Applying the 2D module to Collection Systems

Technical Review
Using InfoWorks 2D for Collection Systems Analysis

InfoWorks 2D is an optional extra, which is available as a module for InfoWorks CS. When activated, it is fully integrated into the application; it does not run as a separate program.

Before you start, you will need a suitable InfoWorks CS model that predicts localised flooding, good quality GIS data of the catchment (to provide houses and urban pathways), and good quality DTM/DEM/BEM data to represent natural depressions and embankments on the ground.

The quality of the GIS / Mapping data is critical to any InfoWorks 2D run. It is needed to ensure urban pathways can be correctly defined in the 2D Mesh. Without buildings, roads and footpaths, the above ground flood routing patterns are of significantly less value. The 'picture' within the GeoPlan is also less meaningful, as you cannot easily identify where properties are at risk of flooding, or the likely flood depths at property boundaries.

Modelling 2D flows is much more computationally intensive than modelling 1D flows, therefore mesh generation should be limited to areas of particular interest. During a 2D simulation depth, velocity and direction of flow will be calculated for each mesh element. Flooding from the collection system to the mesh is modelled at 2D node locations where surcharging of the manhole supplies floodwater to the 2D mesh element in which it resides.
InfoWorks uses a one dimensional simulation engine to calculate all flows below ground level. Once manholes start to flood, the floodwater flows across a two dimensional zone representing the surface of the catchment.

To create an accurate model it is necessary to include features such as buildings, walls, areas of different surface roughness and many of the other features that may exist in an urban environment. The software must also be able to fully represent the hydraulic interactions between the above ground surface and the below ground system as well.

The 2D zone, alternatively described as a mesh, consists of a series of contiguous triangles, whose shared boundaries cover the catchment surface. The height of each point on each triangle is derived from a ground model. This data is often referred to as LiDAR data and is typically supplied in either GRID or TIN based formats.

It is significant that InfoWorks uses an irregular mesh, where the size and orientation of the triangles can be varied. This is much more efficient than using a grid, where the layout is fixed. An irregular mesh allows for much more detail and complexity in locations where it is needed. Each 2D zone will typically include the following elements:

- **Voids** - A polygon into and through which no flow is allowed. These are typically used to represent buildings.
- **Walls** - Lines through which no flow can pass.
- **Break lines** - Lines along which the triangle boundaries must be created. These can be used to define the top of a ridge or the low point in a channel.
- **Mesh zones** - Polygons within the 2D zones where the triangles have different maximum triangle area. They are most commonly used to define smaller triangles in areas of greater complexity.
- **Roughness zones** - Polygons where a Manning’s roughness that differs from the default can be specified.

Each object can be either created within InfoWorks, or created in a GIS and then imported using the Open Data Import Centre.
InfoWorks CS/2D contains many features that will help you work quickly and accurately. The remainder of this document outlines the key aspects included in the product.

**Creating a 2D Zone**

Polygons are used to define areas of particular interest for which detailed 2D analysis is required. A 2D mesh of triangular elements can be generated in one or more 2D zones within the overall network, which is then used to model 2D flows. The 2D mesh in InfoWorks is generated using *Shewchuk Triangle* meshing technology. Heights at the vertices of the generated mesh elements are calculated by interpolation from a specified Ground Model.

In order for meshing to be carried out, a bounding *2D Simulation Polygon* is defined in the InfoWorks model. Voids (regions that will not be meshed), break lines and areas of varying roughness and mesh resolution may also be defined. All of these objects can be imported via the Open Data Import Centre or digitised directly on the GeoPlan View.

**Mesh Polygons**

Mesh Polygon objects are used as part of the mesh generation process carried out when undertaking 2D modelling. They can be used to divide a 2D Simulation Polygon into regions of different mesh resolution or to define zones in which ground level modification is required.

Mesh Polygons can be imported via the Open Data Import Centre or generated via intersection. Polygons can also be digitised directly on the GeoPlan View.
**Voids**

Voids are the regions within the 2D Simulation Polygon that will not be meshed e.g. buildings. Polygons digitised on the GeoPlan View can be used to represent void areas. The Polygon Category is used to identify which polygons represent voids when generating the 2D mesh.

Alternatively, if polygons representing voids are stored in GIS format, the GIS file or layer containing the data can be selected for use at the time of generating the 2D mesh.

**Walls**

Walls can be used to represent one-dimensional features such as brick walls. These objects allow the porosity and height of the feature to be specified, thus allowing the modelling of semi-porous structures, or walls with a set height, over which water can flow. Parameters can also be set for simulating collapse of the wall when the results of 2D mesh elements adjacent to the wall meet defined criteria. Wall lines are always enforced as mesh element edges during mesh generation.

If the wall data is in GIS format, the GIS file or layer containing the data can be selected for use at the time of generating the 2D mesh.

General Line objects can also be used to define walls. The Line Category assigned to the line is used to identify which lines are to be used as walls. General Lines representing walls will be treated as impermeable infinitely high barriers during the 2D Simulation.

**Porous Walls / Polygons**

Porous Walls and Porous Polygons have a line type that represent walls with a specified porosity and height, which is taken into account during the 2D simulation process. Where a Porous Wall is collinear with the boundary of a 2D Simulation Polygon, the Porous Wall will override the boundary condition of the 2D Simulation Polygon.
Collapsing Walls

It is possible to set parameters for the Porous Wall object which will result in the full or partial removal of the wall during a simulation depending on criteria based on the results of 2D mesh elements adjacent to the wall. The removal of the wall represents the collapse of the wall. The criteria to trigger a collapse are as follows:

- **Full**: If collapse of the wall is triggered, the entire Porous Wall object will be removed from the simulation.
- **Partially**: If collapse of the wall is triggered, only the portion of the Porous Wall object adjacent to the mesh elements triggering the collapse will be removed from the simulation.

As well as representing the collapse of buildings and walls, this functionality can also represent the breach of lateral embankments and other flood defence structures or the failure of in-line dam structures.
Break Lines

Break Lines can be used to define a high or low location on the ground surface, such as a ridge or the base of a ditch. Break lines are also useful if you need to represent one-dimensional permeable features such as fences. These lines act as break lines during mesh generation, enforcing the lines as mesh element edges.

Break lines can be created or edited within InfoWorks or imported using the Open Data Import Centre.

Alternatively, if the break line data is in GIS format, the GIS file or layer containing the data can be selected for use at the time of generating the 2D mesh. A Line Category is used to identify which lines are to be used as break lines.

Initial Conditions

Initial values for level, depth and velocity can be applied to mesh elements, such as the area bounded by the 2D Simulation Polygon or a specific Initial Conditions Polygon. By default, a set of Initial Conditions will apply to all mesh elements within the 2D Simulation Polygon. However, it is possible to specify areas where the initial values given supersede those set for 2D Simulation Polygons.

The initial values themselves are defined in a 2D Initial Conditions object, allowing different sets of initial conditions to be applied to the same network.

2D Initial Conditions Polygons can be imported via the Open Data Import Centre or generated via intersection. Polygons can also be digitised directly on the GeoPlan View.

2D Simulation Polygon Boundary Condition

The Normal Condition boundary type assumes that Depth and velocity are kept constant when water reaches the boundary so water can flow out without boundary losses (i.e. slope balances friction forces). However, where this is not the case, two boundary condition objects are available:

- **2D Point Source object**: Used to define the location of a flow-time boundary point within a 2D Simulation Polygon when undertaking 2D Modelling. Inflow associated with the point source discharges to the 2D mesh element in which the point is located.
• **2D Boundary object:** Used to define alternative boundary conditions along a section of a 2D Simulation Polygon boundary. The boundary condition defined at the line will override the 2D Simulation Polygon boundary type where the line and polygon boundary coincide.

Interaction with the Below Ground System

Only manholes with a flood type of “2D” or “Gully 2D” are capable of conveying floodwater onto the 2D mesh. No other flood types interact with the 2D mesh.

A manhole with a flood type of 2D acts as a weir allowing floodwater to pass from the manhole on to the mesh or vice versa. The circumference of the manhole shaft defines the length of this weir. The discharge coefficient of this weir is defined by the *Flooding discharge coefficient*, which can be specified by the user as part of the manhole parameters.

The **Gully 2D** flood type allows a head discharge relationship between the 1D manhole and the 2D mesh to be defined by use of a Head Discharge Table.

It is important that the manhole ground levels match the height of the ground model at the relevant location. Otherwise flow could spill on to the mesh either earlier or later than it reached the manhole ground level. One way to achieve a good match between manhole and ground model ground levels is to infer the manhole ground level from the ground model using the inference tool.
2D Outfalls
The 2D Outfall is used to model exchange of water between an outfall and a 2D meshed area. The 2D Outfall discharges to the 2D mesh element in which it resides.

Rainfall on the 2D Mesh
InfoWorks 2D allows rainfall to be applied directly to mesh elements. Independent rainfall profiles can be applied to selected areas and evaporation from the 2D mesh is accounted for.

The user is given the choice of whether the rainfall will be applied to all mesh elements in the 2D Simulation Polygon or only to those mesh elements whose centroids are not inside a subcatchment boundary. This second option prevents rainfall being applied twice to subcatchment areas.
Ground Model Height Correction
The height correction tool allows adjustment of the heights of regions, or zones of a Ground Model. This functionality may be used to correct for road channel and kerb depths prior to conducting a 2D Simulation.

The zones to be adjusted are defined by Polygons. A selection of Nodes may be used as control points against which the ground model data is compared to calculate an average height error for each zone. Alternatively, a user defined height correction may be applied.

Mesh Polygon Ground Level Modification
It is possible to use Mesh Polygons to define specific zones in which ground levels are to be adjusted in the 2D mesh.

Ground level modification is applied either by raising or lowering the 2D mesh data, or by setting a specific level. The changes are applied to the mesh elements within the Mesh Polygon. The adjustment is carried out during simulation without change to the source ground model.

This functionality could be used to correct for road channels and kerb depths for example.
Displaying 2D Simulation Results

2D Simulation results can be displayed by the following methods:

- On the 2D Polygon Mesh Element Properties Dialog
- As themes on the GeoPlan View and 3D Terrain View
- In Time Varying Results Grids and Graphs
- On Graph Reports
- On a Long Section View

2D results can also be exported to CSV file and are included when Exporting Results to GIS.

Point, Line and Polygon Results

Results can be displayed using a variety of point, line and polygon reporting tools, which are available from the InfoWorks toolbar.

2D Mesh Element Results

Results can be displayed in a grid view, via the Mesh Element Properties Dialog. As well as the standard results such as water depth, velocity and direction, for those doing Risk Analysis, InfoWorks 2D can also report Maximum Hazard, Time to First Inundation and Time to Peak Inundation.
The ability to display the results of an SQL expression as a theme for all 2D triangles is also available. A special *Expression* button on the 2D Triangles tab of the GeoPlan Themes Dialog is used to define an SQL expression, which can be saved for future use.

### 2D Point Results

Report on selected points within the 2D Simulation Polygon.

![2D Point Results Image]

### 2D Line Results

Report on the volume of flow passing a line drawn within the 2D Simulation Polygon.

![2D Line Results Image]
2D Polygon Results
Report on the volume of water enclosed by a polygon drawn on the 2D mesh.

2D Results Export to GIS
All 2D results shown on the GeoPlan view can be exported to MapInfo or ESRI GIS systems in the form of MIF files, SHP files or a Personal Geodatabase. InfoWorks can export either maximum results or full time varying data.
2D Results Export to Google Earth
All 2D results shown on the GeoPlan view can be exported to Google Earth in the form of a KMZ file. InfoWorks can export either maximum results or full time varying data.
Hardware configurations

The minimum specification for InfoWorks is an Intel Pentium D, Pentium Extreme Edition or Dual Core Xeon processor with at least 2Gb of RAM. However, for better performance, an Intel Core2Duo or AMD X2 Athlon processor should be used and the RAM should be increased to 4Gb.

For the very best all round results use the latest Intel (i.e. Core-i5 or Core-i7 or multi-core Xeon) or AMD (i.e. Opteron or Phenom II) processor and install at least 4Gb of RAM. As a general rule, the more cores your PC has the better, although the gains do start to tail off once you reach 8 cores.

Approximate timings are:-
- Single Core to Dual Core = 40% improvement.
- Single Core to Quad Core = 60% improvement.
- Single Core to Dual Quad (8) Core = 65%-70% improvement.

Clock speed is also an important consideration. InfoWorks simulations are CPU intensive so the faster the CPU clock speed, the shorter the simulation times.

You may like to use this site to compare performance of the various processors that are currently available -> [http://www.cpubenchmark.net/cpu_list.php](http://www.cpubenchmark.net/cpu_list.php). It’s not a definitive guide, but they do have benchmarking data for most of the common AMD and Intel processors, which can be compared with those going back two or three years.

The graphics resolution should be at least 1280x1024. The important point for InfoWorks (especially if you want to take full advantage of the 3D rendering in the GeoPlan) is that you have a dedicated graphics card, rather than relying on any on-board graphic capability of the PC’s motherboard. Examples would be graphics cards from manufactures like ATi or NVIDIA. These cards have their own dedicated memory, which is typically either 256Mb or 512Mb. Don't go for less than 256Mb.

You should have at least 20Gb of local hard disk space for working files.

The Simulation Engine in InfoWorks CS is a 32-bit executable, but will run quite happily on a 64bit machine and operating system. Full support is provided for the 64-bit edition of Windows Xp and Windows Vista (Service Pack 1) from v10.0 of InfoWorks. InfoWorks v10.5 was the first version to provide full support for the Windows 7 platform (x86 and x64). Support is also provided for Microsoft Windows Server 2003, 2003 R2 and Windows Server 2008. Under these Operating Systems the software can support more than 2GB of memory, which gives InfoWorks more address space (up to a limit of 4Gb), making it capable of creating much larger 2D meshes.

Please note that InfoWorks is only supported on x64 versions of Microsoft operating systems, not on Itanium versions.

We do not support the operation of InfoWorks on Windows 95, Windows 98, Windows Me, Windows NT v3.51 or Windows NT4 operating systems. This is because Microsoft no longer provides any support for these legacy products.

If you are using Windows 2000, you must have applied Service Pack 3 (or later) to use InfoWorks. We recommend applying Service Pack 3 for Windows Xp, although this is not a system
requirement for InfoWorks to run. Those running Windows Vista should ensure they are at Service Pack 1 (SP1) and that the latest fixes and security patches have been applied.

For large models (greater than 10,000 pipes/nodes), a significant amount of disk space will be required. This is both for simulation results and the numerous temporary files created during the model build / data transfer process. If data and results are to be held on the local machine, make sure you have at least 20Gb of free disk space after installation of the operating system, InfoWorks, and any other software tools (i.e. Microsoft Office or your preferred GIS application). For best performance, use a good quality hard drive and make sure it has fast data access speed.

Large networks, particularly those with ground models containing DTM, DEM or LIDAR data, require a lot more memory and a lot more disk space to manipulate than standard data.

If the database and results files are to be held on a Network Server, make sure the network itself can handle the large amount of traffic that could be created. At a minimum, we recommend Fast (100Mb) Ethernet, but a Gigabit network environment is preferable. If your system is based on a T1 connection (1.5Mb) or even 10Mb Ethernet, you may well find the network is unable to cope with the added demands that InfoWorks could place on the system.