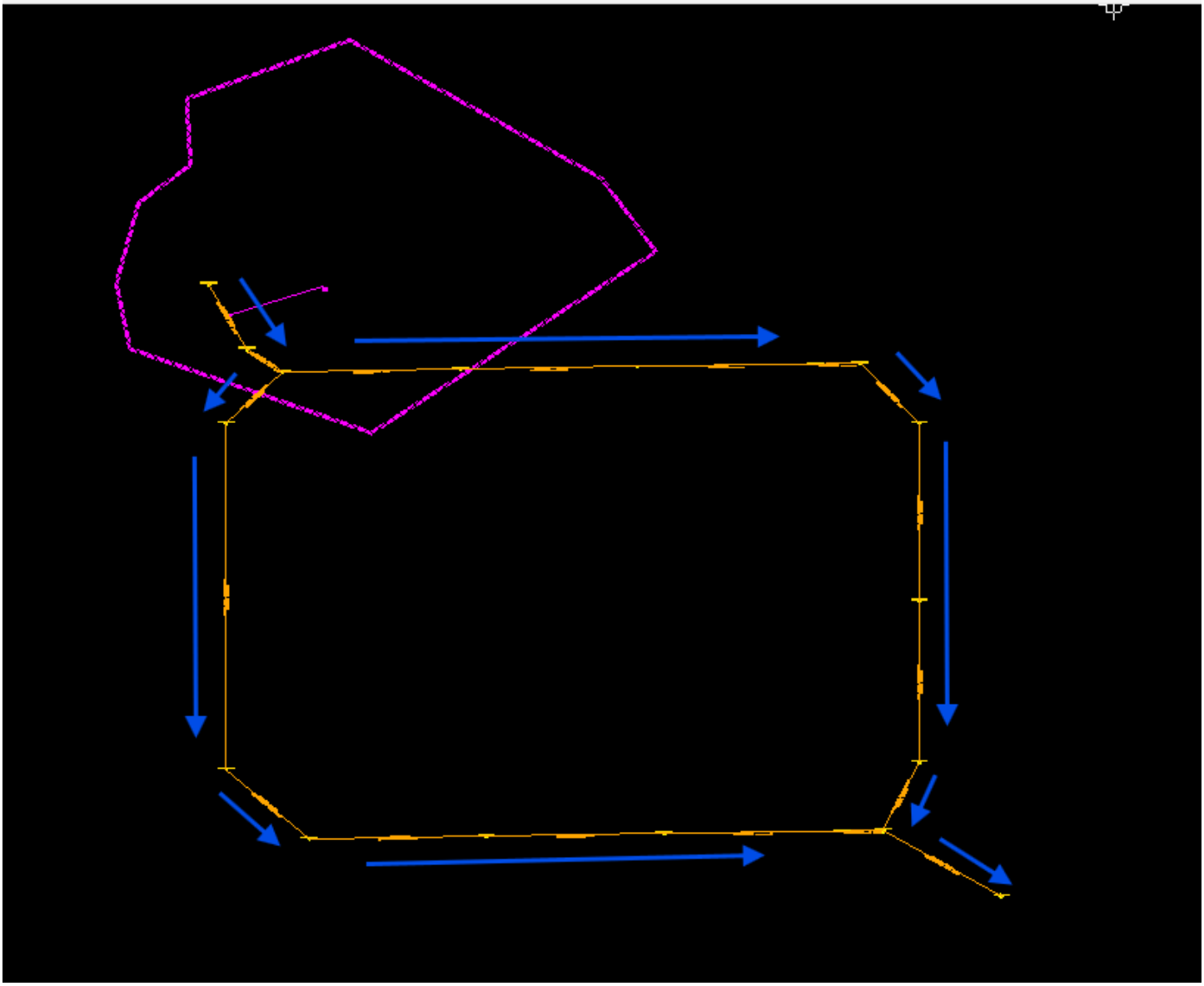
How to design a loop in a drain network

Drain · https://mes100.com/docs/how-to-design-a-loop-in-a-drain-network/

While performing drainage **design** analysis, we may encounter a verification failure stating, '*Drainage network loop not allowed*', as shown in the image below.



This failure occurred due to a drain flow design that formed a closed loop, a configuration that must be resolved for the software to proceed with the analysis.



Reality on the ground

To properly analyze a drainage system with loops, the *steady-state method*, *combined with a simple drain capacity check*, is insufficient. The steady-state equation was developed with the assumption that the drains are connected, "*in-line*".

Therefore, the Design Mode that MiTS has does not make it possible to analyze the looping drainage system. The Design Mode emphasizes ***design***, namely, to determine the drain dimensions, IL quickly; therefore, a steady-state equation is used for this purpose (any other equations more complicated than this one will just make the software unbearably slow). Not only that, but the steady-state

equation also requires **users to manually apportion the flow proportions at junctions** to estimate how the water moves through the system.

To properly obtain the hydraulic characteristics- such as velocity, water depth on a time-by-time basis- of the drainage system, you will need to use the Analysis mode- specifically, the dynamic routing option - in MiTS 3. Dynamic routing will be able to calculate the hydraulic characteristics correctly and **automatically handle the flow distributions**. The drawback is that, because it's computationally intensive, we can't use it to determine the drain dimensions, IL, and other parameters.

The realistic way to go for analyzing a looping drainage system properly is to

1. Use the **Design Mode** to first size up the drain information. Even without the Analysis mode, the result from the Design Mode can still provide an approximation and a preliminary design of the hydraulic characteristics of the drainage system.
2. Only then, use the **Analysis Mode** to calculate the actual hydraulic characteristics.

NOTE:

For users of MiTS 2 versions or those who wish to proceed with analysis using **the current steady-state method**, you may refer directly to the **Design Mode Section**.

For **users of MiTS 3** who are using the Dynamic Wave routing method to **compute time-varying analysis**, you may refer directly to the **Analysis Mode Section**.

Design Mode

Limitations

The Design Mode engine has a limitation when it comes to the computation of in-loop drain connections. To successfully analyze such a system in the Design Mode, break the drain loop by inserting an additional note on the loop to make it sequential.

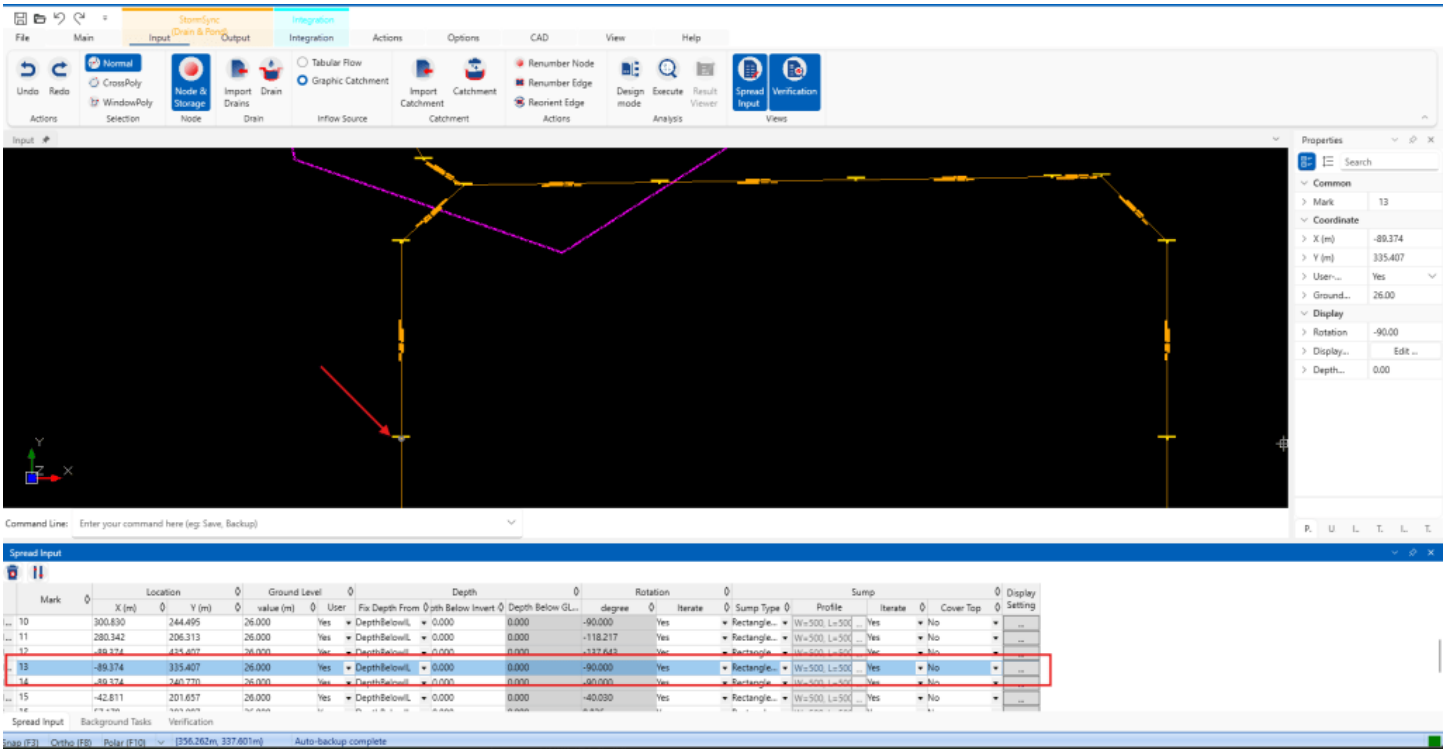
Sample Project File: **Here**

Step 1: Breaking the drain connection

1. Select a breaking point in the loop

Choose a suitable node (sump) within the network to be the breaking point.

Based on the provided sample project file, Node 13 is selected for this purpose.



2. Reposition of the node

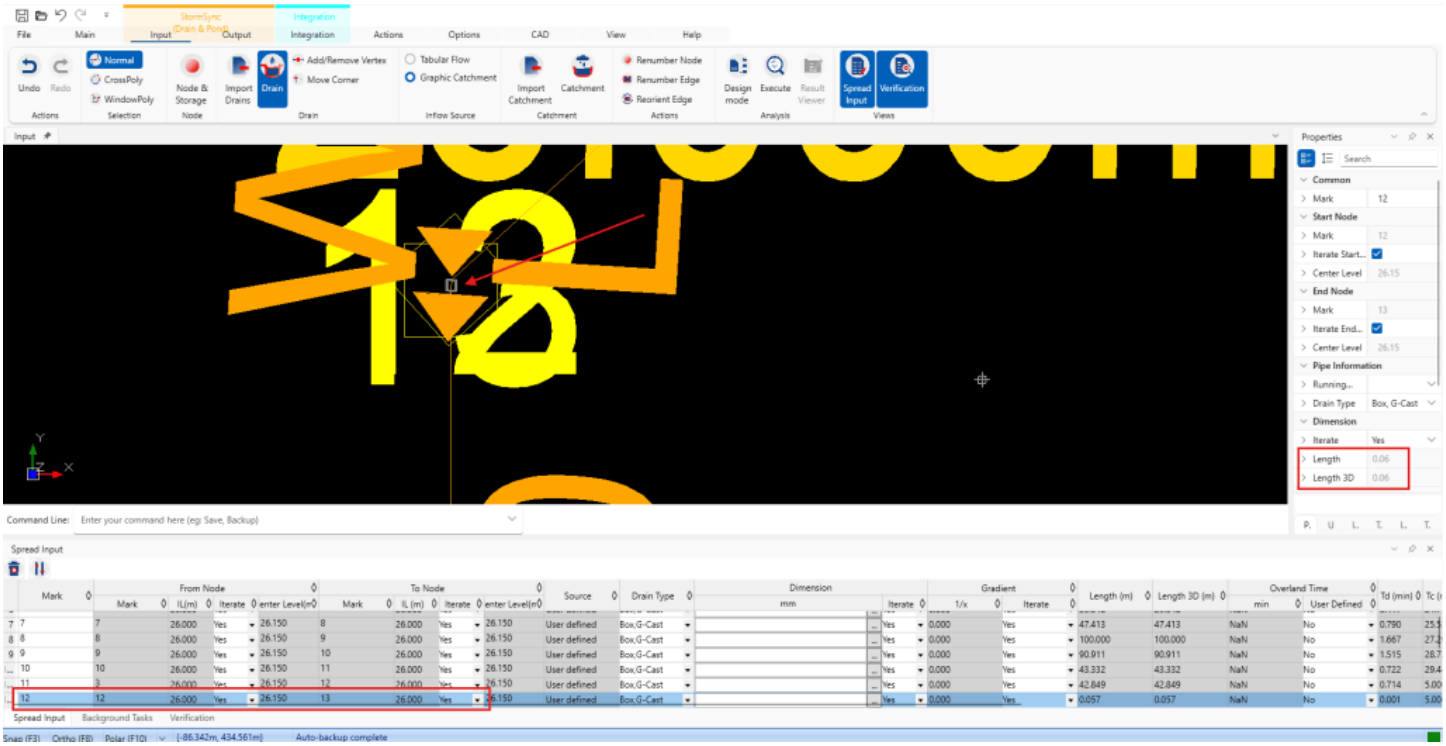
Move Node 13 to be as close as possible to Node 12, effectively placing them "on top" of one another.

Click the Node & Storage button at the top ribbon > Select Node 12

Embedded media:
https://www.youtube.com/embed/T7VIU_Inb4w?version=3&rel=1&showsearch=0&showinfo=1&iv_load_policy=1&fs=1&hl=en-US&autohide=2&wmode=transparent&enablejsapi=1

3. Removing the in-loop connection

Delete the drain that connects Node 12 to Node 13; The deleted drain length should be as short as possible, as in the image shown below.



Step 2: Analysis and level adjustment

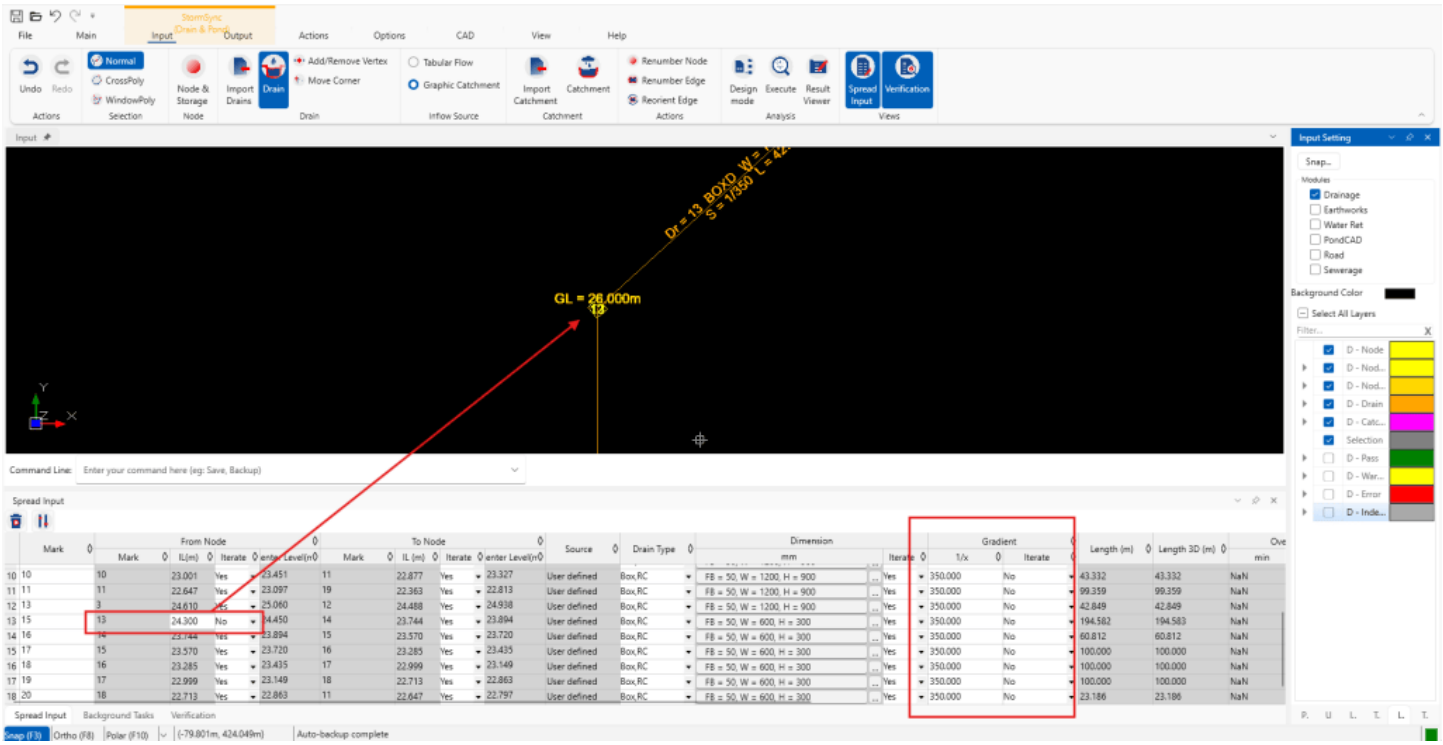
Once the in-loop connection is broken, the software is now able to successfully run the analysis in **Design Mode**. This allows the engine to provide iterated information such as **Invert Levels, Gradients,** and **Drain Dimensions**.

1. Adjusting the Invert Levels

After the analysis, we must manually adjust the Invert Levels at the breaking point (Sump 13) to ensure network continuity that reflects actual on-site construction.

In the sample project file, the Invert Level at Node 13 is fixed at 24.3m, which is slightly lower than the levels at Node 12.

You may also need to adjust the gradients of the network to ensure successful drain analysis.



2. Identifying the previous drain flow

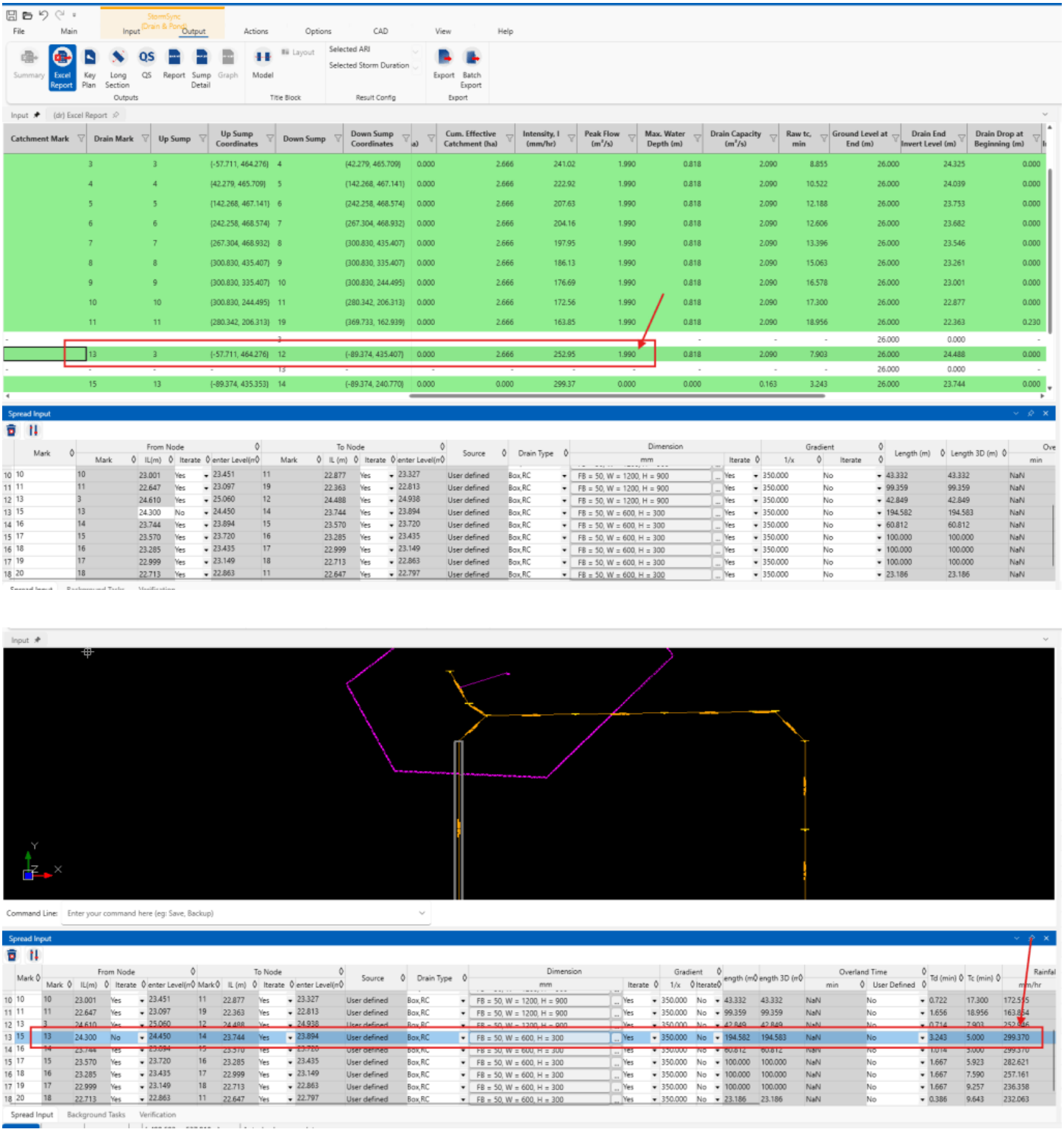
This initial analysis of the drain provides the flow data at Node 12, which is necessary for the next step of replicating the drain flow continuation.

Step 3: Replicating the previous drain flow

To maintain the hydraulic accuracy of the system after breaking the loop, we must replicate the flow from the previous drain (Flow from Drain 11), which is achievable by *applying catchment* for Drain 13.

1. Determining the Catchment Area

The area for this catchment must be calculated manually based on the runoff coefficient, *C*, and rainfall intensity, *i*.



Given data from the initial analysis:

Previous drainflow from Node 12 = 1.990 cms

Properties of Drain 15:

Coefficient = 0.90 (Commercial and Business Centre)

Rainfall Intensity = 299.370 mm/hr

By applying the formula;

Flow,Q=\frac{C.i.A}{360}, rearrange to find A

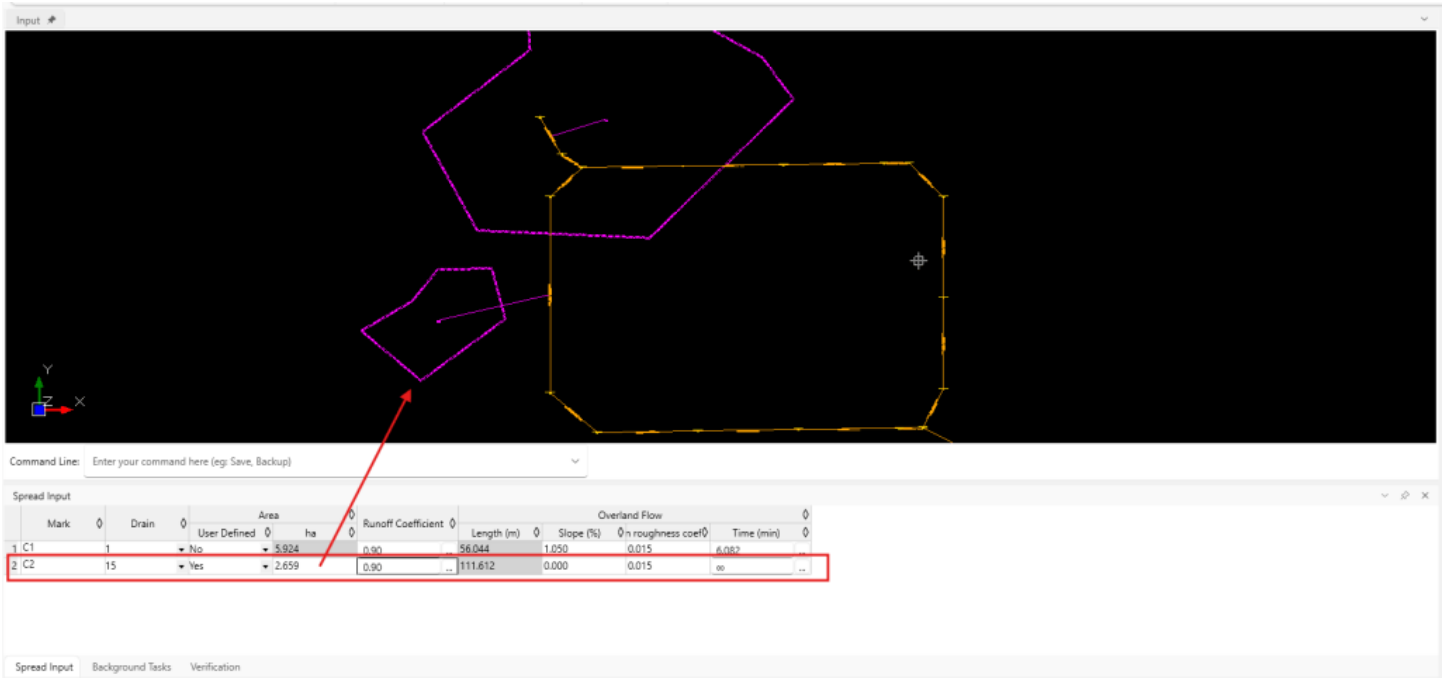
Area,A=\frac{360.Q}{C.i}

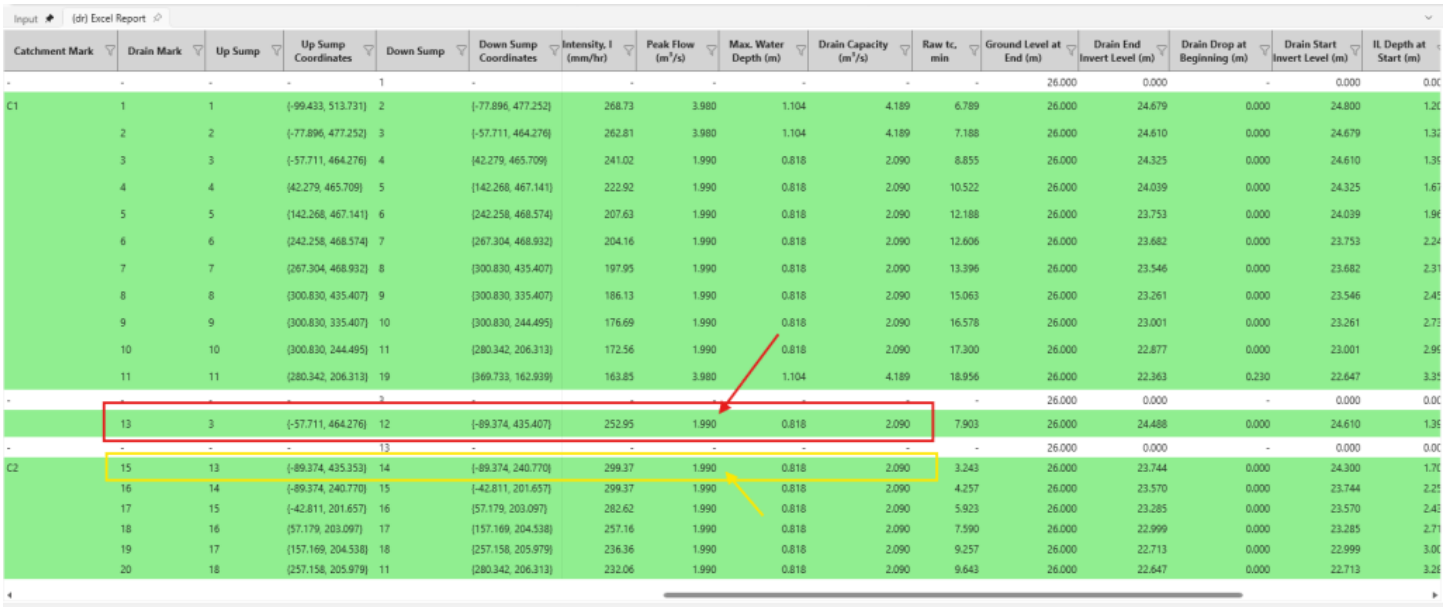
Area,A=\frac{360\times 1.990}{0.90\times 299.370}

Area,A=2.659sqm

With the value computed, create a catchment and set it to be flown into Drain 13.

Set the C to be 0.90 and define the area with the value obtained.





Catchment Mark	Drain Mark	Up Sump	Up Sump Coordinates	Down Sump	Down Sump Coordinates	Intensity, I (mm/hr)	Peak Flow (m^3/s)	Max. Water Depth (m)	Drain Capacity (m^3/s)	Raw tc, min	Ground Level at End (m)	Drain End Invert Level (m)	Drain Drop at Beginning (m)	Drain Start Invert Level (m)	IL Depth at Start (m)
-	-	-	-	1	-	-	-	-	-	-	26.000	0.000	-	0.000	0.00
C1	1	1	(-99.433, 513.731)	2	(-77.896, 477.252)	268.73	3.980	1.104	4.189	6.789	26.000	24.679	0.000	24.800	1.20
	2	2	(-77.896, 477.252)	3	(-57.711, 464.276)	262.81	3.980	1.104	4.189	7.188	26.000	24.610	0.000	24.679	1.32
	3	3	(-57.711, 464.276)	4	(42.279, 465.709)	241.02	1.990	0.818	2.090	8.855	26.000	24.325	0.000	24.610	1.35
	4	4	(42.279, 465.709)	5	(142.268, 467.141)	222.92	1.990	0.818	2.090	10.522	26.000	24.039	0.000	24.325	1.67
	5	5	(142.268, 467.141)	6	(242.258, 468.574)	207.63	1.990	0.818	2.090	12.188	26.000	23.753	0.000	24.039	1.96
	6	6	(242.258, 468.574)	7	(267.304, 468.932)	204.16	1.990	0.818	2.090	12.606	26.000	23.682	0.000	23.753	2.24
	7	7	(267.304, 468.932)	8	(300.830, 435.407)	197.95	1.990	0.818	2.090	13.396	26.000	23.546	0.000	23.682	2.31
	8	8	(300.830, 435.407)	9	(300.830, 335.407)	186.13	1.990	0.818	2.090	15.063	26.000	23.261	0.000	23.546	2.45
	9	9	(300.830, 335.407)	10	(300.830, 244.495)	176.69	1.990	0.818	2.090	16.578	26.000	23.001	0.000	23.261	2.73
	10	10	(300.830, 244.495)	11	(280.342, 206.313)	172.56	1.990	0.818	2.090	17.300	26.000	22.877	0.000	23.001	2.95
	11	11	(280.342, 206.313)	19	(369.733, 162.939)	163.85	3.980	1.104	4.189	18.956	26.000	22.363	0.230	22.647	3.35
-	-	-	-	3	-	-	-	-	-	-	26.000	0.000	-	0.000	0.00
	13	3	(-57.711, 464.276)	12	(-89.374, 435.407)	252.95	1.990	0.818	2.090	7.903	26.000	24.488	0.000	24.610	1.35
-	-	-	-	13	-	-	-	-	-	-	26.000	0.000	-	0.000	0.00
C2	15	13	(-89.374, 435.353)	14	(-89.374, 240.770)	299.37	1.990	0.818	2.090	3.243	26.000	23.744	0.000	24.300	1.70
	16	14	(-89.374, 240.770)	15	(-42.811, 201.657)	299.37	1.990	0.818	2.090	4.257	26.000	23.570	0.000	23.744	2.25
	17	15	(-42.811, 201.657)	16	(57.179, 203.097)	282.62	1.990	0.818	2.090	5.923	26.000	23.285	0.000	23.570	2.43
	18	16	(57.179, 203.097)	17	(157.169, 204.538)	257.16	1.990	0.818	2.090	7.590	26.000	22.999	0.000	23.285	2.71
	19	17	(157.169, 204.538)	18	(257.158, 205.979)	236.36	1.990	0.818	2.090	9.257	26.000	22.713	0.000	22.999	3.00
	20	18	(257.158, 205.979)	11	(280.342, 206.313)	232.06	1.990	0.818	2.090	9.643	26.000	22.647	0.000	22.713	3.28

Analysis Mode

With a more advanced engine similar to EPASWMM, the in-loop drain analysis is no longer a limitation in our software. However, this mode requires known inverse levels, gradients, and drain dimensions to provide an accurate hydraulics analysis.

Therefore, users may need to combine the two modes: **(1) Design Mode** as a coarse method to size up the pipe data, such as dimensions, IL, and so on, and **(2) Analysis Mode** to proceed with in-loop drain analysis to obtain the hydraulic characteristics of the drainage system.

Sample Project File: **Here**

Step 1: Breaking the drain connection

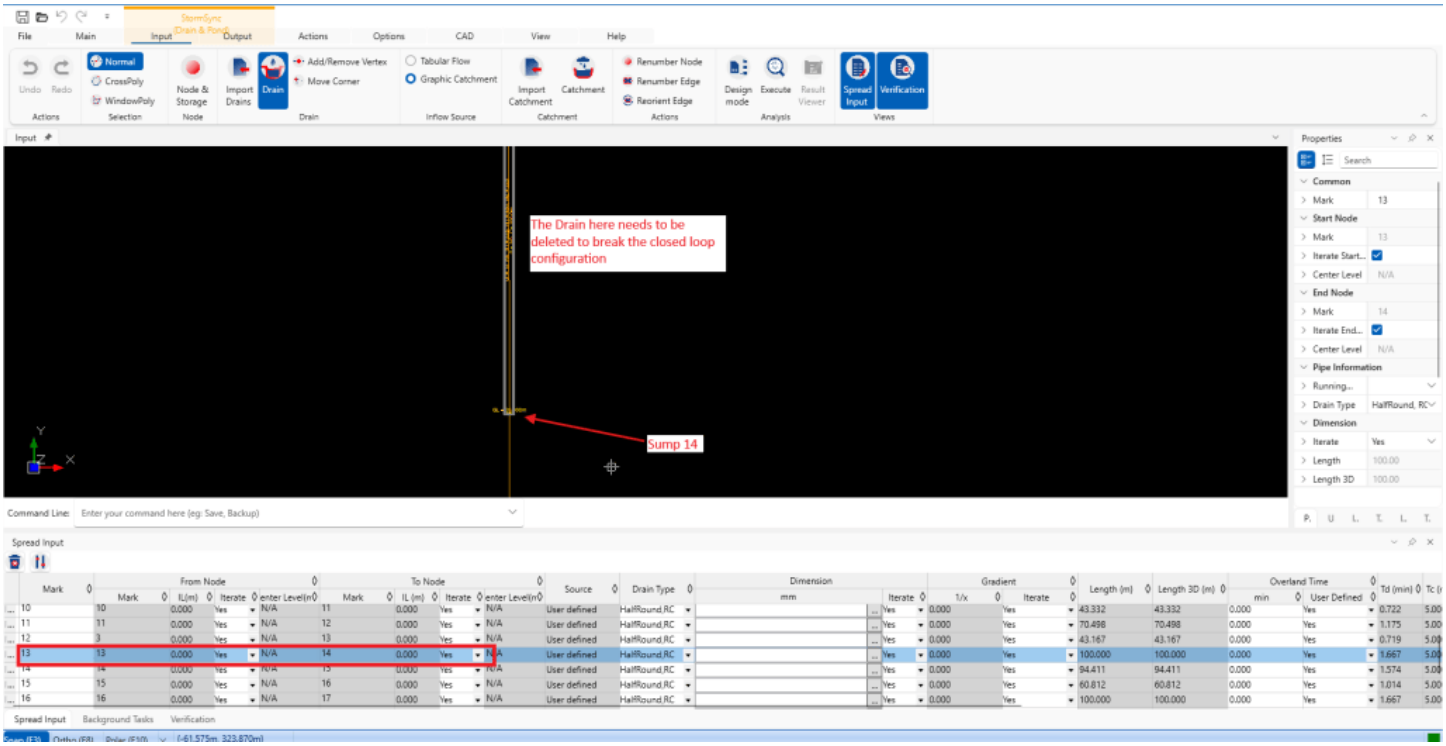
1. Select a breaking point in the loop

Choose a suitable node (sump) within the network to be the breaking point.

Based on the provided sample project file, Sump 14 is selected for this purpose.

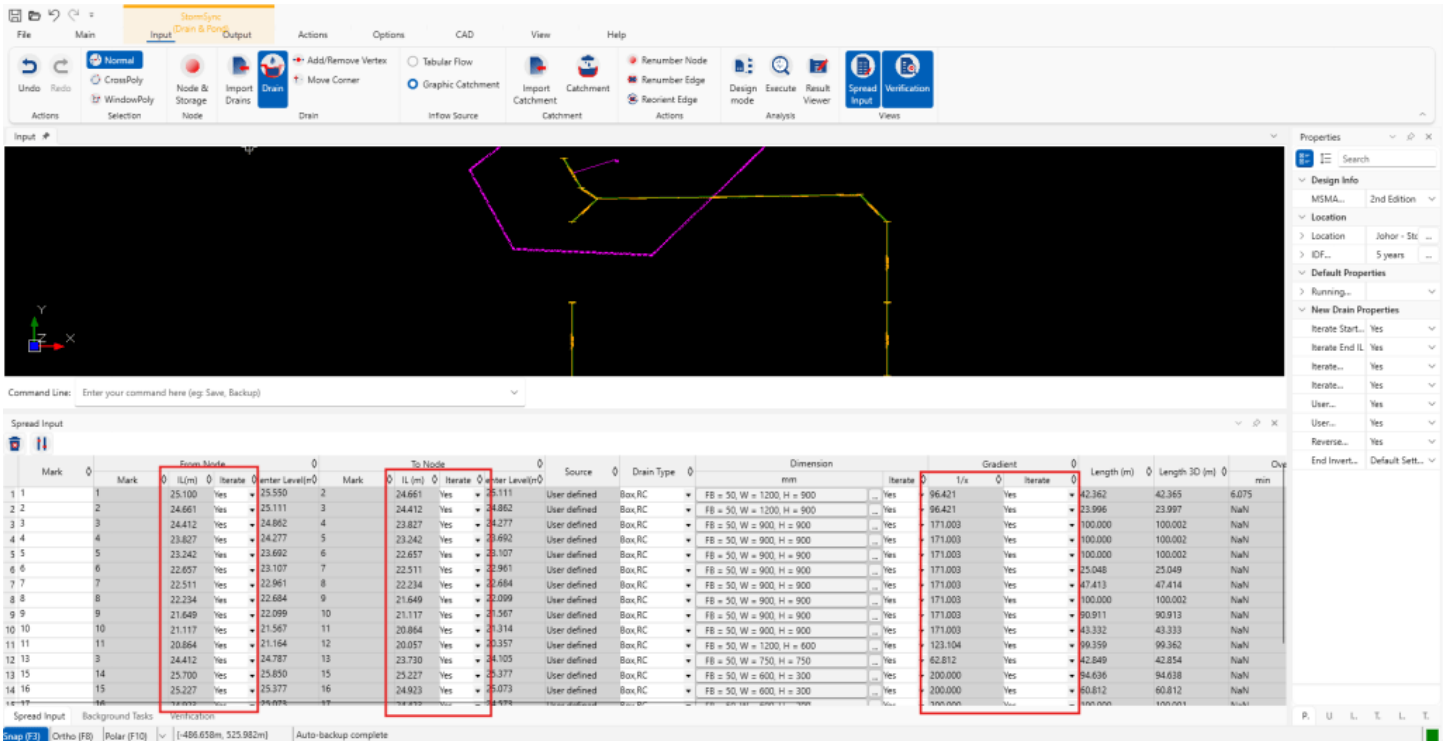
2. Delete the drains connected to the breaking point

Delete the drain before Sump 14 (Sump 14 is the ToNode under Spread Input)



Step 2: Design Mode to size up the pipe info, such as diameter, IL,... and so on

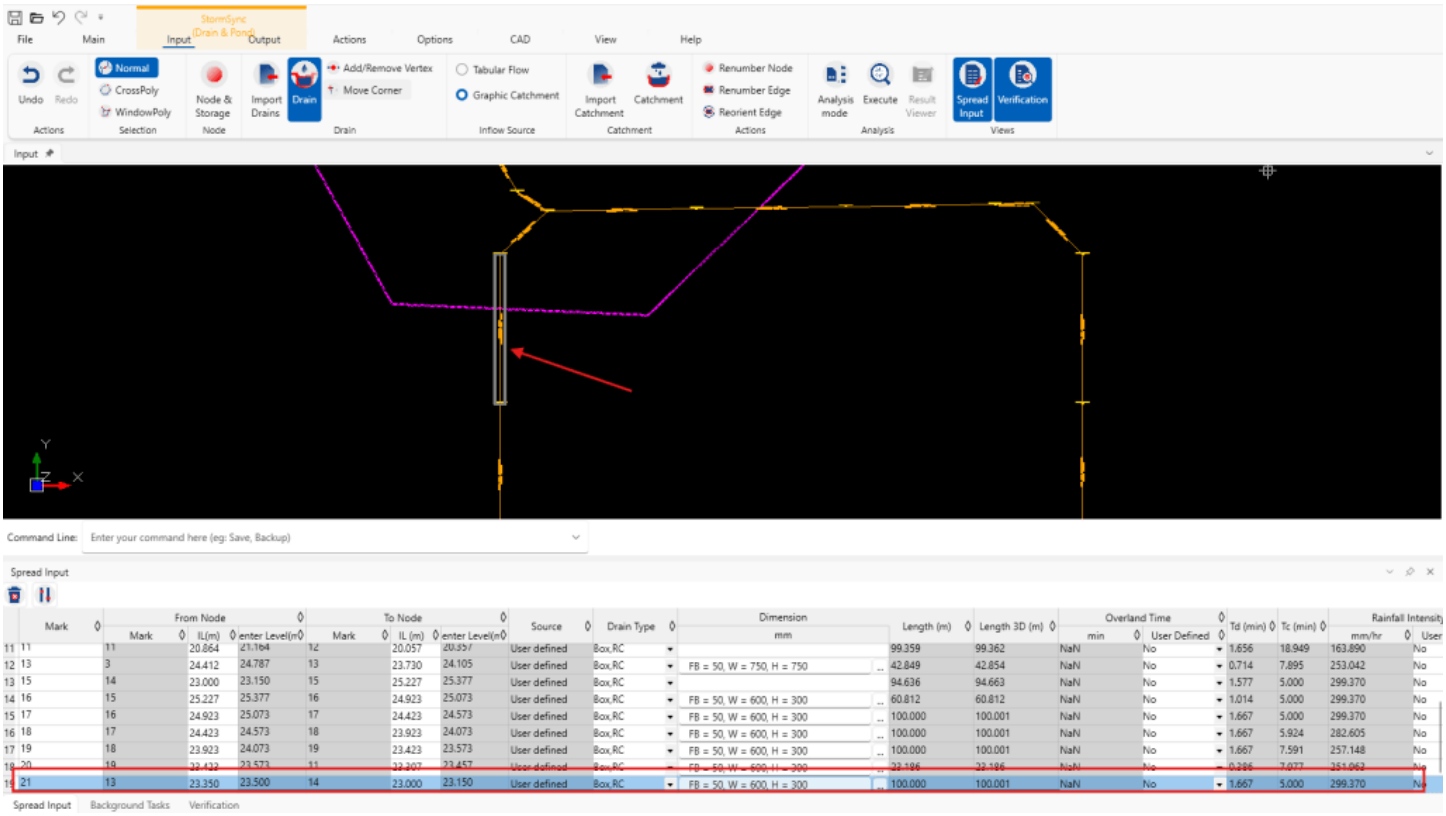
Once the in-loop connection is broken, the software can now successfully run the analysis in **Design Mode**. This allows the engine to provide iterated information such as **Invert Levels**, **Gradients**, and **Drain Dimensions**.



Step 3: Redefine closed-loop configuration

1. Reconnecting Sump 13 to Sump 14

Add a new drain that connects Sump 13 to Sump 14, as shown in the image below.



2. Levels Adjustment

The Invert Levels for the new drain connecting Sump 13 to Sump 14 need to be manually adjusted to ensure continuity of the network.

As an example,

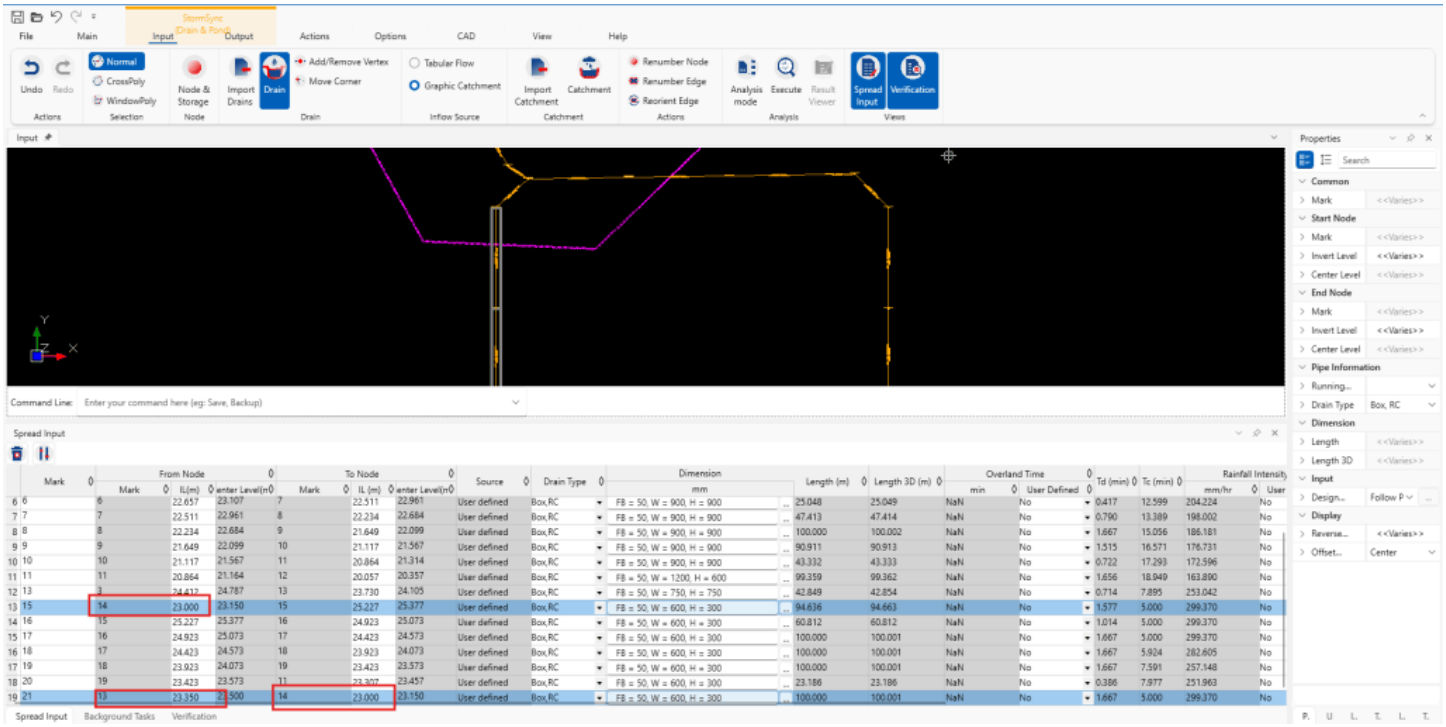
Adjusted properties of the newly added Drain 21:

Start Level, Sump 13 = 23.350m

End Level, Sump 14 = 23.000m

Adjusted properties of the connected Drain 15:

Start Level, Sump 14 = 23.000m



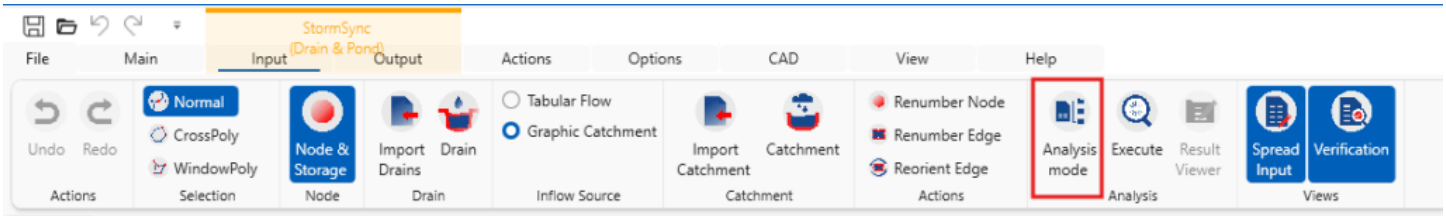
Step 4: Hydraulic analysis with Dynamic Routing

1. Analysis mode configuration

We will need to change the mode of analysis by going to

Input tab of StormSync Module > Go to Analysis section > Ensure the mode is displayed as Analysis Mode, as in the image below

If it is in **Design Mode**, users can click on the Design Mode button once, and the mode will be updated.



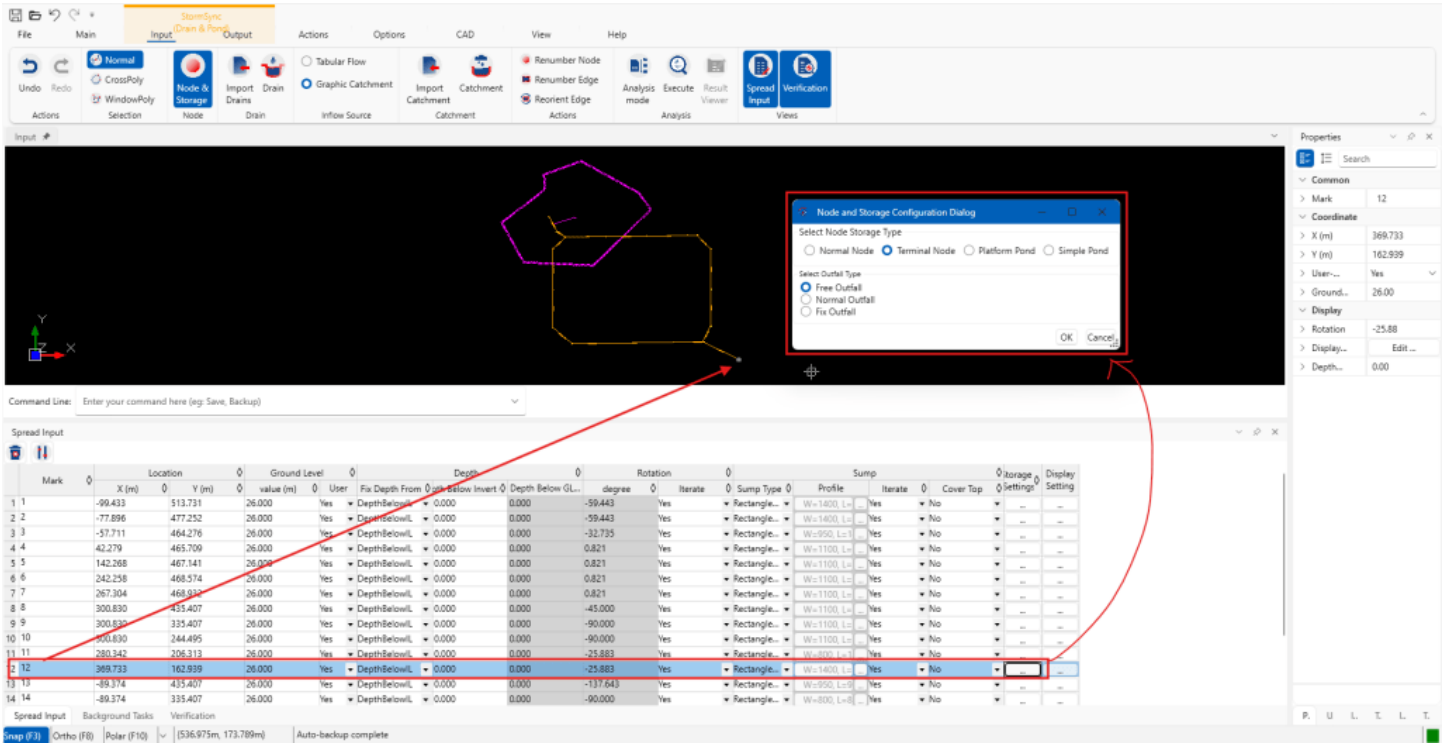
2. Setting up Node and Drain Properties as storage and link

For **Dynamic Wave routing in Analysis mode**, it is required that the network be linked to an outfall; otherwise, an ***error Code 145*** will be received.

An outfall is typically located at the end of the network, where it can either be set as a Terminal Node or a storage. It indicates the boundary condition on how the drain network discharges the water, whether it's discharging directly into the river as if discharging in to a black hole (free fall), or whether that there are some backwater condition (whereby the water depth of the river will affect the discharge rate, or even stopping the water from going into the river, etc).

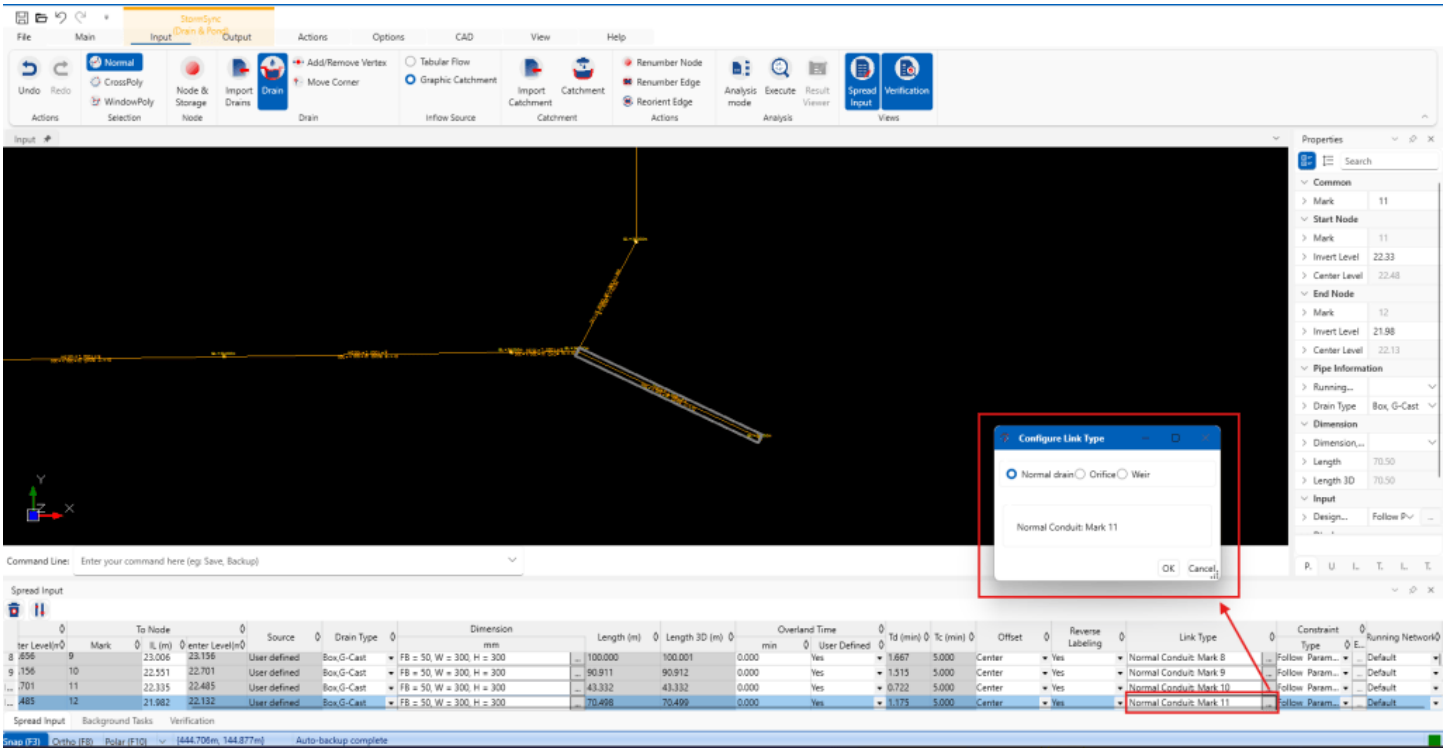
To do this, we may go to

Node Properties > Storage Settings column > Click the '...' button for the Node (in this example, it's Sump 12) > Set it to be a Terminal Node or a storage > Click OK



We can also set the conveyance drain type to Normal drain, orifice, or weir.

If the drain has been set to an orifice or weir, we will need to ensure the drain type is set correctly, or an error, *Drain is Orifice but contains unmatching drain type* will be received.



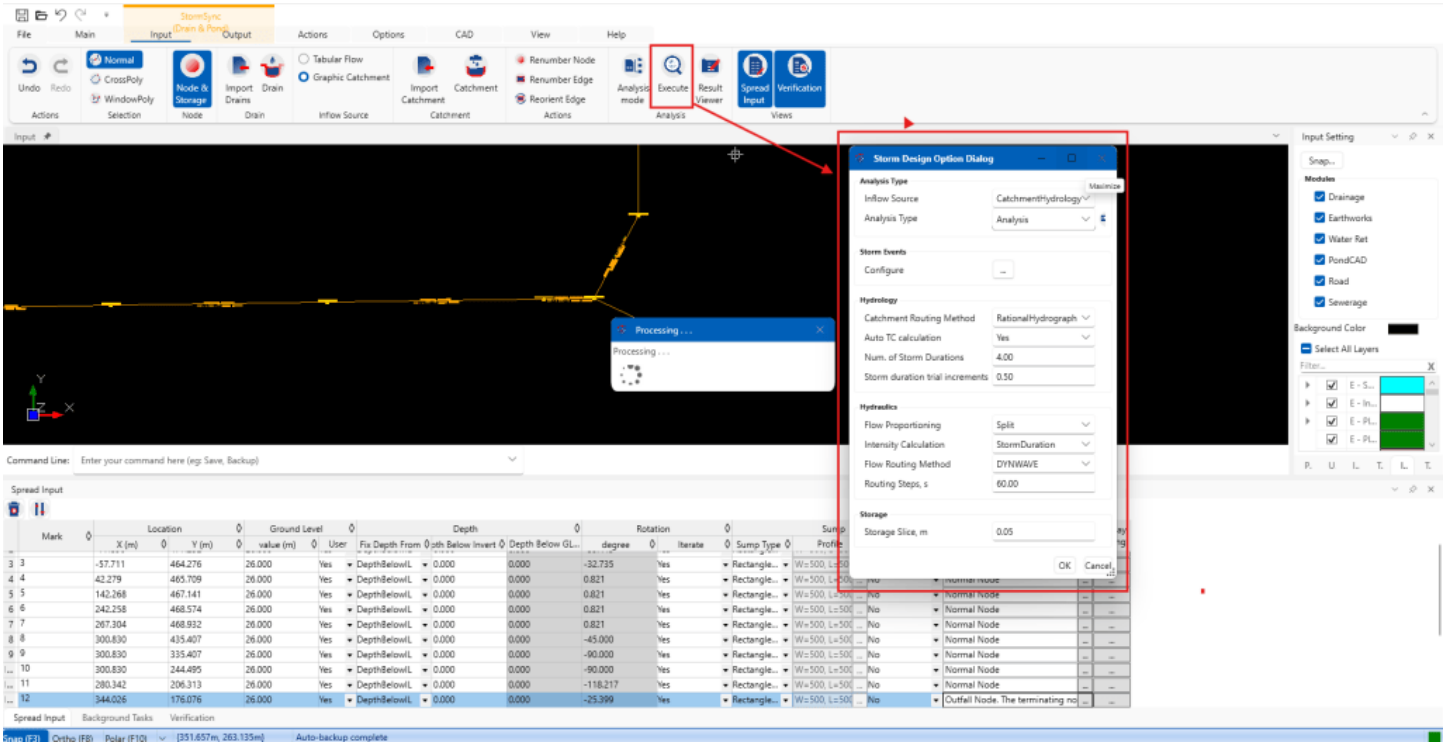
Further explanation of the required Input of the Analysis Mode can be found in our guide, ***Storm Water Design and Analysis Mode*.**

3. Execution of the analysis

To proceed with the analysis,

Click on the Execute button > Set up the Storm Design Dialog Option > Click OK

To know further about the settings for the Storm Design and how it can be applied to your analysis, you may refer to **here**.



Related



Ngu Soon Hui

I'm the Benevolent Dictator for Life for MiTS Software cum Editor of this website. Read more here.

You can also contact me at soonhui@mes100.com