InfoWorks ICM

Stormwater & Flooding Tutorials

Videos:

1. Setting up the Database (5min)
2. Adding project background data (3min)
3. Setting up a basic 2D Model (4min)
4. Adding ARR19 Rainfall (3min)
5. Running a model (2min)
6. Reviewing 2D Results (5min)
7. Adding the 1D network
8. Advanced Meshing
9. Adding Boundary Conditions & Optimising Simulations
10. Exporting Results

Contents

Introduction

InfoWorks ICM is the first software modelling package allowing the full integration of hydrodynamic and hydrological models within a powerful workgroup management platform.

InfoWorks ICM provides a new single simulation engine that fully integrates 1D and 2D simulation of drainage networks, open channels, rivers and floodplains. InfoWorks ICM can be used to model manholes, pipes, inlets, natural channels, man-made channels and rivers for complete integration of above and below ground elements. The resulting model contains common hydrology and can include both catchment and floodplain data.

Enhanced user-friendly interface for more efficient modelling

InfoWorks ICM employs the latest techniques to provide a user interface that is more users friendly and intuitive than ever before, leading to real performance gains as the user can work more efficiently to meet project goals. Features such as undo/redo and recycle bin allow the user to easily modify errors. While the dockable windows and editable property sheets, enable the user to create a more efficient work space. Version control allows multiple modellers to access the database and make edits while maintaining data integrity through auditing, comparing, and conflict resolution. The new model edit strip allows easy access to all common properties, which dynamically update as the user changes selection. Additionally, in-line validation quickly and efficiently highlights any unintentional errors during model building leading to more accurate results in less time.

Powerful scenario manager transforms modelling workflow

An easy-to-use scenario manager allows the user to quickly apply different "what if" scenarios to the base network model. This enables the user to maintain a single model of the drainage system and quickly construct, apply, and evaluate different scenarios as they relate to that model. Scenarios can be cut, copied, and pasted between different branches of the inheritance tree, allowing the user to quickly combine different scenarios to address a particular modelling concern.

New simulation server allows distributed model simulation

InfoWorks ICM supports the running of simulations on standalone workstations, or where more computing resources are available, such as servers or high end computers, model simulation can be distributed to take maximum advantage of those resources. Users can schedule simulations to run on their own local computers as well as remote computers, setting them to run as soon as possible or at a specified time. They can then monitor and control the progress of selected simulations and the queue of simulation jobs on a straightforward user interface. Simulations on remote computers can continue even when users have disconnected their own local computers and the simulation load can be balanced on an individual machine or within named groups of machines, giving equal priority to all users.

The new simulation server also allows users to store results locally or share them on a central server. Simulations can be left to complete, and their results uploaded to the central server without further user intervention.

The goal of this training course is to familiarize new and existing InfoWorks users with the new interface including version control, scenarios, and the new simulation server, as well as introduce the principles and data requirements of 2D modelling. This two-day course will give a basic introduction of InfoWorks ICM, as well as take the modeller through more advanced elements of Integrated Catchment modelling.

Data Management

InfoWorks ICM is a workgroup based modelling and configuration management system that can also be used as a standalone product.

To facilitate both workgroup and standalone operation, InfoWorks ICM maintains data in a centralised multi-user **Master Database**, consisting of a database and additional files.

Local working copies of parts of the master data, (such as networks with changes that have not yet been committed), are stored in each users **Local Working Folder**.

Information that relates to the master database, but is not stored in the database, such as ground model data, is stored in **Remote Roots**. The files containing this data can be very large so it is better to store them outside of the database.

Transportable databases are used to transfer information between master databases.

Master Database Formats

To begin working with InfoWorks ICM, a master database must be set-up. The master database stores all data and information for the network. Databases in ICM can be standalone or workgroup. The database format has no size limit therefore all data and results can be stored in one database. If a model is migrated from InfoWorks CS or SD, it will need to be imported into a new/clean master database. The Master Database provides a flexible hierarchy for managing data.

The top level of this hierarchical structure is the Model Group. All data within the master database must be contained in a model group. A Model Group can contain the following:

- Other Model Groups
- Version controlled items such as networks
- Non version controlled items such as Selection Lists

The database types currently supported by InfoWorks ICM are:

- **Standard Standalone Database** - the default database type for InfoWorks ICM. This database is intended for use by individual users working on a single PC. This database type is only appropriate for use on a standalone machine.
- **Standard (WorkGroup) Database** - intended for use by individuals and groups of users. Requires the use of the Workgroup Data Server software, running as a service on the machine hosting the database.
- **SQL Server (WorkGroup) Database** - you must already have your own Microsoft SQL Server database installation
- **Oracle (WorkGroup) Database** - you must already have your own Oracle database installation.

For more details about the Workgroup Data Server please refer to the Help Topic.

1. Setting up the Database

To begin working with InfoWorks ICM, a master database must be set-up. The master database stores all data and information for the network. Databases in ICM can be standalone or workgroup. The database format has no size limit therefore all data and results can be stored in one database. There are two master database types available in InfoWorks ICM:

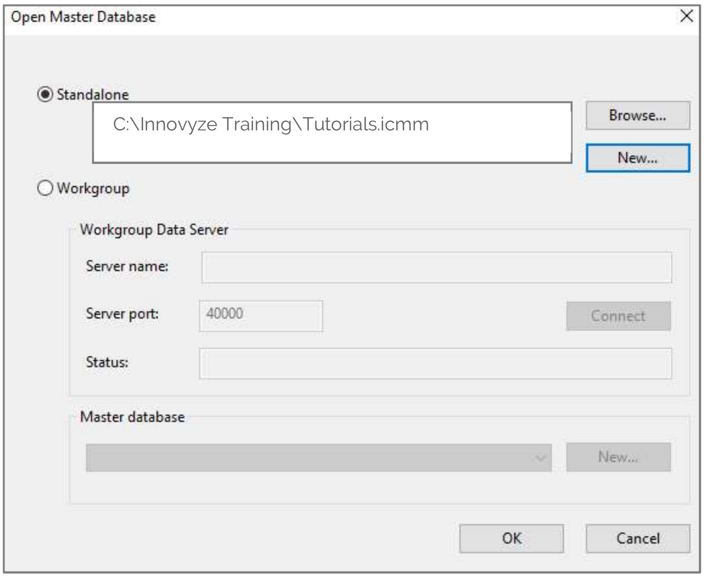
- Standard Standalone Database - the default database type for InfoWorks ICM. This database is intended for use by individual users working on a single PC. This database type is only appropriate for use on a standalone machine.
- Standard (Work Group) Database - intended for use by individuals and groups of users. Requires the use of the Workgroup Data Server

1A. Master Database Settings

1. Create a new standalone master database for the tutorial. Go to File > Open > Open/Create master database… or alternatively click on the Open/Create Master Database icon under the File Toolbar.

2. Select Standalone and click New

3. Create a new folder in C drive called 'Innovyze Training' and call the master database 'Tutorials.icmm'



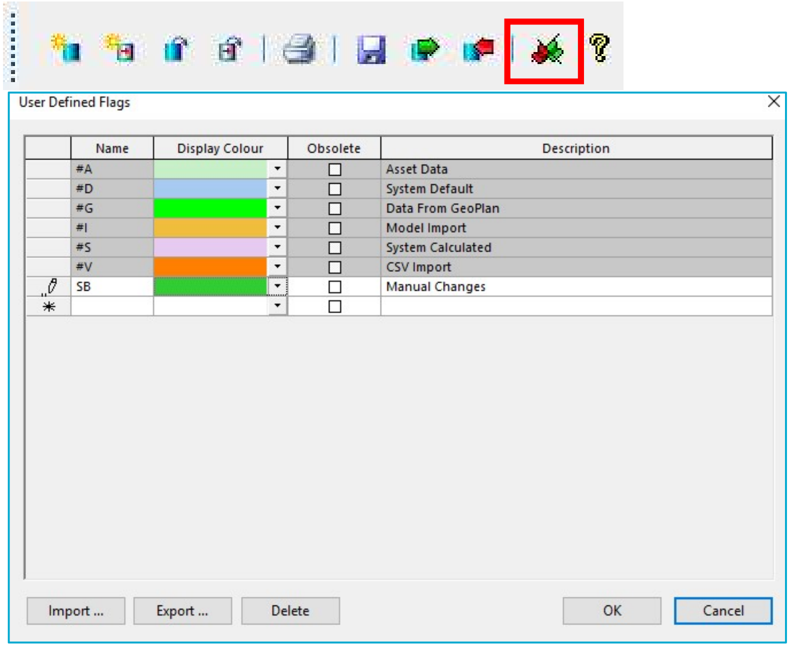
4. The Master Database object should now be visible in the Explorer Window

ⓘ *Note: If you do not see the Group Window, go to Window>New Explorer Window*

1B. Data Flags

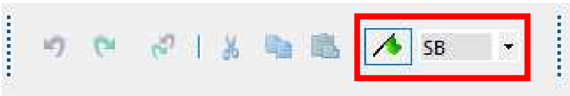
Data Flags are an important part of the model auditing process and are important in determining where data originated. There are some fixed default flags in InfoWorks ICM as shown below:

5. To setup the flags used in the project, go to File > Master database settings > User defined flags or alternatively click on the Data Flags icon under the File Toolbar.



6. Right click in the first empty row under Name and type in your initials, select a display colour and provide the description 'Manual Changes' and click OK.

7. Click on the Use Edit Flag icon under the Edit Toolbar and then select your initials from the dropdown menu. This will flag any manual changes you make in the model with your initials.

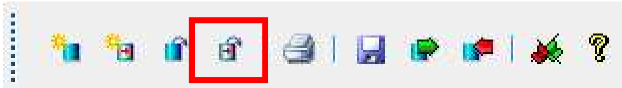


1C. Transportable Databases

To move database content from one master database to another, a transportable database is required. This will allow the data to be zipped up and emailed/transferred to a new location. A transportable database is useful to transfer data between colleagues, to clients or to Innovyze support (support@innovyze.com).

ⓘ Note: DO NOT copy the master database, ALWAYS use a transportable database to move data.

8. For this tutorial, we will use some template data that has been stored in a transportable database. Go to File>Open>Open Transportable Database… navigate to the tutorial files and choose transportable.icmt from the folder and click OK. Alternatively use the Open transportable database icon under the File Toolbar.



9. A new window will appear with model objects inside. **Select all** the objects, right click and **Copy**. In your newly created **master database**, right click on the **master database** and **Paste** the objects.

10. **Close** the transportable database window.

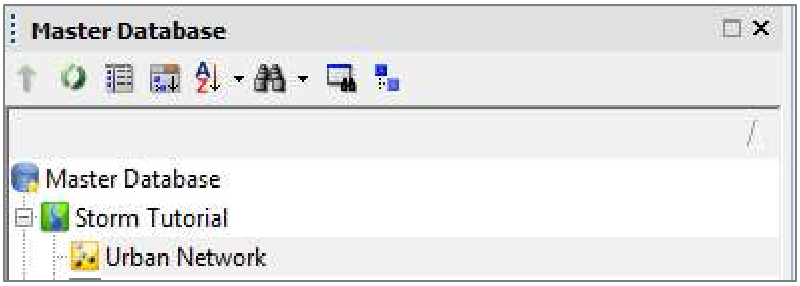
ⓘ Note: To send data to someone else, just do the reverse in a new transportable database. To create a new transportable database, go to File>Open>Open new transportable database.

1D. Model Groups & Networks

A model group stores all modelling and simulation data created in InfoWorks ICM for a project. It is also possible to use model groups to group particular data types together, for example, it is possible to set up a model group for rainfall or inflow files. Just like your C drive on your computer can have many file folders, a master database can have many model groups

11. Right click on the Master Database Object in the Explorer Window and select **New > Model group** and give it the name '**Storm Tutorial**'. This will be the *folder* that contains all your project components (network, rainfall, results etc).

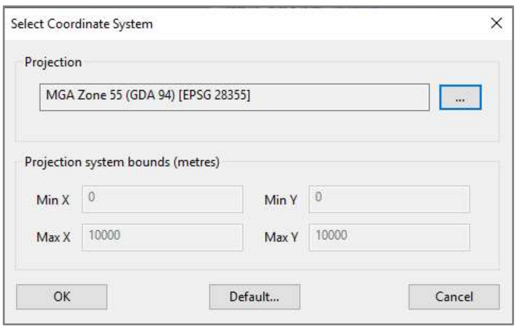
12. Right click on the Model group and select **New>Model network** and type the name '**Urban Network**'



13. Double click on the new network object to open in the GeoPlan and maximise the window.

1E. Projection / Co-ordinate System

14. To set the co-ordinate system for the network go to **GeoPlan>Set co-ordinate system**. Click on the Ellipse icon to see the available projections. Select **MGA Zone 55 (GDA 94)** from the list under the **Map Grid of Australia 1994)**.

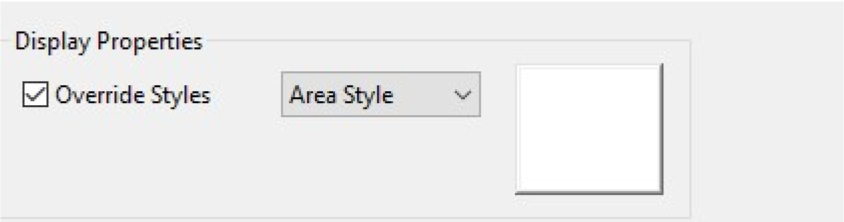


2. Project Data

2A. Background Layers

The InfoWorks Suite interacts closely with a number of GIS packages including ArcGIS and MapInfo. Layers from these GIS packages such as shape files, images and mapinfo files can be added to the GeoPlan as background images. InfoWorks is most commonly installed with MapXtreme. Using MapXtreme will allow background layers of varying types to be inserted on the GeoPlan without need for any additional GIS licences. However, should an ArcGIS licence be available additional interaction with ESRI Geodatabases will be available from within the InfoWorks interface.

1. Right click anywhere in the GeoPlan and select **GIS Layer control…** Select **Add** and search for '**RoadCrown.shp**' in the Raw Data folder. Use the **CTRL** key to select **SurfaceMaterials.shp**' at the same time. The projection MGA Zone 55 should automatically be detected and set from the file.
2. Select the **[SHP] SurfaceMaterials** and click on Properties…
3. In the Display Properties, click on the white square to change the area style.



4. Under the Fill tab, select **N** for no pattern. Then go to the Border Tab, select **magenta** for the line colour. Click **OK**.
5. Next add the '**Aerial.JPG**'. Ensure to select **Raster** from the File type dropdown menu to see the file in windows explorer.
6. Use the **Up and Down** keys to ensure the Aerial is on the bottom of the GIS Layers list, otherwise you won't see the shape files over the top. Click **Apply** and **OK**.
7. Right click again in the GeoPlan and select **View Entire Layer** and choose **Aerial**. The background image should now be visible in the GeoPlan.

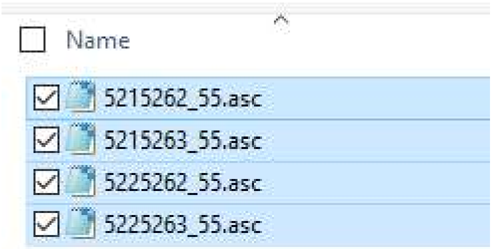
ⓘ *Note: Check the X and Y co-ordinates in the bottom right corner to check you are in the location you expected.*

2B. Importing Ground Models

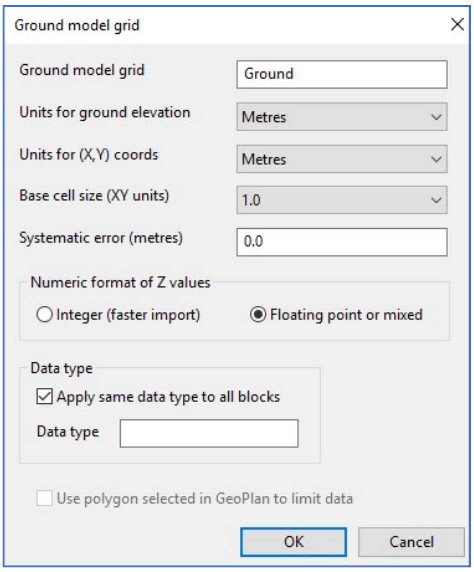
Ground models are a very useful way of looking at and interrogating data. They can be used by the inference tool in order to fill in gaps in the network data and they are necessary for 2D simulations. Ground model are also particularly useful for looking at bridge openings and definition.

8. To add a ground model Right click on the Storm Tutorial model group and go to **Import>Ground model grid…>From ground model grid files…**

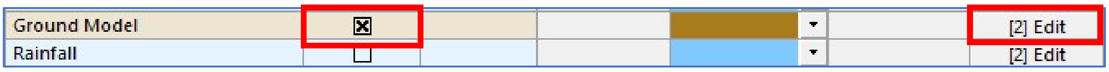
9. Navigate to the Lidar folder in the tutorial data and hold the **CTRL** key down to select all 4 files and select **Open**



10. Type in '**Ground**' for the Ground model Grid name, Choose **Meters** for the elevation units and ensure you use the '**Floating point or mixed**' to include elevations with decimal places.



11. Drag the new Ground object onto the GeoPlan. Right click anywhere in the GeoPlan and select **Properties and themes…**



12. Scroll to the bottom of the **Object Layer list** and tick on the **Display** checkbox for the **Ground Model.** Click Apply.

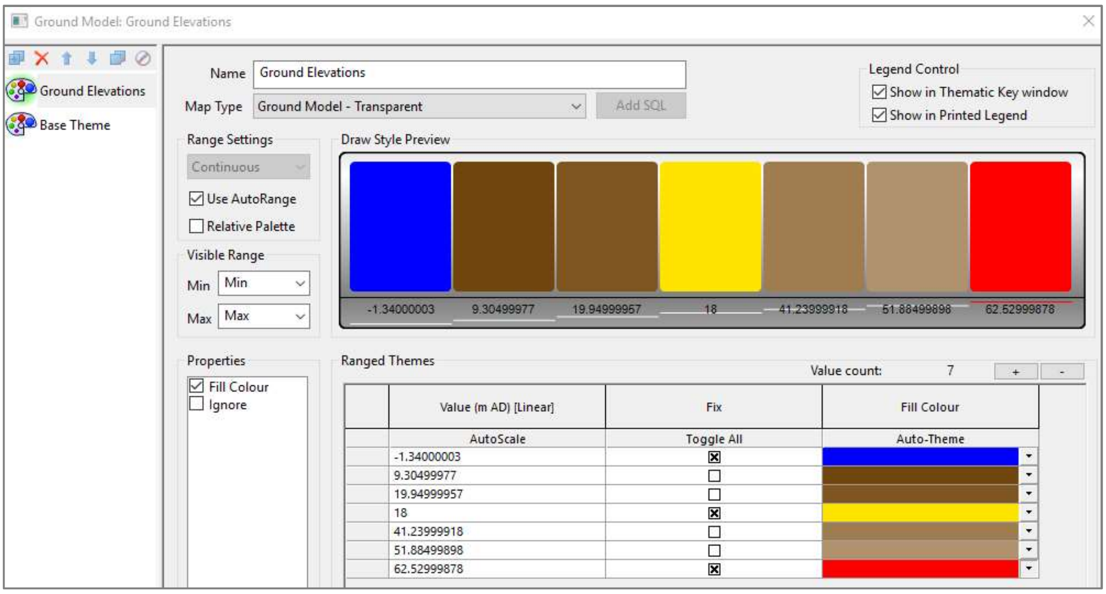
2C. Using Properties & Themes

A significant proportion of the GeoPlan functionality originates from the properties and themes dialog box. A number of visual changes can be made as well as the addition of themes to the network.

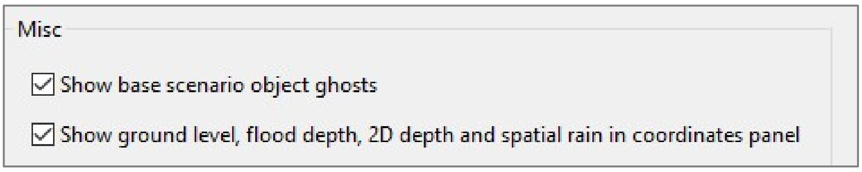
The properties of each asset (e.g. colour, display settings etc.) can be altered using the layers and themes tab of the dialog box. When starting ICM, all assets are assigned a default colour and all are visible in the GeoPlan. It is possible to turn off any of the network objects to improve the visibility of other network objects.

13. Right click anywhere in the GeoPlan and select **Properties and themes…** and click on the **Edit** button for the Ground Model.
14. Give the theme the name **'Ground Elevations'**. Set the Map type to **Ground Model – Transparent** and use the **'+'** button to change the Value Count=7. Fix the 1st, 4th and 7th rows by checking the boxes. Set the value and colours for the 3 Fixed rows:

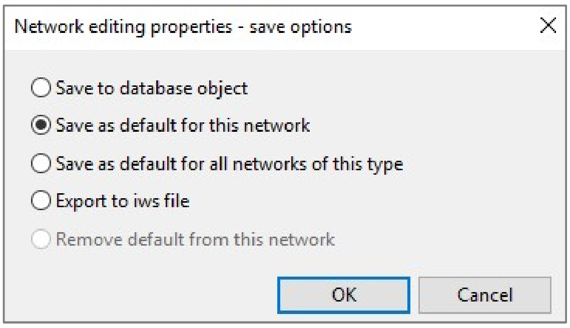
1. 1st row: -1.34 and blue
2. 4th row: 18 and yellow
3. 7th row: 22 and red (note: red is available under custom)



15. Left click on the **Auto Scale** and then on the **Auto-Theme** until the colour scheme grades from blue to green to yellow to red. Click **OK**.

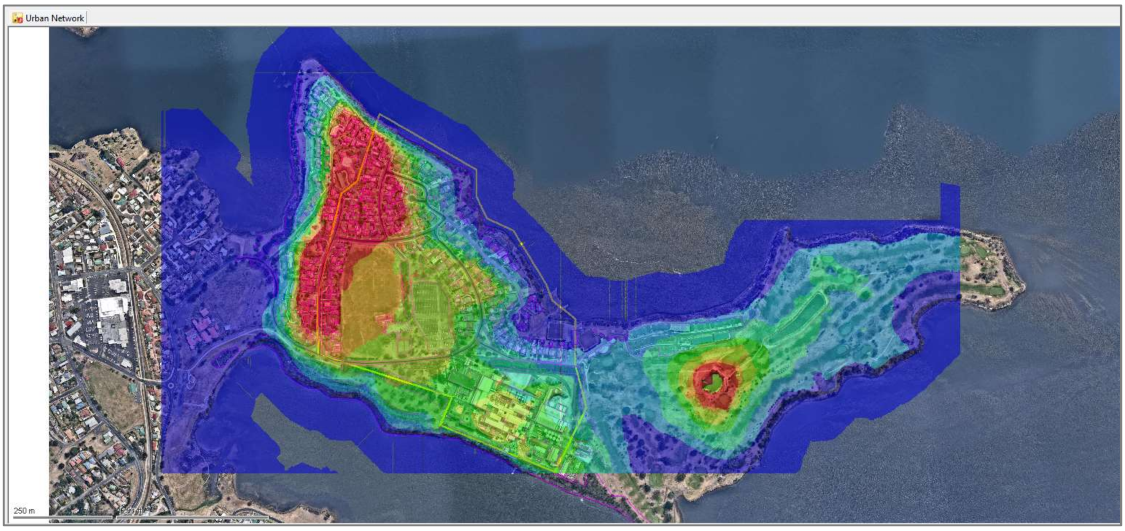


16. Back in the main screen, go to the **Visual** tab and under **Misc** check on the **Show Ground level, flood depths, 2D depth and spatial rain**. This will turn on Z elevations for the cursor location in the bottom right corner of the GeopPlan.
17. Click **Apply** and **Save** and select **save as default for this network**. Click **OK**. The GeoPlan should now look like the below screenshot.



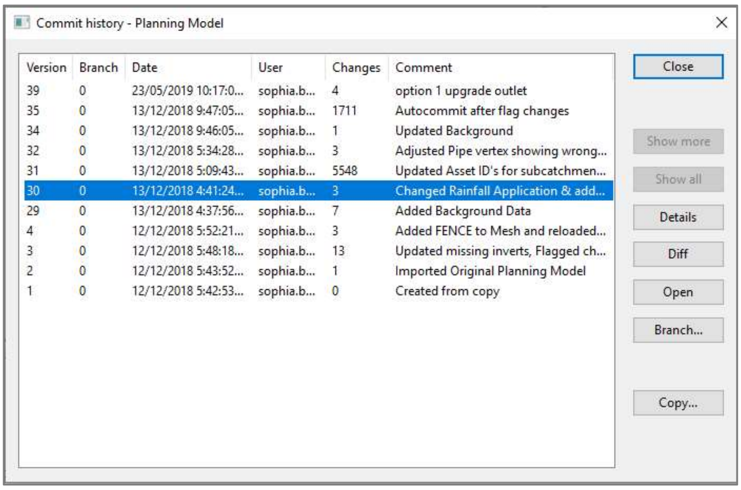
18. Alternatively, use one of the pre-defined colour schemes from the transportable by dragging the 'Theme: Ground Elevations' object into the GeoPlan.

Tip: Remove the Ground model from the network, go to Network>Clear Ground Model.

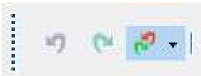


2D. Committing Changes

Every time a change to the network is committed, a new version of the network is created. The Commit History (found under the Network > Version Control menu or by right clicking on the network) details each version of the network. For every commit, there is a corresponding version. It tells the user the version number, date, user and the number of changes made. The notes section should be used as a model log to detail the version number and any changes that have been made. By maintaining detailed commentary, a model log kept outside the software is generally not required.



19. Right click on the **Urban Network object** and select **Commit changes to server**. Add a comment in **'Added Background Layers'** and select OK. We have nothing to validate, so we can choose **No** to Validation and commit without validating. You will notice committing changes will turn the undo/redo greyed out.



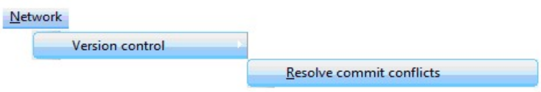
20. Right click again on the **Urban Network** object and select **Show Commit History.** Here we can see the versions committed so far (2).

Tip: Did you know ICM was Multi-User?

ICM allows for more than one user to be working on the same master database (Workgroup) at the same time. While this is available in other InfoWorks products, features such as Version Control, Commit History, etc. make ICM far more efficient at managing multiple users and auditing any and all changes to the master database. With multiple users working on the same network at the same time, there may be a situation where the same asset is modified by two users. If this situation happens, then a conflict will occur when one or the other user commits the changes to the master database.

The conflict is only discovered when InfoWorks ICM is forced to talk to the server i.e. when a commit happens or when the user clicks **Update (get changes made by others)**. Both of these commands prompt ICM to update the master database and therefore, any changes made by other users are found. 

The second user to update the master database will get a warning message if there are conflicts found. They will be prompted to resolve the conflict using the 'resolve conflicts' option. It is advisable to check with the other users involved before confirming a resolution.



As we are using a standalone master database for the purposes of this practical, we will not be experiencing any network conflicts.

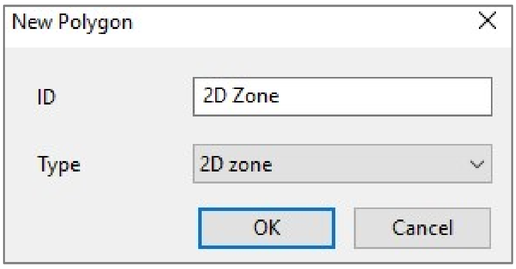
3. Setting up the 2D Zone

3A. Model Extents

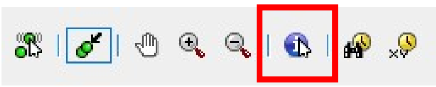
1. Select **Polygon** from the drop-down menu in the GeoPlan toolbar and use the **New Object** icon under the GeoPlan Toolbar to draw a polygon around the catchment area by following the yellow outline on the Aerial. Click once to add a vertex and double click to close the polygon.

Tip: start at the top left corner and work clockwise so you remember where you started.

2. Give the polygon the name **'2D zone'** and select the type **2D Zone**. Click **OK**.



3. The **Properties** for 2D Zone so automatically appear on the left-hand side of the screen. This window can be accessed anytime for any object by double clicking on an object in the GeoPlan or using the **Properties** icon in the GeoPlan Toolbar.



3B. 2D Zone Parameters

Terrain Sensitive Meshing & Mesh Size

Terrain-sensitive meshing is used to increase the resolution of the mesh in areas that have a large variation in height, without increasing the number of elements in relatively flat areas. This functionality can be applied to individual 2D zones by use of the Terrain-sensitive meshing field.

When terrain-sensitive meshing is enabled, the mesh generation process samples the ground model in each candidate triangle. If the range of heights within the triangle exceeds the Maximum height variation specified for the 2D Zone, the triangle is split, increasing the resolution of the mesh in areas where terrain height varies rapidly. This process is repeated until the Maximum height variation is no longer exceeded, or splitting would be likely to result in a triangle with an area smaller than the Minimum element area specified for the 2D Zone.

4. Set the maximum triangle area to 20, the minimum to 2.

ⓘ *Note: the minimum and maximum element areas may need to be adjusted for license limitations. For a 5000 element limit licence use 200 and 25.*

2D Boundary Conditions

When water reaches the edge of the 2D zone, the boundary of that zone behaves as either an outlet to the model or as a vertical wall.

5. Set the **boundary condition** to '**Normal**'. This will allow water to flow freely out of the 2D zone. We will later add the boundary conditions for the downstream outlet (ie. The River).

Terrain Conditions

6. Tick on the **Terrain-sensitive meshing** to ensure the ground elevations are considered in the meshing process

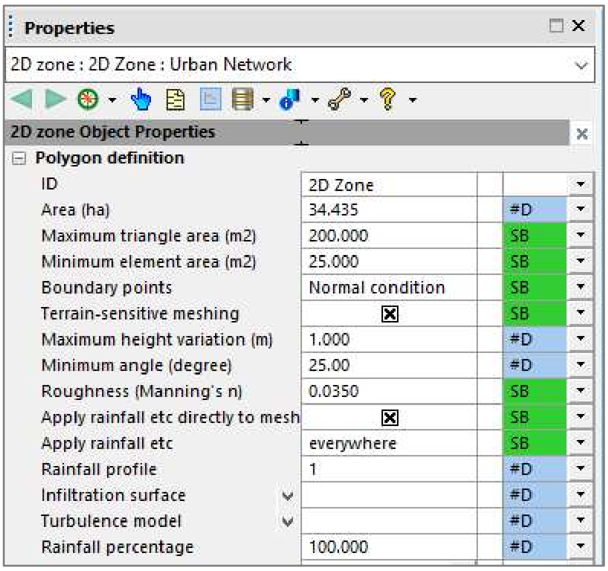
Manning's Roughness

7. Set the **Roughness (manning's n)** to **0.035**. This value will be used for all elements in the 2D zone where no alternate roughness zone is provided.

Using Direct Rainfall & Subcatchment Hydrology

InfoWorks ICM has the option to use traditional Subcatchment hydrology (such as RAFTS), use direct rainfall only or use a combination of the two.

8. We will use the direct rainfall method only, check on the '**Apply rainfall directly to mesh elements**' and choose **everywhere** from the '**Apply rainfall etc**'.

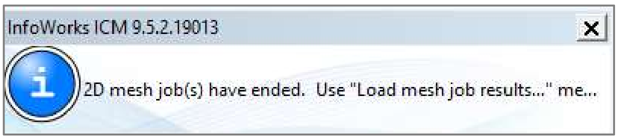


3C. Meshing

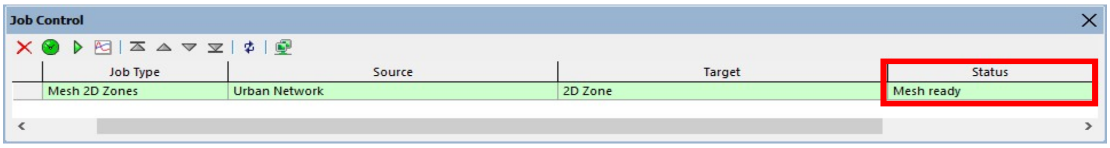
9. Use the select tool  to select the 2D Zone, go to **Model>Meshing>Mesh 2D zones**.. drag the Ground object from the Explorer Window into the Ground Model window and the bottom of the dialog and select **OK** to schedule the job.

10. ICM will notify you when the Meshing is complete with a message box appearing in the bottom right hand corner of the screen. The mesh process can also be monitored from the

Job Control Window which can be opened by going to **Window>Job Control Window** or using the Window Toolbar.

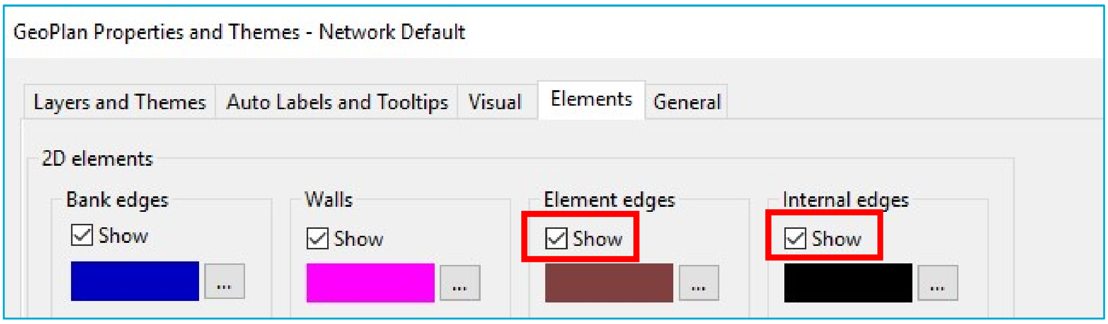


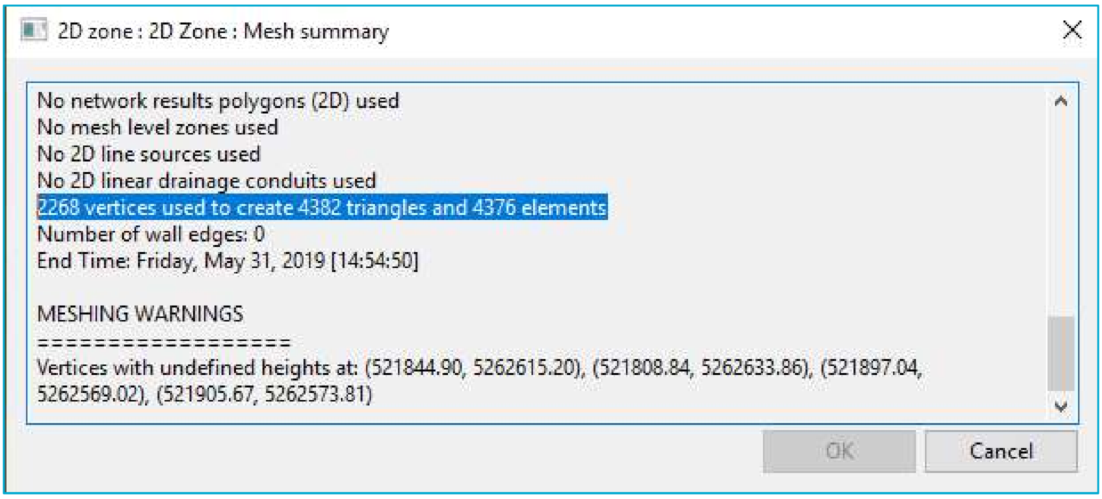
11. To load the mesh into the model, go to **Model>Meshing>Load mesh job results** and select **Load** and then **Close**. Alternatively, Click the **'Mesh Ready'** under the Status in the Job Control window.



12. To view the mesh, open the **Properties and Themes** (right click on GeoPlan) and navigate to the **Elements** tab. Check on the show Element edges and Internal edges. Select **Apply, Save as default for this network** select **OK**.

13. The number of elements in the mesh can be seen in the summary report. To view this, double click in the **2D zone** to bring up the properties and select the 3 dots next to the **Mesh summary** row. Depending on where you have drawn your 2D zone, this may vary.





3D. Committing with Validation

14. Before we commit the changes this time, we want to check we have a Valid Network. To validate, go to **Network>Validate Network** or select the red tick in  the toolbar.

ⓘ **Note**: the validation tool will flag any user-input values that are inaccurate, inappropriate or missing. This ensures a network is suitable to simulate before a run is attempted. These are judged with a range of priorities:

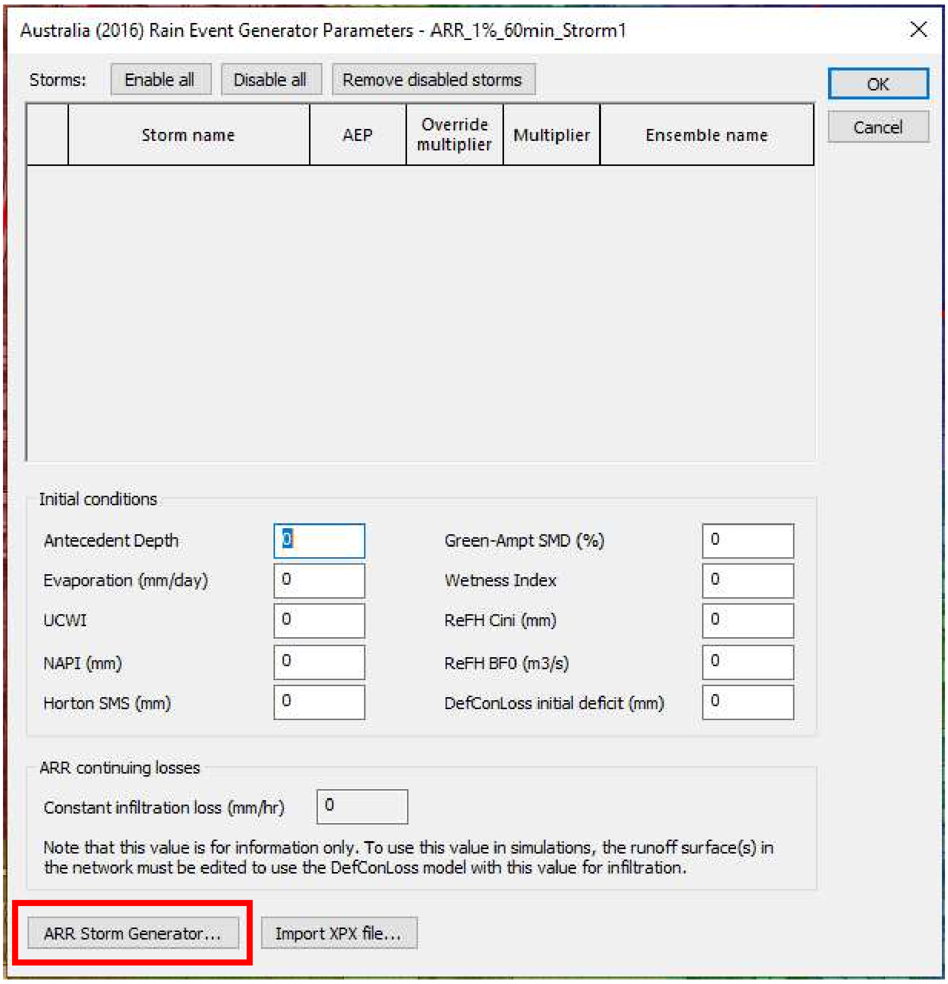
- Red = an error that MUST be fixed is you want to be able to simulate the network
- Yellow = a warning you that something is possibly not right
- Purple = information that you might wish to take note

15. Right click on the Network Model object and select **Commit changes**. Add the comment **'Added 2D zone and Meshed'** and select **OK**.

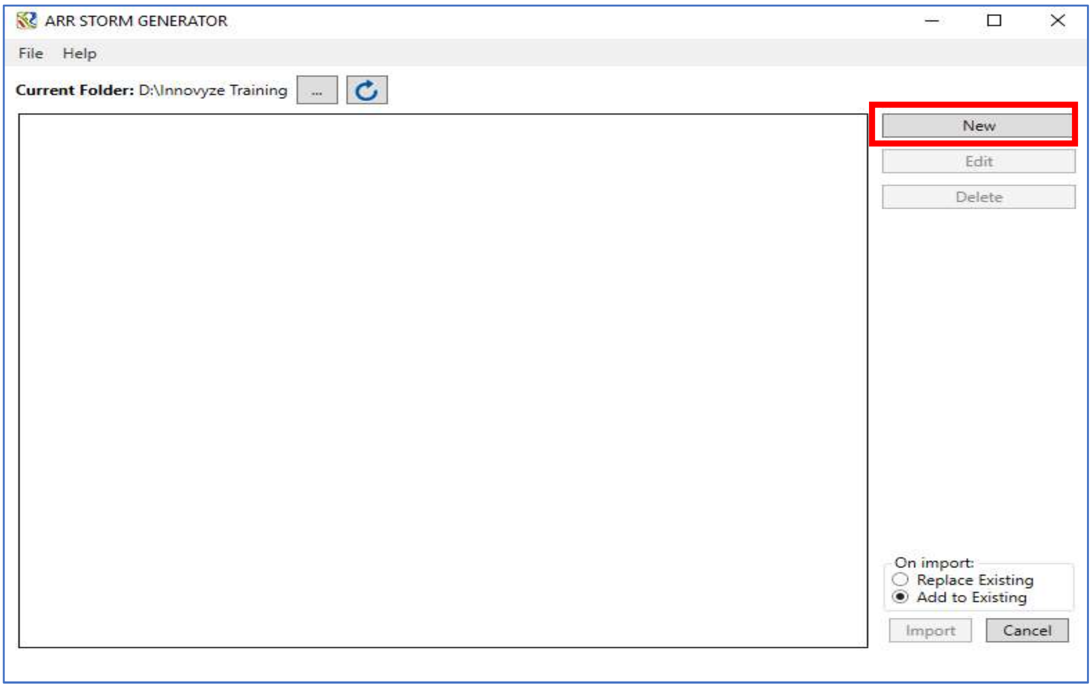
16. Having a valid network means this network can now be simulated!

4. ARR19 Rainfall Data

1. To create a new rainfall object, right click on the Model Group and select New>Rainfall Event and give it the name 'ARR_1%_60min_Strorm1' and click OK. Select Australian (2016) rainfall.
2. Select the ARR Storm Generator button from the bottom left corner.



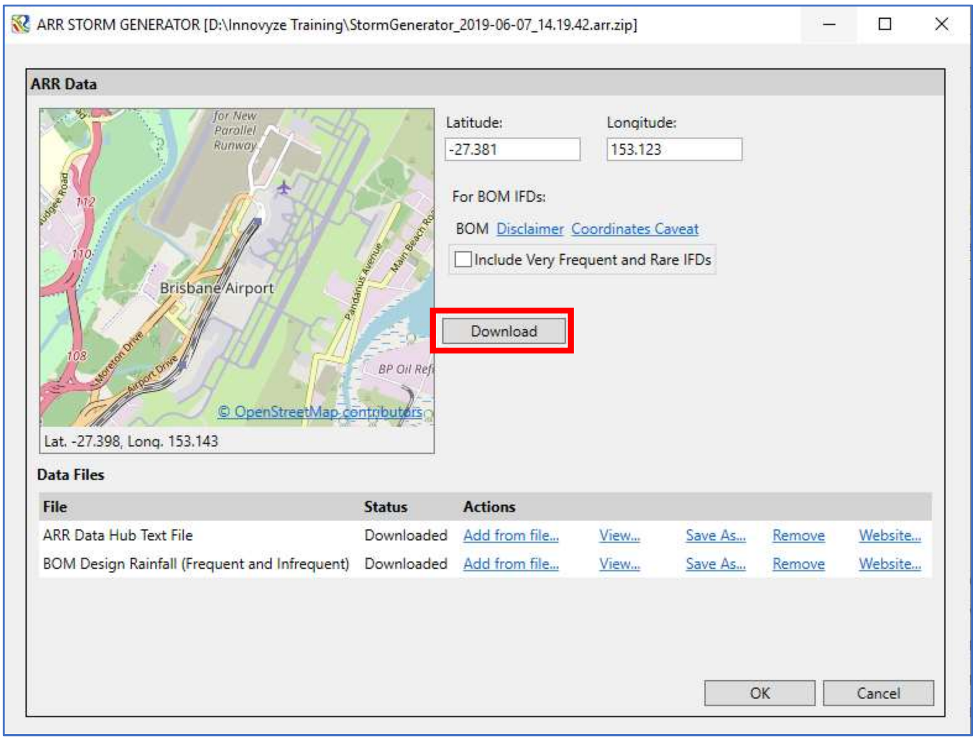
ⓘ Note: by default, the storm generator will access an ARR Storm Generator folder in your working documents. This allows you to keep all your rainfall together in a single folder. If the rainfall is project specific you may want to save it in your project folder or the same file location as the project (.icmm) file (C:\Innovyze Training\).



3. To create a new rainfall data set select New to create a new set of storms. The AEPs and Durations lists should be empty.

4. To populate the storm data (AEPs and Durations), select ARR Data from the top right side of the dialog box.

5. Use the centre scroll button on the mouse to zoom and pan to find the Brisbane Airport. Left click on the Airport to populate the Latitude (-27.381) and Longitude (153.123) of the desired location. Alternatively, you could type these in manually.

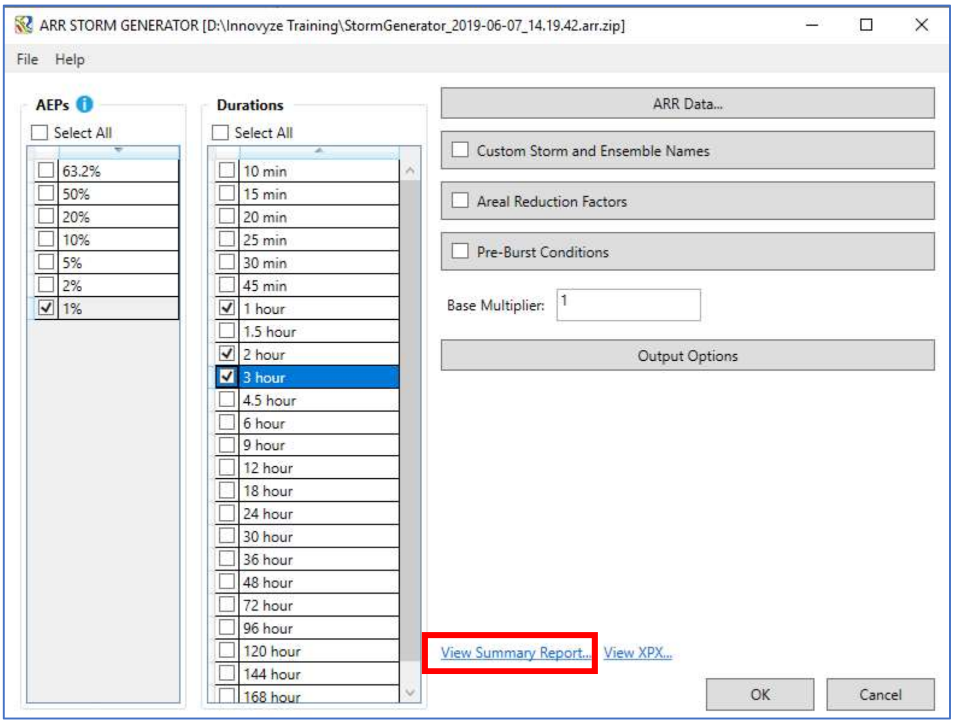


6. Select **Download**. The Data Files status should change to 'Downloaded' once successful. Click **OK**.

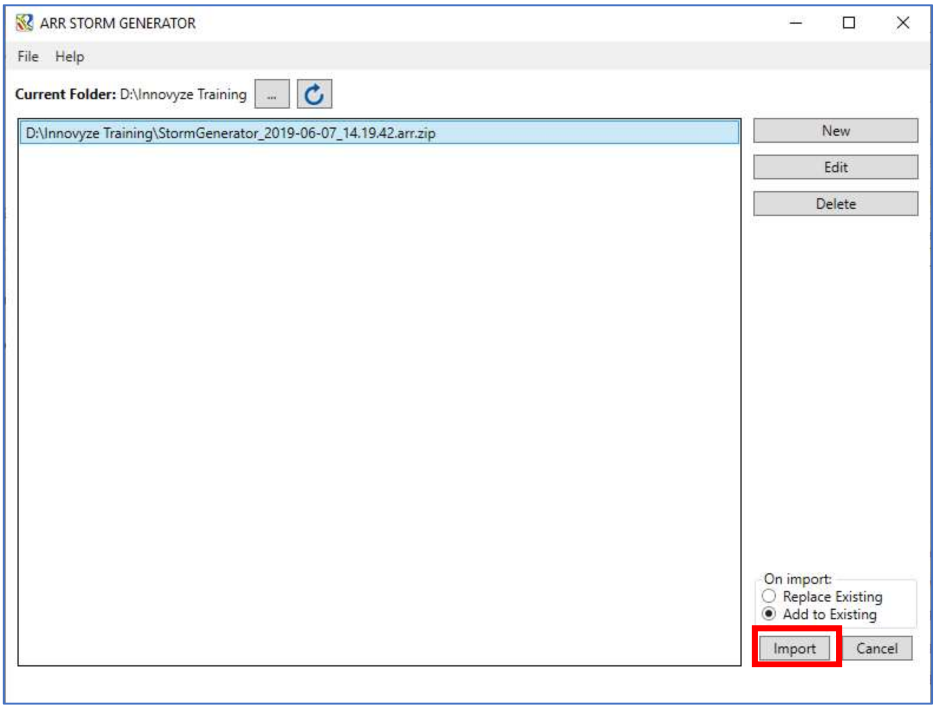
ⓘ **Note**: If no internet connection is available or there is an issue with either the BOM or ARR Data Hub websites, files can be downloaded and the Add from File action can be used to generate rainfall data manually (refer https://help.innovyze.com/display/arr2016/Importing+Storms).

7. The available AEP and Duration lists should now be populated. **Uncheck the Select All box** and the top of both lists to deselect all. Then select the desired events (1% AEP, 1hr, 2hr and 3hr).

8. Click on the **View Summary Report** to check the storms that will be added to the model. There should be 30 (1 ensemble for each of the 3 durations). If you are happy with the summary report, select **OK**.



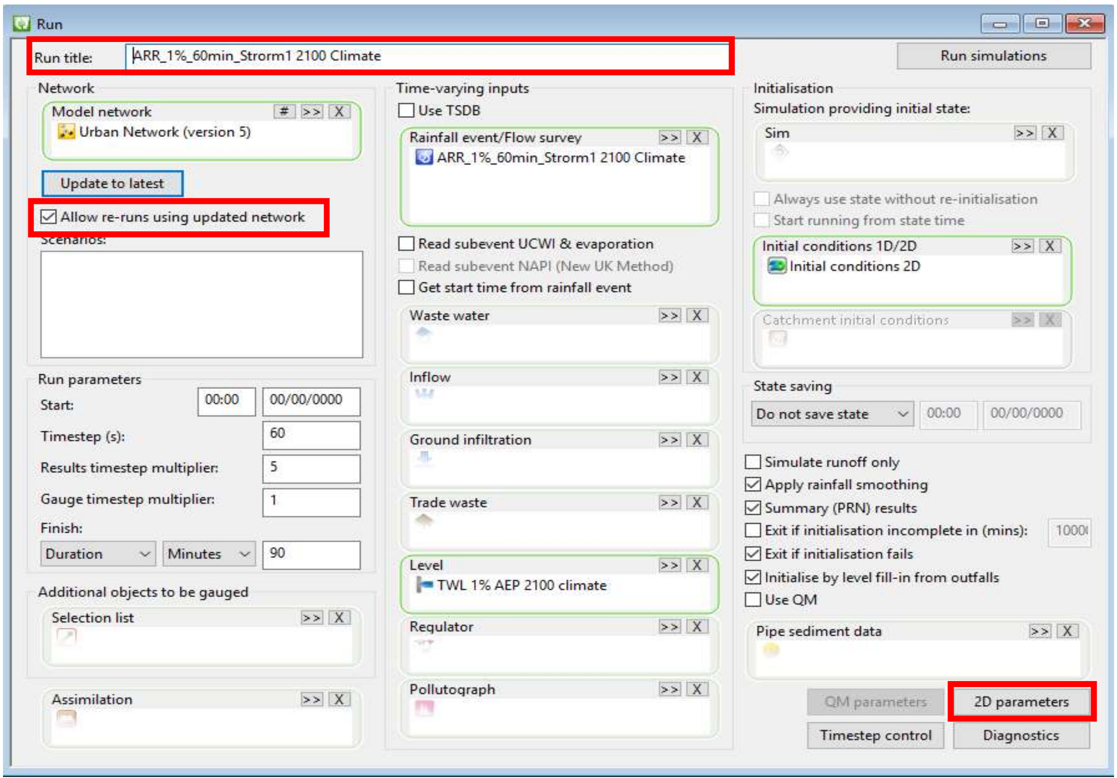
9. You should now see the newly created zip file displayed in the window. This file contains all the data files used to create the storms. Select the new file, use the Add to Existing on import option and click Import.



5. Running a Simulation

5A. Create a New Run Object

1. Right click on the Model Group and select **New>Run**
2. Fill in the Run Title 'ARR_1%_60min_Strorm1_2d'
3. Select the **Urban Network** and **Rainfall** objects from Group Window (Use the CTRL key to multi-select) and drag them onto the Run Window and dropping anywhere within the window.
4. Check the '**Allow re-runs using updated network**' which means we can re-run and overwrite the results rather than creating a copy of the run
5. Change the timestep to 5s, the result timestep multiplier to 12 and the duration to 90.
6. Click the 2D Parameters, under the GPU tab, check the If suitable card is available Click OK.



7. Click **Run Simulations** in the top right-hand corner and click **OK**.
8. Run the simulation on your computer and run it now.
9. The simulation details can be viewed during the run by going to **Window>Job Progress Window** but you will need to be quick to catch this run!

5B. Result Objects

10. Expand the Run object in the Model Group to see the run Results

ⓘ Note: The green coloured diamond symbol for the run indicates the run was successful and had no warnings or errors. There are several colour codes you might see here:

- Blue – the run is simulating
- Yellow – the run is complete and has warnings
- Purple – the run failed to initialise
- Red Cross – the run failed

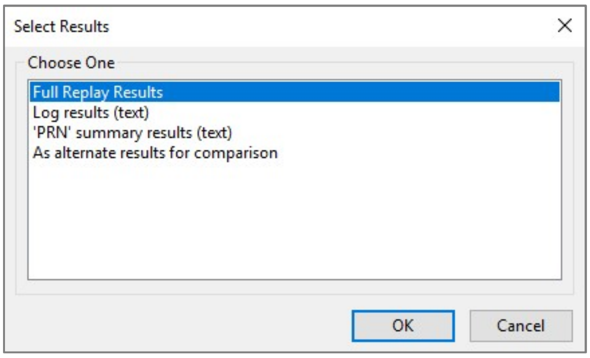
6. Reviewing 2D Model Results

With the simulation complete, we are ready to analyse and present the results. The interpretation of simulation results is key to the hydraulic modelling process. There are many tools in InfoWorks ICM to assist the user in reviewing results.

6A. Log Files

The log file provides a list of all the elements used to run the simulation, the simulation run times, mass balance calculations and any warnings. If the simulation was not successful, this will be the only result file available.

1. When you right click on the Simulation Result object and select **Open As…** the following dialogue appears:



2. Select the **Log Results (text)** from the list and click OK. The log file provides the engine version used to run the simulation and summarises the model details.

```
*********************
2d zone(s) summary
--------------------
2d Zone :                                          2D Zone
Effective area (ha) :                             34.5318
Minimum element area (m2) :                       25.0000
Maximum triangle area (m2) :                     200.0000
Average element area (m2) :                       62.3093
External Boundary Condition :             Normal Condition
Number of nodes :                                    7914
Original number of triangles :                      15620
Number of working elements :                         5542
Minimum element area (m2) :                        7.1216
Maximum element area (m2) :                      199.8982
Average element area (m2) :                       62.3093
Number of working faces :                           11825
Number of internal faces :                          11619
Number of boundary faces :                            206
Number of external boundary faces :                   206
```

Tip: A good check to undertake when using direct rainfall is checking the volume of runoff into the model. As shown in the screen shots, if the 2D zone is ~34ha and the rainfall depth over the simulation timeframe is ~100mm, then the rainfall volume should be ~34,000m3.

```
**********************
Volume balance report
---------------------
2d Zone name : 2D zone
IV -> Initial volume (m3) :                               0.0000
IVC -> Initial volume 2D conduits (m3) :                  0.0000
FV -> Final volume (m3) :                              2346.9899
FVC -> Final volume 2D conduits (m3) :                    0.0000

Inflows :                                                            %
IR -> Rain (m3) :                                     35638.1394   100.00
                                                      ----------   ------
IT -> Inflow total (m3) :                             35638.1394   100.00
```

3. Under the **Filter Index Report**, is the number of Threads used, Elapsed clock time and details of any GPU devices used during the simulation. This is a good way to estimate the time it will take to model say 15 different scenarios or maybe 800 different storm ensembles!

```
**********************
Filter index report
--------------------
2d Zone Name :
Filter index :

 Memory used by simulation engine = 48.2MB
 Threads = 8
 Elapsed clock time = 9s
 Total CPU time = 51.3s
EXITING

Selecting CUDA Device
1 CUDA Devices found
CUDA Device selection based on maximum GFLOPS
Using CUDA Device Quadro K1100M PCI Bus 2 ful
```

Elapsed clock time is the time the simulation took if you were to time it with a stop watch.

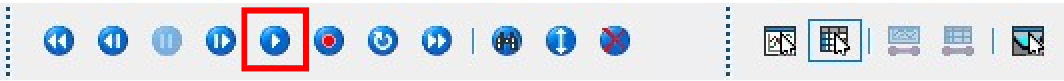
Total CPU time is the time it would take to run the simulation on a single thread (core)

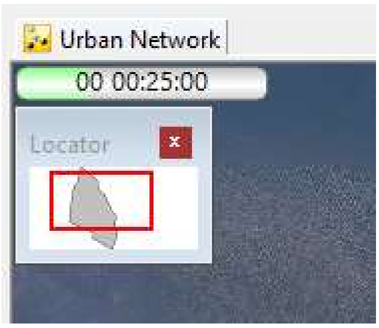
6B. Result Replay

4. Drag the result object into the GeoPlan to add the results to the current view. A new toolbar for the results and replay should appear. The time bar should also appear in the top left corner of the Geoplan.

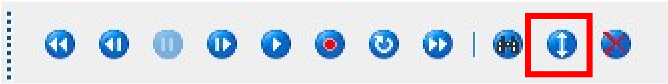
5. Drag on the '**Theme: 2D Results (depth+velocity)**' object from the Transportable Files and then click the **Play** button in the results toolbar.

4.



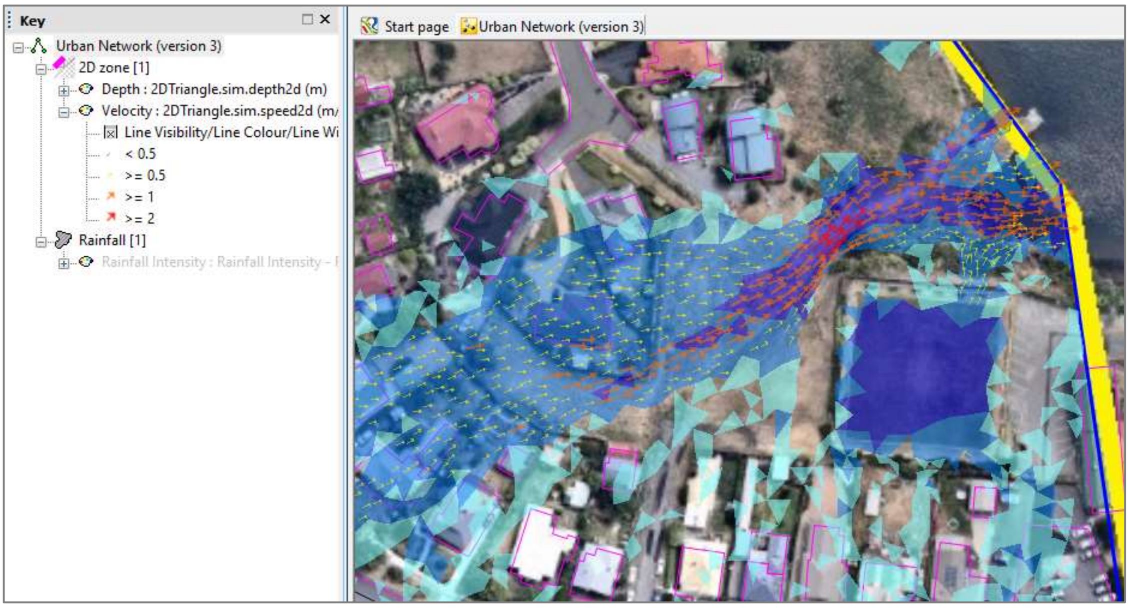


6. Click on the Show Maxima icon to show the worst result for each element across the simulation. This is often what is used to produce static flood maps.



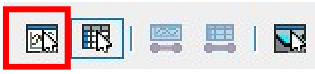
7. To turn of the theme for the Ground model and see the results clearly, right click and select **Property and Themes** and uncheck the display for the ground model.

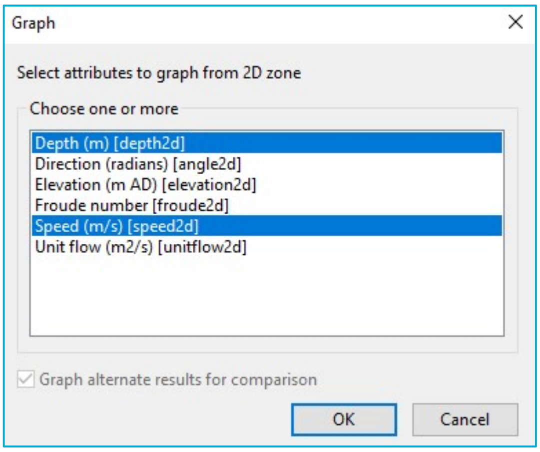
8. Turn on the **Thematic Key** by going to **Windows>Thematic Key Window** to see the legend of the flood depth and velocities. You can double click on any of the themes visible in the window to go directly to editing the **Properties and themes**.



6C. Graphing Tools

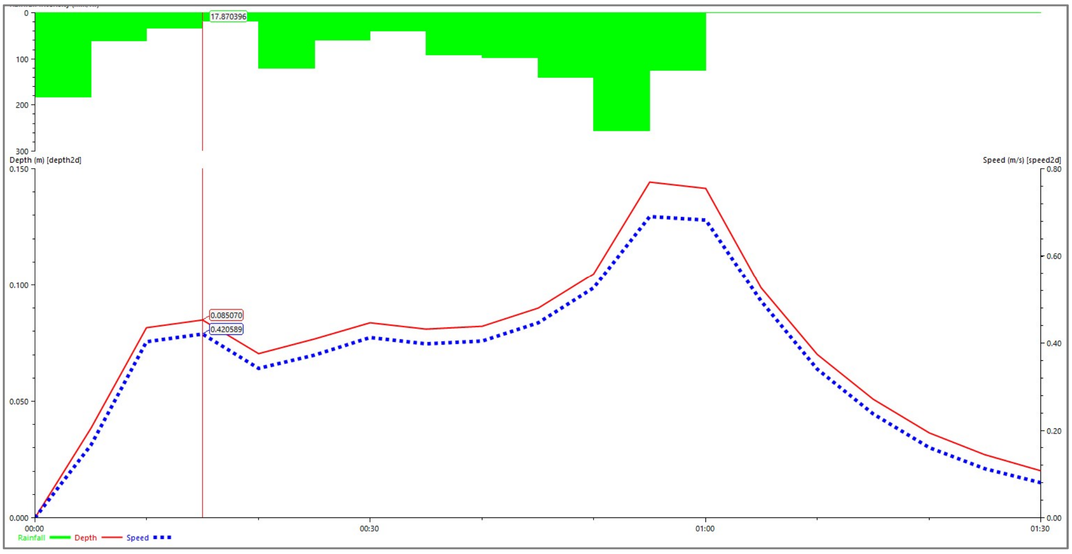
9. Select the **Graph** tool from the toolbars and click on one of the mesh elements. Use the **CTRL** key to Select **Depth (m) and Speed (m/s)**.





10. Right click on the graph and select **Graph Properties**. On the Traces tab, change the display of the Depth and Speed by double clicking on the current style for each. Once happy, Click **Default** and yes to saving as default then **OK**.

11. Right click on the graph again and select **Auto-label traces**. The will add the result value to the graph. To see the graph and the Geoplan together, go to **Window>Tile Vertically**. Now press the Run button again.



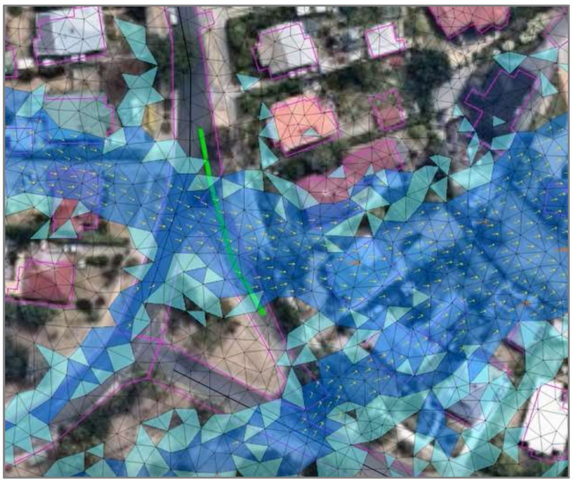
6D Saving Workspaces

12. To save this layout with the windows open, we can save the layout as a workspace. Right click on the Storm Tutorial and select **New>Workspace** give it the name 'Results Graph'

13. To check the workspace saved, you can close all open windows and then double click on the new workspace object to reopen.

6D Result Lines

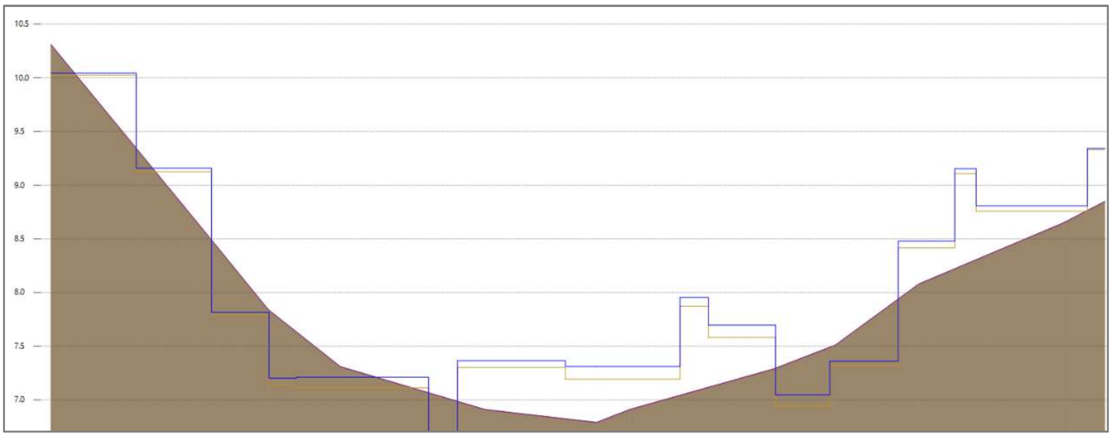
14. Select Line and click the New object tool to create a line across an overland flowpath. Give it the name '**Bournville Road**' and select **Result section** from the **Type** dropdown list.



15. Select the **Graph** tool and click on the New **result section line** and then select '**Flow through line**'. To save the Bournville Cres result section for use later on, right click on the Storm Tutorial and go to **New>Results analysis** and give it the name **'Bournville Cres Results'**.

6E. Flood Sections

16. Select the **New Flood Section** tool and draw a section through one of the overland flowpaths. Right click on the section and tick on **'Show Result Ground Level'** from the Flood sections plots. Use the **Play** button to see the water levels change over the simulation.



ⓘ **Note**: As we have not turned on the sediment deposition model, there should be no change in the Result Ground Level. This is a good way to check if the 2D mesh provides a good representation of the underlying terrain. From the above example, you will notice the invert of the channel is not represented in the Mesh.

17. Click on the **Clear Results** tool to clear the results from the Geoplan.

7. Adding 1d Pipe Network

7A. Setting node and link defaults

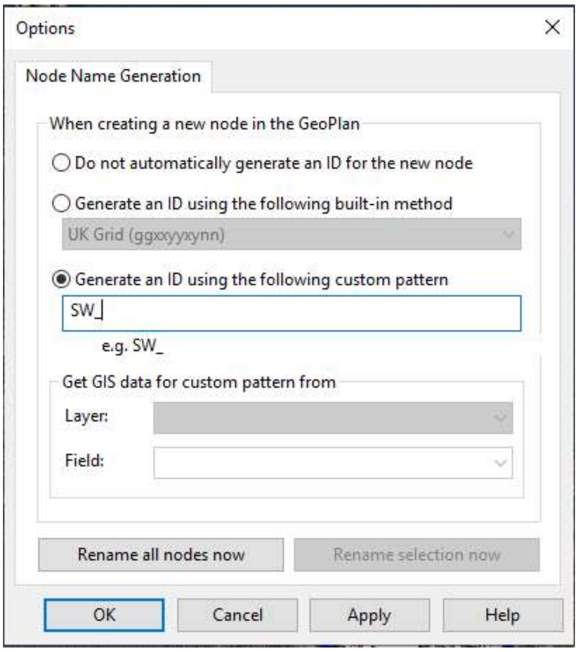
1. Go to **Network>User defined defaults** and select **Conduit**. Under the cross-section roughness parameters, set the **type** to 'N' and type '0.013' into both the top and bottom roughness Manning's n values.

Cross section roughness parameters	
Roughness type	N
Top roughness CW (mm)	1.500
Bottom roughness CW (mm)	1.500
Top roughness HW	
Bottom roughness HW	
Top roughness Manning's 1/n	25.000
Bottom roughness Manning's 1/n	25.000
Top roughness Manning's n	0.013
Bottom roughness Manning's n	0.013

2. Go to Network>User defined defaults and select Node. Set the Flood type to 'Gully 2D' and set the 1D-2D linkage basis to 'Elevation'.

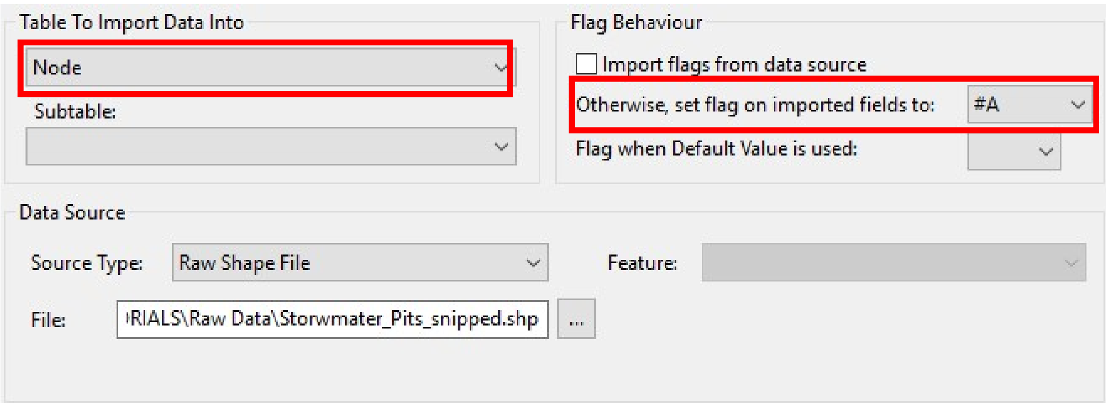
Default node parameters	
Flood type	Gully 2D
Flood area 1 (%)	10
Flood area 2 (%)	100
Flood depth 1 (m)	1.0
Flood depth 2 (m)	99.0
2D element area factor	1.0
Benching method	Full Benching
1D-2D linkage basis	Elevation

3. Go to Network>Node naming Options. Select Generate an ID using the following custom pattern and type in 'SW_', Select Apply and OK.



7B. Importing 1D Data

4. Go to Network>Import>Open Data Import Centre. Select Node from the dropdown list of tables to import data into and choose #A (for asset database) for the default flag. Choose Raw Shape File from the source type and navigate to the 'Storwmater_Pits_snipped.shp' file in the provided data.



5. Click the Load Config… button and navigate to 'node_import.cfg', this will map the fields in the shape file with the fields used in ICM for node objects. Click Import.

6. Repeat the process for the conduits. Select **Conduit** from the dropdown list of tables to import data into. Choose **Raw Shape File** from the source type and navigate to the **'Storwmater_Pipes_snipped.shp'** file in the provided data.

7. Load the 'pipe_import.cfg'. Click **Import**.

8. Click OK to the warning about multiple pipes at the same location.

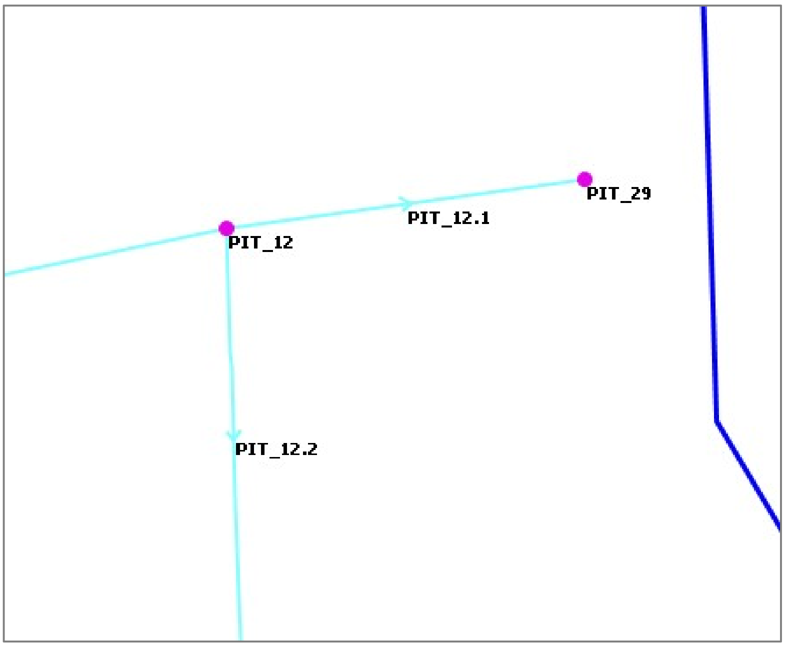
ⓘ **Note**: links exit PIT_12. They will have the same from node and will therefore be provided a different suffix.

7C. Connectivity and Pipe Direction

5. Drag on the **Theme: Storm Conduits** object onto the GeoPlan to show the pipe direction. Right click in the GeoPlan and go to **Properties and Themes** and tick on the **Auto label** for Nodes and Conduits.

ⓘ **Note**: Turn off the visibility of the background layers to see the 1d data clearly. To do this Right click in the GeoPlan and go to **GIS Layer Control**. **Uncheck** the Visible box for all.

6. To find the node that generated the warning, go to the **Find in GeoPlan** tool and type in 'PIT_12' and click **Find Next**.



9. Notice pipe 'PIT_12.2' needs to be reversed in direction as it currently draining away from the outlet (PIT_29). This could be fixed manually or the ICM tracing tools can be used.

10. Go to GeoPlan>Tracing Tools> Connectivity. This model has 5 sub-networks. Click on each row to highlight the sub-network in the GeoPlan. Sub-network 2 contains the pipes with incorrect direction.

11. With sub-network 2 selected, go to GeoPlan>Tracing Tools>Pipe Direction and click on the most downstream pipe in the network (PIT_12.1). Notice that 2 pipes have been selected, both with incorrect direction.

12. With the 2 pipes selected, go to Selection>Selection Operations>Reverse Links

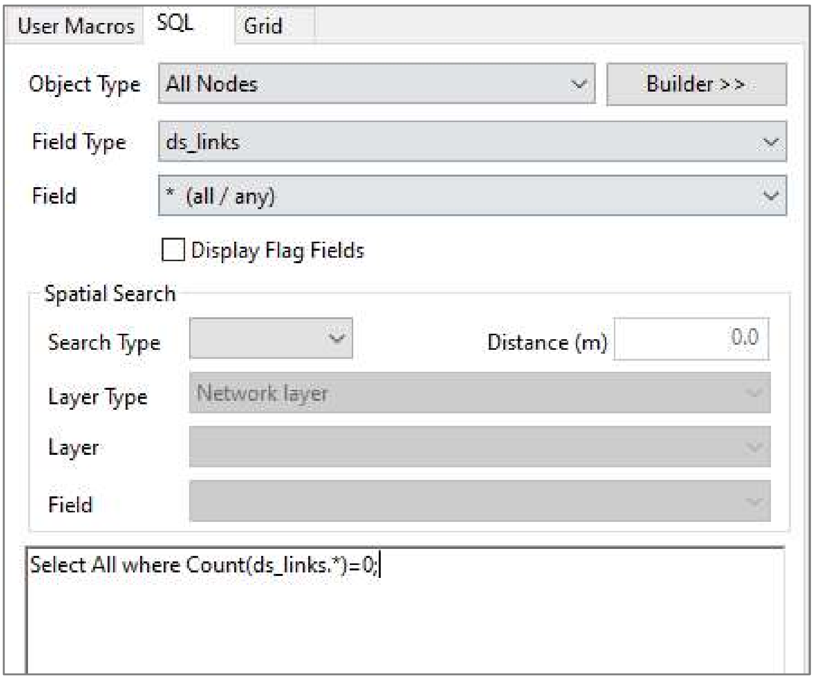
7D. Setting Node types

An underground drainage network will often consist of different node types: sealed access chambers/manholes, field inlets, gully pits, storage and headwalls. Each is treated differently in the hydraulic calculations however all nodes are currently setup as Gully 2D meaning they perform as an inlet gully pit. The GIS data used for this tutorial provided just a lintel size for gully pits (small, medium and large) but provided no further information for headwalls or sealed manholes.

Finding Outfalls with Stored Queries

13. We can identify the outfalls in the network by finding nodes that have no connected downstream links. To do this, go to Selection>SQL Select…

14. In the SQL dialog, type 'Select All where Count(' then select All Nodes from the Object Type, ds_links for the Field Type and *(all/any) and the Field. The finish the text ')=0;'



15. Ensure this is correct by clicking the **Test** button. It should return 5 selected items. Before applying, we will add some additional commands to change the **node type** to allow flows to discharge to the 2D as a headwall would in reality.

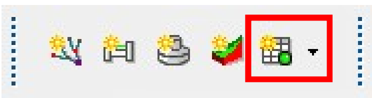
16. Commands in SQL are separated using the semi colon ';'. Add the additional line **'Update Selected Set node_type='Outfall 2D';'** click **Apply**. The 5 most downstream nodes should now be shown as a triangle rather than a circle due to the current theme.

17. Click the **Save** button and save the SQL in the model group with the name **'SQL: Set Outfalls'**.

ⓘ **Note**: Stored queries can be shared and run across different model groups and networks. Keep this one for next time you build your own model!

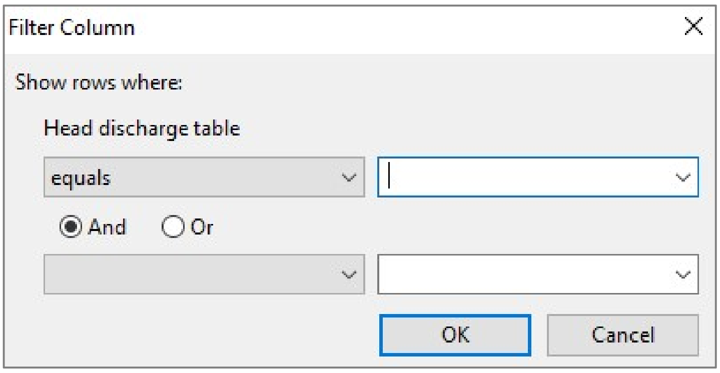
Setting Manhole/Sealed Chambers with Grid Windows

18. Go to **Window>Grid Windows>New Node Window** to view the available node fields as a table. Alternatively use the Grid Windows tool in the Windows toolbar.

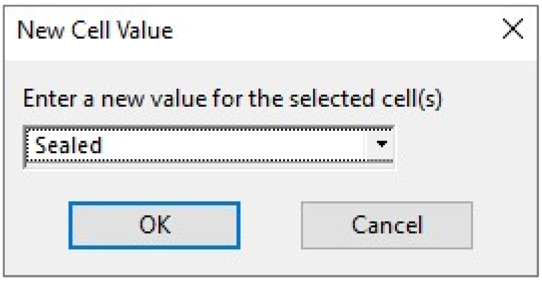


19. Scroll across to the **Head discharge table** column and notice the rows populated as either small, medium, large or null. In this tutorial, any row with a null is not an inlet gully and will be treated as a sealed manhole.

20. Right click on the **Head discharge table** heading and select **Filter**. Show rows where Head Discharge table equals NULL. Select OK.



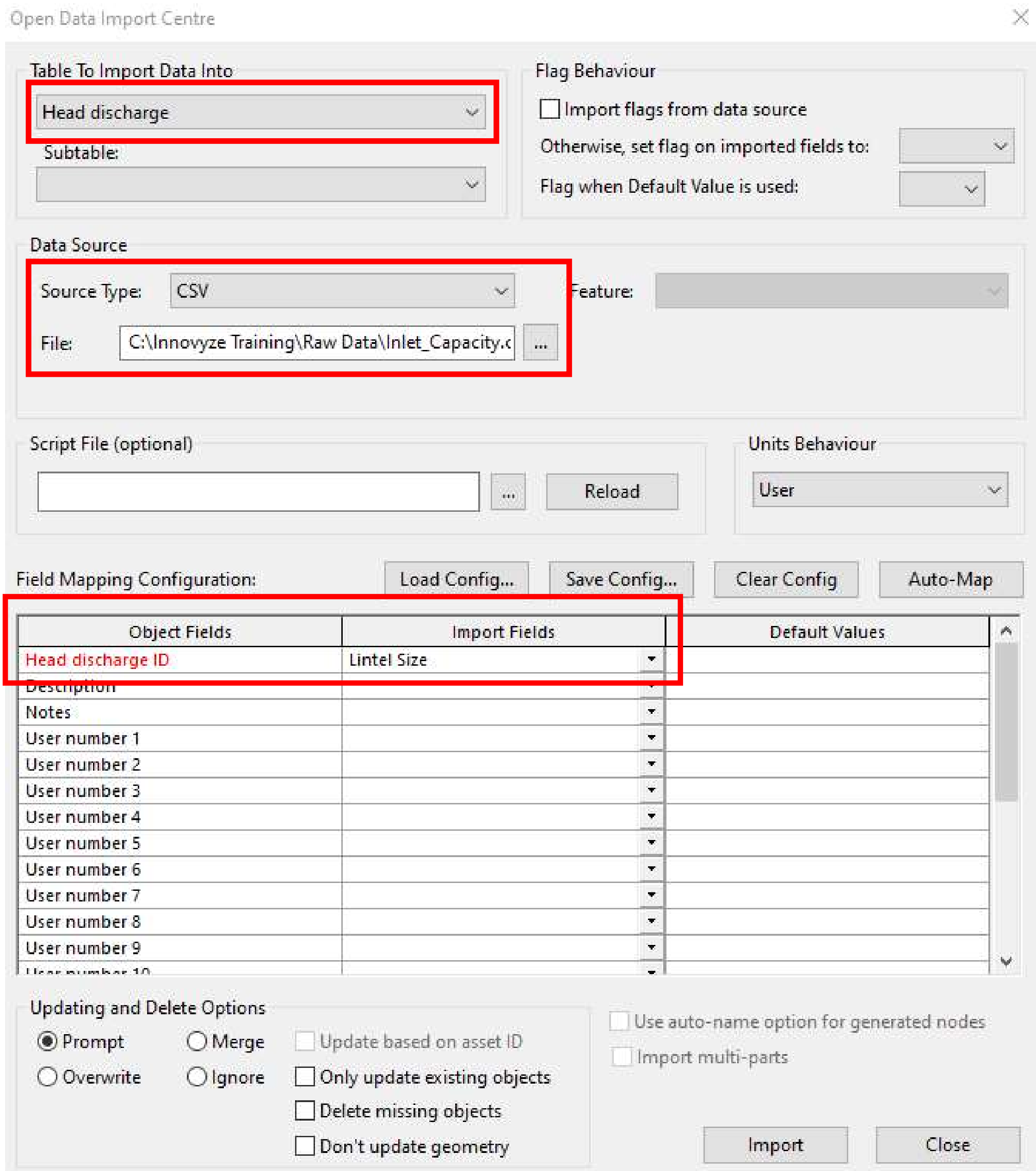
21. Left click on the **Flood type** column heading to select all the rows and then right click in the first row field (where is says Gully 2D) and **select Current cell value>Set New values for Cell**. Select **Sealed** from the list and click **OK**.



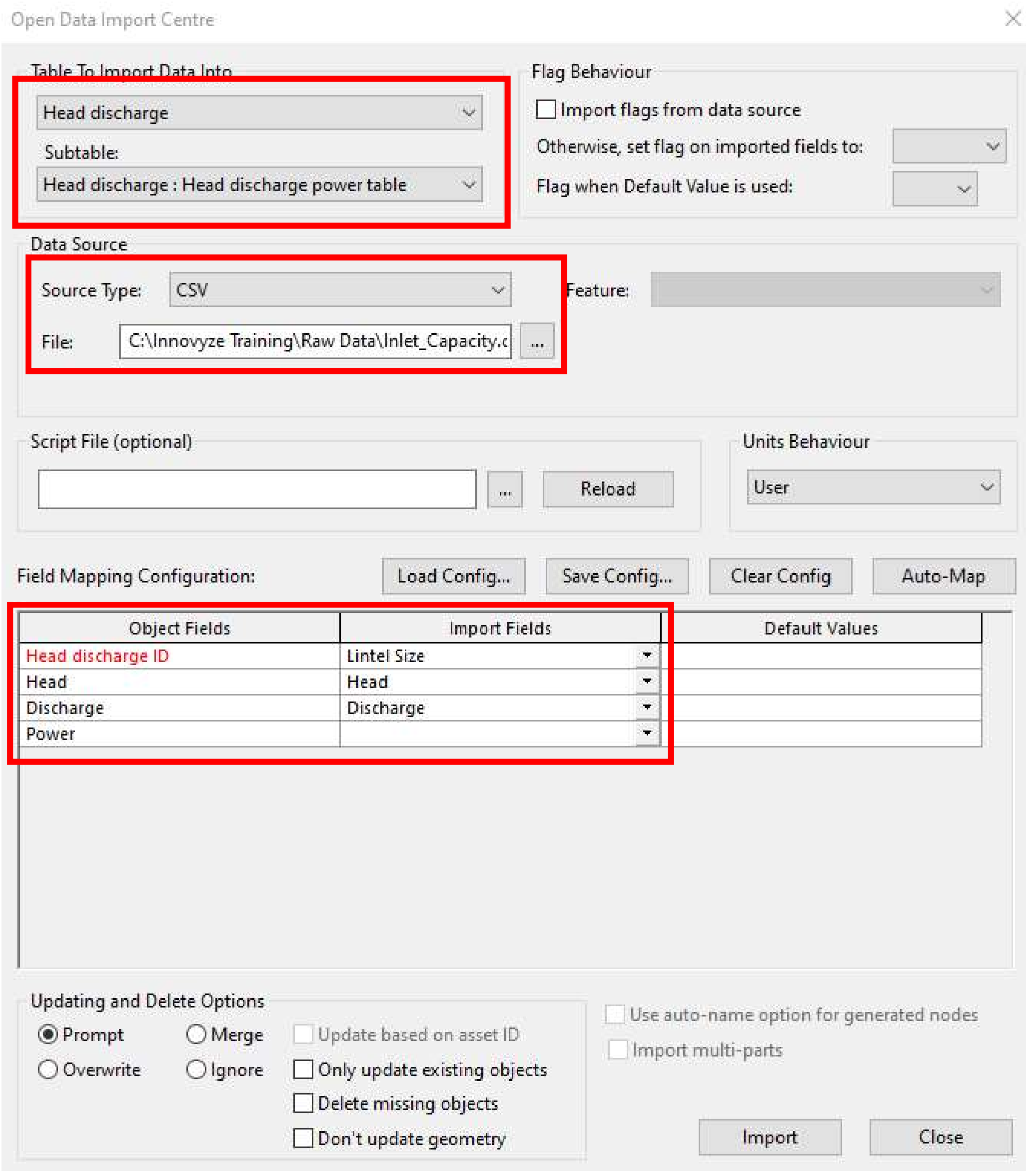
22. Right click on the **Head discharge table** and **clear** the filter.

Importing Inlet Capacity Charts

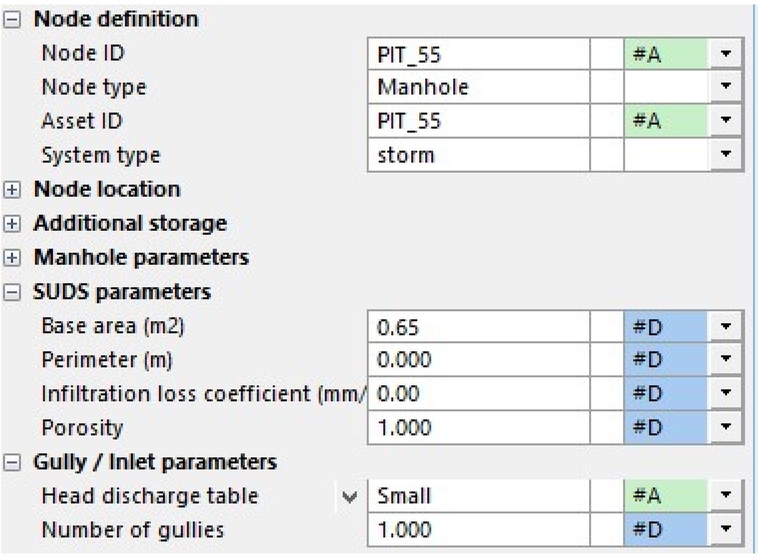
23. To add the inlet capacity chart titles, Go to **Network>Import>Open Data Import Centre**, select **Head discharge** from the table to import data into, choose **CSV** from the source type and select the file **Inlet_Capacity.csv** file from the raw data folder. Map the Object field with the import fields shown below. Select **Merge** for the updating and deleting options. Click **Import**.



24. To populate the depth discharge data for the new head discharge tables, select **Head discharge: Head discharge power table** from the subtable. Map the fields shown below and click **Import**.

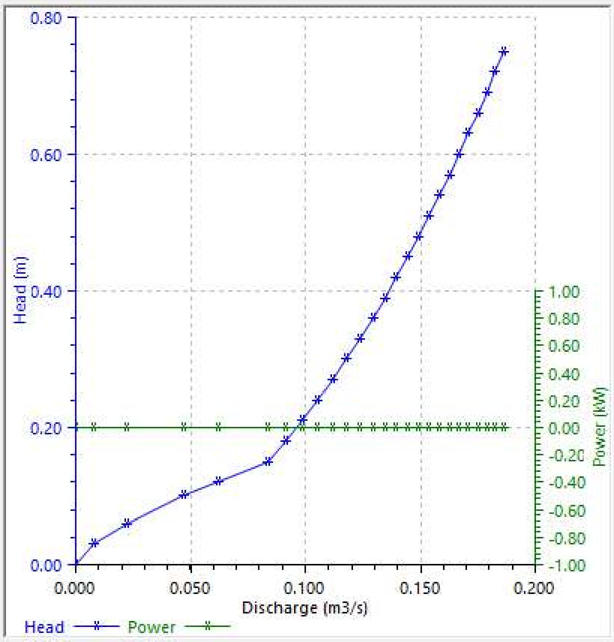


25. Use the **Find** tool to search for **Pit_55** and double click or use the **properties tool** to see the fields. Under the Gully/Inlet Parameters, open the **head discharge table** properties and then click on the 3 ellipses to see the depth discharge graph points.



Head discharge : Small : Head discharge power table

	Head (m)	Discharge (m3/s)	Power (kW)
1	0.0000	0.0000	
2	0.0300	0.0078	
3	0.0600	0.0220	
4	0.1000	0.0472	
5	0.1200	0.0621	
6	0.1500	0.0834	
7	0.1800	0.0913	
8	0.2100	0.0986	
9	0.2400	0.1055	
10	0.2700	0.1119	
11	0.3000	0.1179	
12	0.3300	0.1237	
13	0.3600	0.1292	
14	0.3900	0.1344	
15	0.4200	0.1395	
16	0.4500	0.1444	
17	0.4800	0.1491	
18	0.5100	0.1537	
19	0.5400	0.1582	
20	0.5700	0.1625	
21	0.6000	0.1667	

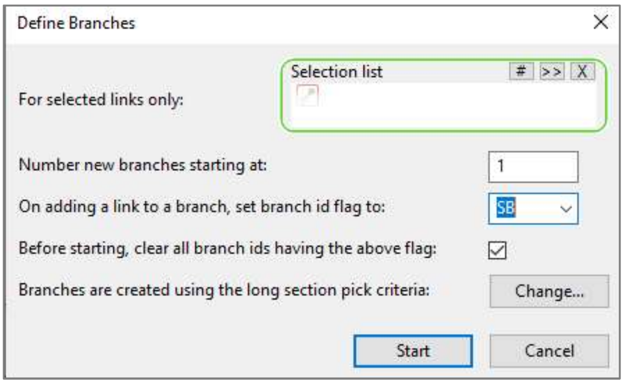


OK Cancel

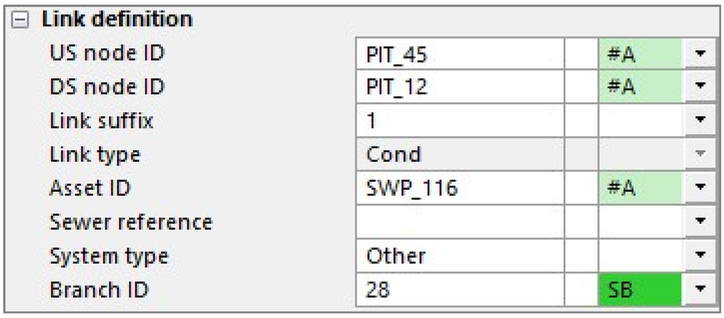
7E. Creating Branches & Long Sections

The Branch ID is a numeric field which can be used to identify to which long section a link is associate. This allows all conduits to be plotted easily.

26. Use the **Select All Objects** tool to select all pipes in the model. With all selected, go to **Model>Define Branches.**. Set the **Branch id Flag** to your initials and click **Start**.

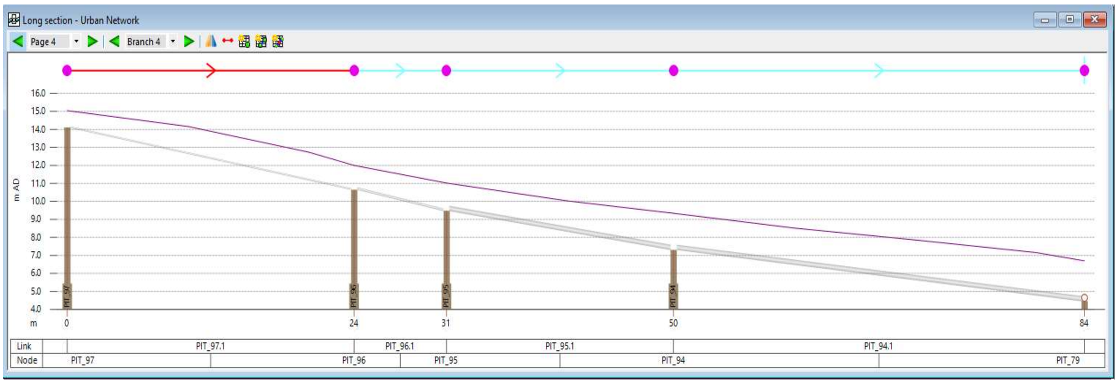


27. View the properties of a link by double clicking or using the **Properties** tool and note the newly populated Branch ID and flag.



28. Open a new Long Section Window by going to **window>New Long Section window** or clicking on the **New Long Section** tool. Tile the windows horizontally.

29. Select all objects in the GeoPlan and then right click in the Long Section window and select **'Create from Selected branches'**. Right click and select **Properties**. Tick on the **'Show Ground Model'** and **'Show 2D Levels'**. Click **Save>Save as default for this network.**



7. Use the **arrow keys** to move through the branches/pages. Note some invert levels are missing from the conduits and the pits currently have no ground/lid levels.

7F. Using the Inference Tool

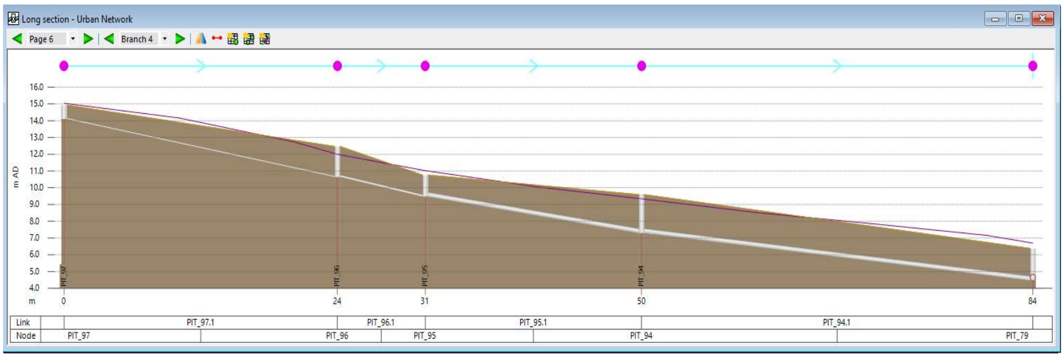
30. Right click on the Storm Tutorial model group and select **New>Inference**. Leave the name **'Inference'** and click **OK**.

31. Click **None** to uncheck all options. Then tick on:

- NODE: Ground level from 2D mesh
- CONDUIT: Invert from invert level
- check 'Use chamber floors if required' and 'Infer invert levels equal to zero'
- CONDUIT: Invert interpolate from inverts
- CONDUIT: Headloss type and coefficient
- check 'Infer headloss with default flags (#D)'

32. Select **your initials** from the Flag inferred values and click **Save**.

33. **Select all objects** in the GeoPlan and then drag the inference object onto the GeoPlan.



34. Right click on the Urban Network and **Commit the changes to the master database** and provide the comment '**Added 1d Network and cleansed for model**' and **select** Yes to validating the Model. The network is now ready to run with the 1d pipe system.

8. Advanced Mesh Options

A single mesh element may be made up of more than one triangle, if a triangle has an area less than the Minimum element area specified for the 2D Zone. Triangles will be aggregated with adjacent triangles until the minimum area is achieved. The mesh elements generated cannot be edited manually, however mesh level zones, mesh zones, bank lines and voids can be added to manipulate the mesh. This is often required to best represent the ground terrain.

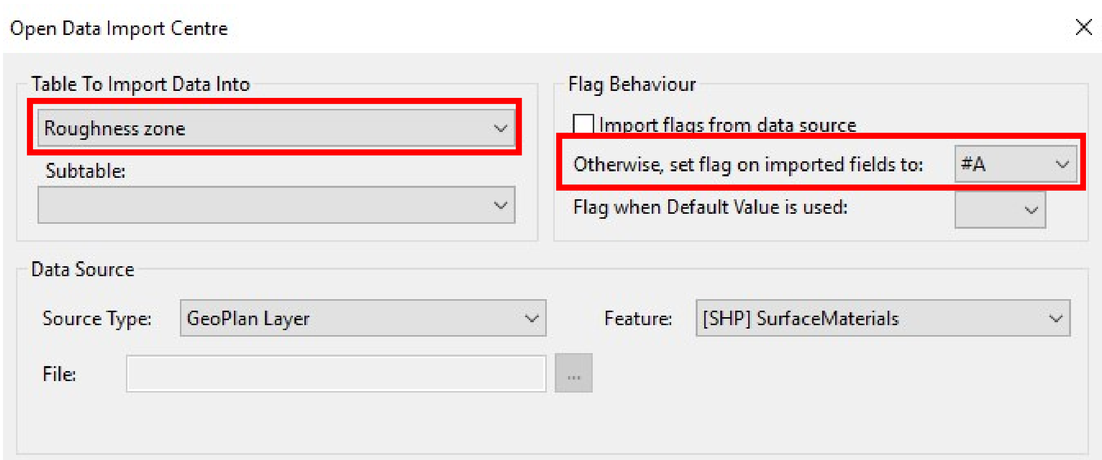
8A. Adding Roughness Zones

1. We will add some roughness polygons using the landuses from the background layer through the Open Data Import Centre. Before we do, lets have a look at the information available. Double click on one of the building outlines and select [SHP} SurfaceMaterials from the selection list. Click OK. The available properties are populated, the object name and the source type but not the manning's roughness value.

Name	Value
name	Building77
source	Building

2. To use the data in the model network we need to firstly import rather than have it as a background layer. Go to **Network>Import>Open Data Import Centre.**

3. Select **Roughness Zone** from the dropdown list of tables to import data into and choose **#A** (for asset database) for the default flag. Choose **GeoPlan Layer** from the source type and **[SHP] SurfaceMaterials** from the Feature dropdown.



4. Map the InfoWorks object fields with the Import Fields (**ID = name, Notes = source**) and Click Import

Field Mapping Configuration: Load Config... Save Config... Clear Config Auto-Map

Object Fields	Import Fields	Default Values
ID	polygon_id	
Area		
Exclude roughness zone boundary whei		
Roughness (Manning's n)		
Notes	notes	
User number 1		

5. Under **Properties and themes**.. tick on the **Display** checkbox for the **Roughness**. Drag the '**Theme: Roughness Zones** object onto the GeoPlan to view the different sources.

6. The roughness polygons can be viewed in GRID view by going to **Window>Grid Windows> New Polygon Window** and click on the **Roughness tab**. To view both the plan and the GRID, go to **Window>Tile Vertically**.

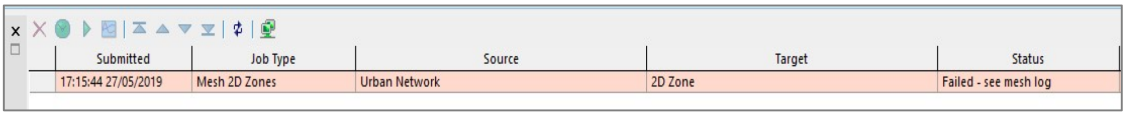
7. Double click on the '**SQL: Set 2D Mannings**' Stored query object from the Model Group. Change the manning's roughness of Buildings from 0.016 to 0.02 and press **Save** and **Apply**. The roughness values should now be updated based on the landuse set in the **notes** column.

```
SELECT ALL WHERE notes="Building";
UPDATE SELECTED SET roughness=0.02
DESELECT ALL;
SELECT ALL WHERE notes="Roads";
UPDATE SELECTED SET roughness=0.018;
DESELECT ALL;
SELECT ALL WHERE notes="Vegetation";
UPDATE SELECTED SET roughness=0.07;
DESELECT ALL;
SELECT ALL WHERE notes="Carpark";
UPDATE SELECTED SET roughness=0.018;
DESELECT ALL;
```

Carpark441	1.412	☒	0.0180	Carpark
Carpark442	0.155	☒	0.0180	Carpark
Carpark443	0.124	☒	0.0180	Carpark
Carpark444	0.236	☒	0.0180	Carpark
Carpark445	1.819	☒	0.0180	Carpark
Vegetation446	0.637	☒	0.0700	Vegetation
Vegetation447	0.240	☒	0.0700	Vegetation
Vegetation448	0.091	☒	0.0700	Vegetation
Vegetation449	1.631	☒	0.0700	Vegetation
Vegetation450	0.660	☒	0.0700	Vegetation
Vegetation451	0.263	☒	0.0700	Vegetation
Vegetation452	0.142	☒	0.0700	Vegetation
Vegetation453	0.997	☒	0.0700	Vegetation
Vegetation454	0.077	☒	0.0700	Vegetation
Vegetation455	0.075	☒	0.0700	Vegetation

8. The mesh needs to be updated to include the new roughness zones. Select the **2D Zone** and go to **Model>Meshing>Mesh 2D zones**.., the Ground model should already be selected as it will be stored in memory of the mesh. Click **OK**.

9. The status of the meshing can be seen in the **Job Control Window**. You should see that this time, the meshing will fail and the row will be coloured red instead of green.



Submitted	Job Type	Source	Target	Status
17:15:44 27/05/2019	Mesh 2D Zones	Urban Network	2D Zone	Failed - see mesh log

10. Click on the **Status 'Failed – see mesh log'** and select **Show Log**. Here you will see any warnings or errors which caused the meshing process to fail. As there is no priority fields in the manning's roughness polygons, overlapping polygons provide conflicting information and therefore return an error.

WARNING	Roughness zones crossing 2D zone boundary: Vegetation450, Roads464, Vegetation456, Roads465, Roads466, Building61, Building23, Building415, Vegetation449, Roads470, Building412, Vegetation446, Vegetation447
ERROR	Roughness zones Building207 and Carpark444 intersect
ERROR	Roughness zones Building327 and Vegetation452 intersect
ERROR	Roughness zones Building417 and Vegetation447 intersect

11. Close the Log and Discard the Mesh. We will need to remove the overlapping polygons before continuing.

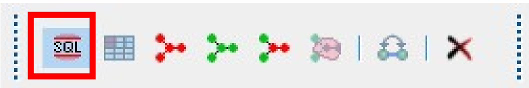
8B. Using the Geometry Tools

The Geometry tools can be used to quickly clean up network data such as overlapping polygons or unnecessary vertices or gaps between objects (slithers). In this tutorial, we will prioritise the Buildings (B in the below example) over the Vegetation, Carpark and Road (A in the below example).

A and B overlapping

A Cropped against B

12. Firstly, we will create the 'A' and 'B' selections. Go to **Selection>SQL Select** and select **Roughness zone** from the object type. Type in the following into the box: **notes="Vegetation" OR notes="Carpark" OR notes="Roads"** click **Test** and then **Apply**.



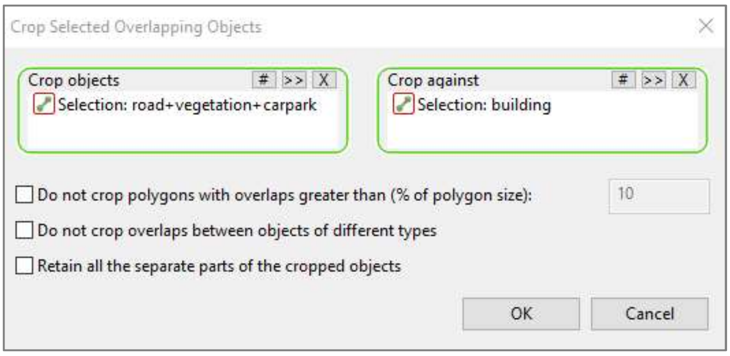
Tip: You don't need to close the SQL dialog box to perform other commands

13. With the roughness polygons selected, go to **Selection>Save Selection…** give it the name **'Selection: carpark+vegetation'** and highlight the **Storm Tutorial** model group to save it in.

14. Now clear the selection by going to **Selection>Clear selection** or using the toolbars 

15. Repeat the process with the Building polygons. Change the text in the SQL to **notes="Building"**, click apply and then save the selection as **'Selection: Building'**

16. To crop the polygons go to Model>Geometry>Crop selected overlapping objects.. Drag the '**Selection: vegetation+carpark**' object into the Crop objects window and the 'selection: buildings' object into the crop against window. Click **OK**.



17. The mesh needs to be updated to include the new roughness zones. Select the **2D Zone** and go to **Model>Meshing>Mesh 2D zones..**, Click **OK**. The Mesh should be successful, **Load** the mesh.

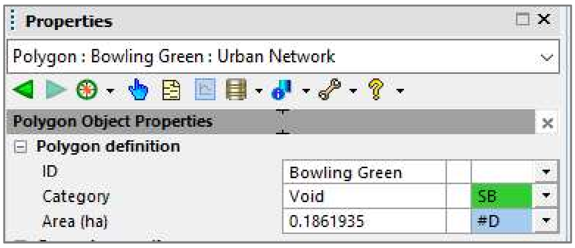
18. Now is a good time to commit the changes, Right click on the **Urban Network object**, select **commit changes** and provide the comment **'Added Roughness zones'** select **Yes** to validating the network.

8C. Adding Voids

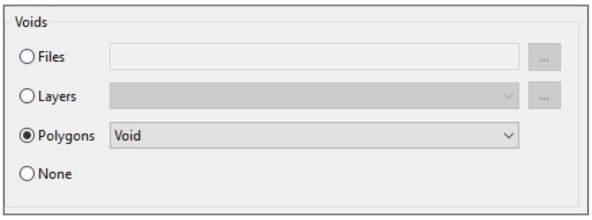
19. In the south-east corner of the catchment, you will notice a bowling green. We will assume this has its own drainage system that discharges outside our area of interest (ie. We will remove from the model). Using the **new object** tool, draw a new **polygon** around the green. Give it the **ID 'Bowling Green'** and **Type: Polygon**.

ⓘ **Note**: you may need to turn the visibility on for polygons (top row) in **Properties and themes…**

20. In the object properties, set the **Category** to **'Void'**.



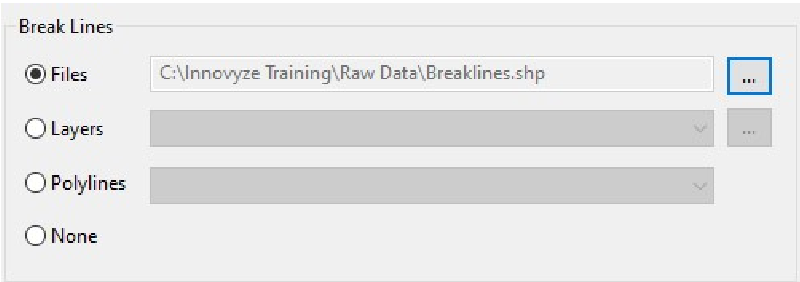
21. Select the 2D zone and go to Model>Meshing>Mesh 2D zone… Select 'Polygons' under the voids section and select OK and OK and then load the mesh when complete.



22. Zoom into the green, you should see no more elements within the bowls green.

8D. Adding Break Lines

23. From the start of the tutorial, you will remember we added the road crows as a background layer. We will now add them to the mesh as break lines to better define the roads. Select the 2D zone, go to Model>Meshing>Mesh 2D zone… and this time under the Break Lines section select the Files and choose breaklines.shp from the raw data folder. Select OK and OK and then load the mesh when complete.



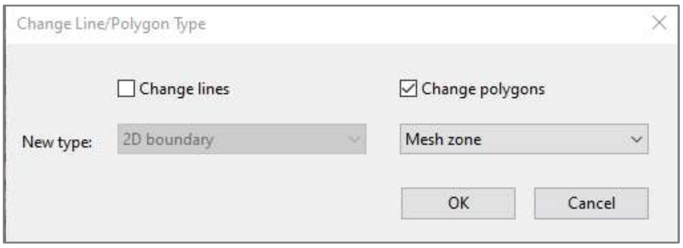
24. If you zoom into a road section, you should now see the elements snapped to the road crown.

25. Commit and validate the change and provide the comment 'Added road crowns & breaklines to Mesh'

8E. Adding Mesh Zones

The mesh elements used in the model to date have been too large to adequately represent the flowpaths in the roads. Mesh zones can be used to change the mesh element sizes locally.

26. Check nothing is currently selected by using the clear selection tool. Then drag on the object 'SQL: Select Road Polygons'
27. To copy the selected objects, go to Edit>Copy objects, then go to Edit>paste objects
28. While the pasted objects are selected, go to Selection>Selection Operations>Change line/Polygon Type and change the polygons to Mesh Zone



29. Open the Grid Window of the polygons by going to Window>Grid Windows>New polygon window and go to the Mesh zone tab.
30. Click on the Maximum triangle area column to highlight the column data. Right click in the first row field and go to current cell value>Set New cell value and type in the value 5 and click OK.
31. Check on the Override 2D zone minimum element area setting and set the Minimum element area to 1.

ID	Area (ha)	Maximum triangle area (m2)	Override 2D zone minimum element area setting	Minimum element area (m2)	Ground level modification	Raise by (m)	Level (m AD)	Notes
Roads295!	0.366	5.000	☒	1.000	None	0.000	0.000	Roads
Roads296!	0.311	5.000	☒	1.000	None	0.000	0.000	Roads
Roads297!	0.320	5.000	☒	1.000	None	0.000	0.000	Roads
Roads298!	0.535	5.000	☒	1.000	None	0.000	0.000	Roads
Roads299!	0.356	5.000	☒	1.000	None	0.000	0.000	Roads
Roads300!	0.220	5.000	☒	1.000	None	0.000	0.000	Roads
Roads301!	0.345	5.000	☒	1.000	None	0.000	0.000	Roads
Roads302!	0.049	5.000	☒	1.000	None	0.000	0.000	Roads
			☐					

32. Remesh and load the new mesh into the GeoPlan. The mesh element density in the roads should now provide much better definition.

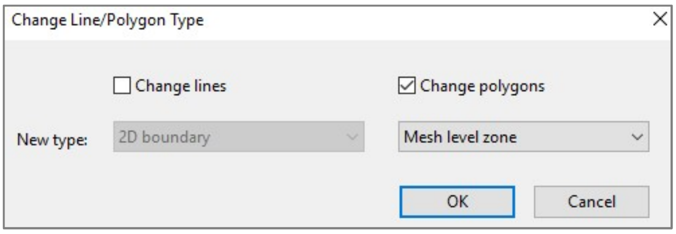
8F. Raising the Buildings in the Mesh

To better represent the overland flowpaths in the model, we will raise the buildings up above the flood level. We will use mesh level zone objects to do this.

33. The building roughness polygons can be reused. Drag the 'Selection: building' onto the GeoPlan.

34. With the buildings selected, go to Edit>Copy object(s), Edit>Past Append object(s)

35. We now need to move the selection from the Roughness object type to the Mesh level zone object type. Go to Selection>Selection operations and select Change Polygons and choose Mesh level zone from the drop down. Click Ok.



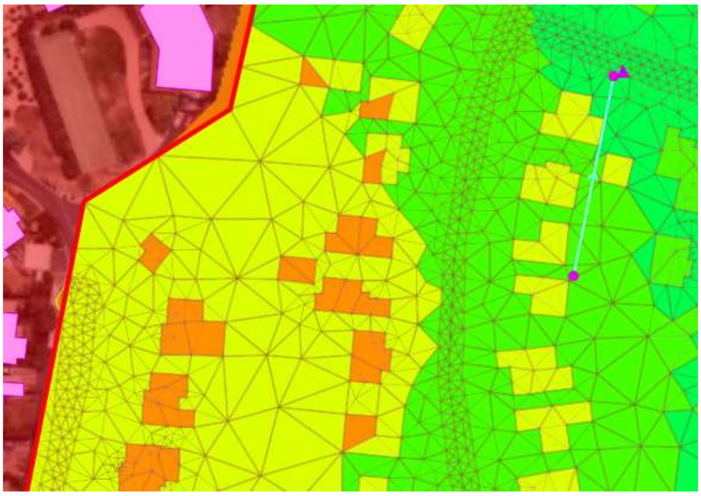
36. We could adjust the vertex of each polygon individually, this would be useful if we were correcting lidar data or perhaps adding a design feature such as a levee. For ease we will just raise all the buildings equally by 2.5m.

37. Go to **Selection>SQL Select**, Select Mesh level zone from the object type and then type in **'SET level_type='Adjust'; SET level_sections.elev_adjust=2.5'** and click **Apply**. Alternatively, drag the **'SQL: Set Building Levels'** object into the GeoPlan

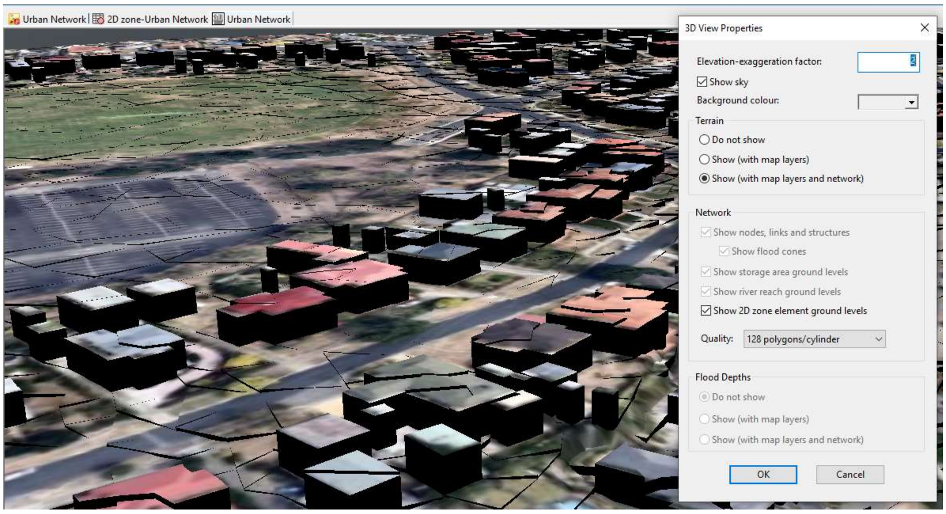
38. Select the **2D Zone**, go to **Model>Meshing>Mesh 2D zones…** Click OK and load the mesh into the network.

39. Commit and validate the change and provide the comment 'Added detailed road mesh levels (max 5m min 1m) & Raised buildings 2.5m from mesh'

40. It is important to check the changes to mesh look and behave as we expected. We can do this in the GeoPlan by using a colour scheme. Drag the **'2D Element Elevations'** object into the GeoPlan. You should notice the buildings will be higher than the surrounding elevations.



41. These changes can be viewed in 3D view. Go to **Windows>New 3D Network Window**. Use the left mouse button to spin the view. Right click and go to **Properties** to change the view settings. Ensure you have **'Show 2D Zone element ground levels'** checked on.



8G. Adding 2D Result Locations

You may remember we used a result Section earlier to review flows in the 2D mesh. For a more detailed result calculated during the simulation and that can be used for statistical analysis such as the ARR ensemble statistics, we want to use Network Result objects.

42. Drag the 'Bournville Cres Results' object into the geoplan to get the location. Create a new object using the Line tool and trace over the existing results section (remember to draw left to right looking downstream – towards the bowling green).

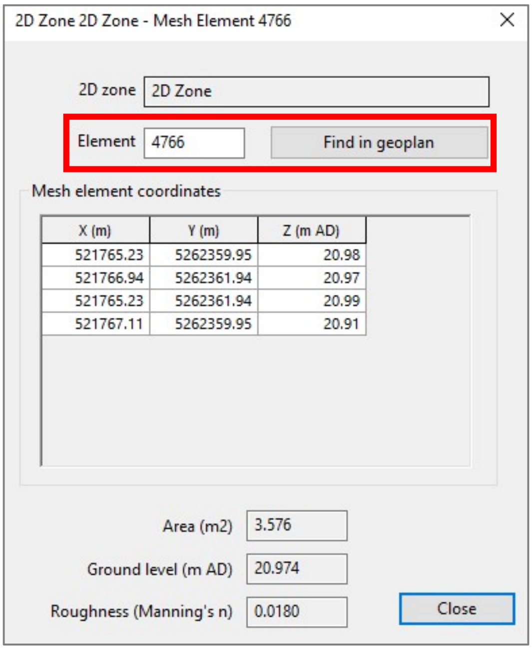
43. Give the line the name "Network Result" and set the type to "Network Result Line (2D)"

8H. Reviewing the Mesh

We have made a number of changes to the mesh. It is important we check the changes we have made has resulted in the desired mesh and check we haven't introduced any problematic elements. Small elements are not desirable as they slow down meshing times can be detrimental to simulation times and can, in some circumstances, lead to mass balance.

44. Last commit and validate, you may have noticed a validation warning (yellow) message 'W2285: Element XXX has area (xxm2) less than 50% of minimum element area (25m2)' appeared in your message box.

45. To find the warning, double click and select any mesh element. Type in the element number in the warning message into the Element text box and click Find in geoplan.



46. Use the Edit Object Geometry tool to adjust the vertices that are forcing the mesh result in a small element.

Tip: Use the CTRL key to add vertices and the ALT key to remove them.

47. Select the 2D Zone, go to Model>Meshing>Mesh 2D zones... Click OK.

48. The Mesh process should be successful, load the mesh into the model using the Model> Meshing>Load mesh
49. Commit the changes and provide the comment “Removed small elements”

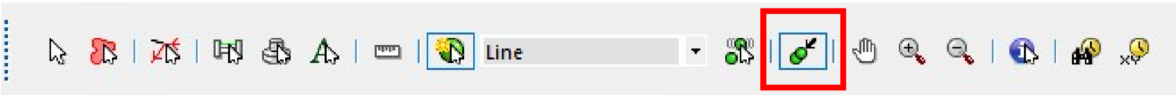
9. Additional Boundary Controls

The network we have developed so far has given no consideration to the environment conditions downstream. From the background aerial we can see the catchment discharges to open water and consideration to the water level should be given.

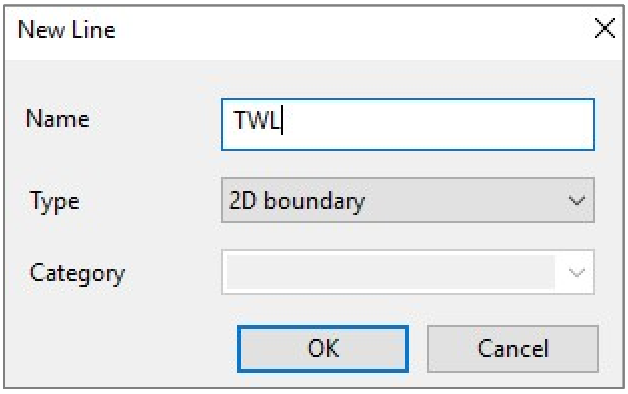
9A. Creating 2D Boundary

1. We will add a Level boundary to control the tailwater level. Select **Line** from the drop-down menu in the GeoPlan toolbar and use the **New Object** Tool to draw a line following the outlet to the River. It is important to snap the boundary to the 2D Zone, ensure the Snapping Option is selected.

   ⓘ **Note**: it is important to snap the boundary to the 2D Zone, ensure the snapping option is selected.



2. Give the new boundary the name 'TWL' and select **2D Boundary** from the Type drop down.



3. Under the properties of the newly created 2D Boundary, Set the line type to **Level**.

ⓘ **Note**: this tutorial will not be running the water quality module as such the bed load and suspended load types are not used during the simulation.

2D boundary Object Properties		
Boundary definition		
Name	TWL	
Boundary line type	Level	SB
Length (m)	1122.779	#D
Bed load boundary type	Pollutograph	#D
Suspended load boundary type	Pollutograph	#D

4. The model now has 2 different boundary types

8. The first is the original boundary type we set on the 2D zone, which was 'Normal Condition' or free outfall.
9. The second is the water level at the catchment discharge point. This 2D Boundary lines overwrite the boundary type set on the 2D zone.

Innovyze®
Empowering water experts

5. Right click on the Network Model object and select **Commit changes to master database**. Add the comment '**Added Downstream TWL Boundary**' and select **OK and Yes** to validating the model before committing changes.

9B. Level Objects

6. Right click on the Model Group and select **New>Level** and give it the name '**TWL 1% AEP**' and click **OK**. The new object will appear under the Model Group, double click to open the object. Type in **1.8** into the first and second row.

ⓘ **Note**: These numbers are made up for this tutorial and do not reflect the tidal values and climate change in the area.

7. Right click on one of the rows you have just typed into, select **Properties** and type in '**TWL**' into the **Node reference**.

Properties

Parameter	Value
Node reference	TWL
Pipe level or datum (m AD)	0.000
Descriptive title	

8. Right click on one of the rows you have just typed into, select **Sub-event Parameters**. Type in 1d (1 day) for the **timestep**. Click **Apply** and **OK**.

Sub-event parameters: 1

Properties

Parameter	Value
Event reference number	0
Start date	00/00/0000
Start time	00:00:00
Timestep (<n>[<u>], u=s,m,h,d)	1d
No. of hydrographs	1
Data type flag (0, 1 or 2)	0

9. Now click **File>Save** to save the new object data.

9C. Setting Initial Water Levels (IWL)

10. Right click on the Model Group and select **New>Level** and keep the name '**Initial Conditions 2D**' and click **OK**. IWL 1% AEP

11. Populate the Zone ID with '2D zone', the zone type with 2D zone, the variable as hydraulic, the level type as Elevation and the Elevation as 1.8.

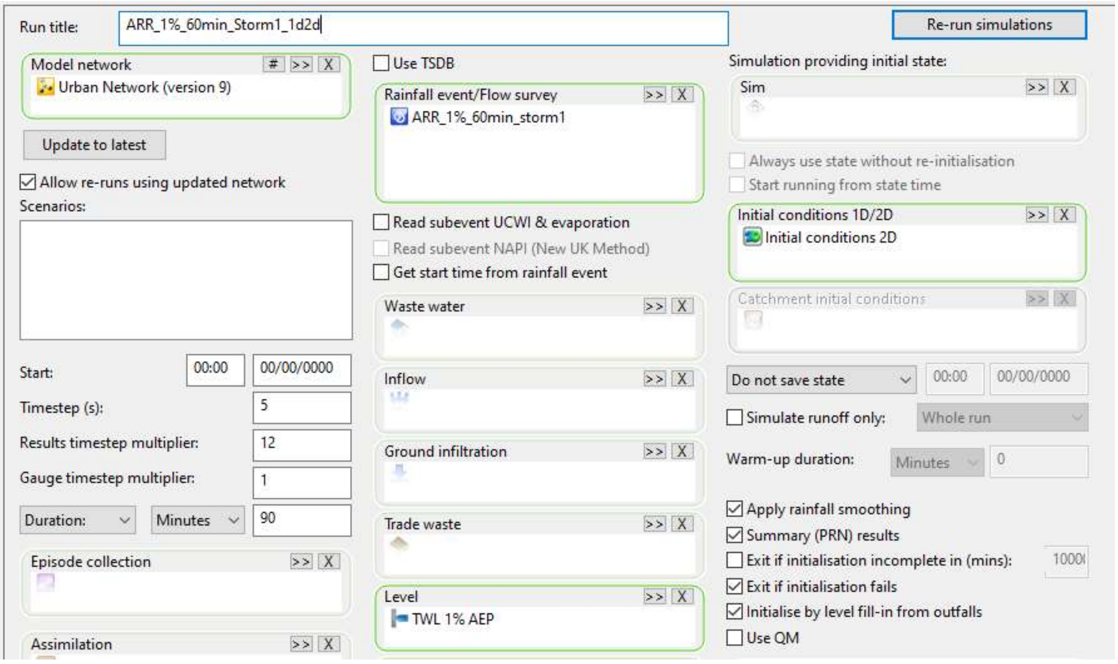
Zone ID	Zone type	Variable	Concentration (mg/l)	Concentration (kg/m3)	Temperature (degC)	Quantity	Horton soil water content (%)	Green-Ampt soil moisture deficit (%)	Level type	Depth (m)	Elevation (m AD)	Speed (m/s)	Direction (radians)
2D zone	2D zone	Hydraulic	0.000	0.000	0.000	0.000	0.000	0.000	Elevation	0.000	1.800	0.000	0.000

ⓘ **Note**: There is no need to set the initial conditions on the 1d network as ICM calculates this during the model initialisation stage.

10. Result Reporting

10A. Re-running Simulations

1. Double click on the original run object 'ARR_1%_60min_Storm1_2D_Only' and change the name to 'ARR_1%_60min_Storm1_1D2D'
2. Click on the Update to Latest to ensure you are using the latest version of the network that includes the 1d pipes, the advanced mesh and the boundary conditions. Ensure you add the Initial Conditions 2D and the TWL 1% AEP objects.
3. Under the 2D parameters, check the 'Link 1D and 2D calculations at minor timestep'. Finally, Re-run the simulation.



4. When the simulation is complete, right click on the result object an open the Log Results.
5. Review the Volume Balance Error % under the Volume Balance Report. Ideally this is close to 0, however generally less than 5% is acceptable.

```
Volume balance summary :
VBE -> Volume balance error (m3) :                          0.6020
VBE = FV+FVC-IT-IV-IVC+OT
VBEP -> Volume balance error % :                            0.0015
VBEP = VBE*100/(IV+IVC+IT)
VBEPIO -> VBE as % of inflow + outflow :                    0.0008
VBEPIO = VBE*100/(IT+OT)
```

6. Review the Elapsed Clock Time, note the difference in run times compared to the original run that lacked the detail of the 1d network and mesh.

7. Review the simulation warning.

```
Warning 1053: One or more nodes had their flow limited from the 2d
zone; see node results grid for more details.
```

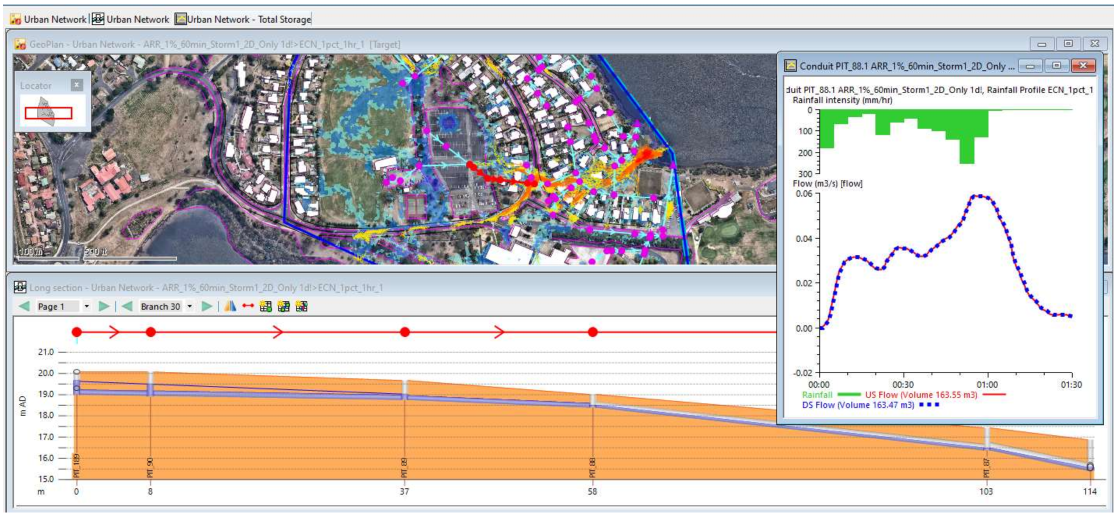
Tip: This warning may just be indicating that the head discharge curves provided in the gully 2d charts is not realistic for the location of the pit and that there is not enough volume on the 2D zone. In this case, the warning can be ignored. However, it is a good idea to check that there are not locations where the element is too small for the connected inlet/outlet.

10B. Using Grid Views

8. To view each node where flow limiting occurred, drag the result object into the GeoPlan. Go to **Window>Grid Windows>New Node Result Window**
9. Select the column **'Total Cumulative Limited Volume (m3)'**, right click and **sort of selected columns descending**. Review any nodes with high volumes for long periods.

10C. Creating Print Layouts

10. First, we will create a new spatial bookmark. To do this zoom in to an area of interest in the model such as the lower right corner near the bowls club. Then go to **GePlan>Spatial Bookmarks>Add From View…** Give it the name *'Print Layout'*
11. Use the **long section pick** to create a long section window and graph the flow through one of the pipes using the **Graph tool**.

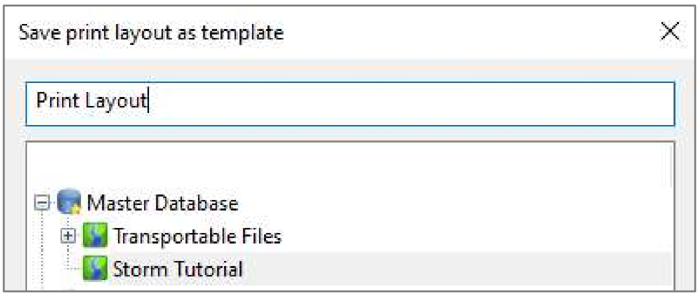


12. Open a new print layout by going to **File>Print Layout**. Click on the **Print Settings** to select a printer, page size (A3) and orientation (Landscape).

13. Select the **Geoplan** from the **Data Views** and drag onto the blank page. Move it into place by using the cover boxes to drag. Do the same with the **long section** and **graph**.

14. Select the **Text** from the **frame types** and drag into the page. **Double click** on the text box add a title.

15. Save the print layout by clicking on the **save** icon. Save as the **Print Layout** under the Storm Tutorial.



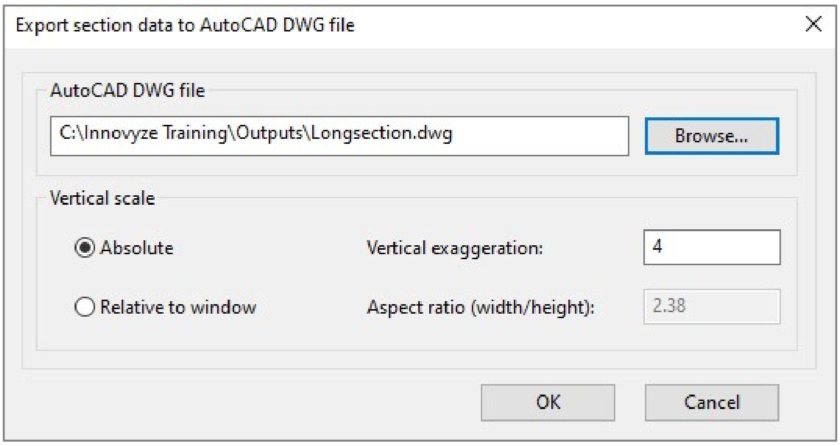
16. Press **print** to print the layout to PDF or to your chosen printer. Close the Print Layout.

10D. Exporting Long Sections

17. Maximise the long section window. **Right click** and go to **Properties** to modify the view. The GRID labels can be added to show different fields of the nodes and links such as the maximum velocity.

18. Click the **Load** button and browse to the **'longsections_plot.iws'** in the Raw Data folder. Select **Load** and note the changes to the link data table at the bottom of the long section.

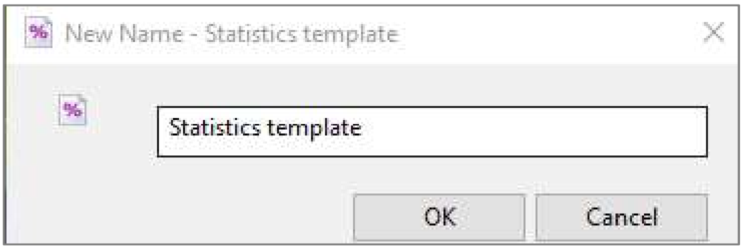
19. Right click and select **Export to AutoCAD DWG**. Select a location and file name for the output and put **4** for the **Vertical exaggeration**.



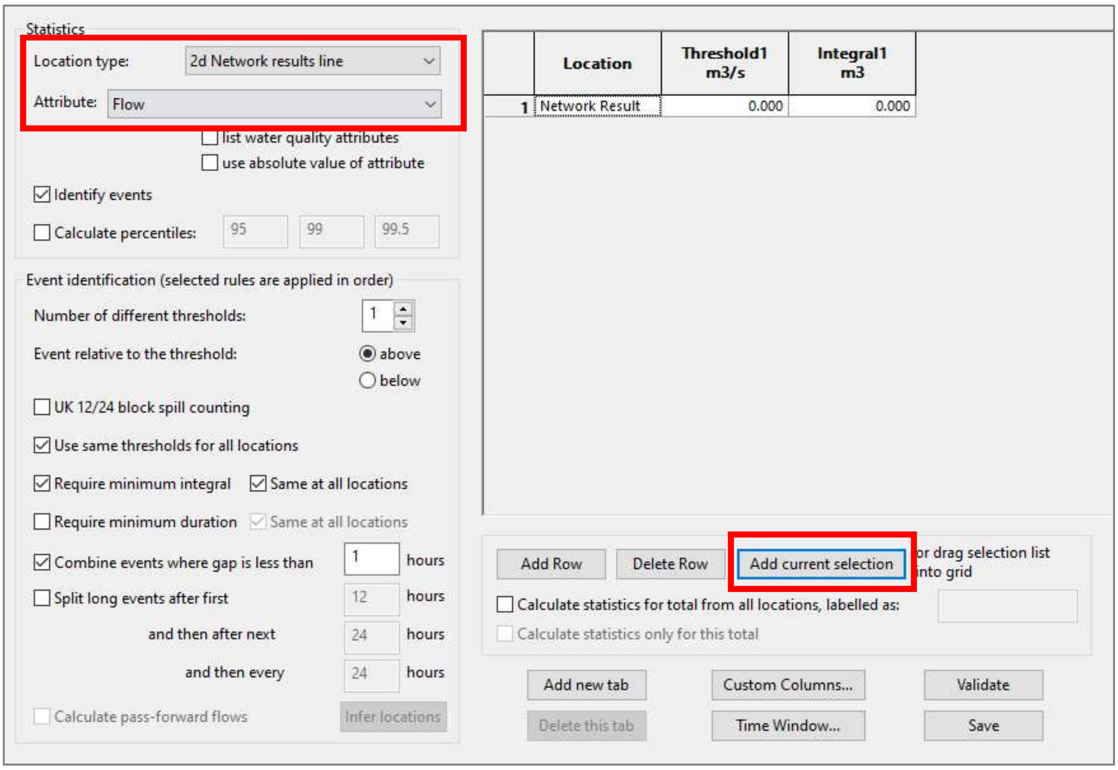
10E. ARR19 Ensemble Statistics

20. First select an element to run the statistics on. This could be any node, conduit or 2D network result object. Select the '**Network Result line (2D)**'.

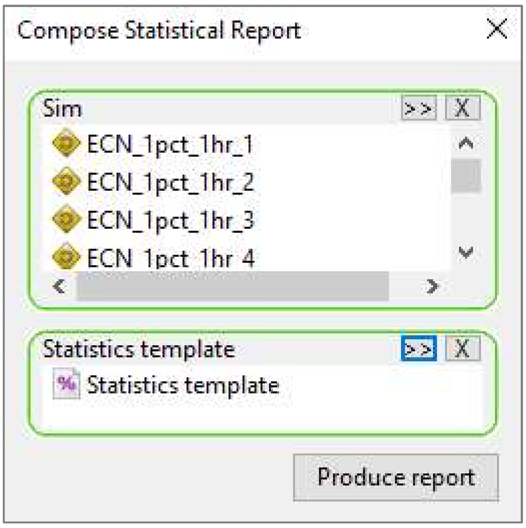
21. Right click on the **Storm Tutorial** Model Group and create a new **Statistical Template**. Keep the default name.



22. Choose **2d Network Result Line** from the **Location type** dropdown. Select **Flow** from the **Attribute** and the click on the **Add Current Selection** to add the location to the list of objects. **Validate** and **Save** the object.



23. To run the statistics, go to the **Results >Statistically Report**. Drag in either the run or selected results to process and the new Statistical Template object.



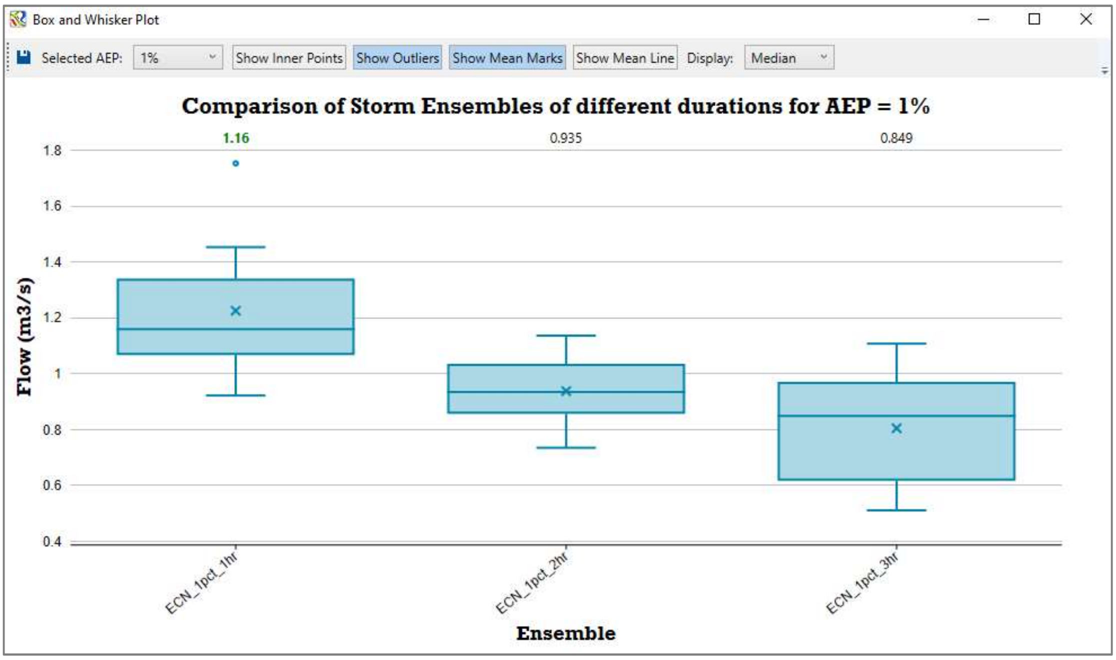
24. Once produces, go the **Ensemble Summary** tab in the report.

Statistical Reports: Statistics template

Attribute	Units	ID	Network	AEP	Ensemble	Ensemble mean	Mean sim	Ensemble median	Median sim	Ensemble min	Min sim	Ensemble max	Max sim
Flow	m3/s	Network Result	Urban Network (version 12)	1%	ECN_1pct_1hr	1.226	ECN_1pct_1hr_4	1.160	ECN_1pct_1hr_9	0.922	ECN_1pct_1hr_5	1.754	ECN_1pct_1hr_1
Flow	m3/s	Network Result	Urban Network (version 12)	1%	ECN_1pct_2hr	0.938	ECN_1pct_2hr_6	0.935	ECN_1pct_2hr_6	0.735	ECN_1pct_2hr_7	1.137	ECN_1pct_2hr_1
Flow	m3/s	Network Result	Urban Network (version 12)	1%	ECN_1pct_3hr	0.805	ECN_1pct_3hr_6	0.849	ECN_1pct_3hr_2	0.511	ECN_1pct_3hr_7	1.108	ECN_1pct_3hr_10

Exceedance detail | Exceedance summary | Simulation summary | Ensemble summary

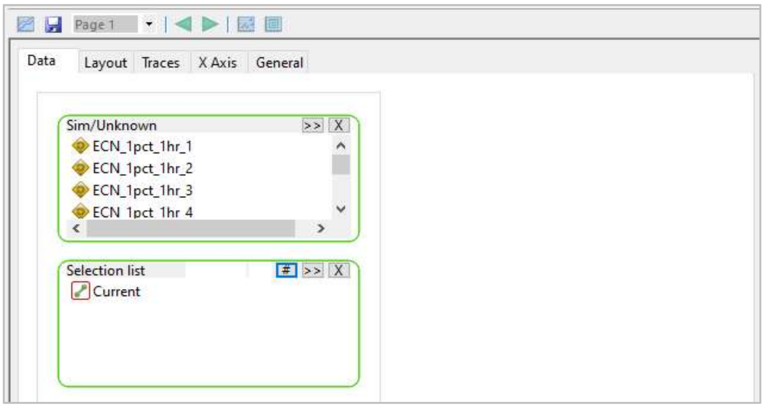
25. To produce a box and whisker plot, go to **Statistics>Export Box Plot** and select a location for the results. Use the **save** button to save the box plot for use in a report later.



10F. Custom Graphs

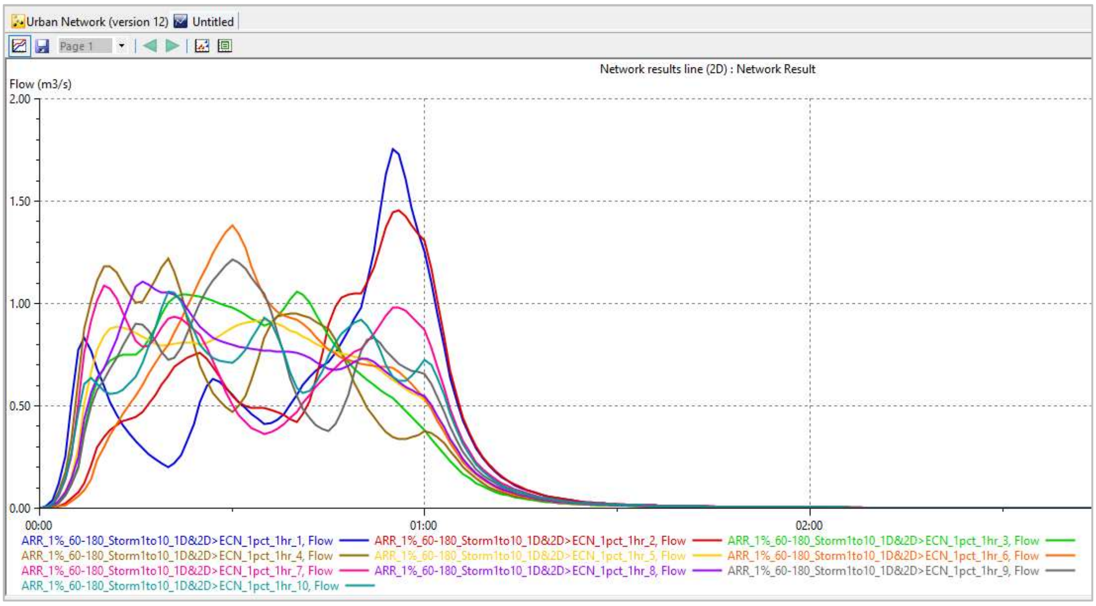
26. While the Network Result line (2D) is still selected, go to **Results>Custom Graphs>Object per page**

27. Drag in the simulations you wish to graph, use the **#** to add the currently selected object (the network result line) to the selection list



28. On the **Layout** tab, select **Flow** and then click on the **Graph button** on the top left corner of the window

29. Right click in the graph and **exclude the title block** and **statistics table**. Use the **Save** icon to save the graph details for later use.



10G. Exporting to GIS

30. If the preference is to produce flood maps in GIS rather than using ICM's Print Layout, the 2d results can be exported to shape, tab or geodatabase. To export, go to **Results>Export maxima to GIS** and select the preferred option.

10H. Creating & Sending Transportables

31. **Open** a **New transportable** file and save it with the name **'transportable_export.icmt'**

32. **Select** the objects in the current Master Database you wish to copy, **right click** and **copy objects**

33. **Right click** in the transportable file and **paste** the data.

ⓘ *Note:* Always close the transportable window before closing InfoWorks ICM.

⌁ Tip: By copying a run object only, ICM will automatically select all required objects and network to copy to enable the run object to be re-simulated. It will also give you the option to copy results.

B1. Combined Hydrology Methods

B2. Pre Vs Post Models

B3. Modelling Basins

12. RAFTS Hydrology