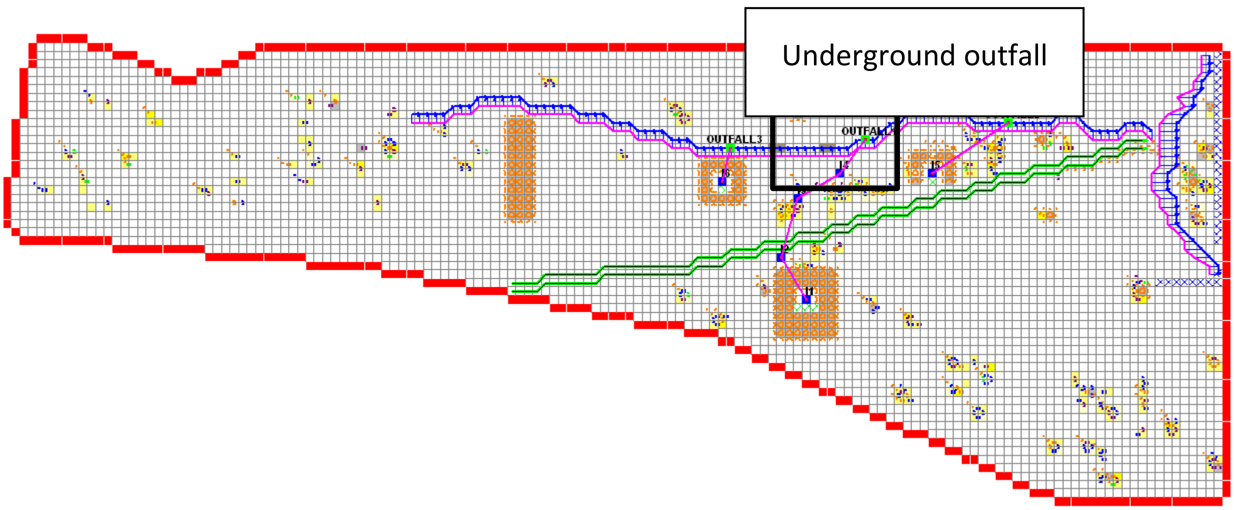


Figure 112. FLO-2D Model for Verification Case 6.

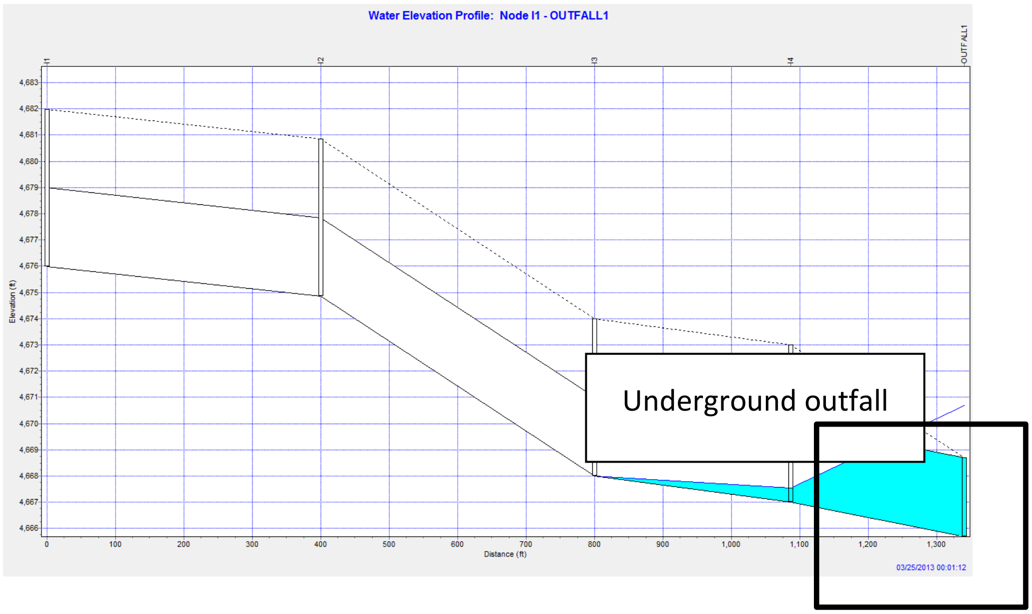


Figure 113. Channel Water Surface Elevation is a Boundary Condition on the Outfall.

Figure 114 indicates that the outflow pipe results oscillate with the upstream and downstream head variation. This verification test demonstrates that an underground outfall can be simulated and that the WSE from the surface layer is correctly imposed as an outfall boundary condition.

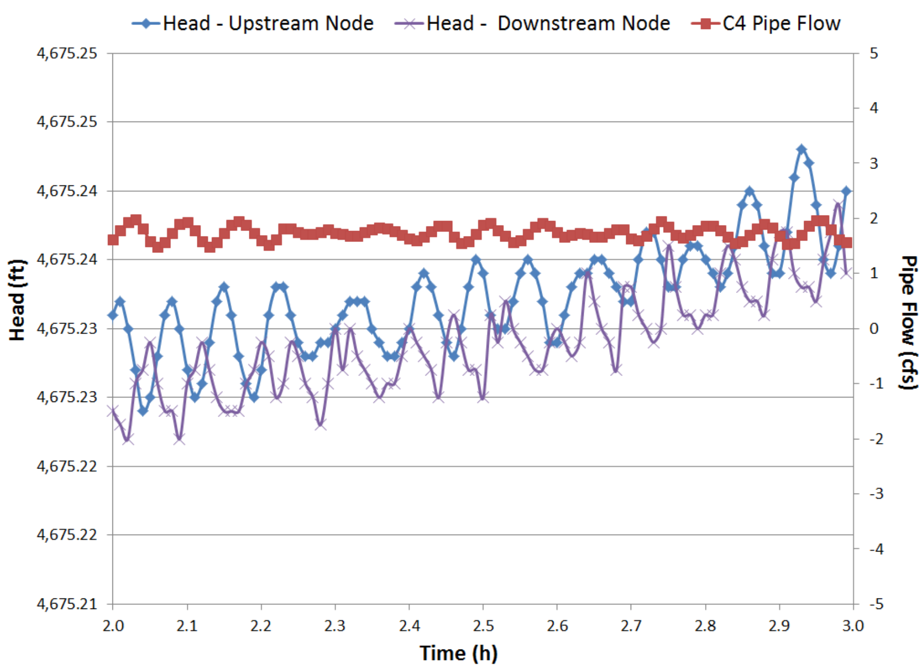


Figure 114. Head Variations Introduce Pipe Flow Oscillations.

References

FLO-2D Software, Inc. FLO-2D Reference Manual, Nutrioso, AZ.

Flood Control District of Maricopa County, 2018. Drainage Policy and Standards for Maricopa County, Arizona. 6th Edition, Phoenix, Arizona.

Johnson, F.L., and F.M. Fred, 1984. Drainage of Highway Pavements (HEC12), Publication FHWA-TS-84-202, Centerville, Virginia.

Rossman, L.A., 2005. "Storm Water Management Model User's Manual Version 5.0", EPA/600/R-05/040, National Risk Management Research Laboratory, U.S. Environmental Protection Agency, Cincinnati, OH.

Rossman, L.A., 2006. "Storm Water Management Model Quality Assurance Report, Dynamic Wave Flow Routing", EPA/600/R-06/097.

Rossman, L.A., 2007. "Storm Water Management Model User's Manual", EPA/600/R-05/040, U.S. Environmental Protection Agency, Cincinnati, OH.

Roesner, L.A., J.A. Aldrich, and R.E. Dickinson, 1992. "Storm Water Management Model User's Manual Version 4: Extraneous Addendum", EPA/600/3-88/001b, Environmental Research Laboratory, U.S. Environmental Protection Agency, Athens, GA.

US Dept. of Transportation, 2013, "Urban Drainage Design Manual HEC-22", Federal Highway Administration, Washington, D.C.