Training Course

Introduction to RAFTS Hydrology in InfoWorks ICM/ ICMOne

Innovyze®

Empowering water experts

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 user 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 workspace. 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.

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:

1. Other Model Groups
2. Version controlled items such as Networks
3. 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**.

Workshop 1: Setting up the Database

To begin working with InfoWorks ICM, a Master Database must be set-up. There are two master database types available in InfoWorks ICM:

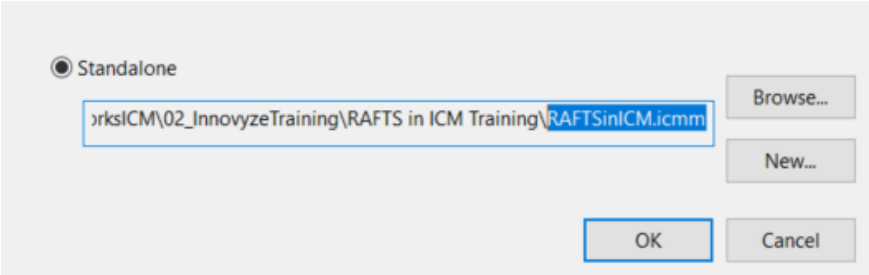
- **Work Group Database** - The default database type for InfoWorks ICM. Intended for use by individuals and groups of users. Requires the use of the Workgroup Data Server.
- **Standalone Database** - This database is intended for use by individual users working on a single PC. This database type is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

1A. Master Database Settings

We will begin by setting up a new Master Database in InfoWorks ICM.

1. To create a new Standalone Master Database, 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 on the C drive called **RAFTS in ICM Training** and call the Master Database **RAFTSinICM.icmm.**



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

NOTE: If you do not see the Explorer Window, go to **Window > New Explorer window**.

1B. Local Folders

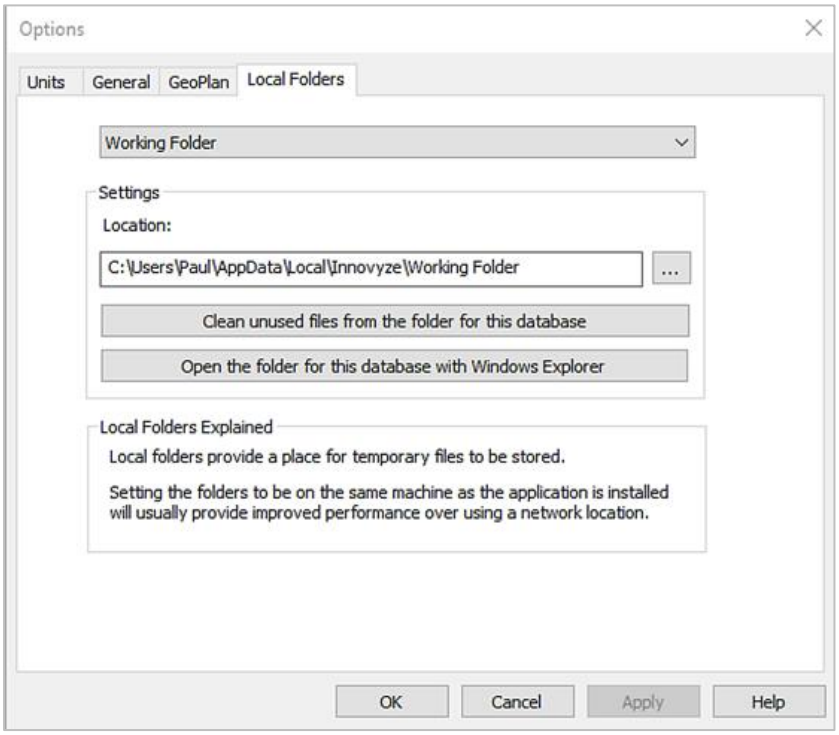
The **Local Folders** is where ICM will store the working files. A new **.wdb** file will be created every time a network is opened in the GeoPlan. This is specific to the user and their working folder and

will not update the Master Database until the changes are committed. The location for this is automatically set by ICM but it can be manually changed. As it is one of the main folders that ICM is reading and writing data in it should be ensured that it is on an appropriate drive.

It is important to manage this folder as it can end up containing a lot of data if it is not maintained regularly. To maintain the size of the working folders there is the button to **Clean unused files from the folder for this database.** This will remove historic network objects that are not currently open which have been committed to the Master Database file so that they can be retrieved in the future.

Review the location of your Local Folders.

5. Go to **Tools > Options…** to open the Options menu and select the **Local Folders** tab. From here you can select either **Working Folder** or **Results Folder** from the drop-down menu. Select **Working Folder** and then click on **Open the folder for this database with Windows Explorer.**



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. **NEVER** copy the master database, **ALWAYS** use a transportable database to move data. A transportable database is useful to transfer data between colleagues, to clients or to Innovyze support (support@innovyze.com).

Workshops in this Training Manual will use template data that has been stored in a Transportable Database.

Open a Transportable Database and copy data to your Master Database.

6. Go to **File > Open > Open transportable database…** navigate to the **RAFTS in ICM data** folder and choose **RAFTSinICM_TD.icmt** from the files and click **Open**. Alternatively use the **Open transportable database** icon under the **File** Toolbar.

NOTE: The model files were developed in InfoWorks ICM 2021.3. You will need to be working with version 2021.3 or higher to open the transportable database.

7. A new window will appear with model objects inside. Select the **RAFTS Part1 – COMPLETE** Model group object, right click and **Copy**. In your newly created Master database, right click on the Master database and **Paste** the objects (with children). Check the options to copy ground models and simulation results.

8. Close the transportable database window.

NOTE: To create a transportable database, go to **File > Open > Open new transportable database…** Again, there will be a dialog prompting you to save the file in a specific location. The file will be given the extension .**icmt**.

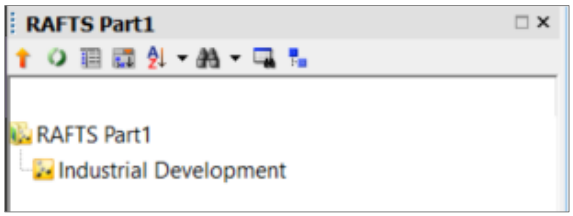
1D. Model Groups & Networks

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 the C drive on your computer can have many file folders, a master database can have many model groups.

Create a new project Model Group and Network.

9. Right click on the Master Database Object in the Explorer Window and select **New > Model group** and give it the name **RAFTS Part1.** This will be the folder that contains all your project components (network, rainfall, results etc).

10. Right click on the Model group and select **New InfoWorks > InfoWorks network** and type the name **Industrial Development.**

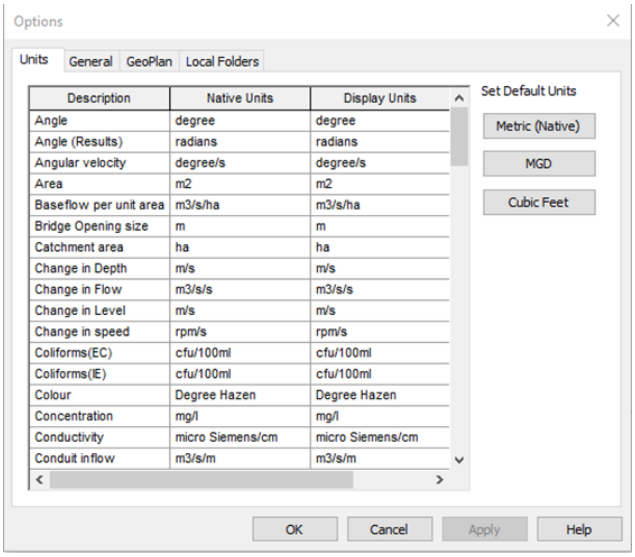


11. Double click on the new Network object **(Industrial Development)** to open in the GeoPlan and maximise the window.

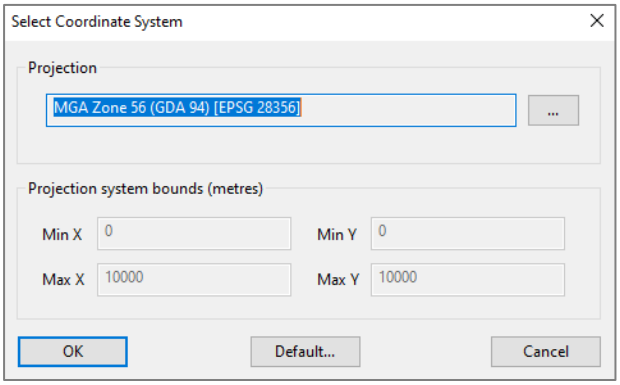
1E. Units & Projection / Co-ordinate System

Set up the Units for the Master Database and the Coordinate System for the project Network.

12. Go to **Tools > Options…** and select the **Units** tab. Notice the units that are assigned to each item in the **Description** column. Set your units to **Metric (Native)** and click **OK**.



13. To set the coordinate system for the network go to **GeoPlan > Set coordinate system…** Click on the ellipsis icon to see the available projections. Select **MGA Zone 56 (GDA 94) [EPSG 28356]** from the list under the **Map Grid of Australia 1994 (MGA94).**



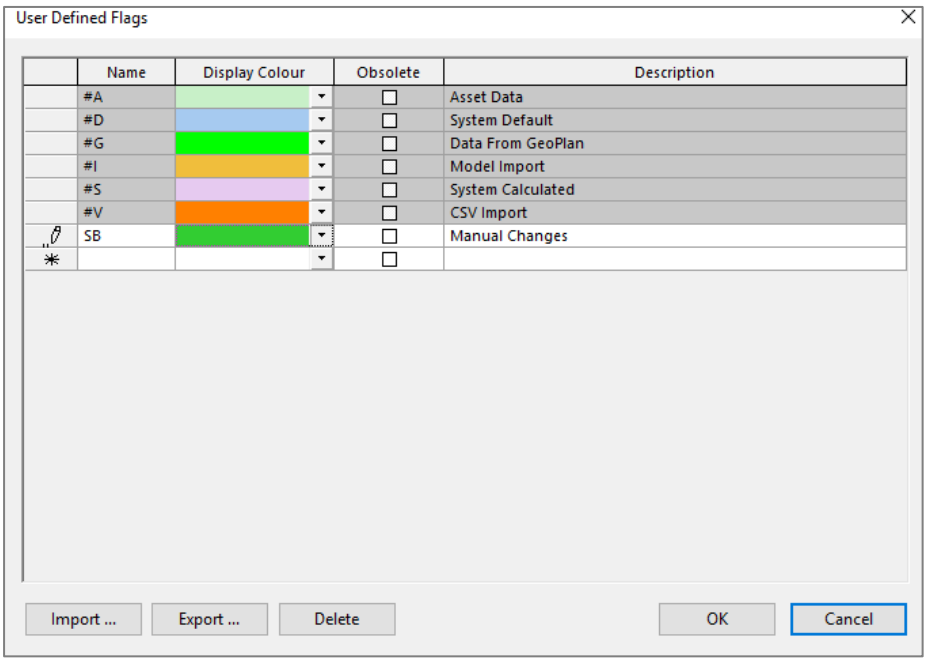
1F. Data Flags

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 in the **User Defined Flags** window below.

Set up user defined flags within the Master Database.

14. Go to **File > Master database settings > User defined flags...** or alternatively click on the **User Flags** icon under the **File** toolbar.

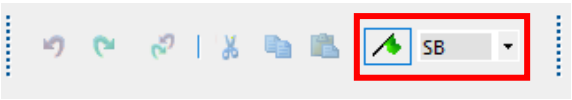
15. Right click in the first empty row under **Name** and type in your initials, select a display colour and provide the description **Manual Changes**.



16. Add an additional 4 user defined flags as described below and click **OK**.

    - FD – Field Data
    - AS – Assumed Data
    - OP – Options Testing
    - IN – Inferred Data

17. 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.



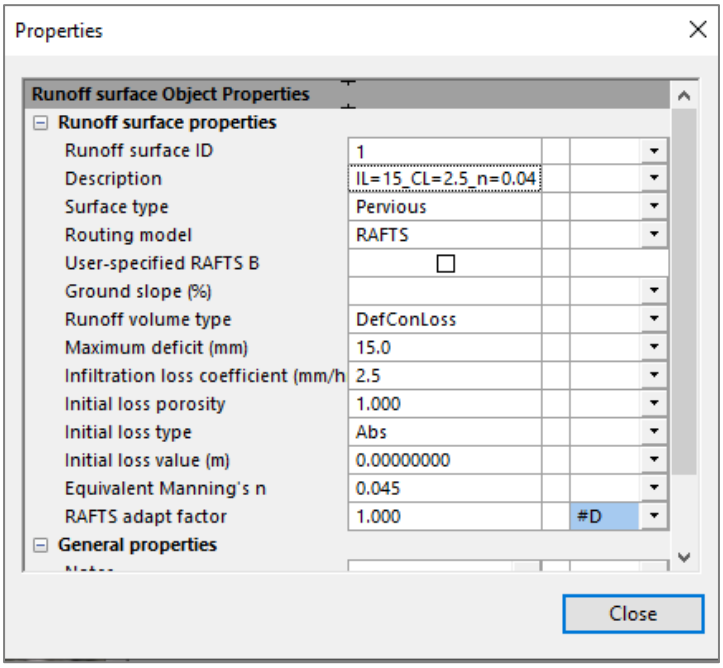
1G. Setting User Defined Defaults

Set the default Land use ID and Runoff surface under Subcatchments for the Network.

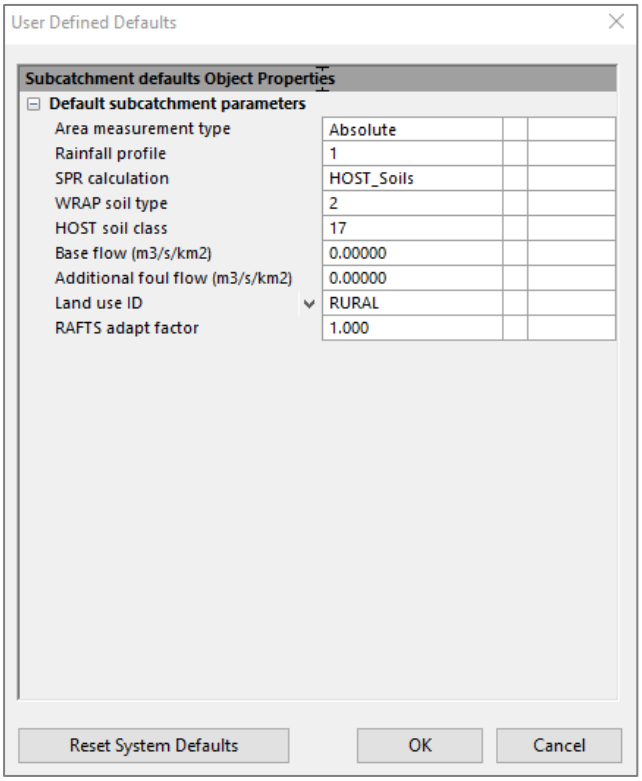
18. Go to **Network > User defined defaults > Subcatchment.** Use the down arrow next to the **Land use ID** to Open Land use DEFAULT.

19. Change the Land use ID to **RURAL**.

20. Use the down arrow next to **Runoff surface 1** to open the runoff surface properties. Populate the **Description** as IL=15_CL=2.5_n=0.045, set the **Surface type** to Pervious, the **Routing model** to RAFTS, the **Runoff volume type** to DefConLoss, the **Maximum deficit (mm)** to 15, the **Infiltration loss coefficient (mm/hr)** to 2.5, the **Initial loss type** to Abs and 0 for the **Initial loss value (m).** Then set the **Equivalent Manning's n** to 0.045. Click **Close** on the Runoff surface window and **Close** the Land use window.



21. Back in the Subcatchment defaults dialog, change the **Land use ID** to RURAL and click **OK**.



1H. ICM Help

Getting started can sometimes be the most difficult hurdle, so we have put together a collection of resources that we think will get you up and running with InfoWorks ICM right away. The InfoWorks ICM Online Help is a great source of information that includes links to the **Innovyze Support Portal** and **Knowledgebase**. The Knowledgebase has a host of articles available that cover topics in more depth, include more "how-to" instructions, and answer common support questions in more detail. The Knowledgebase can be accessed from the Support Portal.

The **ICM Help** is very robust with a wealth of information and detail about the features and functionality of the software. ICM Help can also be accessed from the **Start Menu** if ICM is not open. If it is, you can simply go to **Help** on the menu toolbar.

Access the InfoWorks ICM Help.

22. On the menu bar, go to **Help > Help topics.** Now type **"RAFTS"** into the search bar and select **List Topics.**

23. This will now bring up a list of topics which contain the key words underneath. Select the topic called **RAFTS Routing Model** by double clicking. This will highlight the key words which can be cleared by pressing F5.

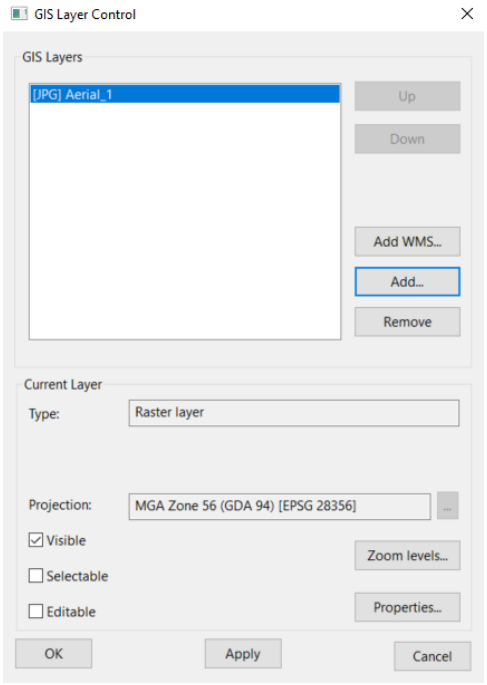
Workshop 2: Setting up Project Data

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.

Add background layers to the GeoPlan.

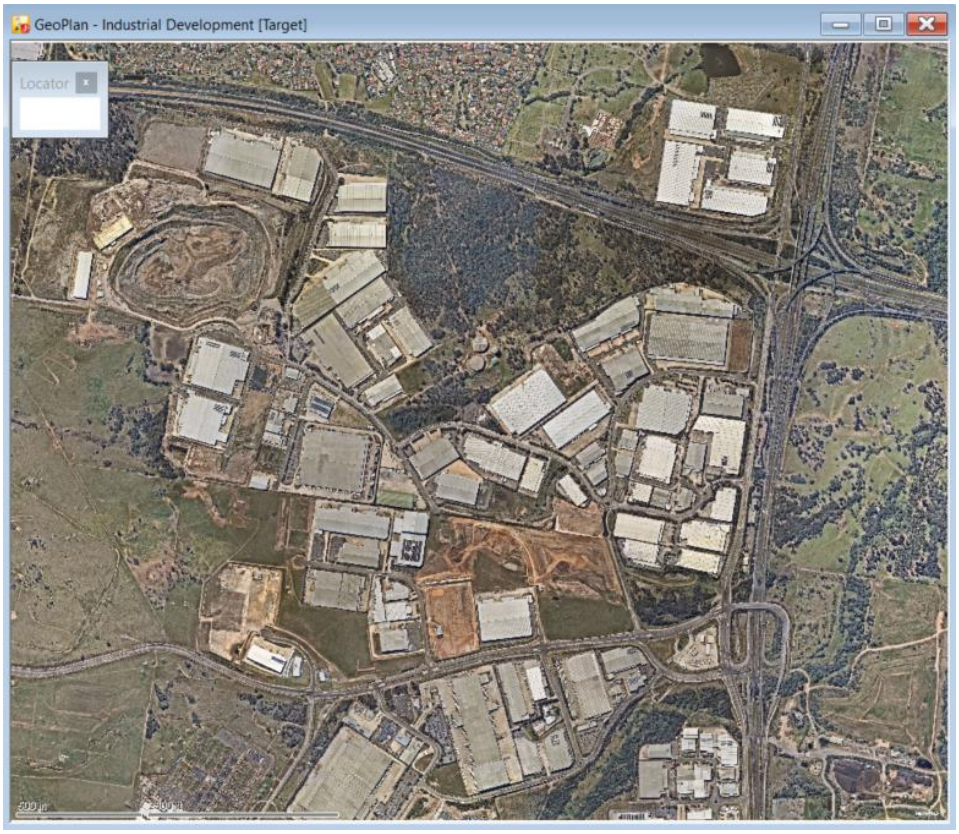
1. Right-click on the Geoplan and select **GIS Layer control…** Click on **Add…** and browse through the **RAFTS in ICM Data\Part 1** folder. Make sure that the File type is set to **Raster Image**, select **Aerial_1.jpg** and click **Open**. Check that the image is in the same projection as our network and click **OK**.



2. Right click in the Geoplan and **View entire GIS layer…** and select the JPG to zoom to the project area. Note the scale bar at the bottom of the Geoplan.

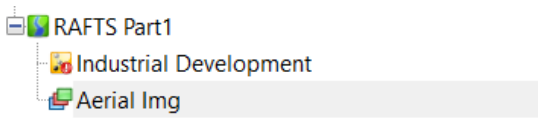
3. Right click in the GeoPlan again and select **View online > Google Maps**. The internet browser should open Google Maps at the same location as the mouse pointer. This is a good check to ensure you have the right projection set for the site.

4. Pan the project site by pressing and holding the mouse wheel and the moving hand appears next to the cursor to pan around the image. Use the mouse wheel to zoom in and out by rolling the wheel forward and backwards, respectively. A second option is to use the **Pan** and **Zoom** icons in the **GeoPlan** toolbar.



Tip *– Press F8 on the keyboard to toggle between select and pan.*

5. The background layers can be saved for use at a later date or in another network. Right click on the Model group and go to **New InfoWorks > Layer list.** Give the object the name **Aerial Img** and select **OK**.

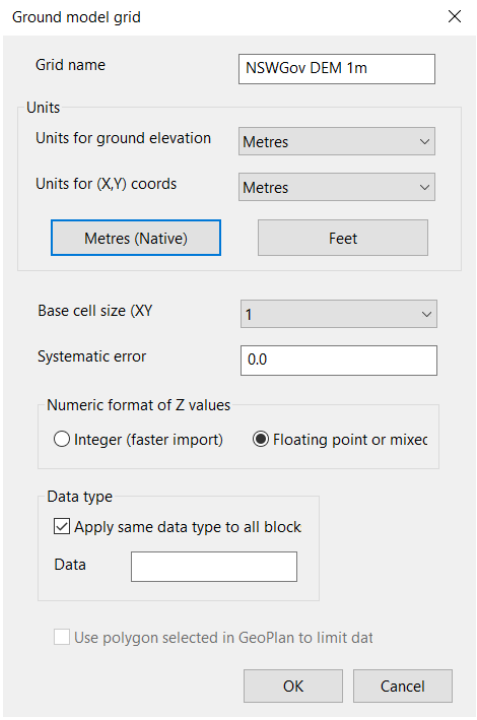


2B. Importing Ground Models

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 models are also particularly useful for looking at bridge openings and definition.

Import Ground Model data into the project Model group.

6. Right click on the RAFTS Part1 Model group and go to **Import InfoWorks > Ground model grid > from ground model grid folder…**
7. Navigate through the following path **RAFTS in ICM data\Part 1\NSW Government – Spatial Services\DEMs** and select the **1 Metre** folder.
8. Type in **NSWGov DEM 1m** for the Ground model Grid name, choose **Meters (Native)** for the Units and ensure you use the **Floating point or mixed** to include elevations with decimal places. Click **OK**.

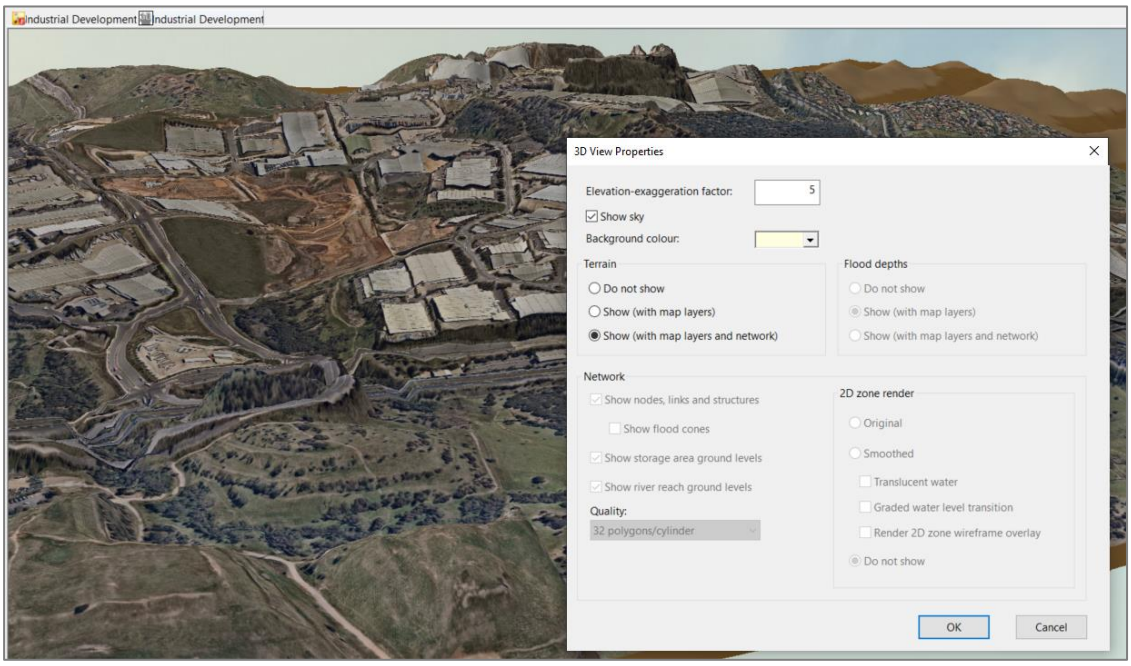


9. Drag the new **Ground model** object onto the GeoPlan to add it to the view.
10. You can view the ground model draped over the background images using the **New 3D network window** icon in the **Windows** toolbar or going to **Window > New 3D network**

window. Right click in the 3D view to bring up the **Properties** dialog. From here you can further exaggerate the vertical factor of the ground model's elevation (see image below).

11. Close the 3D view.

NOTE: You can clear the ground model from the GeoPlan by going to **Network > Clear ground model.**



2C. Using Properties & Themes

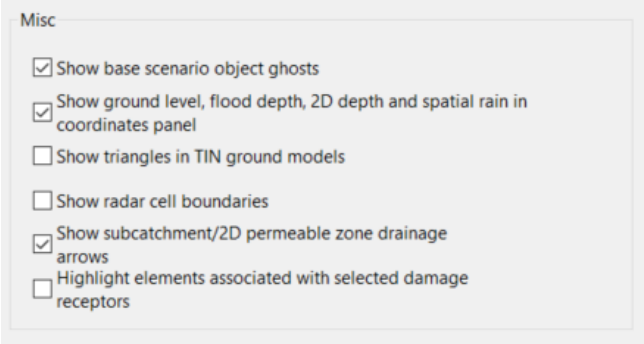
A significant proportion of the GeoPlan functionality originates from the **GeoPlan Properties and Themes** dialog box. A number of visual changes can be made as well as the addition of themes to the network.

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.

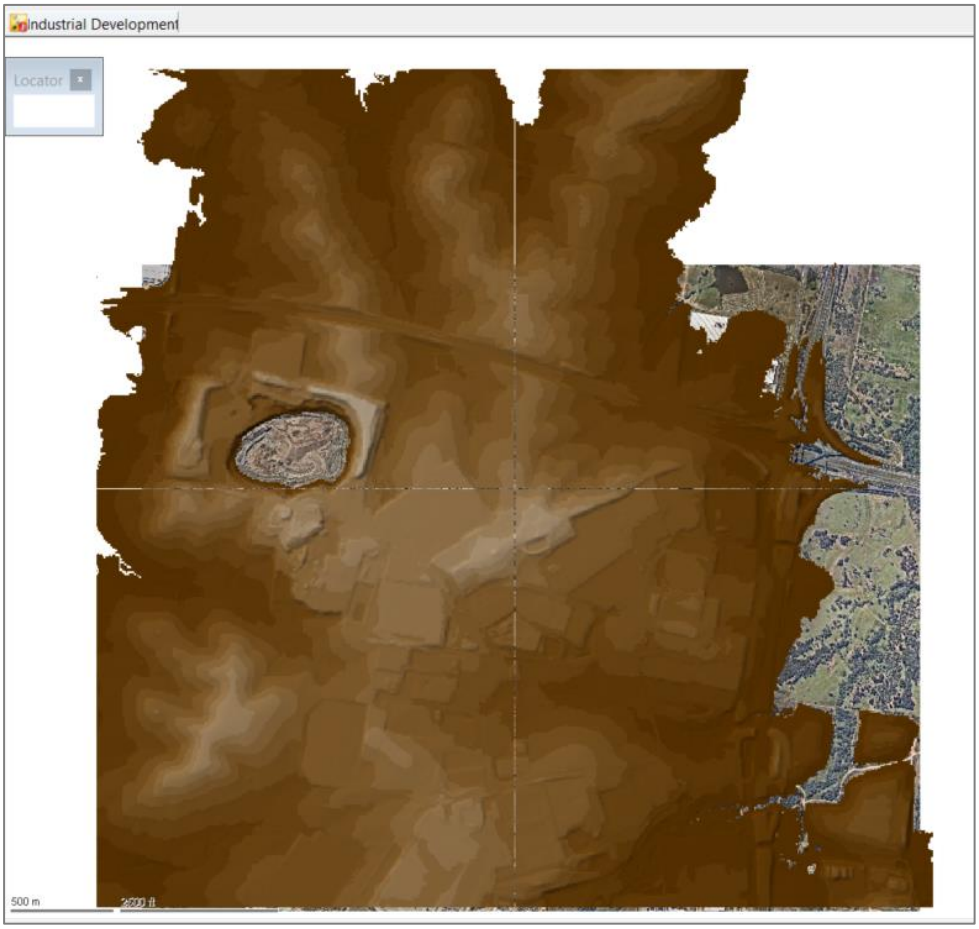
Open the GeoPlan Properties and Themes dialog, save changed settings as a default and load pre-defined themes from the transportable database.

12. Right click anywhere in the GeoPlan and select **Properties & Themes…**

13. Go to the **Visual** tab and under **Misc** check on the **Show ground level, flood depth, 2D depth and spatial rain in coordinates panel**. This will turn on Z elevations for the cursor location in the bottom right corner of the GeoPlan.

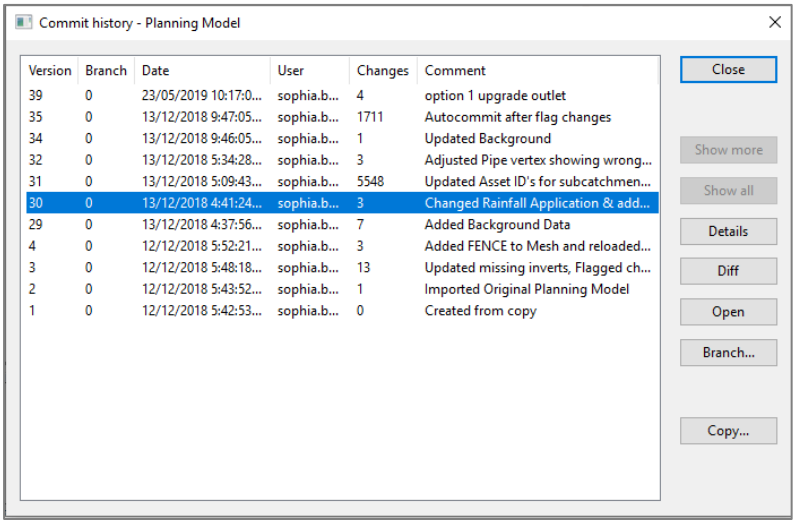


14. Click **Apply** then **Save** and select **Save as default for this network**. Click **OK** to exit.
15. Move your curser over the Ground model to investigate the grid elevations.
16. Locate the **RAFTS Hydrology** Theme object within the transportable files that we copied into our Master Database earlier and drag it onto the GeoPlan. You will notice a change in the Ground model definition.



2D. Committing Changes

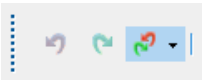
Every time a change to the network is committed, a new version of the network is created. The **Show commit history** (found under the **Network > Version control** menu or by right clicking on the network) details each version of the network. For every commit, there is a corresponding version. It tells 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.



Commit the changes to the Master Database.

17. Right-click on the **Industrial Development** network in the Explorer Window and select **Commit changes to master database.** In the **Comments**, type in a summary of what has been made so far since the last commit. A good example would be **Workshop 2 – Setup GeoPlan and background data. Changes made to Subcatchment defaults.** Click **OK**.

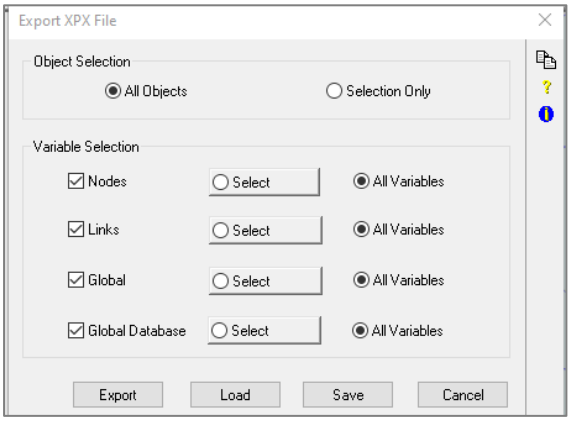
18. Click **NO** in the Validation Window. You will notice committing changes will turn the undo/redo greyed out. This network is now version controlled.



19. Right click again on the Industrial Development Network object and select **Show commit history**. Here we can see the versions committed so far (2).

Workshop 3. XPRAFTS model Import

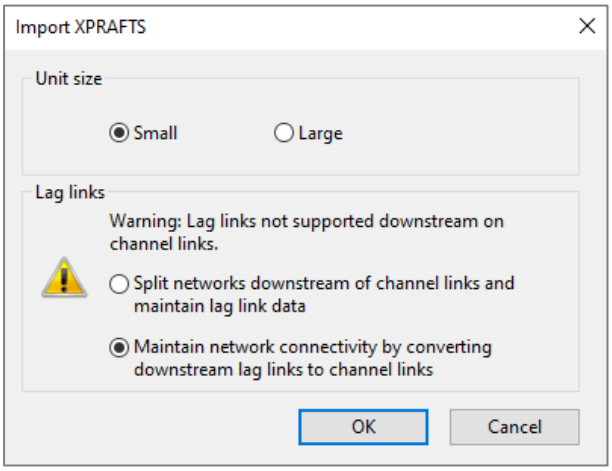
Before importing an existing model into InfoWorks ICM it will first need to be exported from XPRAFTS classic. With the desired XPRAFTS model network open, go to **File > Export Data…** and ensure **All Objects** and **All Variables** are selected. Save and **Export** the XPX file.



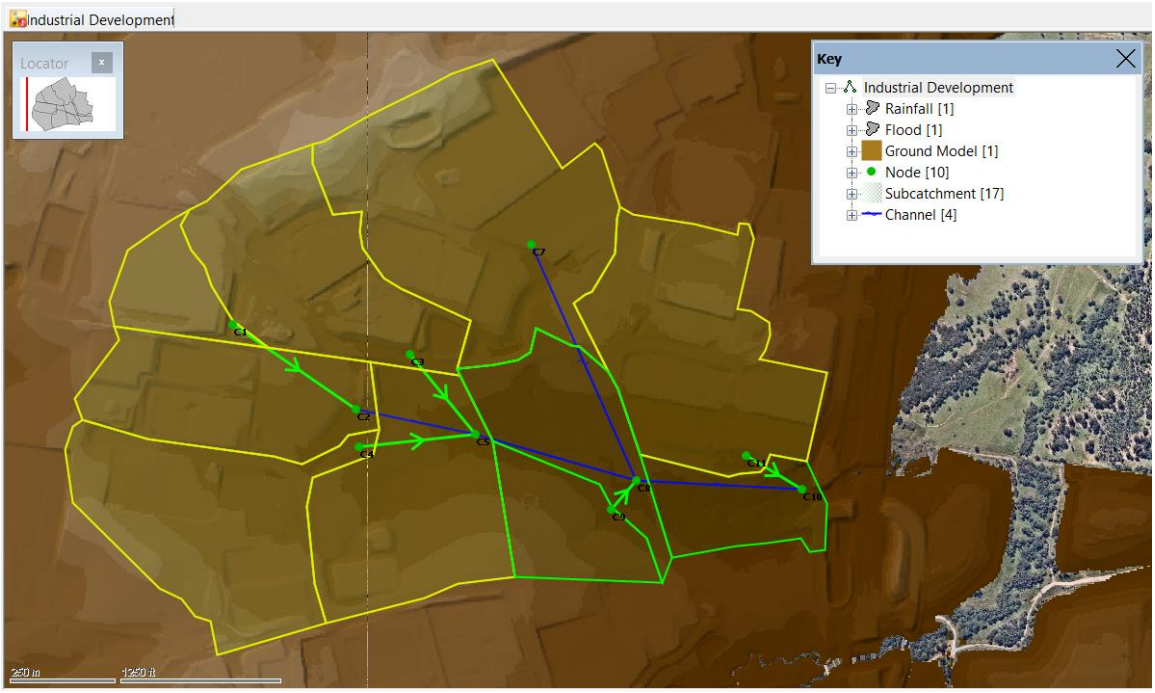
3A. Importing XPRAFTS Network

Use the XPRAFTS importer to import a XPRAFTS Network into ICM.

1. With the Industrial Development Network open in the GeoPlan, go to **Network > Import > Model > from XPRAFTS XPX file…**
2. Navigate to the **RAFTS in ICM Data** folder, select the **Development1.xpx** file and click **Open**. The import options dialog will appear to select the **Unit size** and **Lag links** options. Select **Small** and **Maintain network connectivity by converting downstream lag links to channel.**



3. Close the Import log. Right click in the GeoPlan and select **Zoom to network**. The shortcut F12 can also be used to zoom to the network.

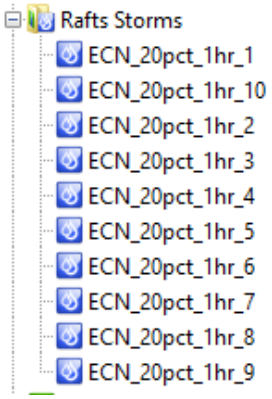


3B. Importing XPRAFTS Storms

Import the Global storms from a XPRAFTS model as Rainfall objects in the project Model group.

4. Right click on **RAFTS Part1** in the Explorer Window and select **Import InfoWorks > Rainfall event > from XPRAFTS XPX file…**

5. Navigate to the **RAFTS in ICM data\Part 1** folder, select the **Development1.xpx** file again and click **Open**. All the available storms in the XPRAFTS model will now be available as individual Rainfall objects under the Model group.



3C. Validating the Network

Before we can commit a network and run a simulation, the network must be validated.

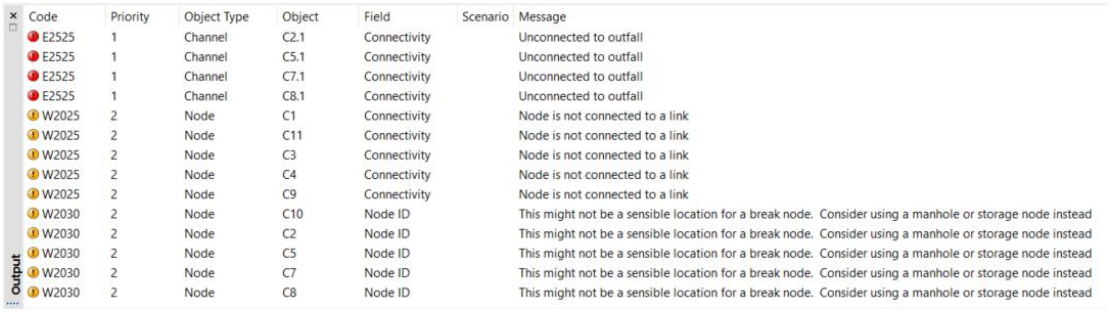
The validation tool will flag any user-input values that are inaccurate, inappropriate, or missing. These are judged with a range of priorities:

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

Validate the Network model and view the Output messages.

6. Go to **Network > Validate network** or select the red tick in the **Validation** toolbar  . Select **OK** in the Network Validation window.

7. The message output box should appear with a number of warnings and errors.



Code	Priority	Object Type	Object	Field	Scenario	Message
E2525	1	Channel	C2.1	Connectivity		Unconnected to outfall
E2525	1	Channel	C5.1	Connectivity		Unconnected to outfall
E2525	1	Channel	C7.1	Connectivity		Unconnected to outfall
E2525	1	Channel	C8.1	Connectivity		Unconnected to outfall
W2025	2	Node	C1	Connectivity		Node is not connected to a link
W2025	2	Node	C11	Connectivity		Node is not connected to a link
W2025	2	Node	C3	Connectivity		Node is not connected to a link
W2025	2	Node	C4	Connectivity		Node is not connected to a link
W2025	2	Node	C9	Connectivity		Node is not connected to a link
W2030	2	Node	C10	Node ID		This might not be a sensible location for a break node. Consider using a manhole or storage node instead
W2030	2	Node	C2	Node ID		This might not be a sensible location for a break node. Consider using a manhole or storage node instead
W2030	2	Node	C5	Node ID		This might not be a sensible location for a break node. Consider using a manhole or storage node instead
W2030	2	Node	C7	Node ID		This might not be a sensible location for a break node. Consider using a manhole or storage node instead
W2030	2	Node	C8	Node ID		This might not be a sensible location for a break node. Consider using a manhole or storage node instead

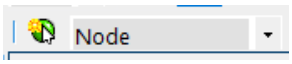
8. The validation tool has alerted the need to include an outfall in the network. There are also several warnings in the output which can be ignored due to the purpose of this model. The errors will be resolved in the following steps.

3D. Adding Objects Manually

XPRAFTS models do not include **Outfall** nodes and in ICM subcatchments wont drain to Outfalls. To remove the validation errors, an outfall will need to be added to the Network.

Add an outfall node and a 'Dummy' weir to validate the Network.

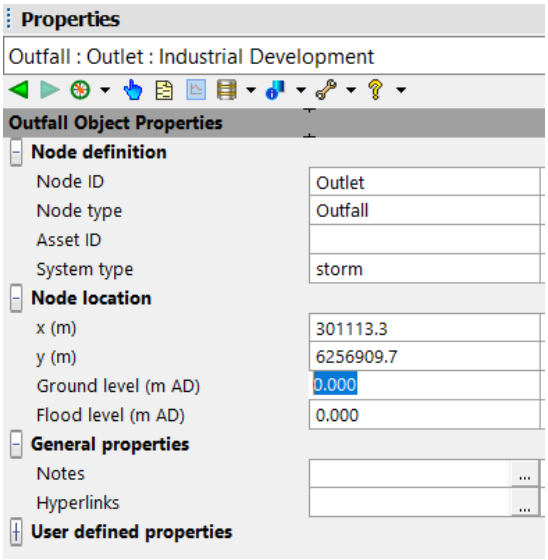
9. Select **Node** from the dropdown box within the **GeoPlan** toolbar. Press the **New object** button to begin creating a node.



10. Create a new node downstream of node **C10**. Give the name **Outlet**, set the node **Type** to **Outfall** and the **System Type** to **storm**.

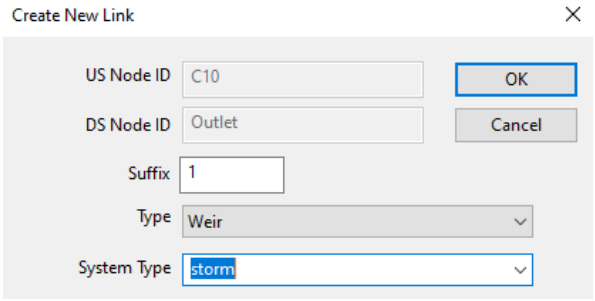
NOTE: The properties dialog should automatically appear for the new node that has been created. You will notice there are some validation messages appearing where critical data for the node is missing. We will need to populate this data before we can validate the network for a simulation.

11. In the properties dialog enter **0** for the **Ground level (m AD)**. XPRAFTS models do not include invert levels however ICM will start the imported network grade at RL 0 m.

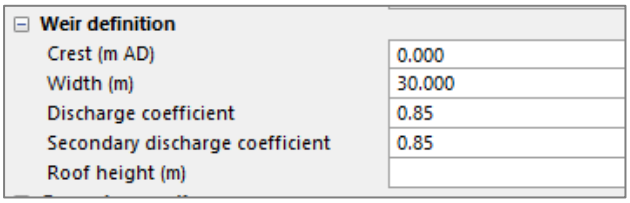


12. Select **Link** from the dropdown box within the **GeoPlan** toolbar. Press the **New object** button to begin creating a link.

13. Draw a link from **C10** to **Outlet.** Double click to complete the link and the **Create New Link** dialog will be shown. Select **storm** as the **System Type** and **Weir** as the **Type.** A default value of **1** will be adopted as the **Suffix**.



14. In the properties dialog, enter **0** for the **Crest (m AD)** and then set the **Width (m)** to **30.**



15. Validate the network. This time no errors should appear in the **Output** window.

3E. Customising the Object Property window

In ICM, Users have the ability to customise the **Object Properties** window to highlight the important input data, group together desired properties or hide irrelevant information just to name a few examples.

Create a new Group within the Subcatchment object properties and group in selected subcatchment properties.

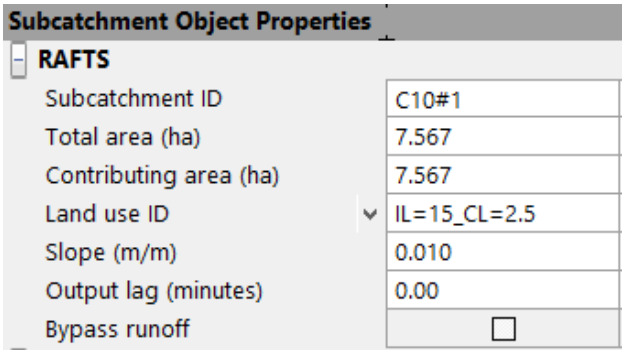
16. Double click on Subcatchment **C10#1** to open the subcatchment object properties.

17. Click on the drop-down next to the **Configuration** icon and select **Layout > Edit.**

18. Right click within the Object properties window and select **Add group**. Call the new group **RAFTS**. Ensure that the new RAFTS group is moved to the top of the window.

19. Find the following subcatchment properties and drag them under the RAFTS group: **Subcatchment ID, Total area (ha), Contributing area (ha), Land use ID, Slope (m/m), Output lag (minutes), Bypass runoff**

20. Click on the drop-down next the Configuration again and select **Layout > Save locally.** Right click within the properties window and select **Close** to exit the edit window.



21. Right click on the **Industrial Development** Network in the Explorer Window and select **Commit changes to master database.** In the Comments, type in a summary of what has been made so far since the last commit. A good example would be **Workshop 3 – XPRAFTS model imported and outfall added.** Click **OK** and select **Yes** to validating the network if required.

Workshop 4. Simulating and Reviewing Results

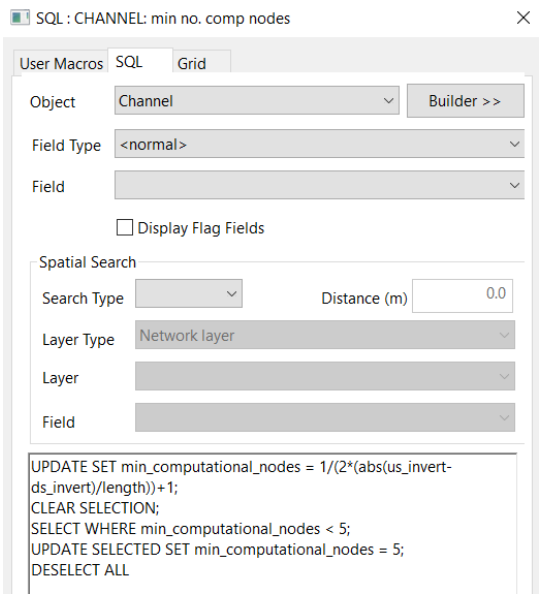
4A. Updating the Channel Definition

By default, the imported Channel links will have a minimum of 5 computational nodes per link when the engine is performing the model calculations in the hydraulic simulation. This is based on global default simulation parameters, chosen for optimum accuracy and performance. The Simulation Parameters can be viewed in **Model > Model parameters > Simulation parameters**.

Editing of the global simulation parameters is not recommended. However, we are able to update the minimum number of computational nodes individually for each channel links without affecting the simulation parameters across the entire network to improve the performance of the model calculations.

Inspect the pre-defined SQL and Run to change the minimum number of computational nodes for each channel.

1. From the Transportable Database, right click on the **CHANNEL: min no. comp nodes** SQL and select **Open**.
2. You can see from the editor that the SQL is going to update the **Minimum computational nodes** field based on the slope of the channel. Press **Close**.



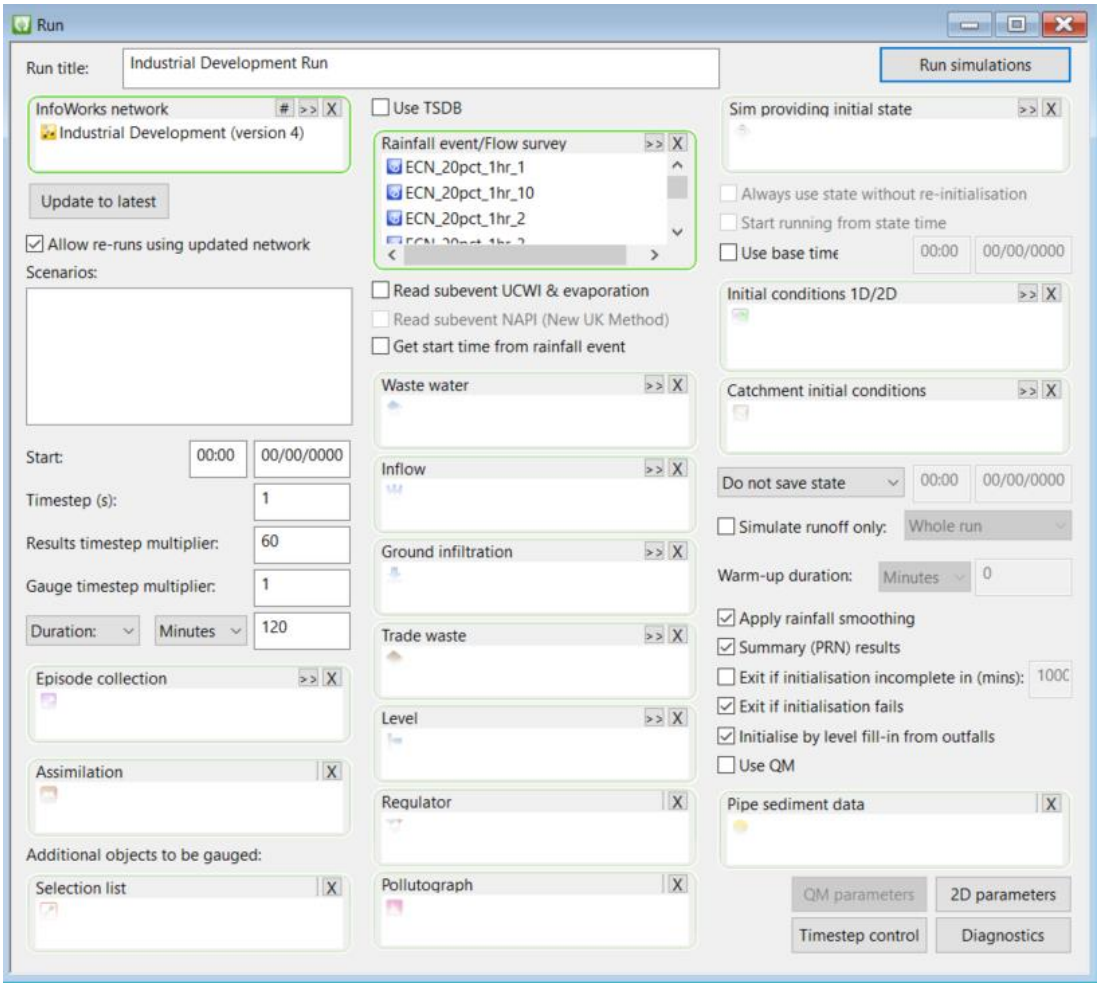
3. Drag the SQL onto the GeoPlan and open the properties for a few of the channels in the model. The min. computational nodes field will be updated from 5.

4. Validate and commit the Network.

4B. Setting up the Run object

Now we are going to set up the simulation data and parameters and run the simulation.

5. Right click on the **RAFTS Part1** Model Group and select **New InfoWorks > Run.**
6. Fill in the Run title as **Industrial Development Run.**
7. Select the **Industrial Development** Network and the **Rafts Storms** group from the Group Window (Use the CTRL key to multi-select) and drag them onto the Run Window dropping anywhere within the grey area of the dialog.
8. Check the **Allow re-runs using updated network** option which means we can re-run and overwrite the results rather than creating a copy of the run later.
9. Change the Timestep (s) to **1**, the Result timestep multiplier to **60** and the duration to **120** minutes**.**
10. Click **Run simulations** and **OK**.



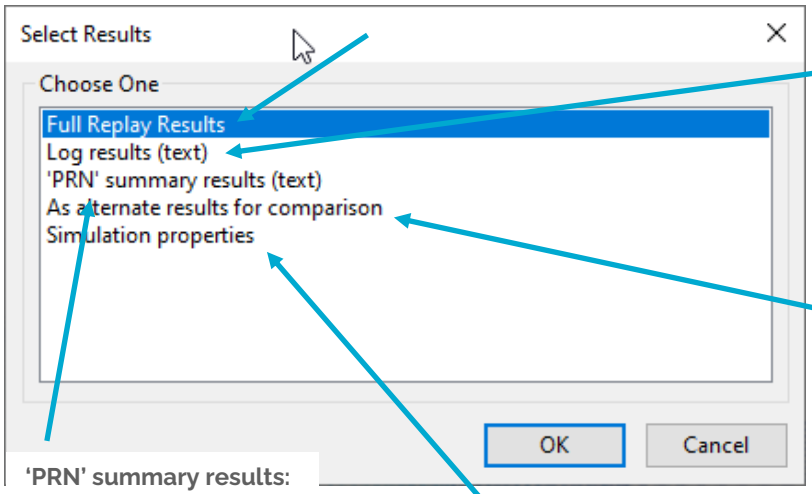
4C. Simulation Results

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

Firstly, we will look at the summary results and text output files. Then open the time-varying results and explore the different methods of interrogation including setting up custom graph templates.

11. Right click one of the simulation icons and go to **Open As...** The following dialog appears:

Full Replay Results:
Visual GeoPlan results
as explained above.



Log results: A breakdown of the simulation, any simulation errors are listed here; this is especially useful if the model is not completely successful.

As alternate results for comparison: Open two GeoPlan results for comparison.

'PRN' summary results: This is a summary of the networks inputs and outputs at all locations.

Simulation properties: Summary properties for the simulation. Can be used to re-

12. Open one of the simulation's **PRN** and **Log** files and investigate any messages that may appear.

An example section of the **PRN** results is shown below. It is used to get an overview of the network results containing maximum results for nodes and links.

```
Industrial Development (version 5)                    Event -        1  WS77771300PM                    Produced 2020-11-18

                                        ********** Link data **********

                                                  <              Upstream              >  <            Downstream              >
     Link      D/S Pipe  Pipe  Sed  P.Full  Invert   Max     Max     Max     Total     Invert   Max     Max     Max     Total
 Reference     Node  Len  Hgt  Dpth   Flow   Level   Depth   Flow    Vel     Flow      Level   Depth   Flow    Vel     Flow
                     (m)  (mm)  (mm)  (m3/s) (m AD)   (m)   (m3/s)  (m/s)    (m3)     (m AD)   (m)   (m3/s)  (m/s)    (m3)
     C10.1   OUTLET                      0.000  0.376  18.382          64857.7   0.000  0.000  18.382          64857.7
      C2.1       C5  170  1727     0 537.733  6.400  0.238   3.282  0.969   9208.5   3.000  0.478   3.158  0.287   9129.1
      C5.1       C8  250  1727     0 359.250  3.000  0.478  11.860  1.012  35660.4   0.500  0.502  11.454  0.897  35271.9
      C7.1       C8  300  3257     0 1476.11  6.500  0.260   4.781  1.228  11757.4   0.500  0.502   4.563  0.612  11503.8
      C8.1      C10   50  1392     0 167.275  0.500  0.502  15.266  0.919  52601.7   0.000  0.417  15.226  1.343  52477.6

+ after total flow indicates a conduit surcharged by flow and depth at that end.
x after total flow indicates a conduit surcharged by depth only at that end.

NOTE :
  (i) Maximum elevations, depths, volumes, velocities and discharges are selected from the values at each
      time increment and will be in general more extreme than the maximum values in the time varying results.
 (ii) Maximum elevations, velocities and discharges are not necessarily calculated at the same time.
(iii) Maximum velocity is not calculated for a conduit unless the depth exceeds the base flow depth
      (by default, this is 5% of height for slopes <= 0.01, 10% otherwise, subject to a minimum of 0.02 m).
```

An example of the **Log** results is shown below. It is used to summarize the input data and performance of the simulation. It will contain any error or warning messages as well as a summary of the model run times.

Useful information will be the summary of total rainfall, runoff, inflow, outflow and lost volume. These can be used to ensure that the 1D model is behaving as expected.

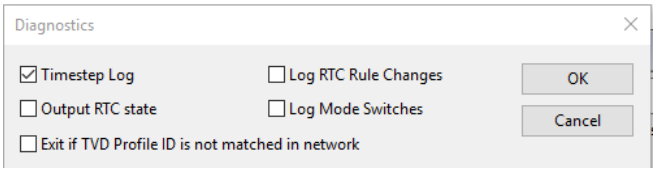
```
Event details: ECN_20pct_1hr_1
  Event label -                          1
  Start time -            0 day(s) 00:00:00
  Requested duration (min) -             120

SIMULATING
WRITING
  Message 253: Run finished for event 1.

  Memory used by simulation engine = 5.7MB
  Threads = 16
  Elapsed clock time = 2s
  Total CPU time = 30.1s
EXITING
18/11/20 15:27:23 Simulation engine exited code 0 (0x00000000)
18/11/20 15:27:23 Starting SimPostP engine on BRISLC2X0P13.xps.xpsolutions.com as user Matthew.Piggott
"C:\Program Files\Innovyze Workgroup Client 2021.1\simpostp.exe" 824
SimPostP 12.0.250
FileLDS: C:\ProgramData\Innovyze\InfoWorksAgent\SA_2A53266B-C672-48EB-A288-F65FFBB66DBD\NNET387#5.wdb
File(S): C:\Users\matthew.piggott\AppData\Local\Innovyze\Results Folder\2A53266B-C672-48EB-A288-F65FFBB66DBD\sim418.iwr
File(A): C:\Users\matthew.piggott\AppData\Local\Innovyze\Results Folder\2A53266B-C672-48EB-A288-F65FFBB66DBD\sim418.iwr
Writing: C:\Users\matthew.piggott\AppData\Local\Innovyze\Results Folder\2A53266B-C672-48EB-A288-F65FFBB66DBD\sim418.iwrcache
```

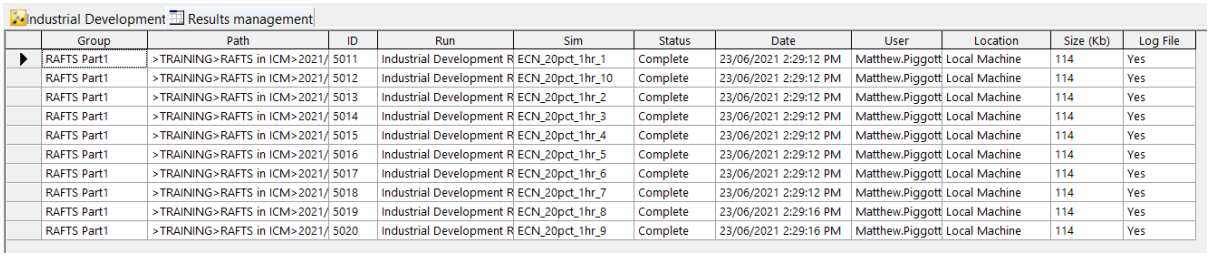
Should the simulation fail then the **Log** file will be the only results text file available. This will allow you to analyse the failed simulation for problems.

NOTE: You can turn on more detailed log files by using the **Diagnostics** tool which is built into the simulation window. To activate select the **Timestep Log** option.



There are many messages that can appear within the **Log** files which can provide information as to why the model simulation may have been unsuccessful. It is beyond the scope of this course to go through them all and knowing how to deal with each one will come with experience. While building your InfoWorks ICM experience it may be useful to use the blog posts and knowledge articles which can be accessed through the **Support Portal**.

13. Right click on the **RAFTS Part1** model group and select **Open results manager**. This will provide a summary of the results under that model group including the ID, simulation results size and location.


Industrial Development | Results management

Group	Path	ID	Run	Sim	Status	Date	User	Location	Size (Kb)	Log File
RAFTS Part1	>TRAINING>RAFTS in ICM>2021/	5011	Industrial Development R	ECN_20pct_1hr_1	Complete	23/06/2021 2:29:12 PM	Matthew.Piggott	Local Machine	114	Yes
RAFTS Part1	>TRAINING>RAFTS in ICM>2021/	5012	Industrial Development R	ECN_20pct_1hr_10	Complete	23/06/2021 2:29:12 PM	Matthew.Piggott	Local Machine	114	Yes
RAFTS Part1	>TRAINING>RAFTS in ICM>2021/	5013	Industrial Development R	ECN_20pct_1hr_2	Complete	23/06/2021 2:29:12 PM	Matthew.Piggott	Local Machine	114	Yes
RAFTS Part1	>TRAINING>RAFTS in ICM>2021/	5014	Industrial Development R	ECN_20pct_1hr_3	Complete	23/06/2021 2:29:12 PM	Matthew.Piggott	Local Machine	114	Yes
RAFTS Part1	>TRAINING>RAFTS in ICM>2021/	5015	Industrial Development R	ECN_20pct_1hr_4	Complete	23/06/2021 2:29:12 PM	Matthew.Piggott	Local Machine	114	Yes
RAFTS Part1	>TRAINING>RAFTS in ICM>2021/	5016	Industrial Development R	ECN_20pct_1hr_5	Complete	23/06/2021 2:29:12 PM	Matthew.Piggott	Local Machine	114	Yes
RAFTS Part1	>TRAINING>RAFTS in ICM>2021/	5017	Industrial Development R	ECN_20pct_1hr_6	Complete	23/06/2021 2:29:12 PM	Matthew.Piggott	Local Machine	114	Yes
RAFTS Part1	>TRAINING>RAFTS in ICM>2021/	5018	Industrial Development R	ECN_20pct_1hr_7	Complete	23/06/2021 2:29:12 PM	Matthew.Piggott	Local Machine	114	Yes
RAFTS Part1	>TRAINING>RAFTS in ICM>2021/	5019	Industrial Development R	ECN_20pct_1hr_8	Complete	23/06/2021 2:29:16 PM	Matthew.Piggott	Local Machine	114	Yes
RAFTS Part1	>TRAINING>RAFTS in ICM>2021/	5020	Industrial Development R	ECN_20pct_1hr_9	Complete	23/06/2021 2:29:16 PM	Matthew.Piggott	Local Machine	114	Yes

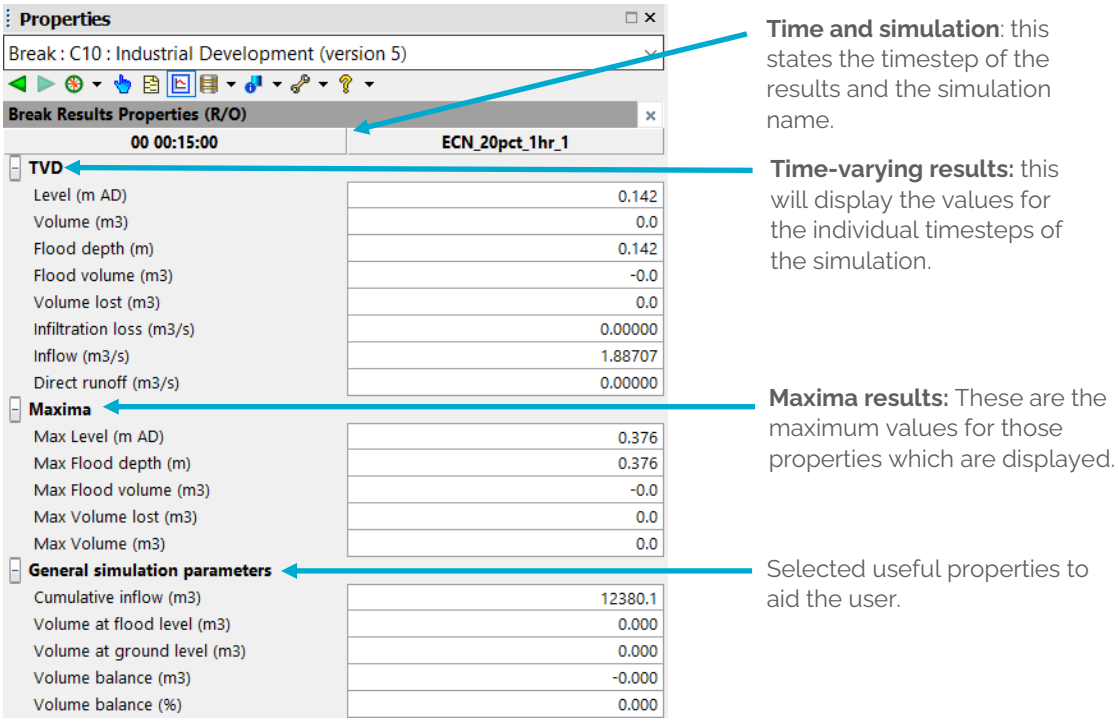
This information can be extremely useful as it can be used to estimate how much storage a full set of results may use, help you understand why you cannot view simulation results or identify which simulation results are associated to the runs.

NOTE: Using the **Results manager**, you can move results from your local results folder to the remote root and vice versa. This means that you can have those results locally that you require

fast, read/write access and then move them once reviewing is complete. This can cut down on speed issues resulting from slow network connectivity.

The properties dialog which is opened using the tool will automatically open on the properties results page when results are active. You can toggle between the object properties and results using the icon. The time is displayed on the left-hand corner.

NOTE: The results that can be viewed are snapshot intervals taken at the timestep and results multiplier. Therefore, unless the maximum and minimum values occurred at a major timestep the graph and object results will differ. This should not however be significant. If there is a significant difference it may indicate that the model is having convergence problems, and this should be investigated further.



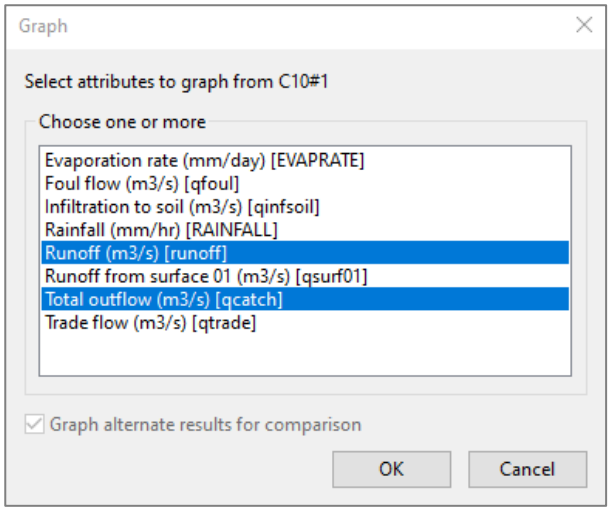
The results properties will show a range of result parameters at the current timestep. When animating time-varying results the results value will change as the results timestep changes (as will the grids and any themes you have active).

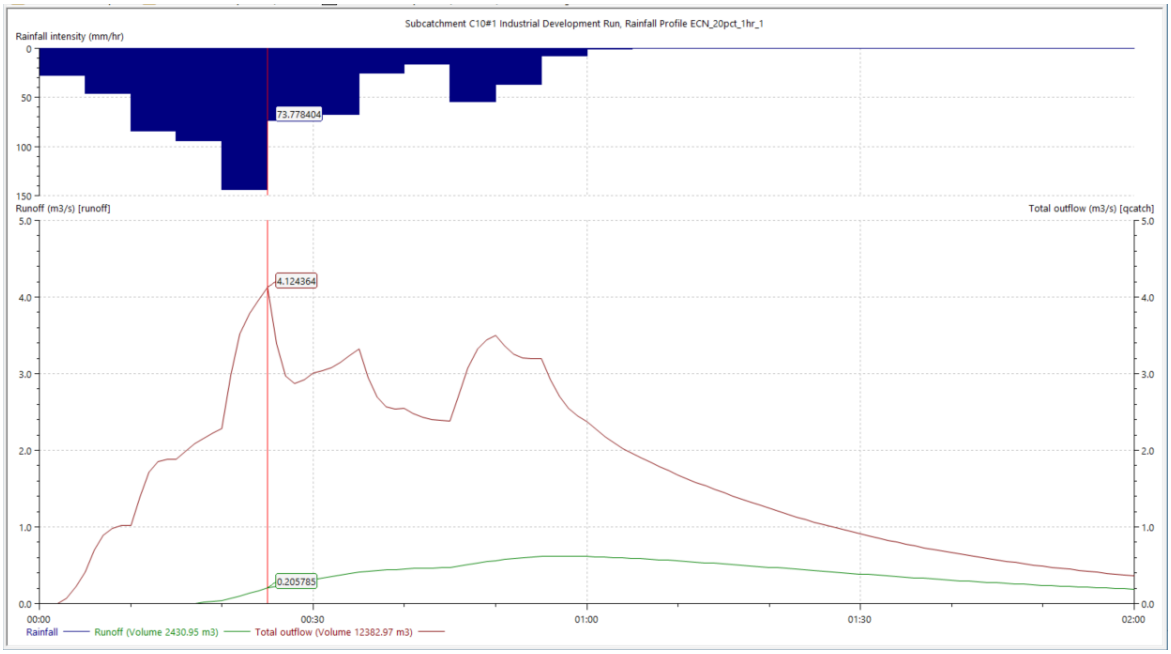
14. Open the **ECN_20pct_1hr_1** simulation results by right clicking **Open** or dragging into the GeoPlan. Select a node or channel of your choice using the **Properties** tool in the **GeoPlan** toolbar.

4D. Results Grids, Themes & Graphs

Results can be graphed using either the quick **Graph** tool or the **Graph selected objects** tool which has the red bar underneath .

15. Use the **Graph** tool and click on Subcatchment **C10#1**. Use the CTL key to select both **Runoff (m3/s)** and **Total outflow (m3/s)** from the attributes to graph list and click **OK**.

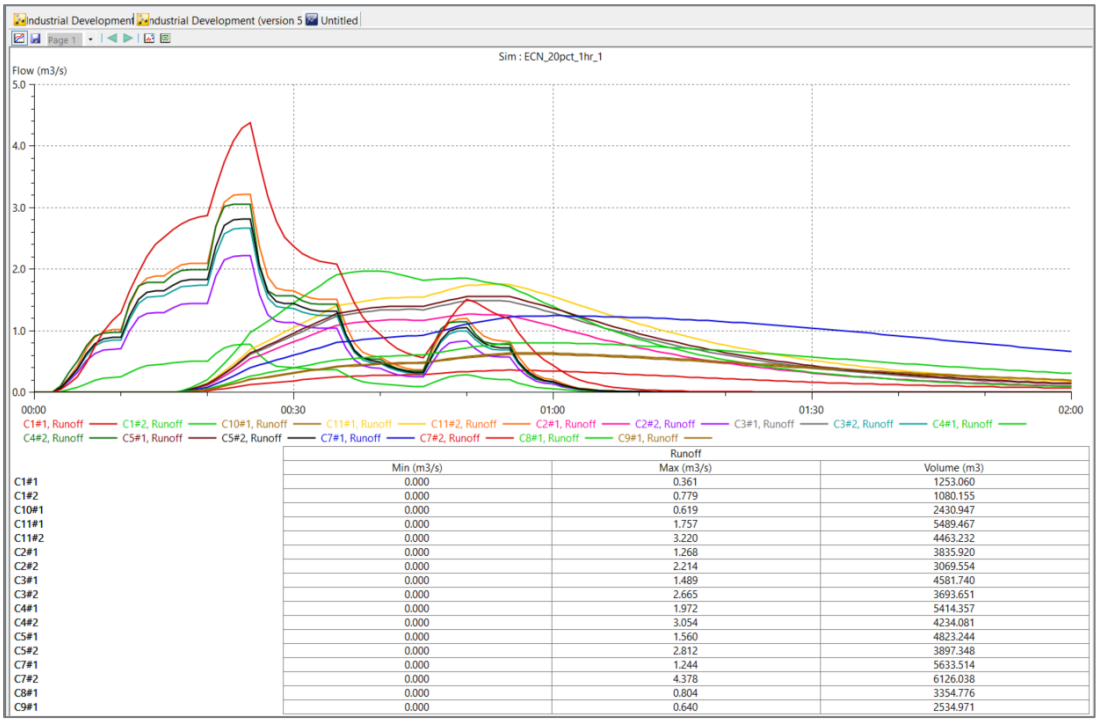




16. Select all objects in the GeoPlan using the **Select all objects** tool  . Then go to **Results > Custom graphs > Simulation per page report...**

17. Drag the completed Run object into the **Sim** window and press the **#** key in the **Selection list** to use the current selection. Move to the **Layout** tab and select **Runoff (Subcatchment)** in the **Attribute for axis (Left)** dialog and select all the Subcatchments (17 in total) from the **Plot attribute at these locations** dialog.

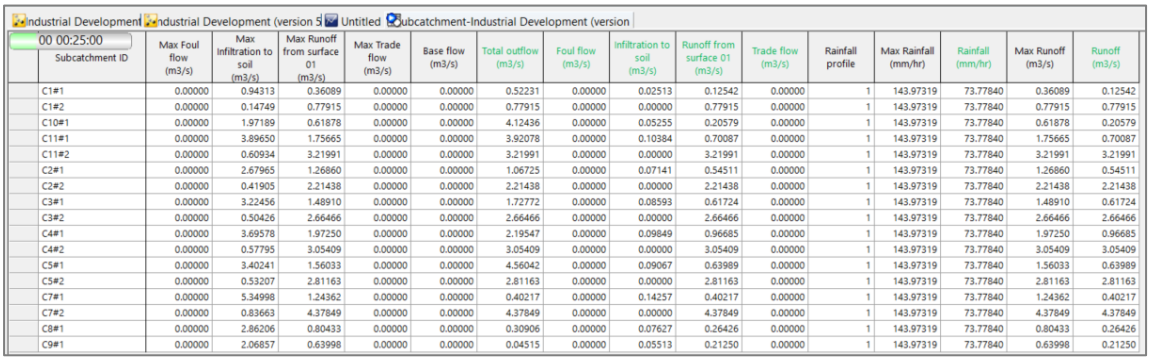
18. Click on the **Graph view** tool in the top left corner of the dialog to view the Subcatchment runoff results.



	Runoff		
	Min (m3/s)	Max (m3/s)	Volume (m3)
C1#1	0.000	0.361	1253.060
C1#2	0.000	0.779	1080.155
C10#1	0.000	0.619	2430.947
C11#1	0.000	1.757	5489.467
C11#2	0.000	3.220	4463.232
C2#1	0.000	1.268	3835.920
C2#2	0.000	2.214	3069.554
C3#1	0.000	1.489	4581.740
C3#2	0.000	2.665	3693.651
C4#1	0.000	1.972	5414.357
C4#2	0.000	3.054	4234.081
C5#1	0.000	1.560	4823.244
C5#2	0.000	2.812	3897.348
C7#1	0.000	1.244	5633.514
C7#2	0.000	4.378	6126.038
C8#1	0.000	0.804	3354.776
C9#1	0.000	0.640	2534.971

The result grids tabulate all the result information for the individual objects. You will see the default layout when we open the results grids. This can be customised and saved so that the most useful information is moved into the left-hand columns for easier viewing.

19. Grids are a good way to review the object results all together. Go to the **Window > Grid windows > New Subcatchment results window.**



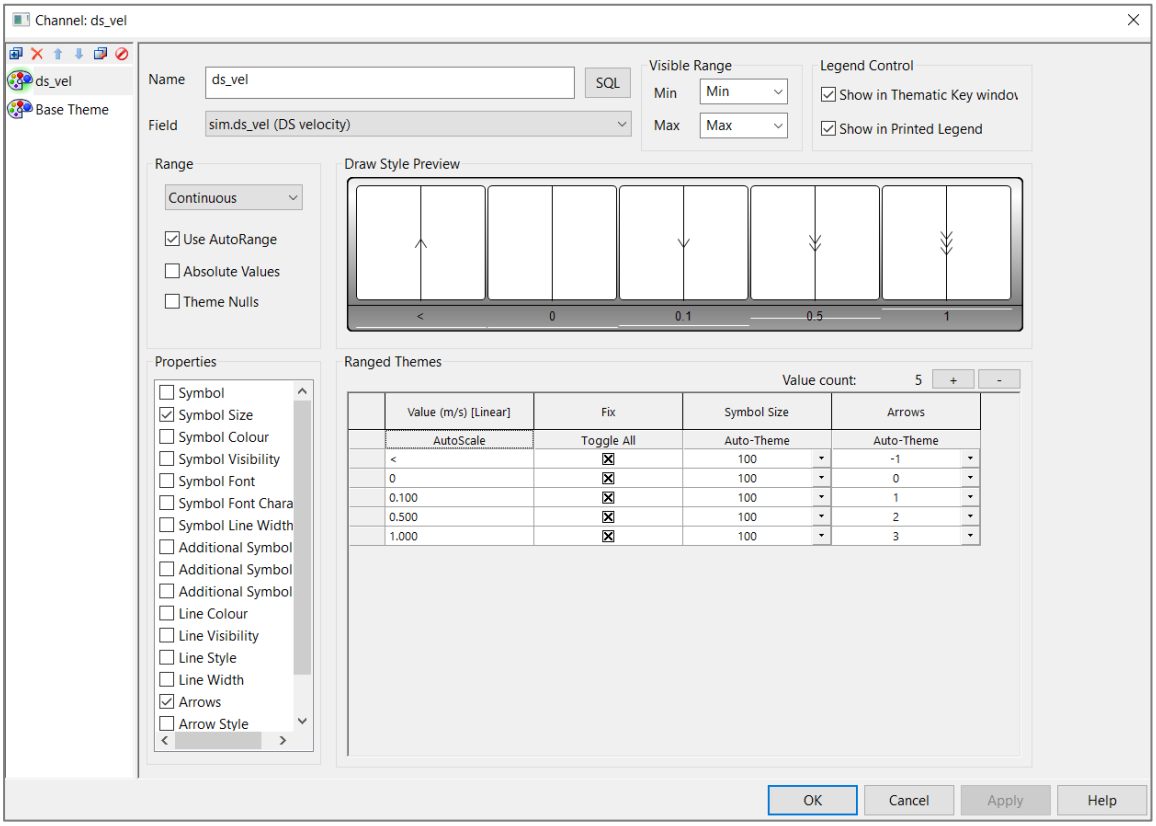
Subcatchment ID	Max Foul flow (m3/s)	Max Infiltration to soil (m3/s)	Max Runoff from surface 01 (m3/s)	Max Trade flow (m3/s)	Base flow (m3/s)	Total outflow (m3/s)	Foul flow (m3/s)	Infiltration to soil (m3/s)	Runoff from surface 01 (m3/s)	Trade flow (m3/s)	Rainfall profile	Max Rainfall (mm/hr)	Rainfall (mm/hr)	Max Runoff (m3/s)	Runoff (m3/s)
C1#1	0.00000	0.94313	0.36089	0.00000	0.00000	0.52231	0.00000	0.02513	0.12542	0.00000	1	143.97319	73.77840	0.36089	0.12542
C1#2	0.00000	0.14749	0.77915	0.00000	0.00000	0.77915	0.00000	0.00000	0.77915	0.00000	1	143.97319	73.77840	0.77915	0.77915
C10#1	0.00000	1.97189	0.61878	0.00000	0.00000	4.12436	0.00000	0.05255	0.20579	0.00000	1	143.97319	73.77840	0.61878	0.20579
C11#1	0.00000	3.89650	1.75665	0.00000	0.00000	3.92078	0.00000	0.10384	0.70087	0.00000	1	143.97319	73.77840	1.75665	0.70087
C11#2	0.00000	0.60934	3.21991	0.00000	0.00000	3.21991	0.00000	0.00000	3.21991	0.00000	1	143.97319	73.77840	3.21991	3.21991
C2#1	0.00000	2.67965	1.26860	0.00000	0.00000	1.06725	0.00000	0.07141	0.54511	0.00000	1	143.97319	73.77840	1.26860	0.54511
C2#2	0.00000	0.41905	2.21438	0.00000	0.00000	2.21438	0.00000	0.00000	2.21438	0.00000	1	143.97319	73.77840	2.21438	2.21438
C3#1	0.00000	3.22456	1.48910	0.00000	0.00000	1.72772	0.00000	0.08593	0.61724	0.00000	1	143.97319	73.77840	1.48910	0.61724
C3#2	0.00000	0.50426	2.66466	0.00000	0.00000	2.66466	0.00000	0.00000	2.66466	0.00000	1	143.97319	73.77840	2.66466	2.66466
C4#1	0.00000	3.69578	1.97250	0.00000	0.00000	2.19547	0.00000	0.09849	0.96685	0.00000	1	143.97319	73.77840	1.97250	0.96685
C4#2	0.00000	0.57795	3.05409	0.00000	0.00000	3.05409	0.00000	0.00000	3.05409	0.00000	1	143.97319	73.77840	3.05409	3.05409
C5#1	0.00000	3.40241	1.56033	0.00000	0.00000	4.56042	0.00000	0.09067	0.63989	0.00000	1	143.97319	73.77840	1.56033	0.63989
C5#2	0.00000	0.53207	2.81163	0.00000	0.00000	2.81163	0.00000	0.00000	2.81163	0.00000	1	143.97319	73.77840	2.81163	2.81163
C7#1	0.00000	5.34998	1.24362	0.00000	0.00000	0.40217	0.00000	0.14257	0.40217	0.00000	1	143.97319	73.77840	1.24362	0.40217
C7#2	0.00000	0.83663	4.37849	0.00000	0.00000	4.37849	0.00000	0.00000	4.37849	0.00000	1	143.97319	73.77840	4.37849	4.37849
C8#1	0.00000	2.86206	0.80433	0.00000	0.00000	0.30906	0.00000	0.07627	0.26426	0.00000	1	143.97319	73.77840	0.80433	0.26426
C9#1	0.00000	2.06857	0.63998	0.00000	0.00000	0.04515	0.00000	0.05513	0.21250	0.00000	1	143.97319	73.77840	0.63998	0.21250

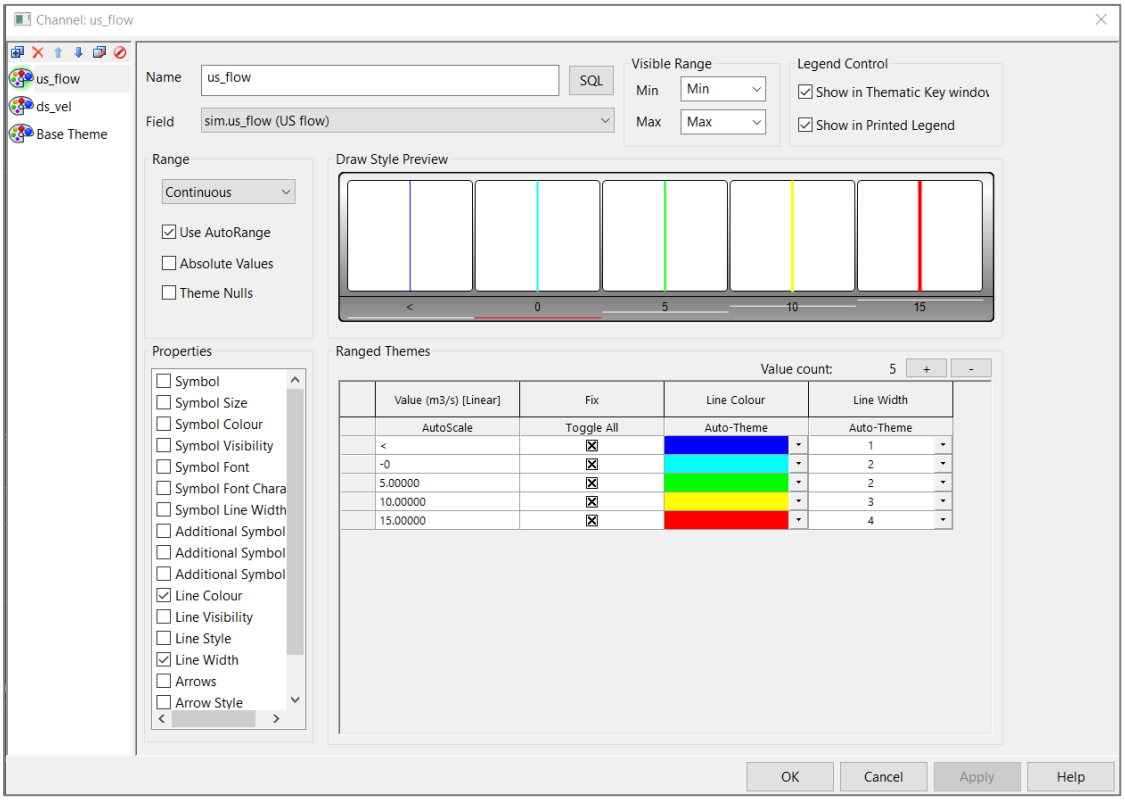
In the Subcatchment results grid you will notice green column headings which represent the time-varying results for the time step you are on and black column headings which represent the maximum results.

The process is the same for theming results and data input (as we did previously). However, when we have results open, we will get an additional set of options from the drop-down menus. During this part of the exercise, we will set-up some basic themes that will make viewing the model easier, as well as presenting some basic results.

20. Right click in the GeoPlan and go to **Properties & Themes…** Click on **Edit** for the **Channel** Object Layer under **Theme** and add two new subthemes by pressing the **Add New** box in the top left-hand corner of the dialog window.

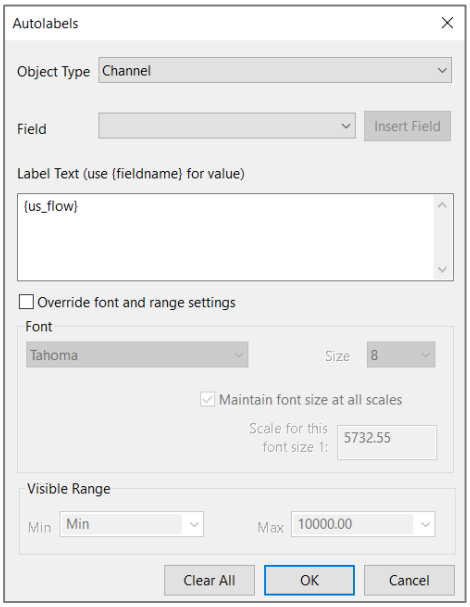
21. Edit the subthemes as per the figures below.





22. Click on the **Auto Labels and Tooltips** tab. Check the option to **Rotate Link Labels** then select **User** under the **Label Field** and click **Set**.

23. Change the **Object Type** to **Channel**, choose **US flow** in the **Field** drop-down and press **Insert Field**. Click **OK**.

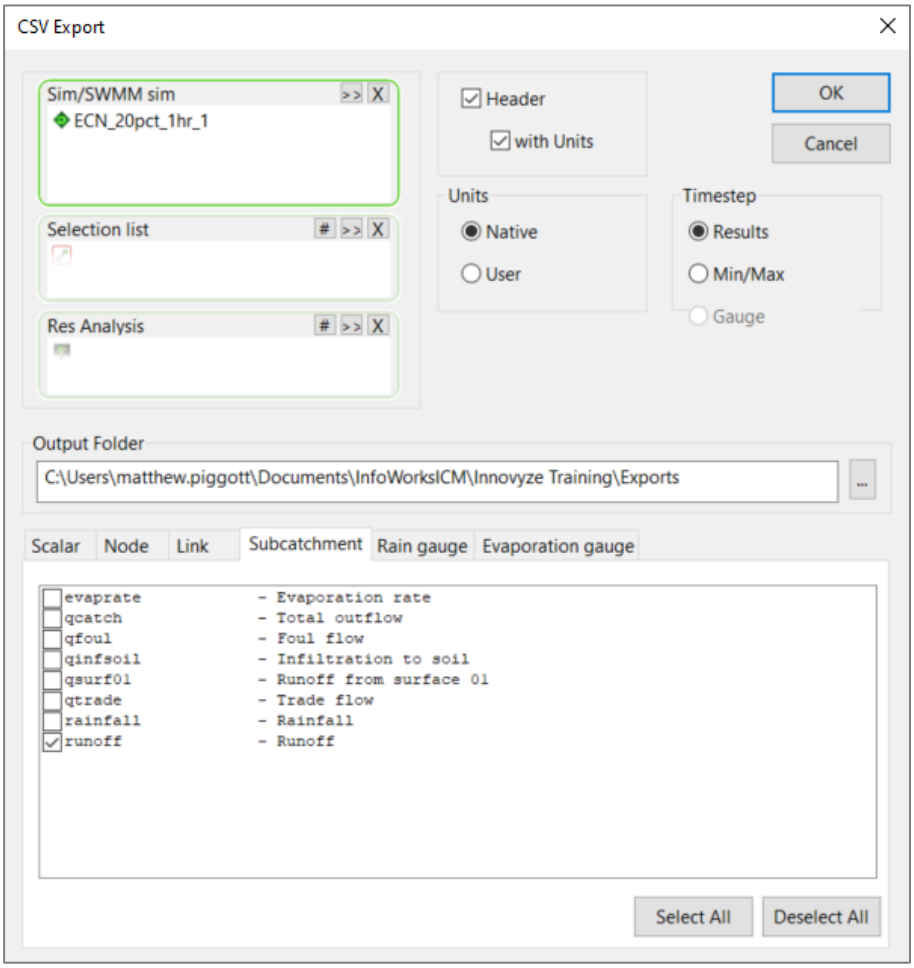


24. Go back to the **Layers and Themes** tab and check on the box for **AutoLabel** next to **Channel**.
25. Save the themes under **Save to database object** to the RAFTS Part1 model group with the name **RAFTS Sim Results.**

4E. Exporting Results

Export subcatchment hydrographs to CSV.

26. Go to **Results > CSV export…** drag in the ECN_20pct_1hr_1 simulation object, check **runoff** under the **Subcatchment** tab and set the **Output Folder** to an appropriate location. Click **OK**.

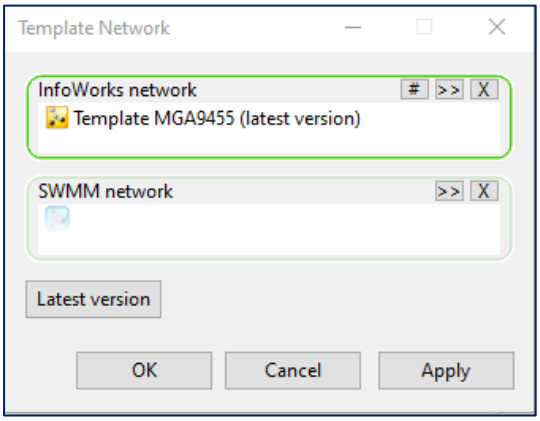


Workshop 5. Creating a New Network

5A. Using Network Templates

We will use a Template Network to create a new network to shortcut setting up the background information and settings from workshop 1 and 2. Any network created from a template will contain all the data and settings from the templated network. In this example, the template contains some default node, link and Subcatchment settings, themes and the projected co-ordinate system. It also has a development site and existing ground contour information.

1. Open up the **RAFTSinICM.icmt** transportable database from the training data folder again and copy the **RAFTS Part2 - Template** Model group in the master database.
2. Go to **Tools > Template Network**. Drag in the **Template MGA9455** from the files copied from the transportable database and click **Apply** then **OK.**



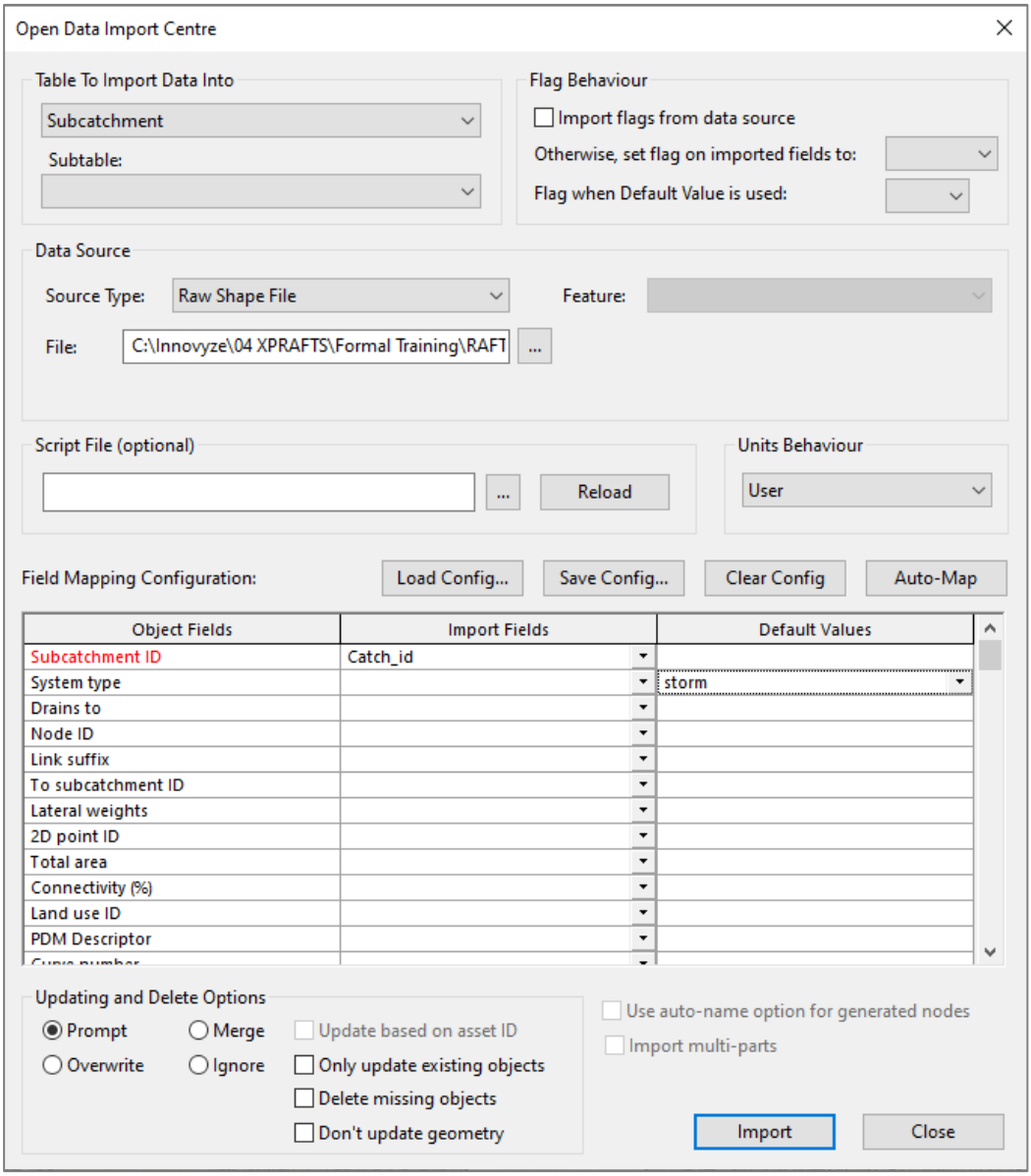
3. Right click on the Master Database Object in the Explorer Window and select **New InfoWorks > Model group** and give it the name **RAFTS Part2.** This will be the folder that contains all your project components (network, rainfall, results etc).
4. Right click on the Model group and select **New InfoWorks > InfoWorks network**, type the name **Urban Development** and check on **Use template network.** Press **OK.**
5. Double click on the new **Network object (Urban Development)** to open in the GeoPlan and maximise the window.
6. Right-click on the GeoPlan and select **GIS layer control…** Press **Add…** and select **Aerial_2.jpg** from the Part 2 data folder. Note that the background layer is a Raster image type.

7. Save the GIS image to the Model group as a new Layer List: **Aerial Img – P2**. Now drag on the **Label list: Development Site** from the copied transportable database files. The network should look like the below.

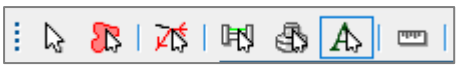
5B. Adding Subcatchments

From the contour information we can see there is an external catchment draining to the top of the development site. We will start by importing Subcatchment data from GIS and manually create any missing subcatchments that drain to the development site.

8. To import the subcatchments go to **Network > Import > Open Data Import Centre**…
9. Set the Table to Import Data Into as **Subcatchment** and set the Data Source Type to **Raw Shape File**. Press the ellipsis button and browse to **UrbanDev_Subcatchments.shp** in the **RAFTS in ICM data/ Part 2** folder path.
10. Under Import Fields, set the **Subcatchment ID** equal to the **CatchID** and the **System type** to **storm** from the drop down under **Default Values**. Click **Import** and then **Close**. You should get a message outlining that 3 new objects have been imported.

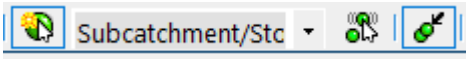


11. Use the **Custom label** icon in the GeoPlan toolbar to add a label to the two subcatchments shown in the image below.



Note that the subcatchments do not cover the full area draining to the top of the development site. We will add the missing Subcatchment manually using the Geoplan tools.

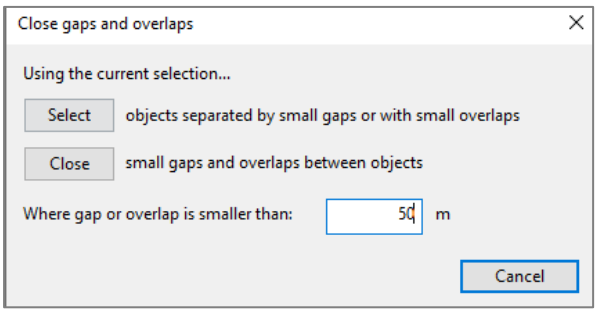
12. To add the missing Subcatchment manually, select **Subcatchment/Storage** from the dropdown menu and click on the **New object** tool to start drawing. Using the background contours for reference, add in the missing catchment area. Ensure that the **Use snap mode** is turned on.



13. Set the **Name** to **C_2** and the **Type** to **Subcatchment.**
14. Add a **Custom label** to the new Subcatchment.

15. We can use the Model tools to ensure the 4 subcatchments have snapped vertices and no gaps or overlaps. Hold down CTRL to select all 4 polygons using the select tool. Alternatively, you can right-click on **Subcatchments** in the **Key** window and choose **Select objects**.

16. Go to **Model > Geometry > Close gaps and overlaps**... Set the area to **50m** and then click **Select** and **Close**. Once there are no more gaps or overlaps select **Cancel** to close the window.

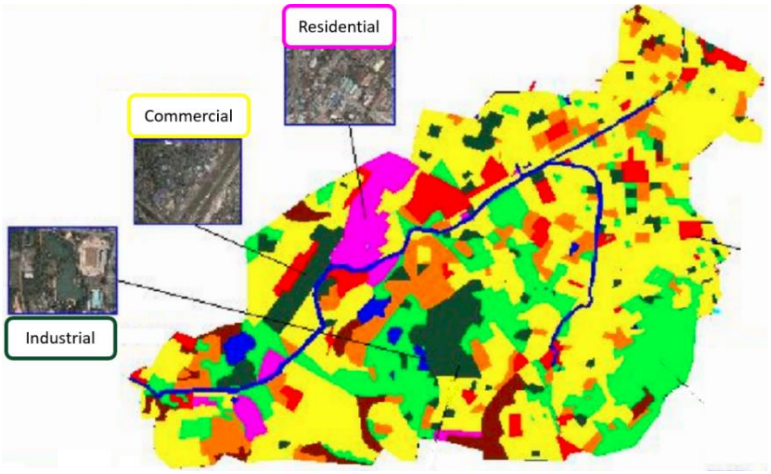


17. For the purpose of the training, we will manually overwrite the area of the drawn subcatchment to ensure consistent results. Open the Subcatchment properties for **C_2** and set the **Total area (ha)** as **4.57**. Note the contributing area should automatically update.

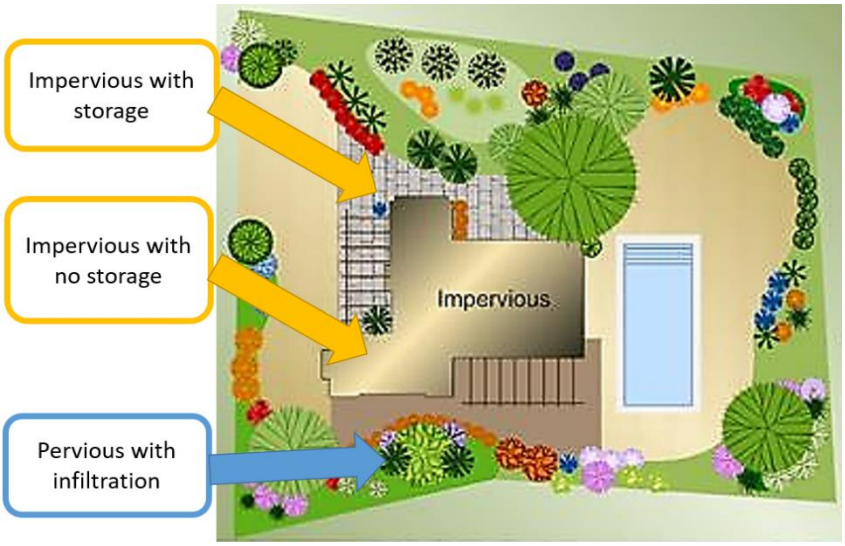
5C. Entering Subcatchment Data

A **Subcatchment** describes an area draining to one or more inflow nodes. Some of the parameters are defined directly in the subcatchment. The specific Runoff Surfaces are defined in a Land Use which is associated with one or more subcatchments. The Land Use also provides some default parameters for the subcatchment. These defaults can be over-ridden in the subcatchment data if necessary.

A **Land Use** contains a list of Runoff Surfaces particular to the type of Land Use being defined. Example Land Uses are Residential, Industrial, Commercial, etc. It also contains default values for several factors, including the percentage of total area for each Runoff Surface, Population Density, and Connectivity. The two typical surface types used with the RAFTS Procedure are Pervious and Impervious. The software allows use of up to 12 Runoff Surfaces per Land Use to support the modelling of more complex mixed surfaces.

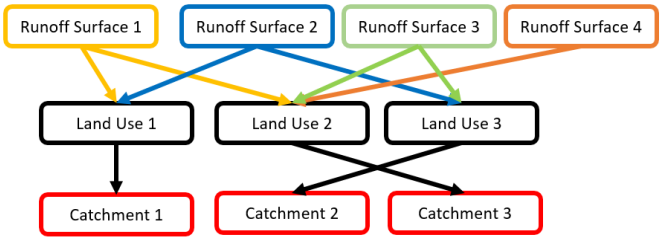


A **Runoff Surface** describes the runoff characteristics of a specific surface type, for example Road, Roof or Pasture. For each surface type individual characteristics can be defined that determine how runoff of rainfall occurs on that surface, the volume of runoff and the rate at which it enters the drainage system. Runoff Surfaces include parameters like **Infiltration**, **Manning's n** and **Routing Model**.



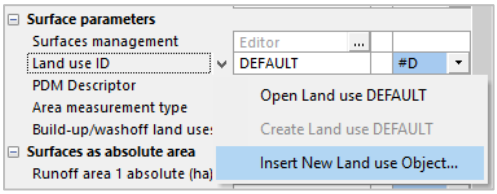
In the example above, we can assume that this typical **Urban Land Use** has 3 types of **Runoff Surfaces:** Impervious with Storage, Impervious with no storage, and Pervious with Infiltration. Each surface has its own characteristics, and runoff subcatchment when rainfall falls on them.

The relationship between Runoff Surface, Land Use and Catchments is illustrated in this example below:

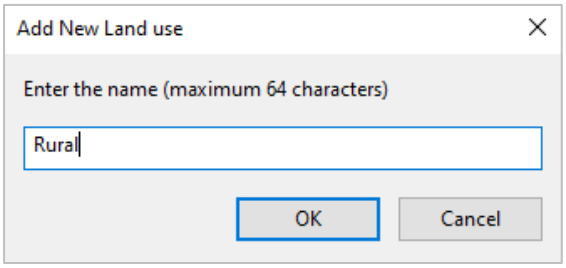


We are going to create a new Land Use with user defined Runoff Surfaces to apply to our subcatchments.

18. Double click on Subcatchment **C_1** to open the properties dialog. Click on the down arrow next to the **Land use ID** under our new **RAFTS** group heading and go to **Insert New Land use Object…**

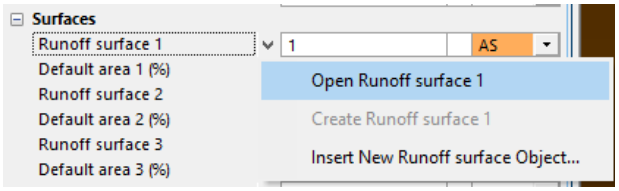


19. Name the **Land use ID**, **Rural** and click **OK**. A land use can be made up of up to 12 different Runoff Surfaces (eg. Roads, buildings, vegetation, waterbody).

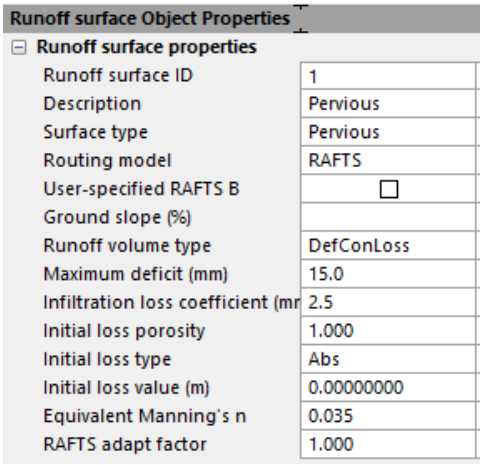


NOTE: When using RAFTS hydrology in Urban areas, a minimum of 2 runoff surfaces are recommended. One for impervious areas and one for pervious areas.

20. From the dropdown menu set the **Runoff surface 1** to **1**. Then use the down arrow key to **Open Runoff surface 1**.



21. Populate Runoff surface 1 as; Description to **Pervious**, Surface type to **Pervious**, Routing model to **RAFTS**, Ground slope (m/m) is **NULL**. Runoff volume type equal to **DefConLoss**, Maximum deficit (mm) to **15**, Infiltration loss coefficient (mm/hr) is **2.5**, Initial loss type is **Abs**, Initial loss value (m) to **0,** Equivalent manning's n to **0.035**.



22. Use the green back arrow in the properties dialog to go back to the **Land use Object Properties.** Set the Default area 1 (%) to **100**. This will set the Rural Land use to be 100% pervious if not overwritten manually on each subcatchment.

Now that we have setup the Runoff Surfaces that make up our Land use, we need to apply the this to the other subcatchments and also set the subcatchment slope.

23. Go to **Window > Grid windows > New subcatchments window**. Highlight the **Subcatchment ID** column and right click on the header to **Freeze Columns**. This will show the Subcatchment ID at all times in the table.

NOTE: To change the order of the data in the grid windows, highlight a column, right click and choose one of the **Sort** options provided or set your own rules.

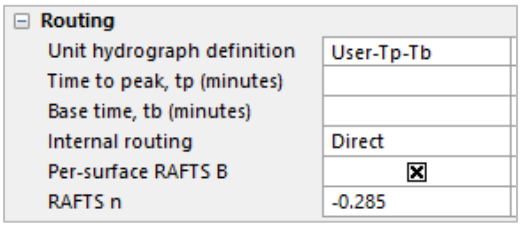
24. Scroll to the **Land use ID**, highlight the column and right click in the first cell and go to **Current Cell(s) Value > Set new value(s) for cell(s).** Set all subcatchments to **Rural** and click **OK**.

25. Scroll further across to the **Slope (m/m)** column. Populate the Subcatchment slopes for each subcatchment as per below:

Subcatchment	Slope (m/m)
C_1	0.40
C_2	0.40
C_3	0.20
C_4	0.04

Note: The Subcatchment slopes in this example are >10%, so only a rough estimate of the slope is needed as the impact of changing this by say +- 10% is unlikely to affect results.

26. Finally, ensure you check on the **Per-surface RAFTS B** option for each subcatchment. This will tell ICM to calculate **B** from the parameters you provided in the **Runoff surfaces** (for example manning's n, Surface Type, RAFTS adapt factor also known as BX) and not from those in the subcatchment.



27. Right click on Urban Development Network and **Commit changes to master database**. Add to the comments **Workshop 5 - Added Land use, Runoff surface and Subcatchment data** and click **OK**. Select **No** to validating the model at this stage.

Workshop 6. Hydrograph Translation

6A. Draining the Subcatchment

ICM allows Subcatchments to drain to a variety of different network elements. The locations a Subcatchment can drain to within ICM are described below.

Database Value	Description
Node	Specify a Node ID to which the subcatchment drains.
Link	Specify a Node ID and Link Suffix to which link the subcatchment drains.
Subcatchment	Specify Subcatchment ID to which the subcatchment drains. This also allows for Bypass flow / lagging links.
Multiple Links	Subcatchment flows can be distributed to multiple links via a list of conduits and weighted flows.
2D Point Source	Specify a Point Source ID to which the subcatchment drains allowing direct distribution to the 2D mesh.

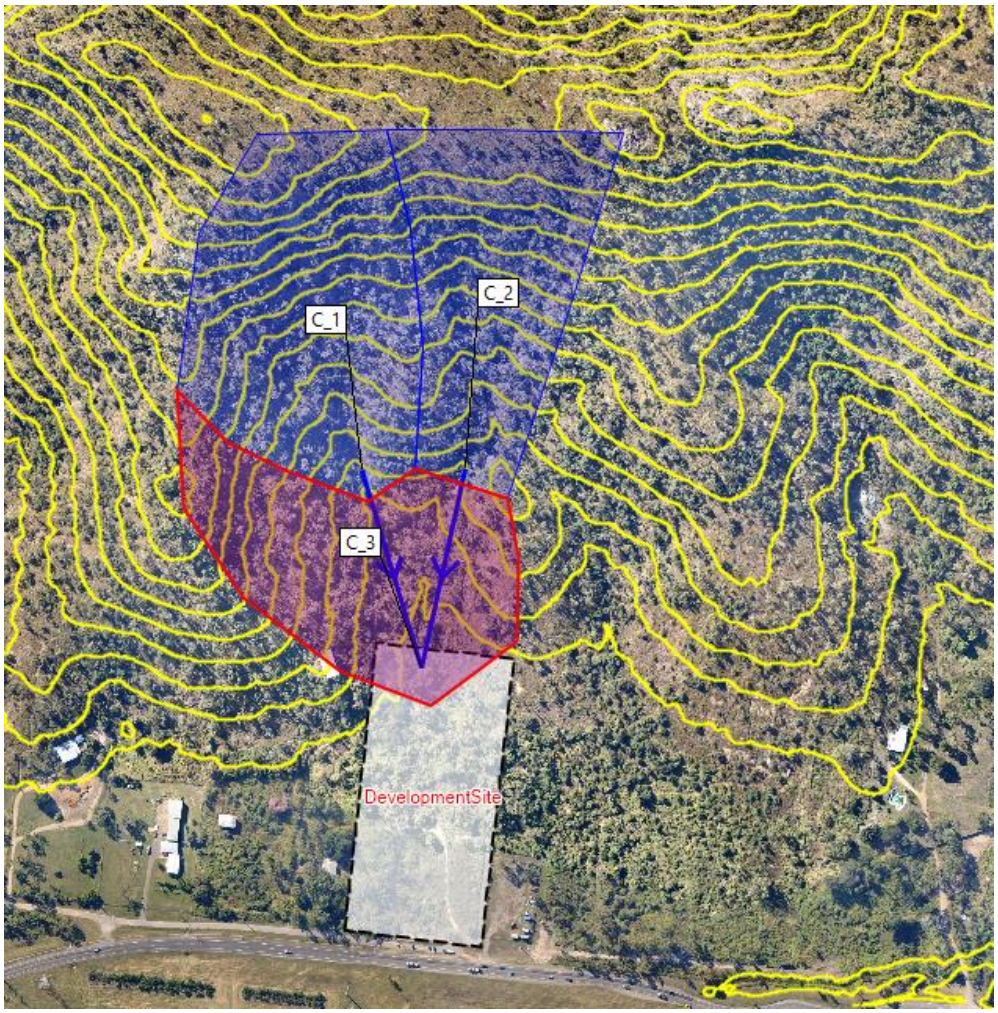
Lagging Links

In this example, the upper 3 subcatchment flowpaths will be represented as lag links. We will drain Subcatchment C_1 and C_2 to the outlet of C_3.

1. Double click on Subcatchment **C_1** to bring up the properties page. Change the **Drains to** to **Subcatchment** and the **To subcatchment ID** to **C_3.**
2. Then repeat the above steps for **C_2.**

NOTE: The output lag on the Subcatchment represents the time it takes the hydrograph to move from the outlet of C_1 and C_2 to the outlet of C_3. We can manually move the drainage arrows from the centroids to the outlets to better represent this connection.

3. Select the **Edit object geometry** tool from the **GeoPlan** toolbar and click on the end of the arrows to show the vertex and then drag the vertex to desired location (see image below).



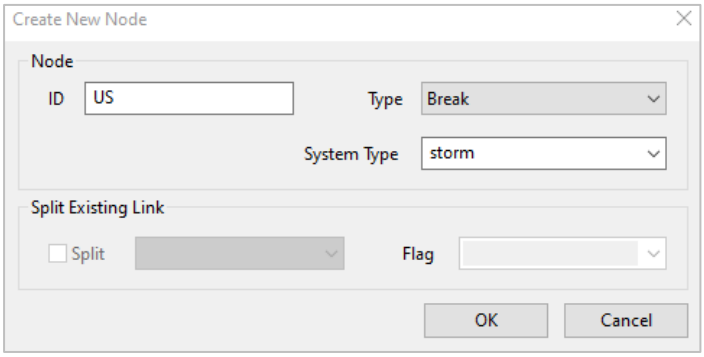
4. Lastly, set the **Output lag (minutes)** to **5** and check on the **Bypass runoff** for both subcatchment **C_1** and **C_2**. This can be done in the property window for each subcatchment or in the subcatchment grid windows.

Output lag (minutes)	5.00
Bypass runoff	☒

Channel Links

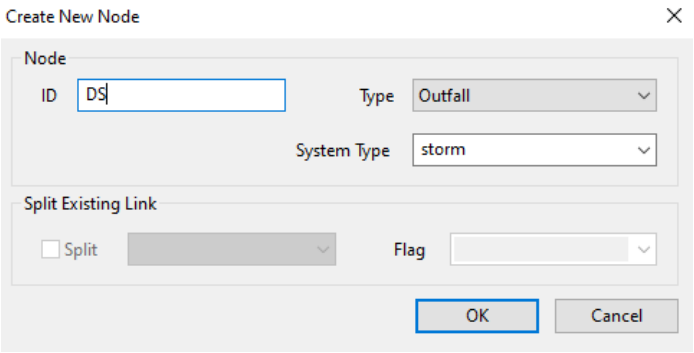
Subcatchment C_3 will outlet to the top of a channel which will be sized as part of the project.

5. Select **Node** from the New object dropdown list and select the **New object** icon from the toolbar. Create a new node at the top of the north end of the development site and set the ID to **US**, the Type to **Break** and the System Type to **storm**. Click **OK**.



6. Press Esc to exit out of the new object pointer and bring up the properties page for **C_3.** Set the Drains to Node ID to **US**.

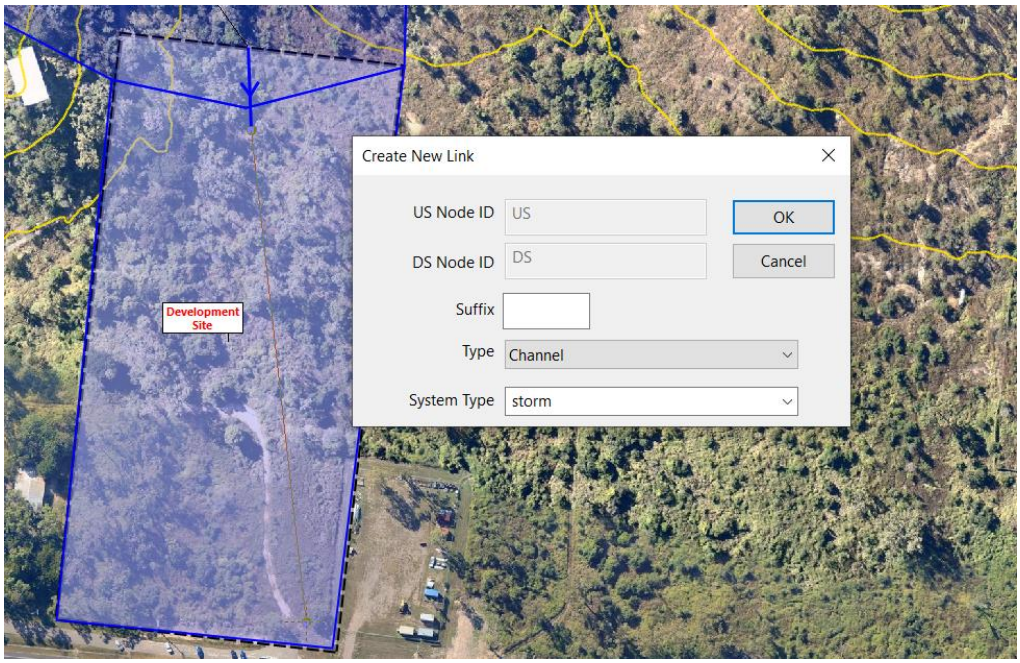
7. Choose the **New object** tool again and draw a new node at the downstream end of the development site. Set the ID to **DS**, the Type to **Outfall** and the System Type to **storm**. Click OK.



8. The properties page for the new Outfall should appear and ICM will provide a red validation error where the **Ground level (m AD)** needs to be populated. Set it to **10**.

Now we will add the channel.

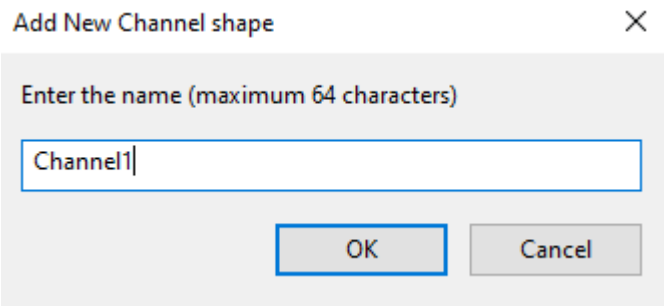
9. Select **Link** from the dropdown list and select **New object** from the toolbar. Create a new link from **US** to **DS** nodes. Set the Type to **Channel** and the System Type to **storm**. Click **OK**. The Suffix with automatically adopt **1**.



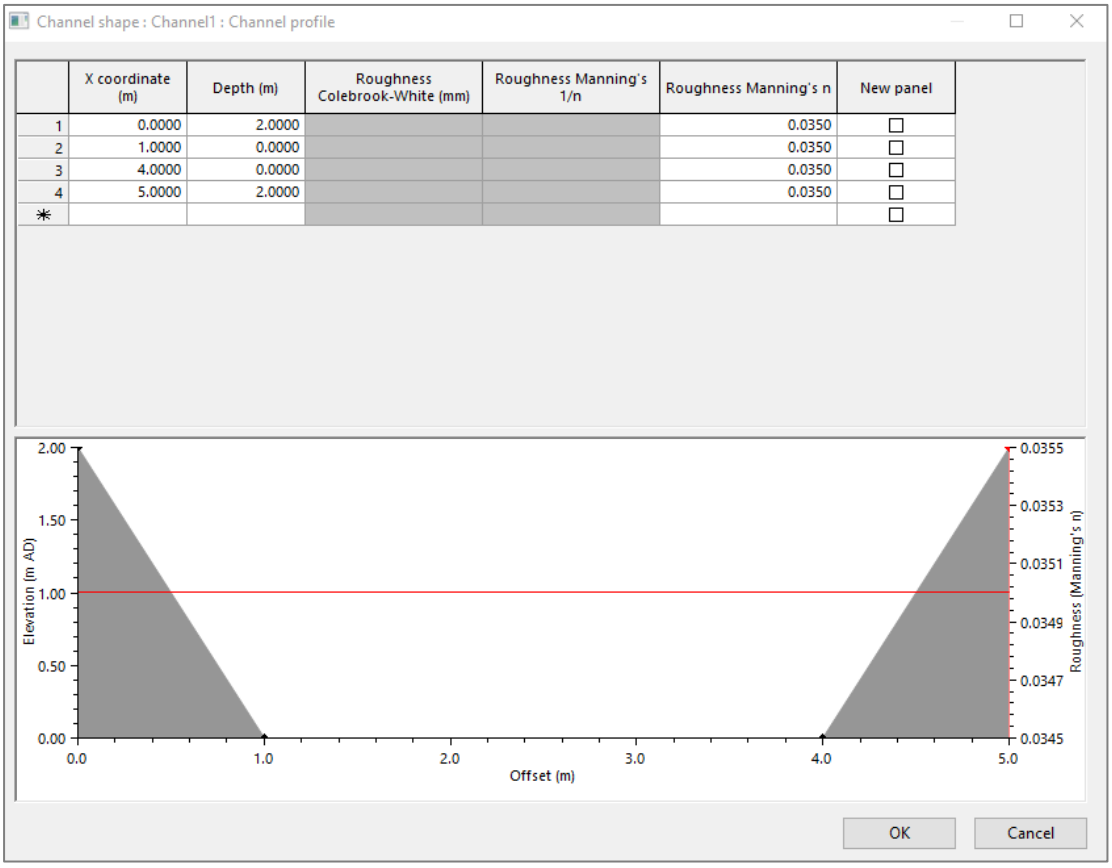
10. The properties page for the new channel should appear and ICM will provide a red validation error where the invert levels and section shape need to be populated. Set the **US invert** to **20** and **DS invert** to **10.**

11. Override the **Length (m)** to equal **250**.

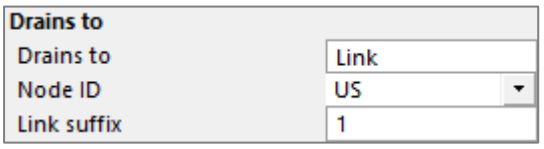
12. Click on the down arrow next to the Shape ID and **Insert New channel shape object…** Give it the name **Channel1**. In the new properties page, set the roughness type to **N** and click on the 3 ellipses to set the **Channel profile.**



13. We will start with a 5m wide, 2m deep channel with a manning's n of 0.035. Set the **Channel shape** up as shown below and press **OK**.



14. The Subcatchment covering the development site will drain to the channel for now. Bring up the properties page for **C_4** and set the Drains to field to **Link,** the Node ID to **US** and the Link suffix to **1.** This will add the flows midway.



15. We are ready to validate the network and commit the changes. Right click on the Urban Development Network and select **Commit changes to master database**. In the comments add: **Workshop 6 – Added lags, lag links and a channel**. Click **OK** and **Yes** to validating the network.

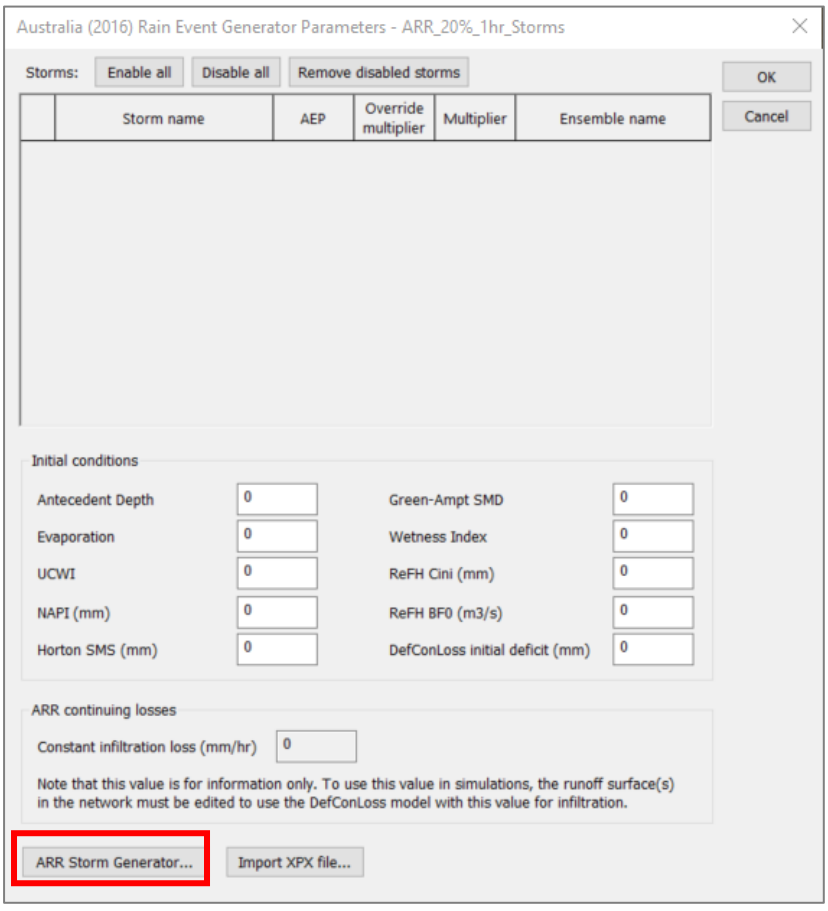
Workshop 7. Rainfall Data

7A. ARR Rainfall Generator

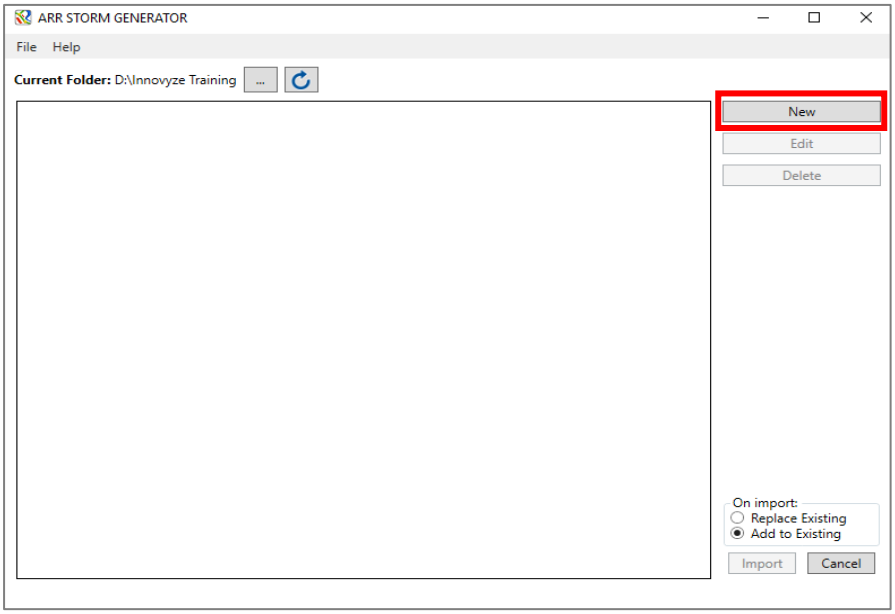
We are going to use the ARR Storm Generator in ICM to create a new rainfall object containing design storms from the ARR Data Hub and BOM websites.

1. To create a new Rainfall object, right click on the Model Group and select **New InfoWorks > Rainfall event.** Give it the name **ARR19_Ensembles** and click **OK**. Select **Australian (2016) Rainfall** and click **OK**.

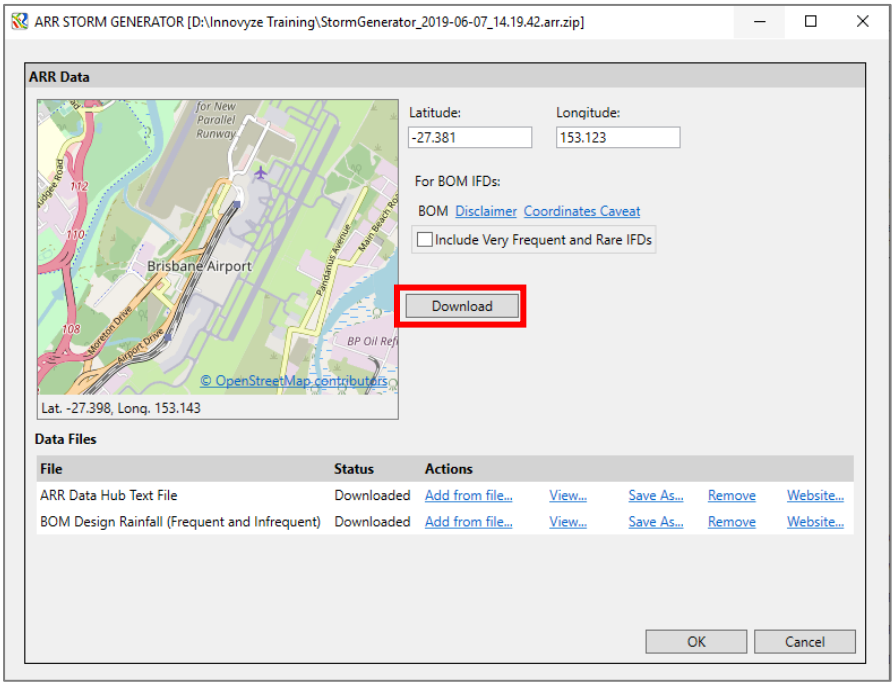
2. Select the **ARR Storm Generator...** button from the bottom left corner.



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:\...\RAFTS in ICM Training\).



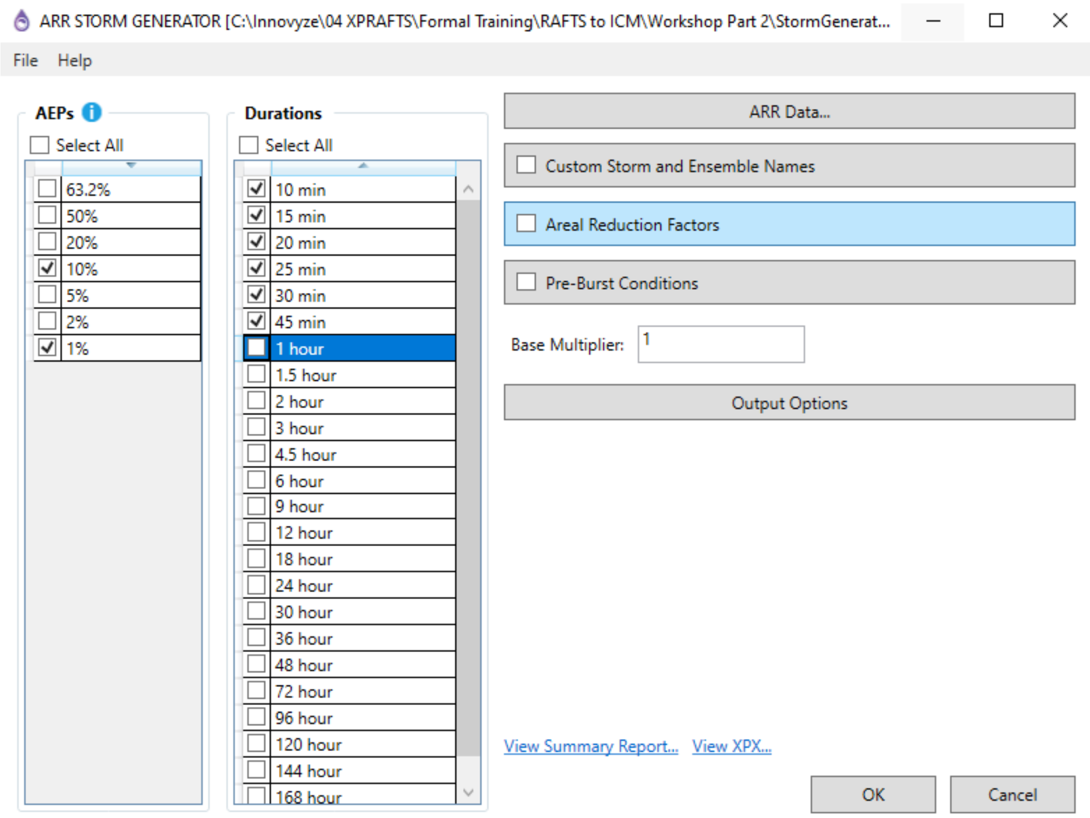
3. Select **New** to create a new dataset of storms. The AEPs and Durations lists should be empty.
4. To populate the storm data (AEPs and Durations), select **ARR Data…** from the top right side of the dialog box.
5. Use the centre scroll button on the mouse to zoom and pan to find the Brisbane Airport. 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 AEPs and Durations lists should now be populated. **Uncheck** the **Select All** box at the top of both lists to deselect all. Then select the desired events **(10% and 1% AEPs, 10-45min durations).**



8. Click on the **View Summary Report...** to check the storms that will be added to the model. There should be 120 (6 ensemble containing 10 patterns for both the 10% and 1% AEPs). If you are happy with the summary report, select **OK.**

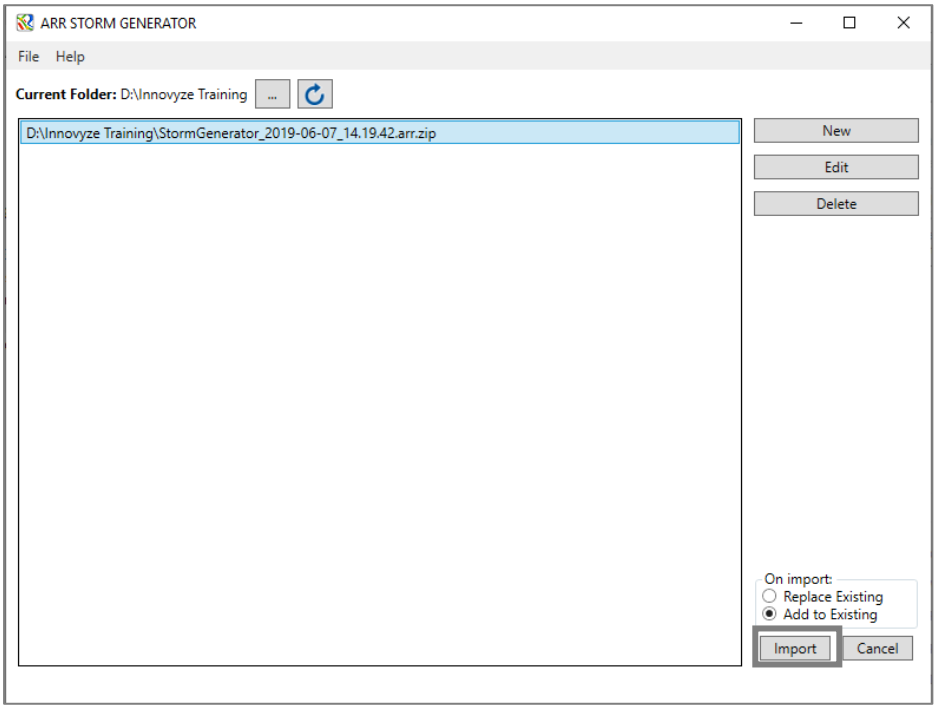
2. INFILTRATION GLOBAL DATABASE

Global Database Name: ARR I=13,C=2.2
Initial Loss: 13
Continuing Loss: 2.2

3. GLOBAL STORMS

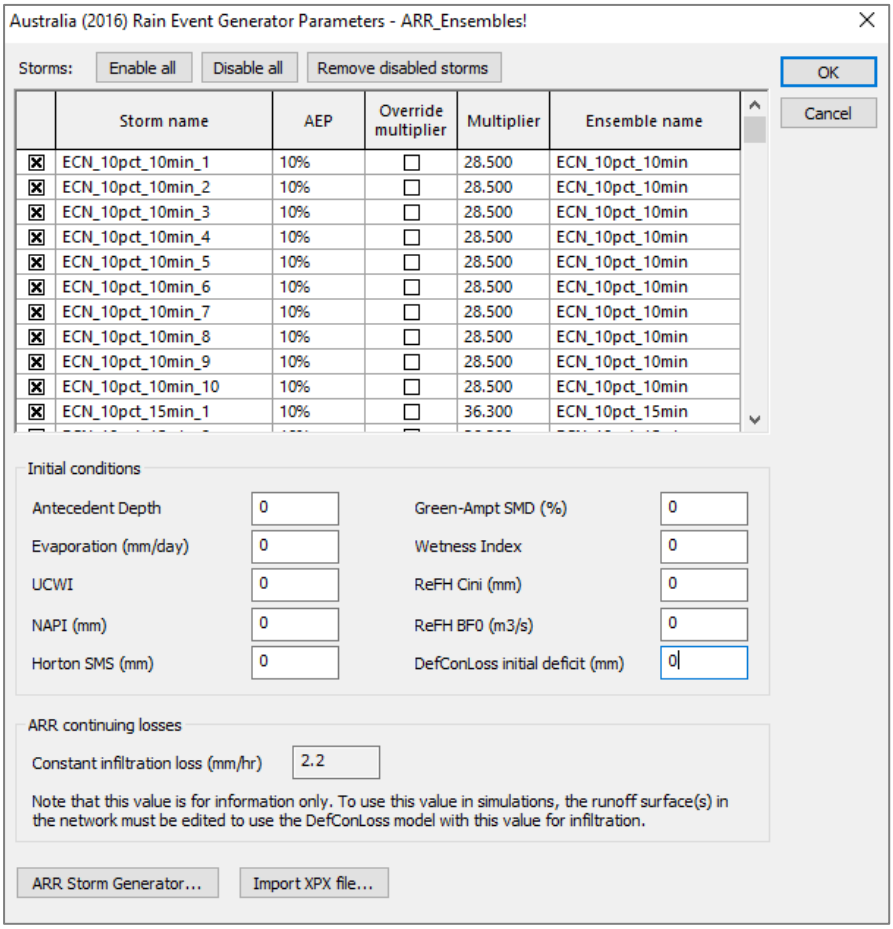
Storm Name	Ensemble Name	TP Region	Design Rainfall Class	AEP	Duration	Pattern Number	IFD	ARF	Base Multiplier	Pre-Burst Ratio	No. of Pre-Burst Depth Timesteps
ECN_10pct_10min_1	ECN_10pct_10min	ECnorthIntermediate		10%	10min	1	28.5	-	1	-	-
ECN_10pct_10min_2	ECN_10pct_10min	ECnorthIntermediate		10%	10min	2	28.5	-	1	-	-
ECN_10pct_10min_3	ECN_10pct_10min	ECnorthIntermediate		10%	10min	3	28.5	-	1	-	-
ECN_10pct_10min_4	ECN_10pct_10min	ECnorthIntermediate		10%	10min	4	28.5	-	1	-	-
ECN_10pct_10min_5	ECN_10pct_10min	ECnorthIntermediate		10%	10min	5	28.5	-	1	-	-
ECN_10pct_10min_6	ECN_10pct_10min	ECnorthIntermediate		10%	10min	6	28.5	-	1	-	-
ECN_10pct_10min_7	ECN_10pct_10min	ECnorthIntermediate		10%	10min	7	28.5	-	1	-	-
ECN_10pct_10min_8	ECN_10pct_10min	ECnorthIntermediate		10%	10min	8	28.5	-	1	-	-
ECN_10pct_10min_9	ECN_10pct_10min	ECnorthIntermediate		10%	10min	9	28.5	-	1	-	-
ECN_10pct_10min_10	ECN_10pct_10min	ECnorthIntermediate		10%	10min	10	28.5	-	1	-	-
ECN_10pct_15min_1	ECN_10pct_15min	ECnorthIntermediate		10%	15min	1	36.3	-	1	-	-
ECN_10pct_15min_2	ECN_10pct_15min	ECnorthIntermediate		10%	15min	2	36.3	-	1	-	-
ECN_10pct_15min_3	ECN_10pct_15min	ECnorthIntermediate		10%	15min	3	36.3	-	1	-	-
ECN_10pct_15min_4	ECN_10pct_15min	ECnorthIntermediate		10%	15min	4	36.3	-	1	-	-
ECN_10pct_15min_5	ECN_10pct_15min	ECnorthIntermediate		10%	15min	5	36.3	-	1	-	-
ECN_10pct_15min_6	ECN_10pct_15min	ECnorthIntermediate		10%	15min	6	36.3	-	1	-	-

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**.



10. The storms can be toggled on and off in the Rainfall Dialog Editor. To apply the initial loss at the Subcatchment level, change the **DefConLoss initial deficit (mm)** back to 0. Click **OK** to accept the current rainfall storms.

NOTE: The continuing losses from the Data Hub are shown in the dialog however they are for information only.



NOTE: Rainfall objects become 'Read Only' once used in a Run. This is to allow Run objects to be re-simulated later without changes to the outcome.

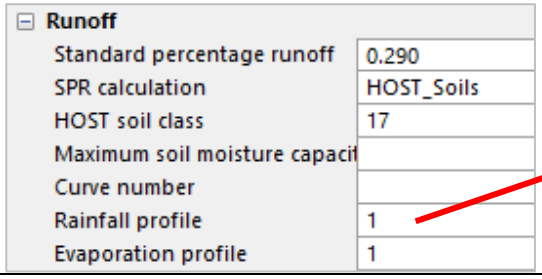
11. You can view each rainfall event details individually by right clicking on the Rainfall object and going to **Open As... > Grid Editor (TVD Read only)** and the selecting an event to view.

	Time	ECN_10pct_10min_1 (mm/hr)
	SUB-EVENT 1	
	00::00:00	181.773000
	00::00:05	160.227000

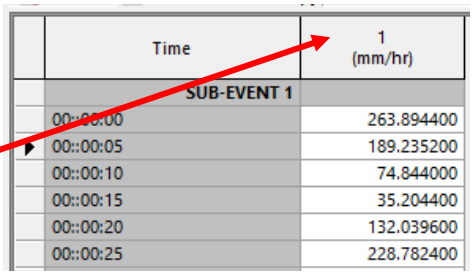
7B. Spatial Rainfall Profiles

By default, all subcatchments in the model will have the Rainfall and Evaporation Profiles set to 1 and is not used unless there are multiple rainfall profiles applied to a simulation. However, these can be manually updated to apply different rainfall profiles to different subcatchments within the same network.

Subcatchment Dialog



Rainfall Object Dialog

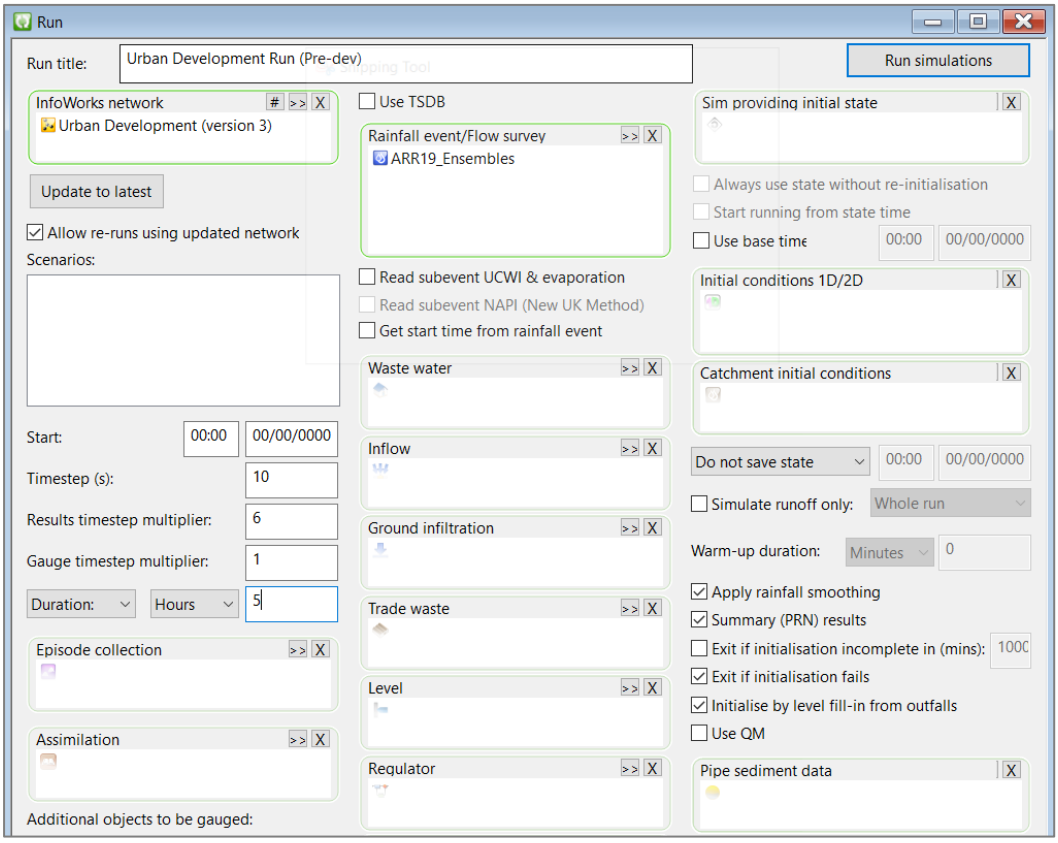


Workshop 8. Running ARR Storms & Analysing Results

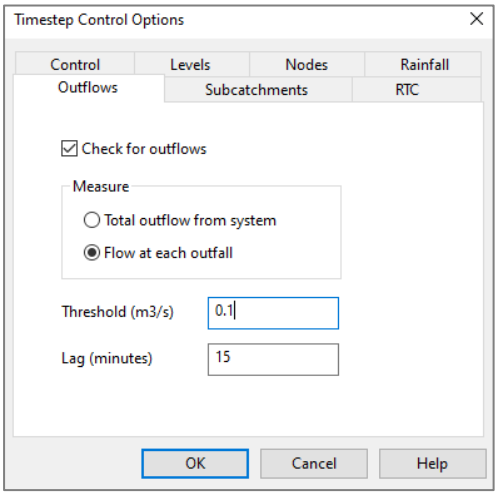
8A. Setting up the Run Object

Now we are going to set up the simulation data and parameters and run the simulation with the new rainfall ensembles.

1. Right click on the **RAFTS Part2** Model Group and select **New InfoWorks > Run.**
2. Fill in the Run title as **Urban Development Run (Pre-dev).**
3. Select the **Urban Development** Network and the **ARR19_Ensembles** object from the Group Window (Use the CTRL key to multi-select) and drag them onto the Run Window dropping anywhere within the grey area of the dialog.
4. Check the **Allow re-runs using updated network** option which means we can re-run and overwrite the results rather than creating a copy of the run later.
5. Change the Timestep (s) to **10**, the Result timestep multiplier to **6** and the duration to **5** Hours.



6. There is no need to simulate a 10min storm for 5hrs as the flows would have passed through the network well before 5hrs. To run the simulations efficiently, click on the **Timestep Control** and go to the **Outflows** tab. Check on the **Check for outflows** dialog and set the flow **Threshold (m3/s)** to **0.1** and the **Lag (minutes)** to **15**. This will stop the simulation 15min after there is less than 0.1m3/s outflow. Click **OK**.



7. Click **Run simulations** and **OK**.

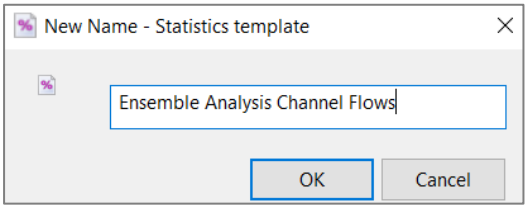
NOTE: Once the simulations are complete, it is a good idea to make sure the flows look OK in the model before running the ensemble statistical analysis. Refer to Workshop 4 for reviewing results.

8B. ARR Statistics

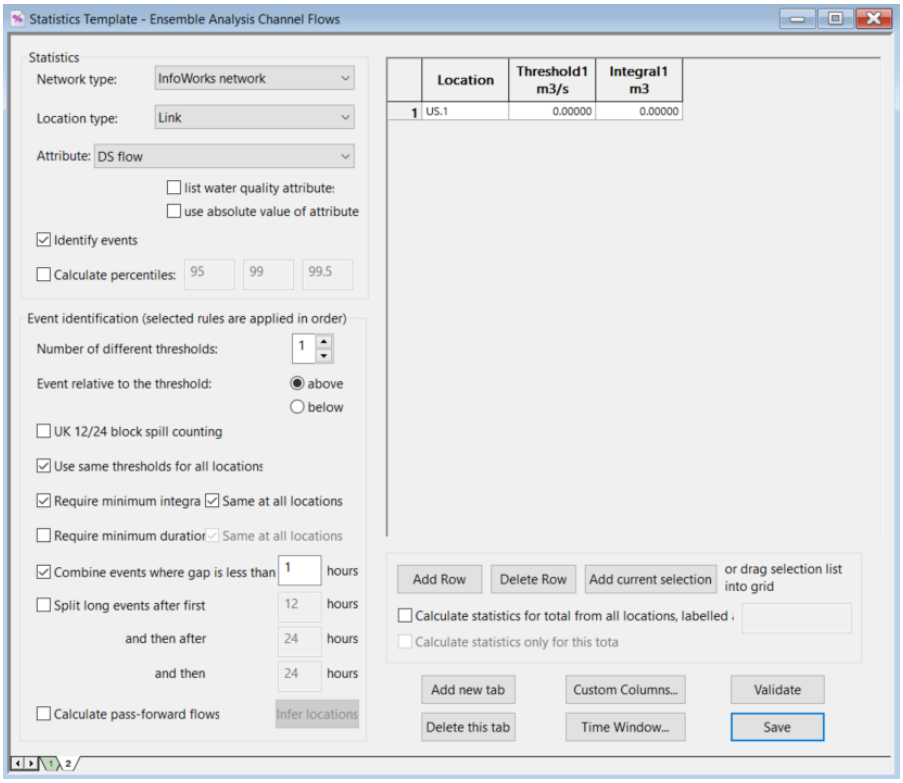
Statistical analysis can be run on a Subcatchment, node, link or 2D network result object for any associated result field. They can include multiple simulation results (such as an ensemble) or a single result such as looking at the average depth over time.

We will create a Statistical Report to analyse the ensemble results at the downstream end of the network.

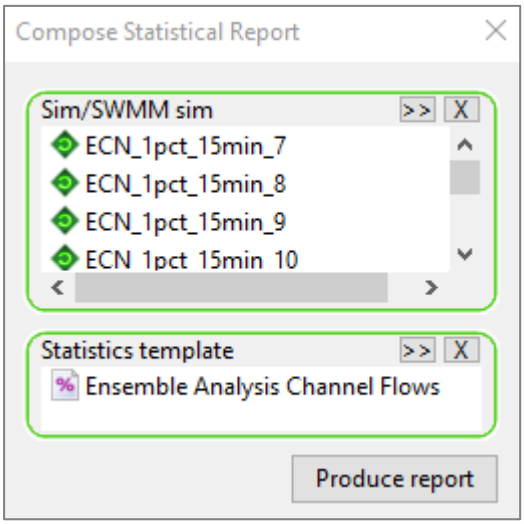
8. Clear any selections on the GeoPlan. Now, using the **Select** tool select the **Channel** link.
9. Right click again on the Model group and select **New InfoWorks > Statistics template**. Name this **Ensemble Analysis Channel Flows** and press **OK**.



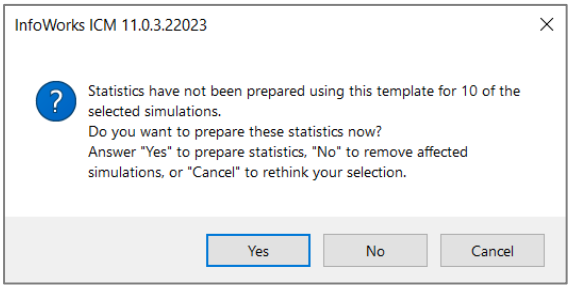
10. We will now define the rules by which we want to undertake the statistics on. Select **Link** from the Location type drown down and **US flow** for the attribute. This will be the flow to be conveyed through the development site from upstream.
11. Click on the **Add current selection** to add the channel location. Then click on **Add new tab** and select **DS flow** from the attribute menu and **Add the current selection**. This will be the peak pre-development site discharges.
12. Then click **Save** and close the dialog.



13. From the top toolbar go to **Results > Statistical reports...** Then populate the boxes with the simulation objects that we have run and the **Ensemble Analysis Channel Flows** Statistics template by dragging and dropping into the relevant windows.



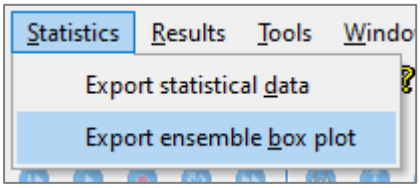
14. Select **Produce report** and then select **Yes** from the pop-up window. This is a warning that it is going to create the statistics as this can take a long time if you have a lot of simulations.



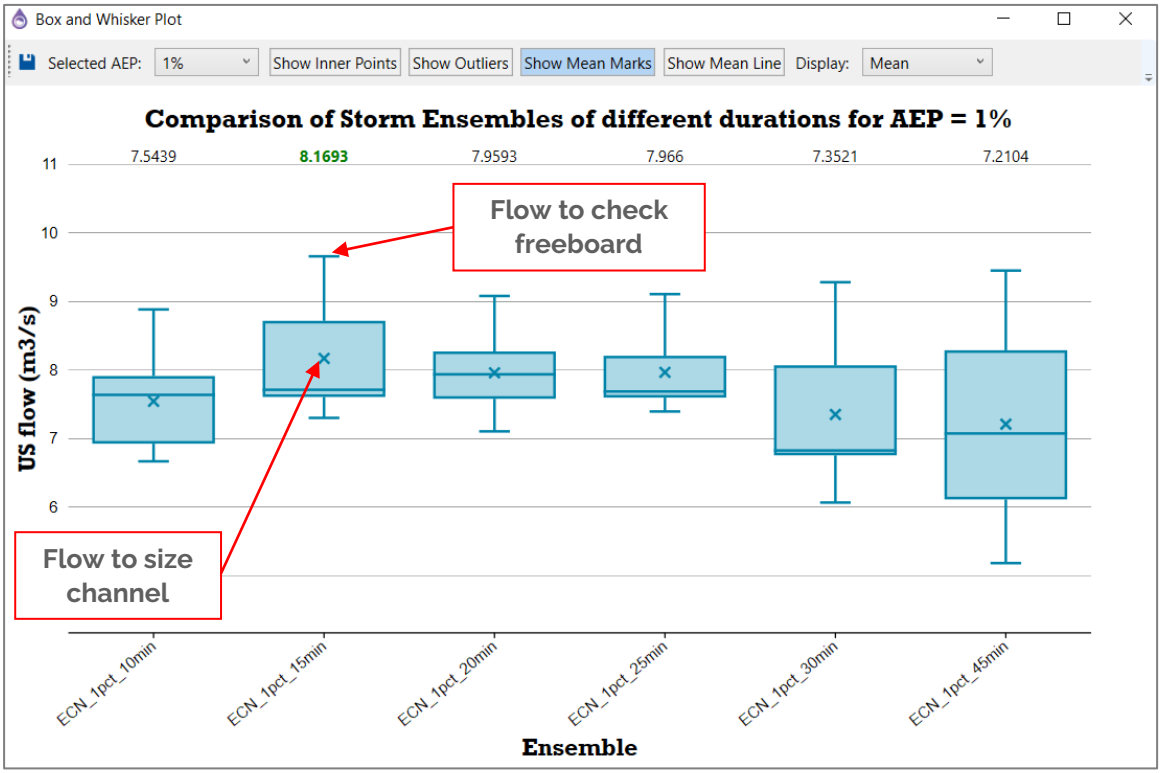
ICM will now generate the statistics report which will contain information for the **Channel – US.1**. The Statistical report will show the peak flow at the US and DS end of the channel for each storm pattern within the ensembles. The **Ensemble summary** tab analyses the Maximum Flow results for each sim in the ensemble and provides the statistical results for Mean, Median, Min and Max and the correlating sim name.

Attribute	Units	ID	Network	AEP	Ensemble	Ensemble mean	Mean sim	Ensemble median	Median sim	Ensemble min	Min sim	Ensemble max	Max sim
DS flow	m3/s	US.1	Urban Development (version 4)	1%	ECN_1pct_10min	5.76835	ECN_1pct_10min_2	5.78210	ECN_1pct_10min_7	5.55632	ECN_1pct_10min_10	6.22732	ECN_1pct_10min_9
DS flow	m3/s	US.1	Urban Development (version 4)	1%	ECN_1pct_15min	7.51783	ECN_1pct_15min_6	7.39961	ECN_1pct_15min_6	6.99364	ECN_1pct_15min_3	8.18005	ECN_1pct_15min_1
DS flow	m3/s	US.1	Urban Development (version 4)	1%	ECN_1pct_20min	7.56452	ECN_1pct_20min_8	7.62985	ECN_1pct_20min_1	6.72093	ECN_1pct_20min_4	8.38654	ECN_1pct_20min_10
DS flow	m3/s	US.1	Urban Development (version 4)	1%	ECN_1pct_25min	7.67884	ECN_1pct_25min_3	7.54120	ECN_1pct_25min_3	6.15274	ECN_1pct_25min_7	9.25880	ECN_1pct_25min_2
DS flow	m3/s	US.1	Urban Development (version 4)	1%	ECN_1pct_30min	6.96709	ECN_1pct_30min_9	6.87093	ECN_1pct_30min_8	5.94171	ECN_1pct_30min_4	8.45849	ECN_1pct_30min_3
DS flow	m3/s	US.1	Urban Development (version 4)	1%	ECN_1pct_45min	6.82994	ECN_1pct_45min_10	6.89939	ECN_1pct_45min_5	4.97792	ECN_1pct_45min_9	8.53641	ECN_1pct_45min_1
DS flow	m3/s	US.1	Urban Development (version 4)	10%	ECN_10pct_10min	2.75237	ECN_10pct_10min_9	2.75244	ECN_10pct_10min_6	2.64378	ECN_10pct_10min_4	2.83617	ECN_10pct_10min_10
DS flow	m3/s	US.1	Urban Development (version 4)	10%	ECN_10pct_15min	4.03257	ECN_10pct_15min_7	4.02368	ECN_10pct_15min_7	3.74610	ECN_10pct_15min_3	4.42229	ECN_10pct_15min_9
DS flow	m3/s	US.1	Urban Development (version 4)	10%	ECN_10pct_20min	4.66072	ECN_10pct_20min_3	4.80680	ECN_10pct_20min_5	3.95741	ECN_10pct_20min_10	5.22774	ECN_10pct_20min_8
DS flow	m3/s	US.1	Urban Development (version 4)	10%	ECN_10pct_25min	4.55762	ECN_10pct_25min_7	4.45683	ECN_10pct_25min_7	3.36361	ECN_10pct_25min_3	5.88474	ECN_10pct_25min_10
DS flow	m3/s	US.1	Urban Development (version 4)	10%	ECN_10pct_30min	4.63967	ECN_10pct_30min_6	4.64015	ECN_10pct_30min_8	3.50427	ECN_10pct_30min_1	5.49316	ECN_10pct_30min_7
DS flow	m3/s	US.1	Urban Development (version 4)	10%	ECN_10pct_45min	4.17578	ECN_10pct_45min_2	4.15446	ECN_10pct_45min_2	3.12340	ECN_10pct_45min_8	5.44799	ECN_10pct_45min_9
US flow	m3/s	US.1	Urban Development (version 4)	1%	ECN_1pct_10min	5.87145	ECN_1pct_10min_2	5.87264	ECN_1pct_10min_7	5.59367	ECN_1pct_10min_1	6.42426	ECN_1pct_10min_9
US flow	m3/s	US.1	Urban Development (version 4)	1%	ECN_1pct_15min	7.61451	ECN_1pct_15min_6	7.50525	ECN_1pct_15min_6	7.08844	ECN_1pct_15min_3	8.27152	ECN_1pct_15min_1
US flow	m3/s	US.1	Urban Development (version 4)	1%	ECN_1pct_20min	7.62336	ECN_1pct_20min_8	7.69649	ECN_1pct_20min_9	6.76162	ECN_1pct_20min_4	8.50777	ECN_1pct_20min_10
US flow	m3/s	US.1	Urban Development (version 4)	1%	ECN_1pct_25min	7.72879	ECN_1pct_25min_3	7.57494	ECN_1pct_25min_3	6.21830	ECN_1pct_25min_7	9.28571	ECN_1pct_25min_2
US flow	m3/s	US.1	Urban Development (version 4)	1%	ECN_1pct_30min	7.02189	ECN_1pct_30min_9	6.93207	ECN_1pct_30min_7	5.96861	ECN_1pct_30min_4	8.52107	ECN_1pct_30min_3
US flow	m3/s	US.1	Urban Development (version 4)	1%	ECN_1pct_45min	6.86164	ECN_1pct_45min_10	6.92759	ECN_1pct_45min_5	5.00647	ECN_1pct_45min_9	8.55607	ECN_1pct_45min_1
US flow	m3/s	US.1	Urban Development (version 4)	10%	ECN_10pct_10min	2.81602	ECN_10pct_10min_6	2.79710	ECN_10pct_10min_6	2.69945	ECN_10pct_10min_4	2.93705	ECN_10pct_10min_10
US flow	m3/s	US.1	Urban Development (version 4)	10%	ECN_10pct_15min	4.08832	ECN_10pct_15min_8	4.06068	ECN_10pct_15min_8	3.76109	ECN_10pct_15min_3	4.55156	ECN_10pct_15min_9
US flow	m3/s	US.1	Urban Development (version 4)	10%	ECN_10pct_20min	4.70137	ECN_10pct_20min_3	4.88371	ECN_10pct_20min_5	3.99701	ECN_10pct_20min_10	5.25401	ECN_10pct_20min_8
US flow	m3/s	US.1	Urban Development (version 4)	10%	ECN_10pct_25min	4.58941	ECN_10pct_25min_7	4.48399	ECN_10pct_25min_7	3.39019	ECN_10pct_25min_3	5.95803	ECN_10pct_25min_10
US flow	m3/s	US.1	Urban Development (version 4)	10%	ECN_10pct_30min	4.66495	ECN_10pct_30min_8	4.65844	ECN_10pct_30min_8	3.52044	ECN_10pct_30min_1	5.50460	ECN_10pct_30min_7
US flow	m3/s	US.1	Urban Development (version 4)	10%	ECN_10pct_45min	4.20122	ECN_10pct_45min_2	4.17430	ECN_10pct_45min_2	3.12677	ECN_10pct_45min_8	5.48927	ECN_10pct_45min_9

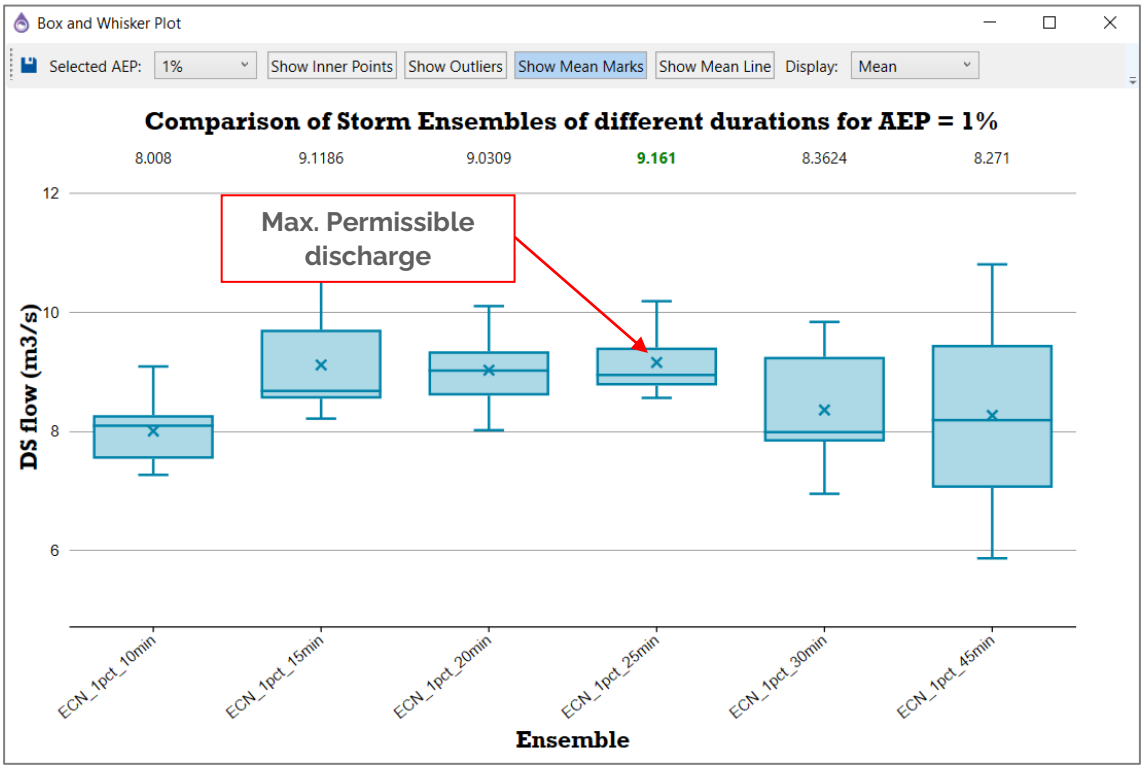
15. You can also export this information to excel or produce a box and whisker plot. With the curser set on an **US Flow** attribute row, navigate to the top toolbar and click to **Statistics > Export ensemble box plot**.



16. Select a location to save the box and whisker plot information and **Save** with the default name. Have a play with the available options and turn on the **Show Mean Marks** and display the **Mean** value at the top of the boxplot. Note the storms that give the Mean peak flows.



17. Repeat the process for the **DS flow** attribute.

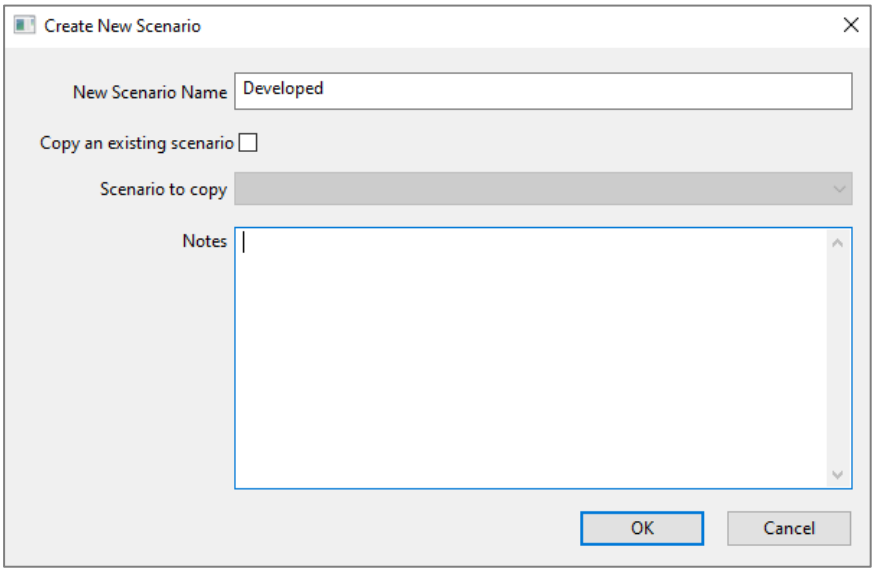


Workshop 9. Developed Case

We will now look at the hydrology of the development site and determine what size storage would be needed to maintain the current peak flows.

9A. Creating a scenario

1. Press **Create scenario** on the **Scenarios** toolbar and type **Developed** for the New Scenario Name.

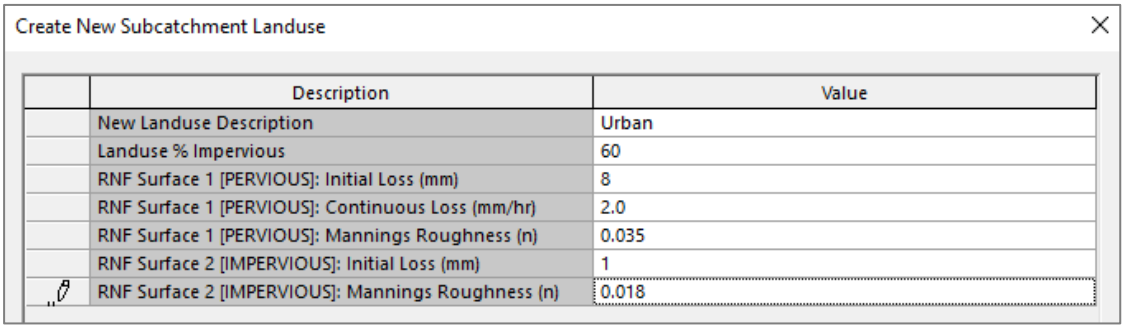


2. Ensure that you are working in the **Developed** scenario within the GeoPlan.

9B. Using Stored Queries

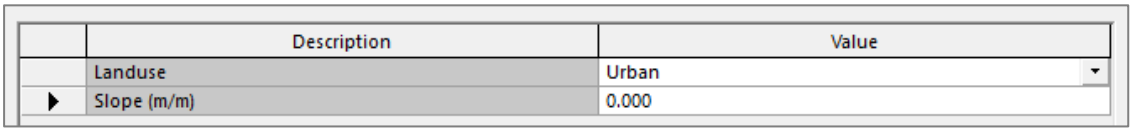
In Workshop 5 we created the Rural landuse and associated runoff surface manually. In this example we will use a stored query to help automatically populate new Landuse data for us.

3. From the transportable files, drag in the **Stored Query: Create New Landuse** into the GeoPlan. Populate the dialog for an **Urban** Landuse as below and then click **OK**.



4. Next, we will set the Development Subcatchment (**C_4**) to the **Urban** landuse. Select **Subcatchment C_4** and drag on the object **Stored Query: Apply Landuse to Subcatchment** into the Geoplan.

5. Choose **Urban** from the dropdown menu and leave the Subcatchment slope as **0**. The stored query is setup not to update the Subcatchment slope if it is set to 0.



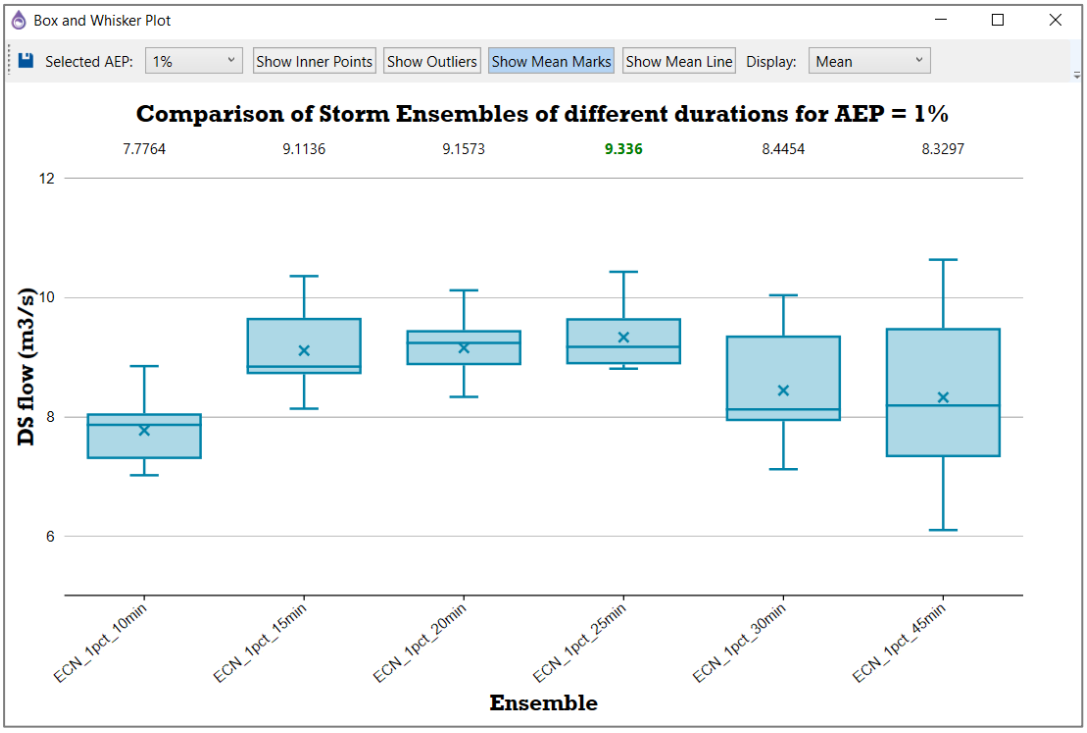
6. It is always a good idea to check your updates in the model after using stored queries. Open the properties dialog for **Subcatchment C_4** and check the Landuse is set to Urban. Then open the two new Runoff surfaces and check the losses and manning's roughness.

7. Once satisfied the model is correct, **validate and commit the changes**. Set the comments to **Workshop 9 - Added Developed scenario**.

9C. Running scenario simulations

8. Double click on the **Urban Development Run** object. Click on **update to latest**, change the name to **Urban Development Run (Post-dev)**, choose the **Developed** scenario then select **Run simulations**.

9. We will now repeat the **ARR statistics** for the downstream end of the channel. Note that the mean peak discharge has increased only by 0.2m³/s for the 1% AEP.

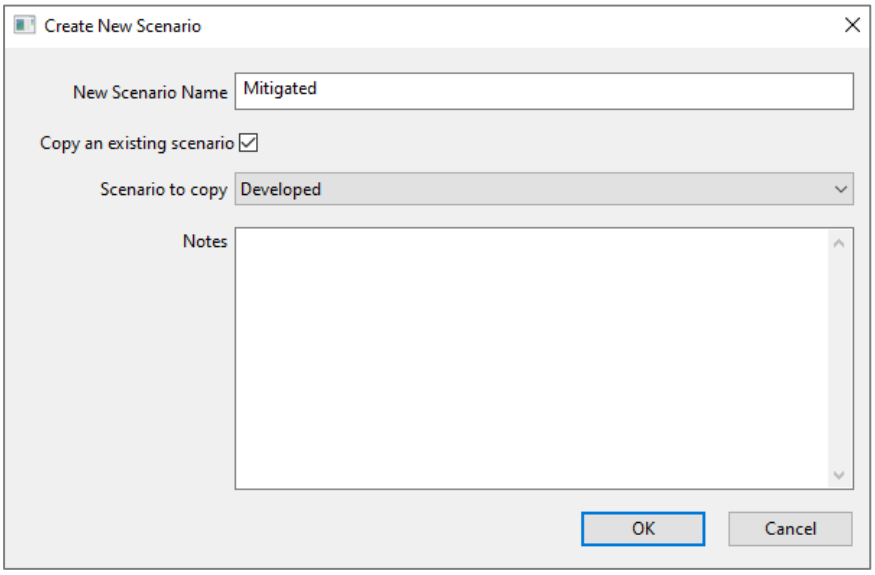

Box and Whisker Plot
Selected AEP:
1%
Show Inner Points
Show Outliers
Show Mean Marks
Show Mean Line
Display:
Mean
Comparison of Storm Ensembles of different durations for AEP = 1%
7.7764
9.1136
9.1573
9.336
8.4454
8.3297
12
10
8
6
DS flow (m3/s)
ECN_1pct_10min
ECN_1pct_15min
ECN_1pct_20min
ECN_1pct_25min
ECN_1pct_30min
ECN_1pct_45min
Ensemble

Workshop 10. Detention Basins

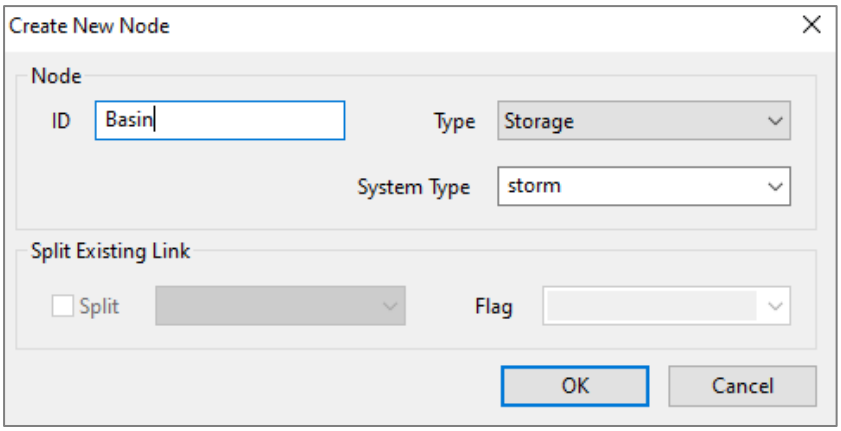
In this workshop we are going to look at adding storage to the model to reduce the site discharge post development.

10A. Adding a storage node

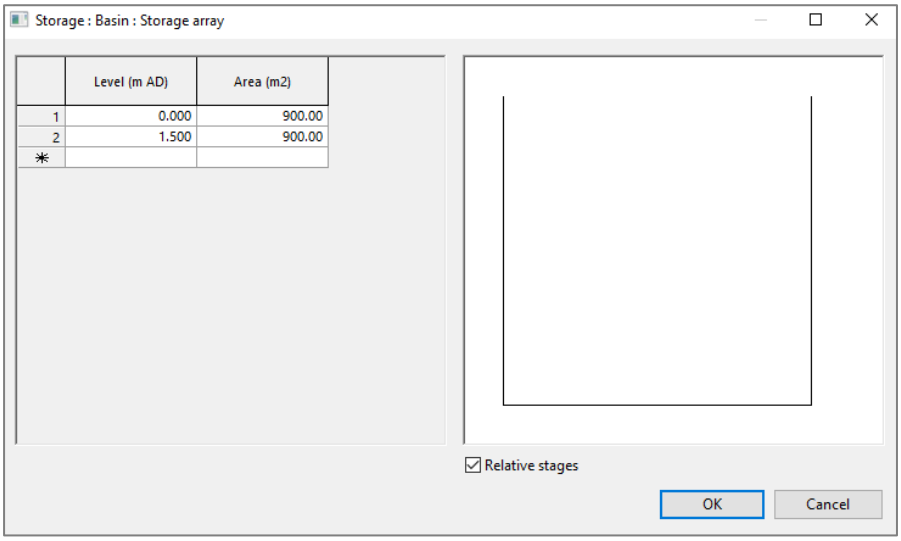
1. **Create a new scenario** with the name **Mitigated**. Check on the option to **Copy** an existing scenario and select the **Developed** scenario.



2. Select **Node** from the **New object** drop down menu and draw a new node near the development outlet. Set the ID to **Basin**, the Type to **Storage** and the System Type to **storm**.



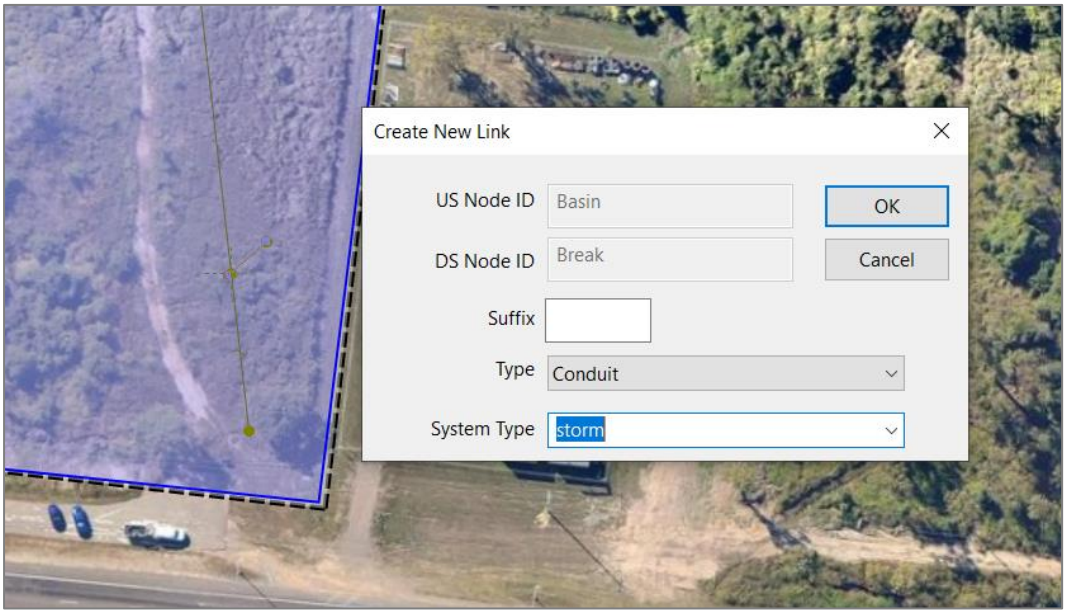
3. In the properties dialog, set the Ground level (m AD) to **12**. Check on **Relative stages** to use depth-based storage values and the click on the ellipses button to set the **Storage array**. Set the storage array Area (m2) to **900** and the Level (m AD) **1.5** high.



4. Use the **New object** tool again to insert a node into the Channel close to the DS node. Set both the **ID** and the **Type** to **Break**. Ensure the **Split** checkbox is selected. Click **OK**.

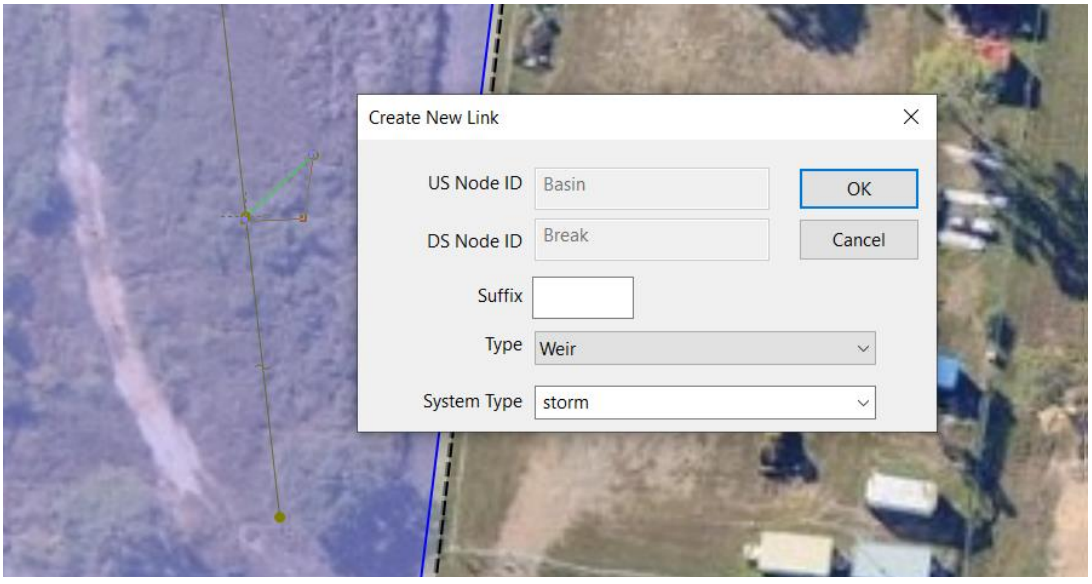
10B. Adding Storage Outlets

5. Select **Link** from the **New object** dropdown menu and create a new link between the Basin and the Break nodes. Set the **Type** to **Conduit** and the **System type** to **storm**.

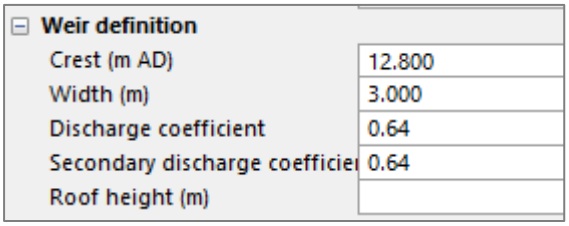


6. In the properties dialog, set the **Width (mm)** to **300**, the **US invert level (m AD**) to **12** and the **DS invert level (m AD)** to match the channel invert at the break point. Open the Channel properties to view the level.

7. Next, draw a second link connecting the same nodes. It is a good idea to insert a vertex halfway (using the Alt key) so the two links can be seen (not on top of each other). This time set the **Type** to **Weir** and the **System type** to **storm**.



8. In the properties dialog, set the Crest (m AD) to **12.8** (0.8m above the storage invert), the Width (m) to **3** and the Discharge coefficient to **0.64**.



NOTES:

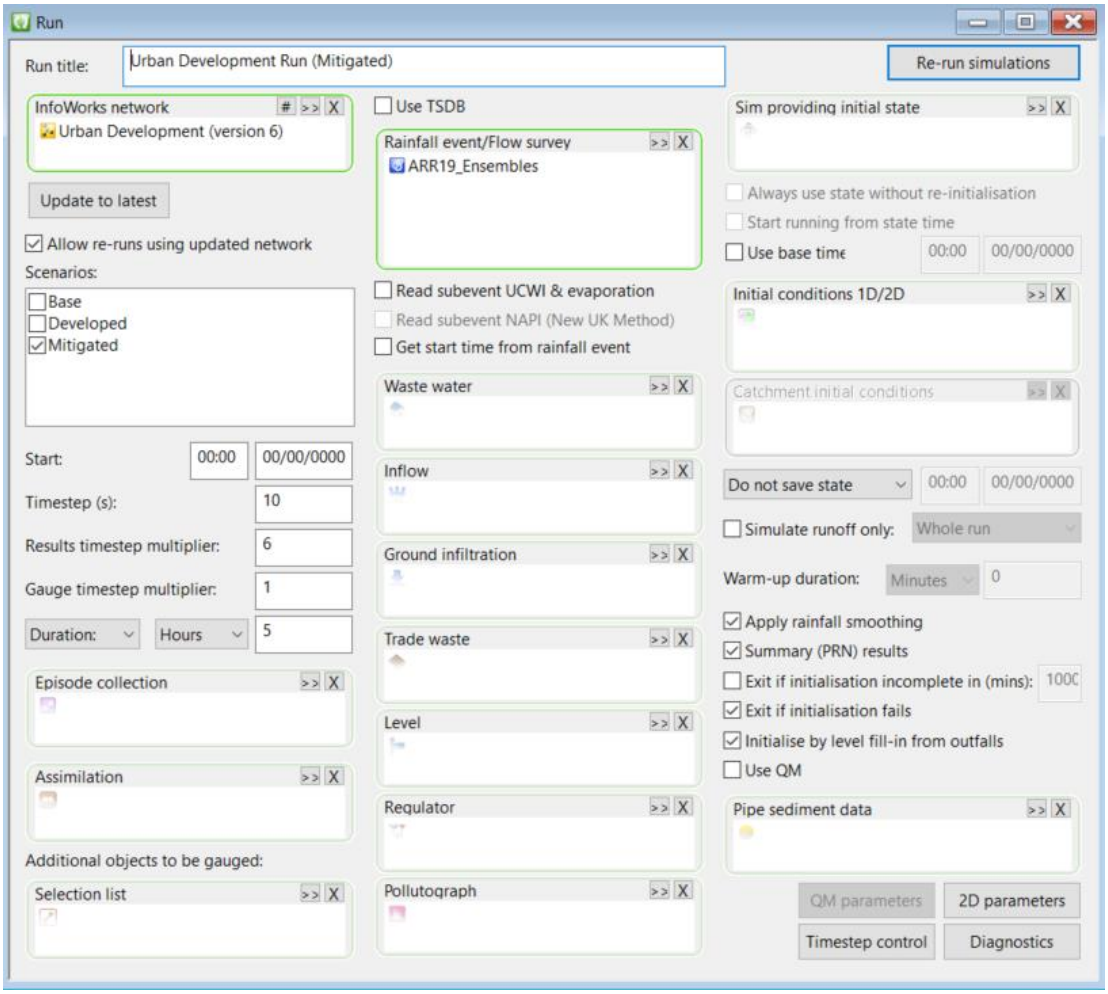
1. The Discharge coefficient in ICM is not the same as the commonly documented Weir Coefficient. To convert a standard Weir Coefficient to a Discharge coefficient for ICM, divide the value by the square root of gravity. For example, a Weir Coefficient of 2 would equal a Discharge coefficient in ICM of 0.64.
2. The Secondary discharge coefficient is used only if a Roof height (m) is specified and the flow becomes pressurised. At this stage, the orifice equation is used.

10C. Re-routing flows

9. Now the Basin is setup, we need to re-route the Subcatchment flows through the basin before getting to the outlet channel. Bring up the properties of **Subcatchment C_4** and change the Drains to back to **Node** and set the Node ID to **Basin.**
10. Validate and commit the network and add the comments **Workshop 10 - Added Mitigated scenario.**

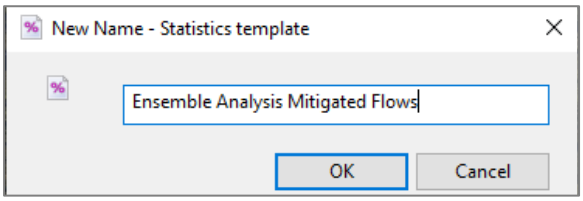
10D. Running scenario simulations

11. Double click on the **Urban Development Run (Post-dev)** object to open. Click on **update to latest**, change the title to **Urban Development Run (Mitigated)** and choose only the **Mitigated** scenario.
12. Keep all other fields the same and click **Run Simulations**.



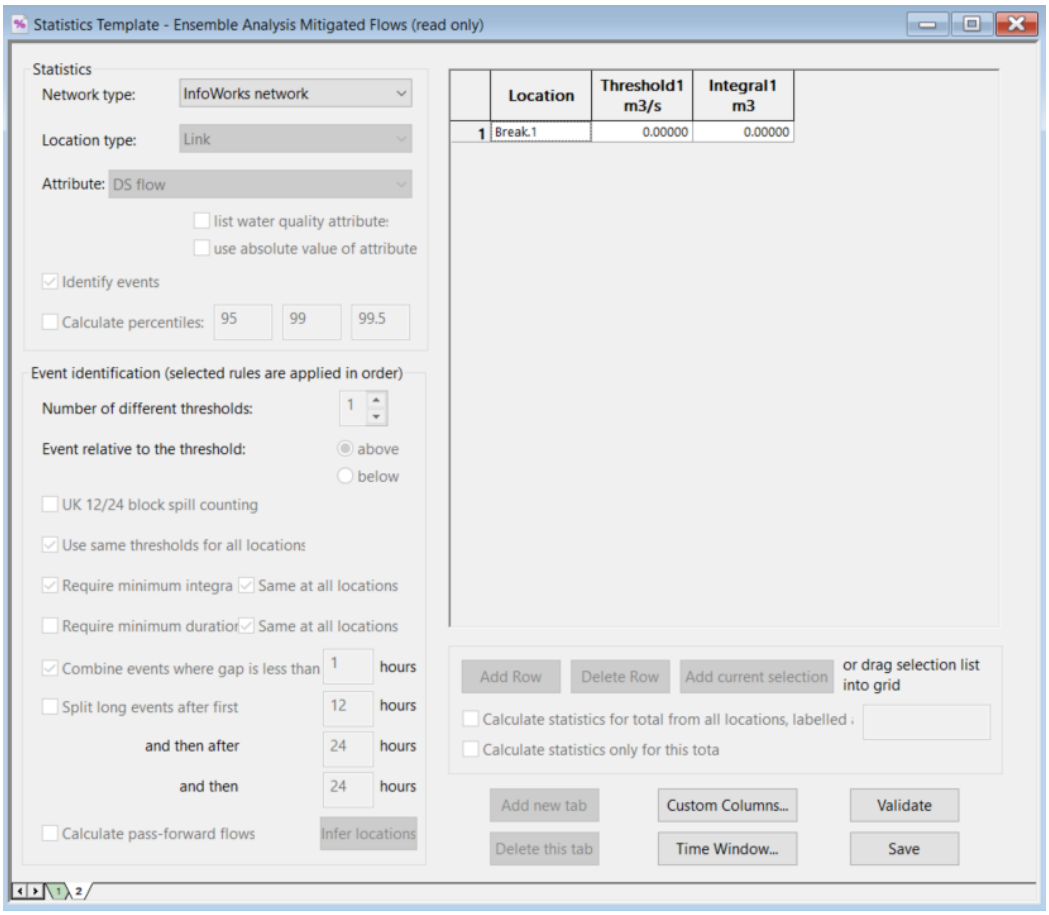
13. We will now repeat the **ARR statistics** for the downstream end of the channel. Note that we need to create a new statistics template now that we have reconfigured the network. Firstly, **select the channel** between the Break node and the DS outlet.

14. Right click again on the Model group and select **New InfoWorks > Statistics template**. Name this **Ensemble Analysis Mitigated Channel Flows** and press **OK**.

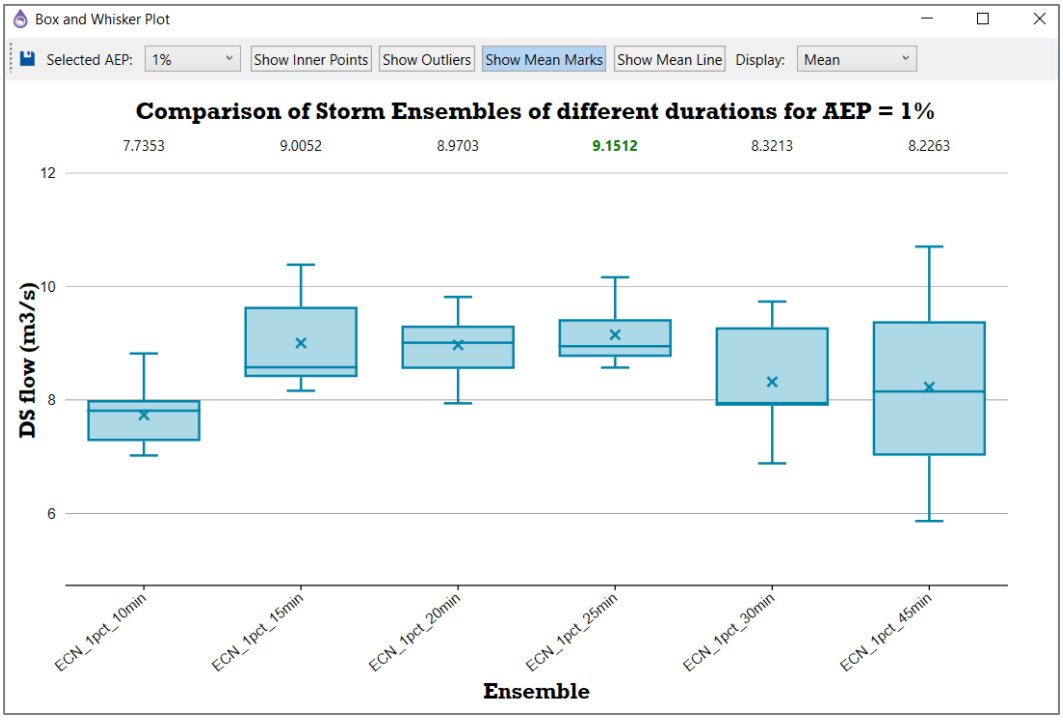


15. We will now define the rules by which we want to undertake the statistics on. Select **Link** from the Location type drown down and **DS flow** for the attribute.

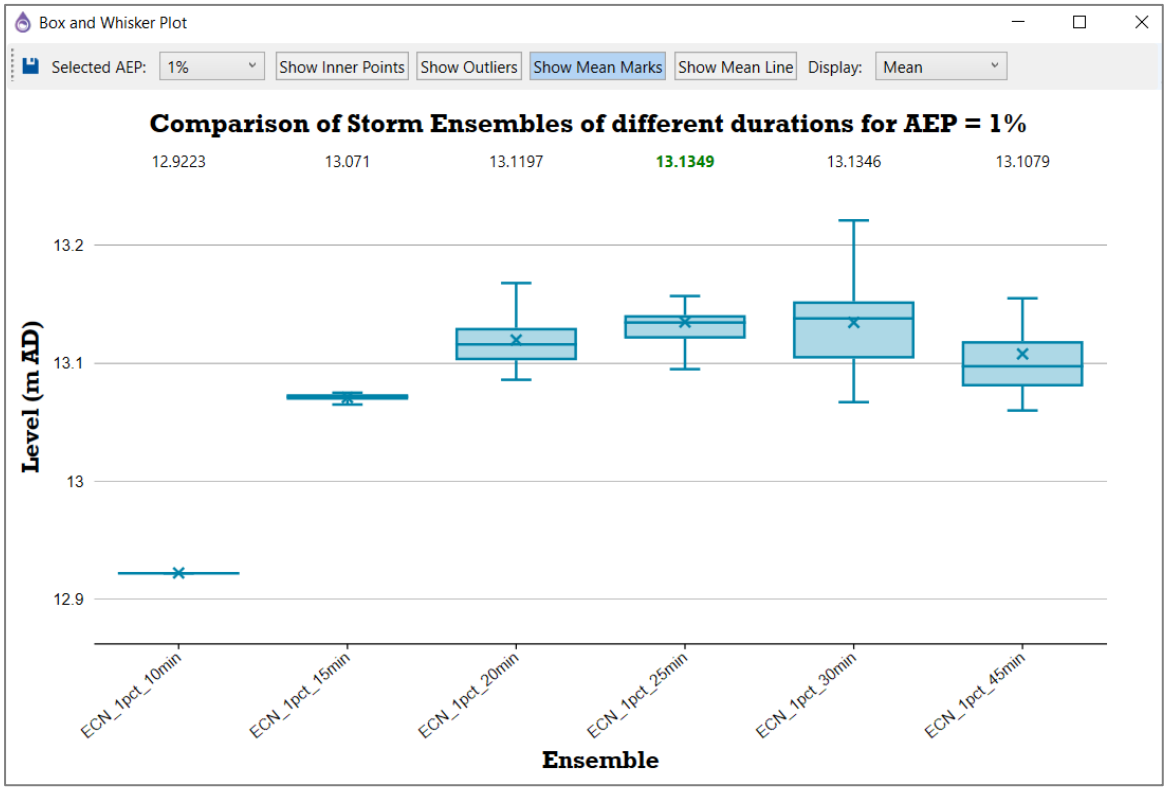
16. Click on the **Add current selection** to add the channel location.
17. Then click **Save** and close the dialog.



18. Go to **Results > Statistical reports…** and add the mitigated run results and the new statistical template. After producing the results from the dialog, you can add the box and whisker plot by going to **Statistics > Export box and whisker plot**.



19. We can do the same ensemble statistics for the depth in the basin. Firstly, select the **Basin** node in the GeoPlan.
20. Right click again on the Model group and select **New InfoWorks > Statistics template**. Name this **Ensemble Analysis Mitigated Basin Levels** and press **OK**.
21. We will now define the rules by which we want to undertake the statistics on. Select **Node** from the Location type drown down and **Level** for the attribute.
22. Click on the **Add current selection** to add the Basin node. Then click **Save** and close the dialog.
23. Go to **Results > Statistical reports…** and add the mitigated run results and the new statistical template. After producing the results from the dialog, you can add the box and whisker plot by going to **Statistics > Export box and whisker plot**.

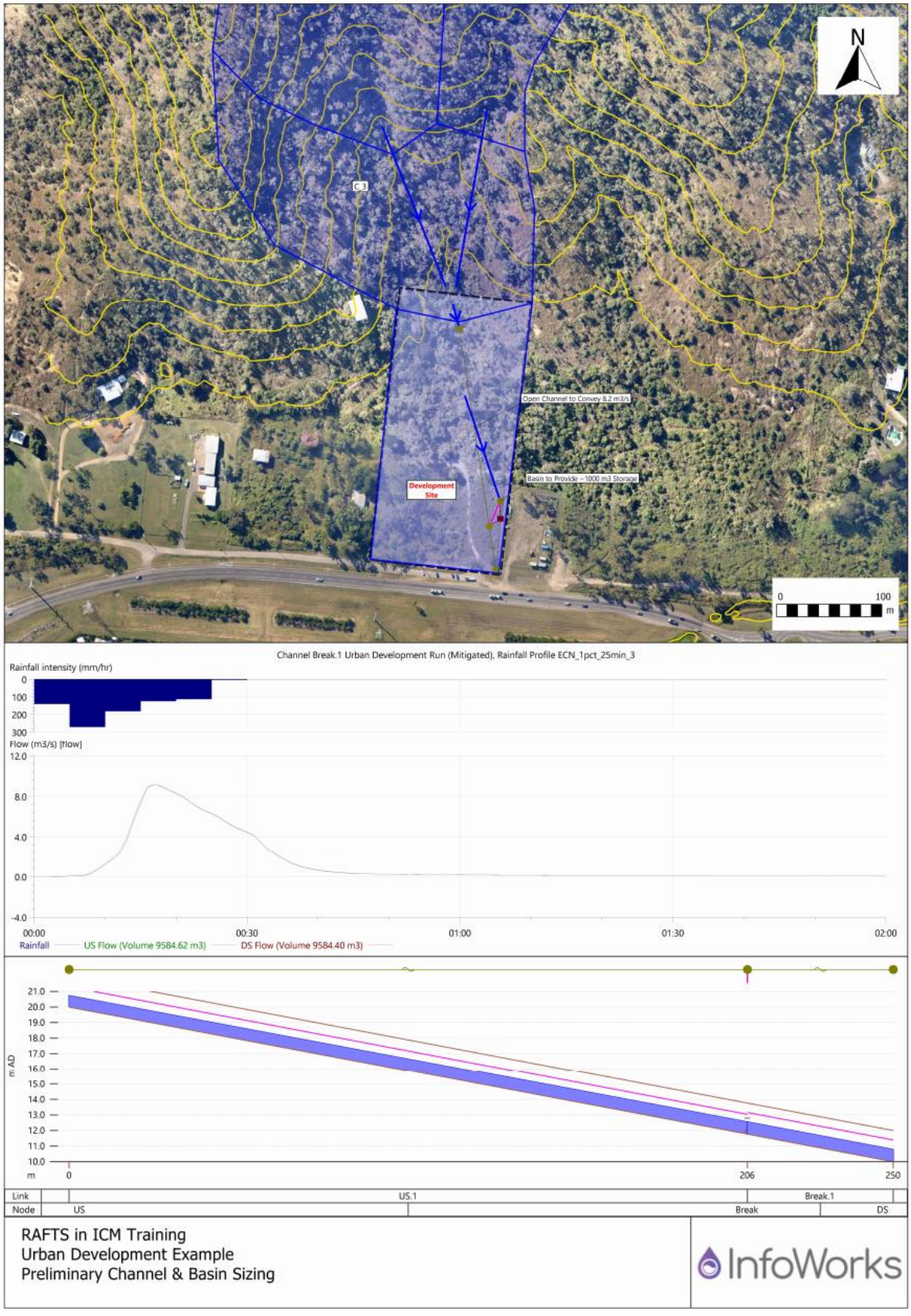

Box and Whisker Plot
Selected AEP:
1%
Show Inner Points
Show Outliers
Show Mean Marks
Show Mean Line
Display:
Mean
Comparison of Storm Ensembles of different durations for AEP = 1%
12.9223
13.071
13.1197
13.1349
13.1346
13.1079
Level (m AD)
13.2
13.1
13
12.9
ECN_1pct_10min
ECN_1pct_15min
ECN_1pct_20min
ECN_1pct_25min
ECN_1pct_30min
ECN_1pct_45min
Ensemble

Workshop 11. Displaying Results

In this workshop we will look at how we can layout our model results easily on a single plan. This is a good way to pass on the information we have gathered for the development site to the Civil Designer.

11A. Print Layout

1. In Workshop 8 we determined the flowrate needed to size the conveyance channel through the development site. Click on the **Custom label** button  and add a label to Channel US.1.
2. Double-click on the new label, click on the **Free text** radial button and type in **Open Channel to Convey 8.2 m3/s.**
3. Go to the **Formatting** tab and set the Font Size to **9** and change the Callout Line Width to **2**.
4. Then do the same for the **Basin** node, except add the text **Basin to provide ~1000m3 storage**.
5. We will now add a print layout. First position the network how you want it to be viewed in the print layout then go to **GeoPlan > Spatial Bookmarks > Add from View…** Set the name to **Plot View.**
6. Create a new print layout by going to **File > Print Layout**.
7. Drag in the **GeoPlan** from the **Data views** options and fit to desired size. Then double click on the View and go to **GeoPlan Scale** tab. Set it to **Fit Geospatial Bookmark** and choose **Plot View** from the dropdown list. Click **OK**.
8. Add a **North** point, a **Scale** bar some **Text** and an **Image**. You can also add any graphs, long sections or grid results that are open in the working window.
9. Then press **Save As…** icon in the toolbar and save to the working model group with the name **Plot Layout: Mitigated**. From here you can print to either PDF or a printer.
10. Close all tabs in the GeoPlan once you are finished.

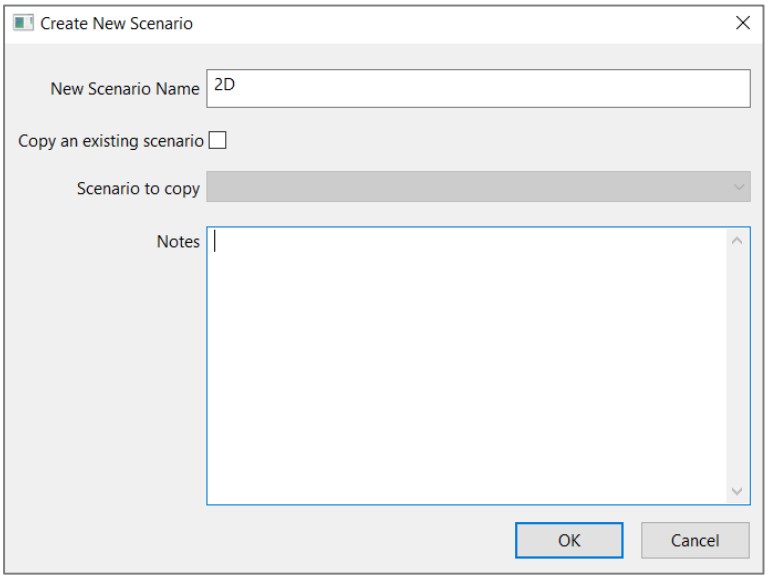

N
C 3
Open Channel to Convey 8.2 m3/s
Development Site
Basin to Provide ~1000 m3 Storage
0
100
m
Channel Break.1 Urban Development Run (Mitigated), Rainfall Profile ECN_1pct_25min_3
Rainfall intensity (mm/hr)
0
100
200
300
Flow (m3/s) [Flow]
12.0
8.0
4.0
0.0
-4.0
00:00
00:30
01:00
01:30
02:00
Rainfall
US Flow (Volume 9584.62 m3)
DS Flow (Volume 9584.40 m3)
m AD
21.0
20.0
19.0
18.0
17.0
16.0
15.0
14.0
13.0
12.0
11.0
10.0
m
0
206
250
Link
US.1
Break.1
Node
US
Break
DS
RAFTS in ICM Training
Urban Development Example
Preliminary Channel & Basin Sizing
InfoWorks

Workshop 12. Adding the 2D Zone

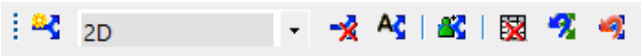
Finally, we will look to create a new scenario and add a 2D zone so that we can model the subcatchment hydrographs as overland flow in the 2D engine. We will drain the subcatchments to 2D point sources, import a mesh level zone to act as channel for flow diversion, re-run the model and compare the hydrographs leading to the development site.

12A. Creating a scenario

1. Open the **Urban Development** network to the GeoPlan.
2. Press **Create scenario** on the **Scenarios** toolbar and type **2D** for the New Scenario Name.



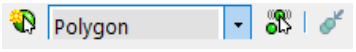
3. Ensure that you are working in the **2D** scenario within the GeoPlan.



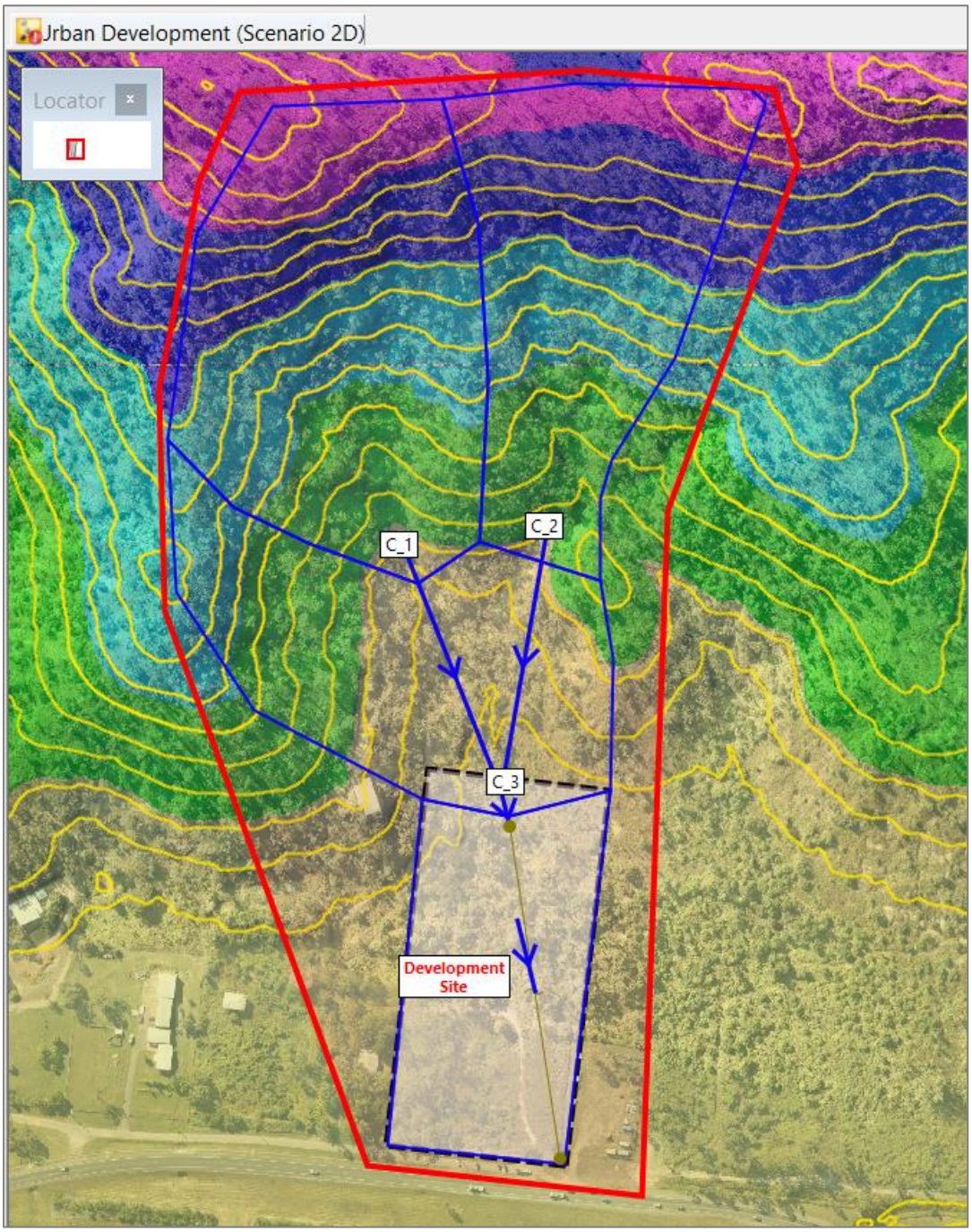
12B. Setting up the 2D zone

4. Right-click on the tutorial model group and select **Import InfoWorks > Ground model grid > from ground model grid files...** Navigate to the Part 2 folder within the training files and select both **Ground1.asc** and **Ground2.asc**. Press **Open**.
5. From the Ground model grid dialog, name the Grid as **DEM_1m**, ensure the Units are set to **Metres** and check on the **Floating point or mixed** radio button. Press **OK**.

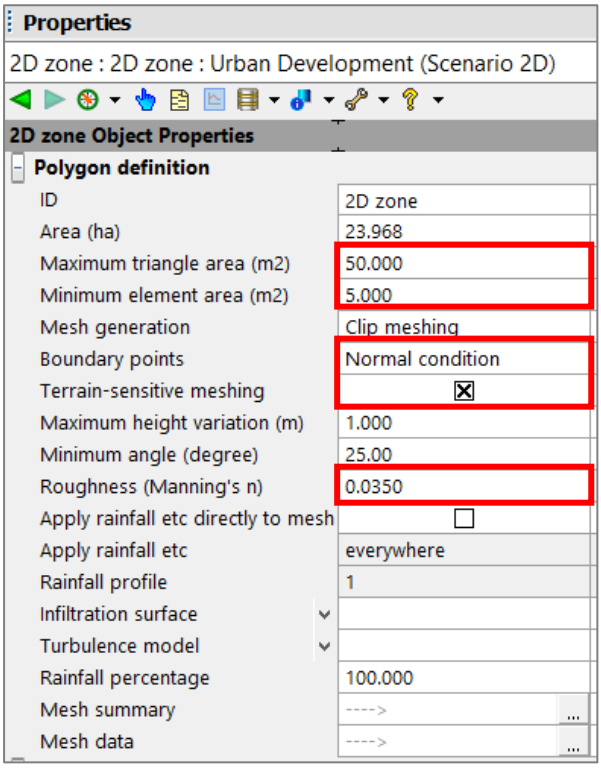
6. Drag the newly created Ground Model onto the GeoPlan to open. Also drag on the **2D Theme** object from the template files.

7. Now we will draw the 2D zone. Select **Polygon** 

from the new object dropdown and digitise a 2D zone around the existing subcatchments and development site. Set the Type and ID to **2D zone**.



8. The 2D zone Object Properties should now be open. Set the highlighted 2D zone fields as per the figure below.



9. The next step is to create the 2D mesh elements. With the 2D zone selected, go to **Model > Meshing > Mesh 2D zones…** Drag in **DEM_1m** to the Ground Model field and Run the mesh generation on your machine.

10. Once the meshing process is complete (progress can be viewed in the Job Control window) go to **Model > Meshing > Load mesh job results…** and with the completed mesh job highlighted select **Load Mesh**. You will now see the completed mesh within the 2D zone.

11. A summary of the mesh can be viewed. Open the properties for the 2D zone and select the ellipsis in the **Mesh summary** field. Take note of the number of elements and triangles created.

12C. Draining Subcatchments to the 2D zone

2D Point Source objects can be used to define the location of a flow-time boundary point. Inflow associated with the point source discharges to the 2D mesh element in which the point is located. It should be ensured that the mesh element is an appropriate size to cope with the inflow.

12. Select **Point** 

from the new object dropdown and digitise four **2d point source** objects close to the subcatchment outlets and name them as per the corresponding subcatchment e.g., **C_2**.

13. We now need to drain our subcatchments to the 2D point sources so that the hydrographs can be applied to the 2D mesh. Open the properties for subcatchment **C_1** by double clicking on the object. Set the Drains to field to **2D point source** and the 2D point ID to **C_1**.

14. From the transportable template files drag the **Stored Query: Drain Subcatch to 2D** onto the GeoPlan to assign the remaining subcatchments to their respective 2D point IDs.

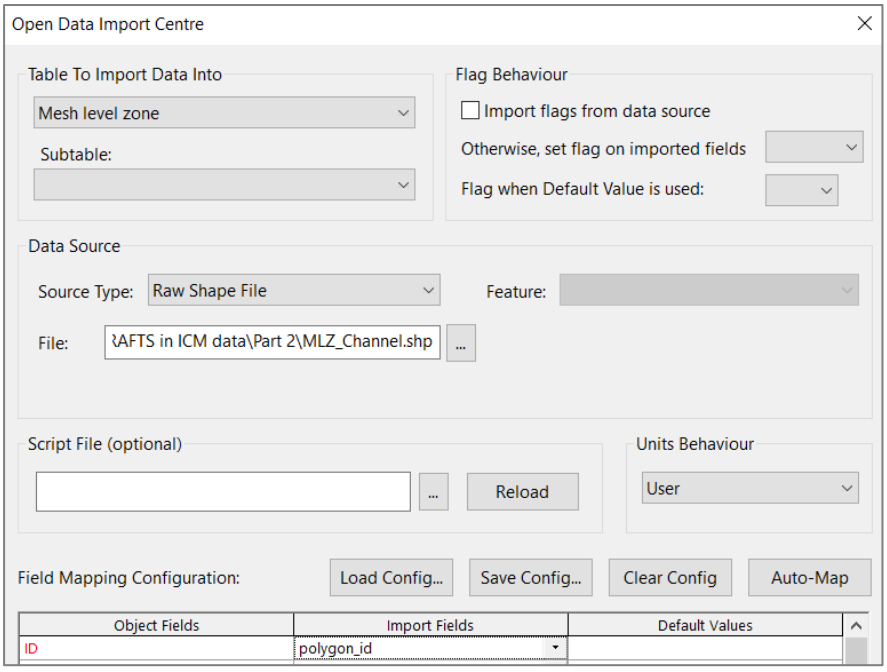
15. As this is a 2D scenario we will need to remove the 1D network within the Development Site. This time from the transportable template files, drag the **Stored Query: Remove 1D Objects** onto the GeoPlan.

16. Validate the network and Commit the changes.

12D. Meshing techniques

We will be using a Mesh Level zone to define a diversion channel through the development site and build a Network result line into the mesh for inspection of flow results later.

17. Go to **Network > Import > Open Data Import Centre…** and select **Mesh Level Zone** as the Table, **Raw Shape File** as the Source and the **MLZ_Channel.shp** file from the Part 2 folder. From the Import Fields choose **polygon_id** for ID and click **Import**.



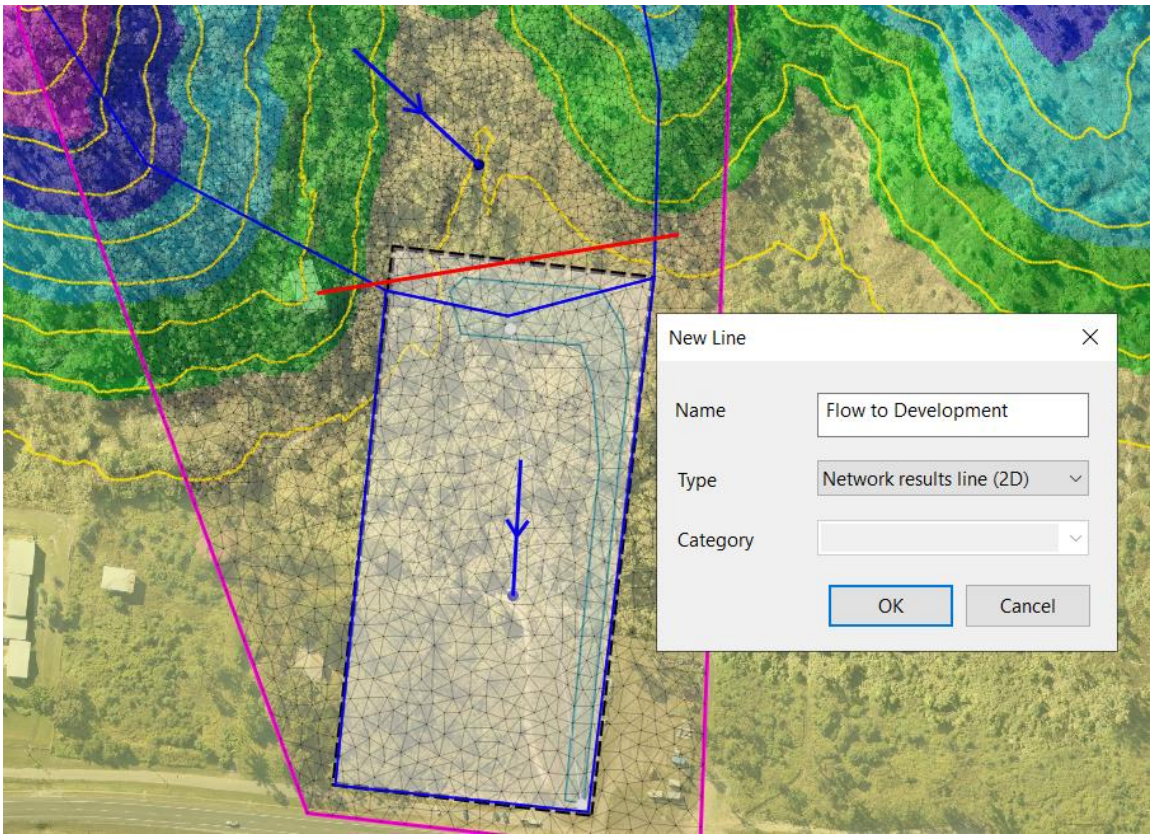
18. There should be 1 new object imported. **Close** the Open Data Import Centre.
19. Open the object properties for the newly imported Mesh level zone and select the ellipsis within the **Vertices** field.
20. Click on the **Vertex elevation type** column header to highlight the column. Right-click on the one of the highlighted rows and select **Set new value(s) for cell(s).** Choose **Set** from the dropdown and press **OK**.
21. Fill out the **Elevation (m AD)** for each vertex as per the figure below. Selecting a row will highlight which vertex you are editing in the GeoPlan.

Mesh level zone : Channel : Vertices

	X coordinate (m)	Y coordinate (m)	Vertex elevation type	Elevation (m AD)	Elevation adjustment (m)
▶	472371.699	7867708.531	Set	17.800	0.000
	472437.542	7867703.115	Set	17.200	0.000
	472448.373	7867685.728	Set	16.600	0.000
	472450.939	7867660.360	Set	15.900	0.000
	472429.561	7867456.846	Set	11.500	0.000
	472420.440	7867457.701	Set	11.500	0.000
	472437.257	7867619.030	Set	16.100	0.000
	472433.837	7867657.510	Set	16.600	0.000
	472428.136	7867676.037	Set	17.600	0.000
	472370.844	7867681.738	Set	17.700	0.000
	472365.428	7867702.260	Set	17.500	0.000
✳					

22. Now select **Line** 

from the new object dropdown and digitise a line through the downstream end of Subcatchment C_3. Make sure that the line is drawn left-to-right looking downstream as this will ensure that the flow through the line is positive when we view the simulation results.
23. Set the Type to **Network results line (2D)** and the Name as **Flow to Development**.



24. Re-mesh the 2D zone to incorporate the diversion channel and network result line.

NOTE: Opening a New 3D network window can help to visualise the 2D mesh changes imposed by Mesh zones, Mesh level zones, etc.

25. Validate the Network and Commit the changes.

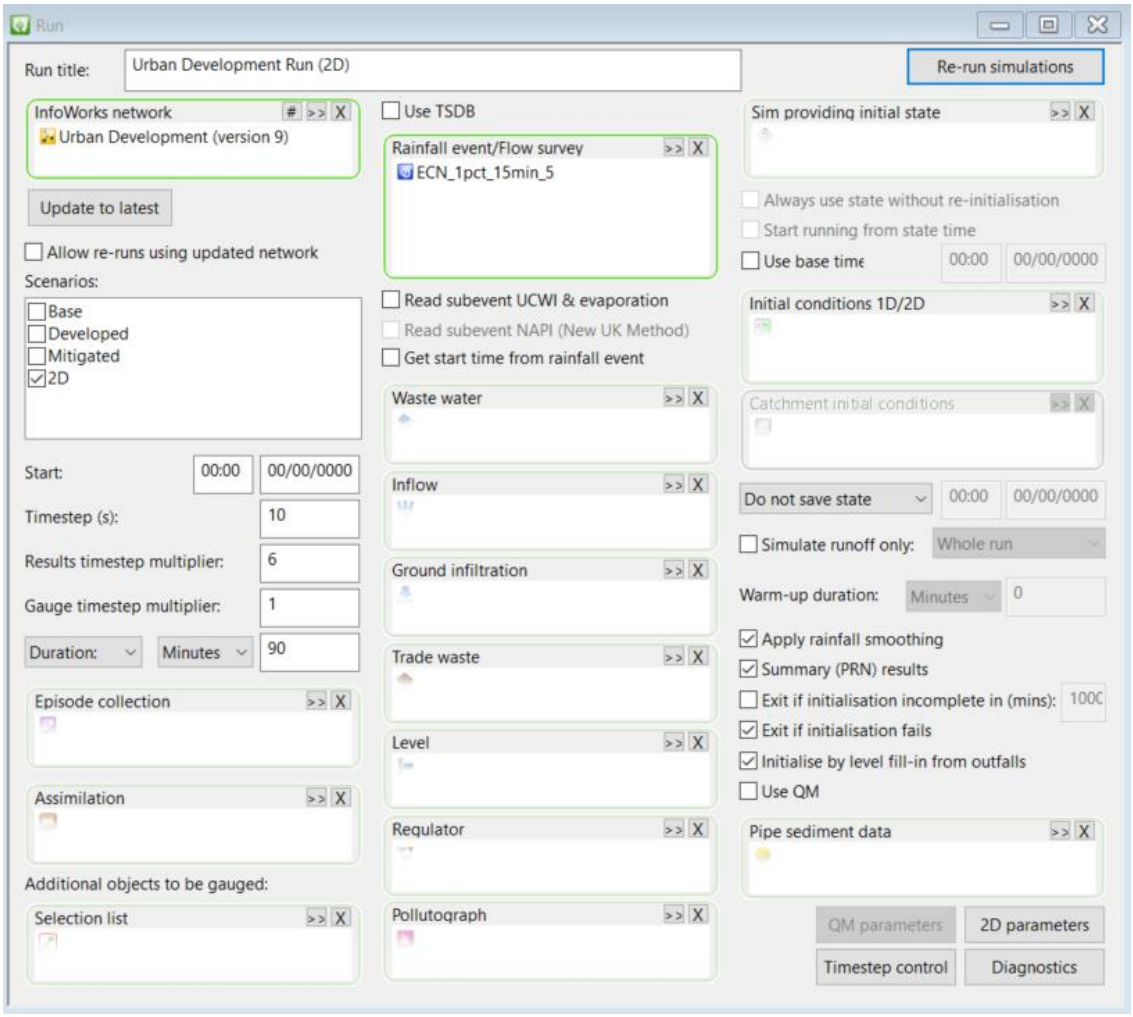
12E. Running the 2D zone scenario

Lastly, we will run the critical storm for flow to the development site from our 1D model, inspect the 2D results and compare the incoming flow hydrographs.

26. Right-click on the **ARR19_Ensembles** rainfall object, select **Copy** then Right-click on the tutorial model group and **Paste**. This will create a copy of the rainfall object that is no longer Read Only (R/O).

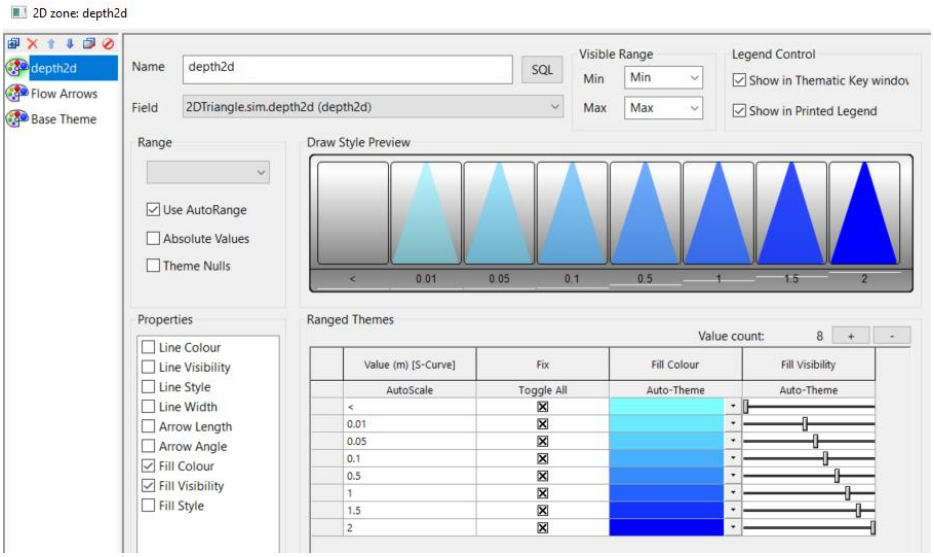
27. Rename the copy **ECN_1pct_15min_5** and open. Select the **Disable all** button and then scroll down and check on the **1% 15min Storm 5**. Click **OK** to save.

28. Create a new Run object by right-clicking on the model group and selecting **New InfoWorks > Run.** Set up the **Run** object as per the figure below and run the simulation.

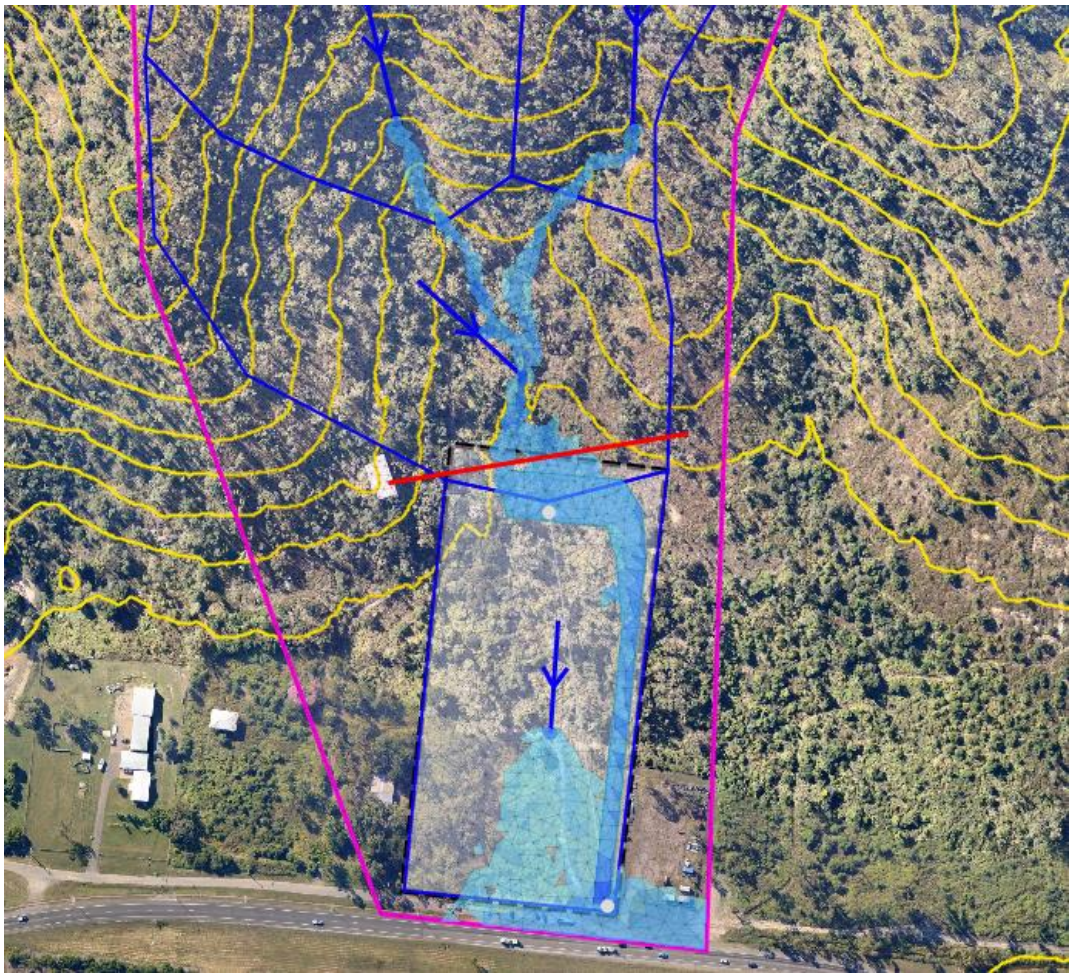


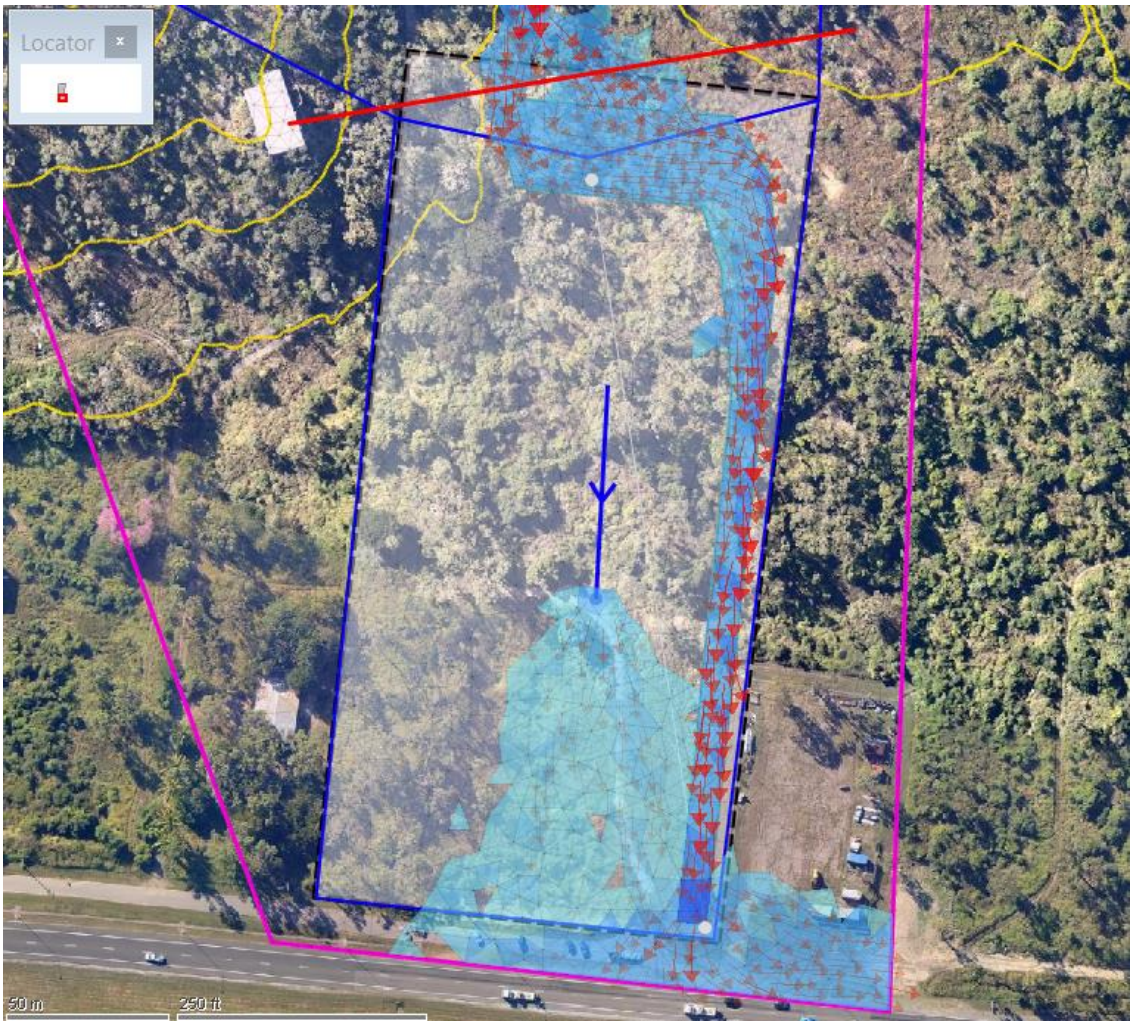
NOTE: 2D calculations can be accelerated significantly by accessing any available GPUs. To use a GPU card for the 2D calculations click on **2D Parameters** in the Run object window and navigate to the **GPU** tab. If 'Always' is selected and no GPU is present the run will fail.

29. Once the simulation is complete drag the result onto the GeoPlan and enable the 2D zone depth and velocity themes set in the **2D Theme** object via the network Properties and Themes if they are not already turned on.

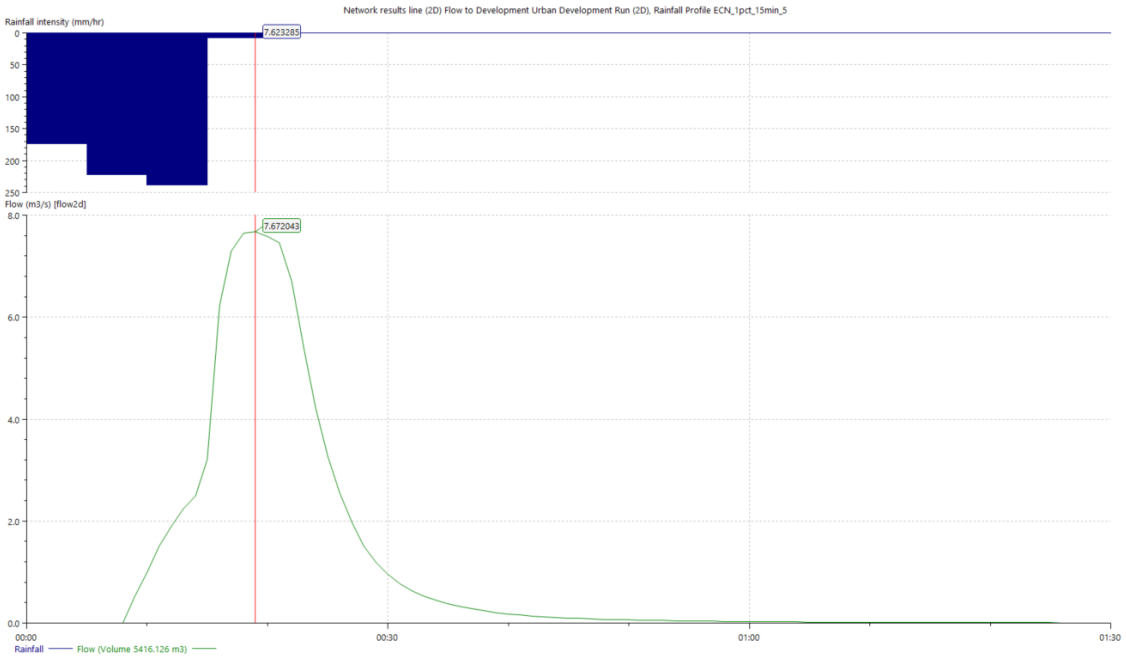


30. Use the Replay toolbar  to play through the simulation results and view the maxima.

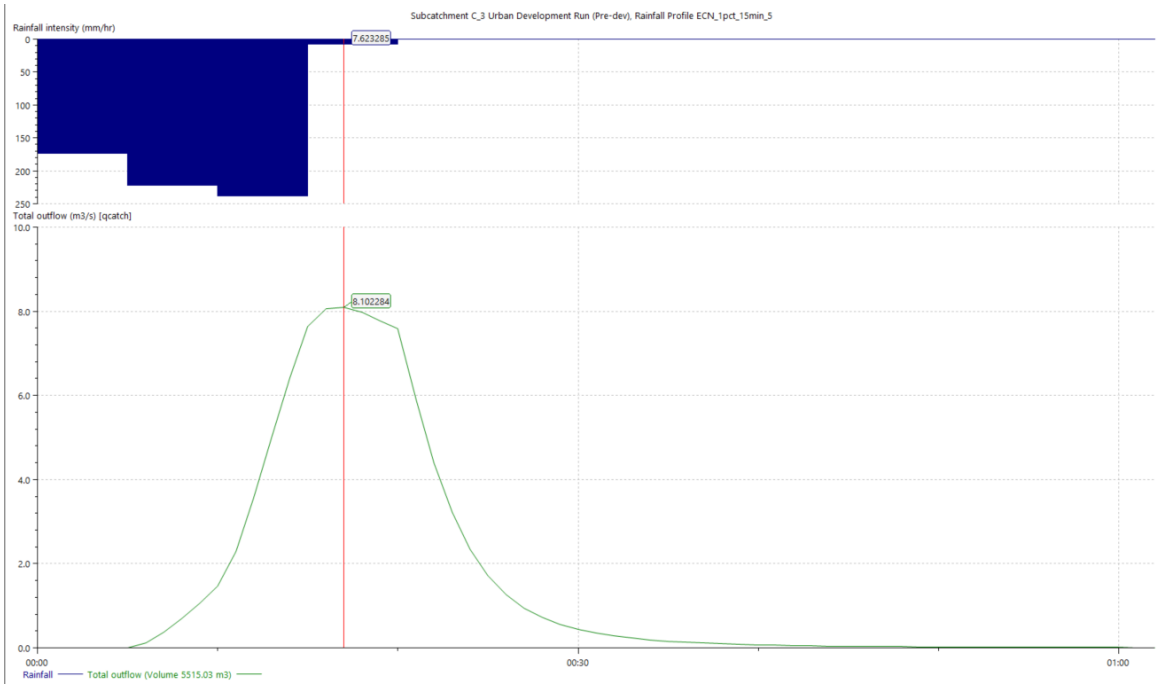




31. Next, use the Graph tool  to select the 2D Network result line and select the **Flow (m3/s)** attribute. Here we can see a peak flow of around 7.7m3/s occurring at a time of 19mins from the beginning of the storm.



32. Compare the result with the same storm simulation completed in the **Urban Development (Pre-dev)** Run. Graph the **Total outflow (m3/s)** attribute for Subcatchment **C_3**. The estimated peak flow is around 8.1 m3/s occurring 17mins after the beginning of the storm.



33. What are some factors that could cause the 2D hydrograph to estimate slightly lower and delayed flows?