APPENDICE A
INFOWORKS

Product Overview

InfoWorks is used to model open channel and overbank flows in any network of channels. Any sensible looped or branched network can be modelled using InfoWorks, which incorporates a wide range of hydraulic network objects including a variety of conduit types, hydraulic structures, and so on.

InfoWorks computes flow depths and discharges using a method based on the equations for shallow water waves in open channels - the Saint-Venant equations.

InfoWorks can be used to solve systems under both steady and unsteady flow conditions. Steady solutions are discussed further in the Steady Flows topic. For unsteady solutions InfoWorks uses the governing hydraulic equations for each network object. These equations are inevitably a combination of empirical and theoretical equations many of which are non-linear. The non-linear equations are first linearised and the solution to the linear version of the problem is then found via matrix inversion. An iterative procedure is used to account for the non-linearities.

The Preissmann four-point implicit finite difference scheme is employed for the channel equations and the matrix is inverted using a powerful sparse matrix solver.

B.1 Steady Flows

To start an unsteady flow run, an estimate of the initial conditions (flow and stage) is required at every model node. This is most often obtained by carrying out a steady state run at the proposed start time.

Two methods are available within InfoWorks to compute steady flows. The Pseudo-Timestepping Method uses the Preissmann 4-point scheme, whereas the Direct Method uses a quite different and highly accurate solution technique.

<u>Direct Method</u>

The main solution method is called the Direct Method. It overcomes some of the difficulties associated with the Pseudo-Timestepping Method. It is faster and more accurate and requires very little initial data. For steady state conditions the Saint-Venant equations can be reduced and written as ordinary differential equations; these are solved for individual reaches. This is used with an automated distance step size control option which tells you where extra interpolated sections should be inserted into the model. The method can be thus seen as having an adaptive grid.

The remaining problem is to solve the network so that Kirchoff's law is satisfied and equal water elevations are obtained at junctions. This is done by an iterative scheme to solve the correction of the flow splits at channel confluences and bifurcations. Convergence is achieved when the maximum correction to a flow split is less than 0.1% and the maximum elevation difference is less than 1mm at a junction.

The method is quicker than the pseudo-timestepping method and is able to pinpoint data problems and sensitive areas in a particular model. The automatic identification of local problem areas is of great advantage.

The method deals accurately and consistently with the problem of cross section spacing. During the computation, the method checks whether the solution is "grid dependent", and if necessary will add extra interpolated nodes implicitly. The user is informed where this has been done so that extra surveyed sections can be added to the model if available, or extra nodes interpolated between the existing sections. If a large number of interpolated sections have been added between two cross-sections, this indicates either that the channel properties of the two sections are significantly different (alerting the modeller to potential data discrepancies) or there is large surface curvature in the channel (for example as the Froude number approaches unity).

Pseudo-Timestepping Method

The Pseudo-Timestepping Method requires initial estimates for flow and stage at each model node specified in the Initial Conditions. These initial conditions are used for the steady state run, with the boundary conditions held constant for the time at which the solution is required.

The initial timestep is input at the start of the run, but after each five pseudo-timesteps the user is asked whether it should be changed. If the initial conditions are very approximate, the flows may be highly unsteady or unstable and a very small timestep (may be as low as 10 seconds) should be used. The timestep can be increased as the flow becomes less unsteady (measured by the flow ratio). The model is run until all the irregularities and inaccuracies in the guessed initial conditions have propagated or been dissipated out of the system. This can be a time consuming process, especially when the model contains large reservoirs which have long response times.

The monitoring parameters are the flow and head ratios, which are automatically printed on the screen. When these values become very small (usually less than 0.5 x 10-3) and the timestep is large (usually greater than 200-500 seconds), then a steady solution should have been attained. This can sometimes be difficult to achieve since some models may be sensitive to sudden increases in timestep. It should be noted that when the timestep is changed the solution may appear to diverge before starting to converge again.

Method Comparison

In general, the direct method will solve a subset of problems that can be solved by the pseudo-timestepping method. This subset includes most problems of practical interest. The main restrictions or differences are outlined below.

The direct method is primarily applicable to in-bank problems and will not solve side spills. If a channel would have flooded over a side spill, a warning is given in the Simulation Diagnostics detailing where this would have occurred.

The direct method will cope with most of the hydraulic units available within InfoWorks. However sluice gates cannot be solved in 'water remote' mode and the method will not yet cope with negative flows through structures.

The direct method will not accept negative flow boundaries (except at junctions), downstream flow-time boundaries or upstream head-time boundaries as these may lead to indeterminacy of the equations.

The cross-sections of a geometrically defined conduit (Circular Conduit or Rectangular Conduit for example) are not permitted to vary within a conduit reach. The open channel sections should be input from upstream to downstream, as is the normal ISIS Flow procedure.

No initial conditions, except for initial estimates of flows at confluences, are required by the direct method. Faster convergence is achieved with better initial conditions.

In general, when there is zero flow in certain reaches, these should be included in the initial conditions. It is possible that in certain reaches with zero flow the water levels cannot be calculated. Such a situation may arise between two structures with closed gates for example. In which case, the stages at nodes in the reach will be written by the direct method as -9999.0 and written as such to the Simulation Results. These must be changed before proceeding to an unsteady run.

Formally, the direct method is fourth order in accuracy and is thus more accurate than the pseudo-timestepping method, which is second order accurate at best. Inevitably there may be a small difference in the results produced by the two methods. This may cause unsteady model runs not to start smoothly and, for sensitive problems such as steep channel networks, instability may result. This can be dealt with by using the steady state results file rather than the initial conditions in the datafile for the initial conditions of the unsteady run. In this case some smoothing iterations are performed prior to the run. Alternatively results from the direct method can be used as initial conditions for a pseudo-timestepping run, the results of which would be used to initiate the unsteady run.

Another reason why the direct method may produce results that are different to the pseudo-timestepping method is the addition of automatically interpolated cross-sections, which will occur when a solution cannot be found to the required accuracy using the section spacing in the datafile. Details of how many (and at which locations) extra sections have been added by the direct method can be found in the Simulation Diagnostics. Experience indicates that for most problems, it is worth adding extra sections to the network using Interpolate units when the direct method has added seven or more sections.

B.2 Supercritical Flows

Supercritical flow can be modelled approximately in InfoWorks. This is achieved by neglecting the $\delta A/\delta x$ part of the convective momentum term in the momentum equation when the Froude number exceeds a specified upper value. Between this upper value and a specified lower value, the term is gradually phased out so that a smooth transition is achieved.

For steady supercritical flow in a uniform channel, the method should be acceptably accurate but will become more approximate as the channel becomes more non-uniform.

This approach is adequate for problems where supercritical flow prevails locally in isolated areas of a model and when low flows are required as initial conditions for an unsteady run.

Since InfoWorks solves the differential form of the momentum equation, the solution at a hydraulic jump or bore can never be accurate. Instead of a sharp change in stage, the change will be spread over several nodes.

Direct Method Transcritical Solver

The Direct Method steady state solver incorporates an optional accurate supercritical and transcritical flow solver which has the capability of modelling hydraulic jumps and supercritical flow more accurately. The full St Venant equations are used, with the numerical scheme reversing the direction of integration to upstream-to-downstream in supercritical parts of reaches. Momentum considerations are then used to establish where the supercritical and subcritical regimes meet, thus determining the locations of hydraulic jumps.

To use the accurate transcritical solver, check the Direct Method Transcritical Solver box on the Run Options Dialog - General Options Page. You access the Run Options Dialog by clicking the Options button on the Schedule Run View.

Scope:

- The Transcritical Solver is limited to the Direct Method, and cannot be used for unsteady, steady timestepping or Flood Routing models.
- Levels at upstream nodes of reaches (for example immediately downstream of junctions) may default to critical depth if no appropriate supercritical depth can be calculated by the solver.
- The flow equations at many structures contain implicit assumptions about subcritical flow and may not be appropriate for supercritical flow. Furthermore, the Direct Method Transcritical Solver calculates a downstream water level from the upstream level, which is not necessarily unique.

B.3 Theoretical Basis

River modelling involves the interaction of the channel and external and internal controls, taking into account the requirement that the mass and momentum of the body of water should be conserved.

Hydrodynamic Channel Flows

The motion of a body of water flowing in open channels can be described by the so called shallow water or St Venant equations, which express conservation of mass and momentum. Conservation of mass leads to the continuity equation which establishes a balance between the rate of rise of water level and wedge and prism storages. Conservation of momentum leads to the dynamic equation which establishes a balance between inertia, diffusion, gravity and friction forces. Some other forces, such as the effect of wind or meanders, may also be included but usually these are small.

The governing equations are the continuity equation:

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

where q is lateral inflow ($m^3/s/m$).

The governing equations are the continuity equation:

$$\frac{\partial Q}{\partial t}+\frac{\partial}{\partial x}\left(\frac{\beta Q^2}{A}\right)+gA\frac{\partial H}{\partial x}-gAS_f=0$$

where S_f is the so called friction slope:

$$S_f=\frac{Q\,|Q|}{K^2}$$

where K is the channel conveyance calculated according Manning's equation:

$$K^2=\frac{A^2R^{\frac{4}{3}}}{n^2}$$

and

$$R=\frac{A}{P}$$

where:

- R is the hydraulic radius
- P is the length of the wetted perimeter
- n is Manning's roughness coefficient.

The equations are described in mathematical terms as a pair of one-dimensional non-linear hyperbolic partial differential equations. In general, the solution of any system of differential equations depends on the existence, uniqueness and stability conditions of that solution. By transforming these equations into characteristic form, Courant R. and Lax P. (1959) have shown that they satisfy the Lipschitz condition ensuring the existence of a unique solution. Stability of the solutions has also been widely investigated. The interested reader is referred to standard texts for further details, such as Mahmood and Yevjevich (1975) and Cunge J A et al (1980).

External Boundaries

The characteristic form of the equations can be used to show that under sub-critical conditions, disturbances to the flow are propagated in both directions whereas under super-critical conditions disturbances can only propagate downstream. This implies that for wholly sub-critical flow, an independent boundary condition is required at both upstream and downstream ends of the model whereas for wholly super-critical flow two independent boundary conditions (usually flow and stage) are required at the upstream extent of the model.

For sub-critical flow, which is of primary interest, these boundary conditions are specified as discharge-time, stage-time or stage-discharge (rating curve) relationships. The following are known to lead to a properly posed system of equations:

- discharge hydrograph upstream and stage hydrograph downstream
- discharge hydrograph upstream and rating curve downstream

although other combinations may work in certain circumstances. The specification of a discharge hydrograph at the downstream end may lead to problems and Cunge J A et al (1980) noted that this can cause instabilities even though Amien M. and Fang C.S. (1970) claim to have implemented a model specifying discharge hydrographs at both ends.

In general the boundary conditions must be correctly specified since the uniqueness of the solution depends upon them. Irregularities in data can cause flow oscillations and it is often advisable to use a smoothed hydrograph, particularly at downstream boundaries.

Internal Boundaries

In a one-dimensional network representation of a river (or conduit), reaches are separated by internal boundaries which may be control structures, losses, reservoirs or junctions (bifurcations or confluences). These boundary conditions impose a relationship between the stages and discharges at the nodes involved.

Control Structures

A wide variety of structures can be used to control flow in open channels each of which imposes a different relationship between flow and stage. In free mode, the general form of the equation is:

$Q = ah^b$

where:

- h is the water depth
- a is some coefficient dependent on the structure
- b is usually greater than or equal to 1.5 for weir type flows and greater than or equal to 0.5 for free flow under a sluice gate.

These relationships are semi-empirical in nature and may require calibration. In general most structures can be categorised as weir or orifice types.

Weirs may be natural sills but are usually man-made structures transverse to the flow direction. They may span the full width of the channel or only part of it. InfoWorks has the capability of modelling the following types using empirical or semi empirical equations from the literature:

- Sharp Crested Weir
- Round Nosed Weir
- Crump Weir
- Spill (Jagged) Weir
- Notional Weir
- Triangular Profile Weir
- Gated Weir (time varying crest elevation)

These structures may operate in dry mode (no flow), or free or drowned mode according to the modular limit.

Sluices operate in a variety of flow modes including weir equations when the gates are out of the water and the obvious orifice type flow for normal sluice operations. InfoWorks considers many possible flow modes for some of the sluice types including no flow, free and drowned weir flow, free and drowned gate flow, free and drowned flow over the top of a sluice gate and combinations of flow both under and over gates.

It is possible to control automatically the opening of sluice gates during a run, such as:

- according to pre-specified times
- according to upstream or downstream water levels
- to relate gate openings to water level at a remote node to simulate actions initiated by a flood warning.

The types of sluice gates available within InfoWorks are:

- Vertical Lift Sluice
- Radial Gated Sluice

You can also input a rating curve for a channel control that is not described by any of the standard structures listed above, such as a flume or a general channel constriction, by using a Flow-Head Control. This network object also has the option of free and drowned modes according to a specified modular limit.

Reservoirs

Reservoirs are defined as large (or sometimes small) storage areas with a flat water surface where the dynamic effects are negligible. This need not necessarily be a formal reservoir, but could be a depression on the flood plain. It is also very often acceptable to use a reservoir to model parts of the floodplain (or channel, in some instances) if dynamic effects are negligible.

The balance of inflow and outflow of the reservoir is related to the rate of change of the head as follows (in a simplified form):

$$\frac{hnew - hold}{\Delta t} = \frac{Qnet}{Area}$$

The reservoir together with the Spill adds a great deal of flexibility into InfoWorks for schematisation of complex natural open channel systems.

Losses

The discrete energy losses such as those caused by a sudden contraction or expansion in the channel can be represented by a Bernoulli Loss which relates the head loss to the upstream velocity head:

$$\Delta H = k\,\frac{V^2}{2g}$$

where k is an empirical loss coefficient which can be used as a calibration parameter.

Bridges can be modelled explicitly using either the US Bureau of Public Roads method (US BPR Bridge) or the Arch bridge method devised by HR Wallingford (Arch Bridge).

Junctions (Bifurcations and Confluences)

In looped or branched systems junctions are an obvious requirement. In InfoWorks junctions are represented by simply equating water levels at the nodes of the junction and conserving mass by applying Kirchhoff's Law to the flows.

Storage effects are neglected so each junction node must be effectively at the same position. They should also be at the same bed elevation. It is possible to include storage effects by replacing a junction with a reservoir.

If the velocity is significant at any of the junction's nodes then total heads should be investigated. Total head junctions are not included explicitly in InfoWorks. If total heads are important then the Bernoulli Loss unit can used on some or all of the junction arms.

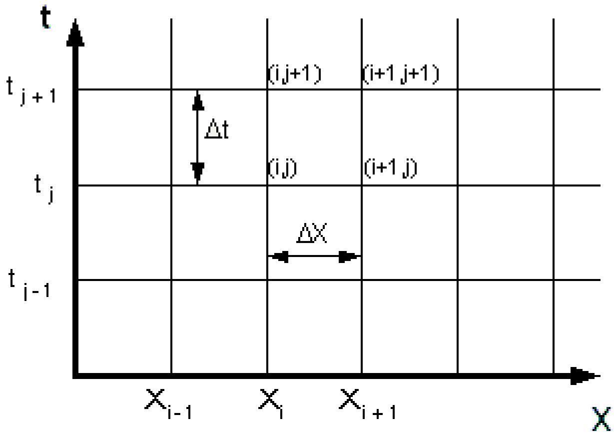
Discretisation

It is not possible to solve the Saint-Venant equations analytically - hence the need for numerical solution. As long ago as 1958 Isaacson E. et al (1958) demonstrated the feasibility of using the full Saint-Venant equations to obtain reliable results for practical engineering purposes. Their work was the first computational model implemented in the field of open channel flow and used an explicit finite difference scheme.

The first computational model using an implicit finite difference scheme was implemented by Preissmann A. (1961). In the course of the following years various different numerical schemes were proposed and implemented by other authors, but it appears now that the Preissmann scheme has more or less become the accepted standard.

InfoWorks also employs the Preissmann implicit scheme - which is popularly referred to as the 4-point Box scheme. The scheme is outlined below.

Let f be the value of depth or discharge or a function of depth or discharge at point $(i + ½, j + \theta)$ as shown in the figure below:



The value of *f* or its continuous derivatives with respect to time or space can be discretised as:

$$f(x,t) = \frac{1}{2}\left[\theta\left(f_{i+1}^{j+1} + f_{i}^{j+1}\right) + (1-\theta)\left(f_{i+1}^{j} + f_{i}^{j}\right)\right]$$

$$\frac{\partial f}{\partial x} = \frac{1}{\Delta x}\left[\theta\left(f_{i+1}^{j+1} - f_{i}^{j+1}\right) + (1-\theta)\left(f_{i+1}^{j} - f_{i}^{j}\right)\right]$$

$$\frac{\partial f}{\partial t} = \frac{1}{2\Delta t}\left[\left(f_{i+1}^{j+1} - f_{i+1}^{j}\right) + \left(f_{i}^{j+1} - f_{i}^{j}\right)\right]$$

where:

- θ is a weighting factor lying between 0.5 and 1
- f_i^j is the value of *f* evaluated at the point (x_i , t_i)

Using the above, both Saint-Venant equations can be transformed into the linear form:

$$aQ_{i}^{j+1} + bH_{i}^{j+1} + cQ_{i+1}^{j+1} + dH_{i+1}^{j+1} = e$$

The values *a, b, c, d* and *e* are calculated for each iteration and each node in the open channel and depend on variables calculated at the previous iteration or timestep.

The coefficient matrix, which comprises largely of the a, b, c, d and e values described above must be inverted to solve the set of simultaneous difference equations for *Q* and *H* at the following iteration or timestep. InfoWorks takes advantage of the banded structure of this matrix by employing a powerful sparse matrix solver.

B.4 Mobile bed sediment transport module

This section is a technical overview of the InfoWorks mobile bed sediment transport module. InfoWorks can predict sediment transport rates, bed elevations and amounts of erosion/deposition throughout a channel system. The module is run as part of an InfoWorks hydraulic simulation and requires the inclusion of a Sediment Data object.

The basic capabilities of the mobile bed module are to predict sediment transport rates, bed elevations and amounts of erosion/deposition throughout a channel system. In summary, this is achieved with the following calculations at each timestep:

1. calculate the hydraulic variables of flow, stage, velocity in the usual way
2. starting at the upstream end of the system, loop around the nodes calculating the sediment transport capacity and solving the sediment continuity equation for depth of erosion/deposition
3. update the channel conveyance tables to allow for any calculated deposition or erosion ready for the next timestep

Various options are available including:

- specification of dredging;
- cohesive sediment transport;
- rigid beds.

The main restrictions on the applicability of the software are that:

- 'local' effects may not be simulated (eg scour at bridge piers);
- dunes and ripples are not explicitly modelled and therefore the effects of changes in form roughness on the hydraulic resistance is not simulated;
- flow reversals are not accommodated at present;
- the following network objects are currently not permitted in a mobile bed module run: Spill, Storage Area, Interpolated Section, Replicated Section, Pump, Muskingum Routing, Abstraction, Flow-Head Control;
- reaches with zero flows are not permitted at present;
- sediment transport computations in compound channels may be inaccurate due, in part, to cross section averaged velocities being unrepresentative of main channel velocities.

Sediment Continuity Equation

For channel units the form of the sediment continuity equation used is:

$$(1-\lambda)\,W\frac{\partial z}{\partial t}+\frac{\partial G}{\partial x}=0 \qquad (1)$$

where:

- λ = bed porosity
- W = water surface width
- z = bed elevation
- t = time
- G = sediment transport rate (m^3/s)
- x = distance in flow direction

Equation (1) assumes that the rate of change of sediment entrained in the flow is negligible in comparison with the terms expressing rate of change in bed elevation and change in transport rate.

Equation (1) is discretised in the following form:

$$(1-\lambda)\, W_{k+1} \frac{\Delta z_{k+1}}{\Delta t} + \frac{G_{k+1}^{i+1} - G_{k}^{i+1}}{\Delta x} = 0 \qquad (2)$$

where:

- k = position index, which increases in the downstream direction
- i = time index
- Δz = change in bed elevation over timestep Δt

Equation (2) can be solved explicitly for Δz as G_{i+1}^{k+1} is calculated by the sediment transport capacity equation and G_{i+1}^{k} will have been determined previously.

At upstream boundaries the user must specify the sediment inflow as rate against time, concentration against time or concentration against flow rate. The bed elevation is free to move at both the upstream and downstream boundaries (this includes the nodes immediately upstream and downstream of junctions and structures such as cross regulators). To achieve this requires a special implementation of equation (2) at the upstream boundary nodes and at the nodes immediately downstream of structures and junctions. At these locations, position index k-1 does not exist, so the change in cross section area is set equal to the change at location k+1. The change in cross section does not directly relate to any change in G_k and there is no length of river upstream to convert the change in area to a change in volume. Therefore, these section changes at upstream boundaries are not recorded as a deposition/erosion volume in the output.

At junctions the sediment outflow is equal to the sum of the sediment inflows (mass is conserved). All inflow node sediment transport rates are determined before the outflow nodes are calculated. If there is more than one outflow node then the concentrations at all outflows are assumed equal.

All hydraulic structures are considered simply as two noded junctions and thus the outflow sediment transport rate equals the inflow sediment transport rate.

Sediment Transport Equations

Four sediment transport equations are available: Engelund-Hansen, Ackers-White, revised Ackers-White and Westrich-Jurashek. All include a calibration factor which has the default value of unity to give the published form of the equations.

You can input a global sediment size distribution for the bed material, which consists of a table of size against proportion present of that size for between 1 and 10 sizes. The sediment transport is calculated for each size and the resulting total transport rate is calculated by multiplying the proportion of the size in the bed material by the calculated rate. The reported transport rate is the summation of the individual rates. Further details on graded sediments are given below.

Engelund-Hansen (1967) Total Load Equation

The Engelund-Hansen equation was developed by equating the work done by the drag forces of the flow to the potential energy gained by particles as they move up the face of a dune. The form of the equation used is:

$$G = K \frac{0.05\, W\, V^2\, h^{1.5}\, S^{1.5}}{(s-1)^2\, D\, \sqrt{g}}$$

where:

- G = volumetric sediment transport rate
- K = calibration coefficient (=1 for standard equation)
- W = width of flow;
- V = water velocity
- h = flow depth
- S = water surface slope
- s = specific gravity of sediment
- D = sediment diameter
- g = acceleration due to gravity

The factor 0.05 in the equation was set using empirical data.

The suggested applicability of the Engelund-Hansen equation is for $\sqrt{(D_{75}/D_{25})} < 1.6$ and for a mean diameter greater than 0.15 mm.

Ackers-White (1973) Total Load Equation

The Ackers-White equation was developed by determining the appropriate form of the equation from physical considerations and dimensional analysis, but using empirical data to determine the various coefficients. Both the original and the updated versions of the coefficients are available in the model. The calculation procedure for the Ackers-White equation is described below.

1. Determine the dimensionless sediment diameter, D_{gr}

   $$D_{gr} = D\left(\frac{g(s-1)}{v^2}\right)^{1/3}$$

   where:

   - D = particle diameter (Ackers and White advise the use of the D_{35} size)
   - g = acceleration due to gravity
   - s = specific gravity of sediment
   - v = kinematic viscosity of water

2. Determine the transition exponent, n, the initial motion parameter, A, and the coefficient and exponent in the sediment transport function (c and M respectively)

   For $D_{gr} > 60$:

$n = 0$

$A = 0.17$

$M = 1.5$

$c = 0.025$

For $1 \le D_{gr} \le 60$:

$$n = 1 - 0.56\log_{10}D_{gr}$$

$$A = \frac{0.23}{\sqrt{D_{gr}}} + 0.14$$

$$M = \frac{9.66}{\sqrt{D_{gr}}} + 1.34$$

$$c = 10^{(2.86\log_{10}D_{gr} - (\log_{10}D_{gr})^2 - 3.53)}$$

3. Determine the particle mobility, F_{gr}:

$$F_{gr} = \frac{V_*^n}{\sqrt{gD(s-1)}}\left(\frac{V}{\sqrt{32}\log_{10}(10h/D)}\right)^{1-n}$$

where:

- V^* = shear velocity (= $(g\ h\ S)^{0.5}$)
- V = mean flow velocity
- h = depth of flow

4. Determine the dimensionless sediment transport rate, G_{gr}:

if $A < F_{gr}$

$$G_{gr} = c\left(\frac{F_{gr}}{A} - 1\right)^M$$

if $A \ge F_{gr}$

$$G_{gr} = 0$$

5. Determine the volumetric sediment transport rate, G:

$$G = K\frac{Q\,G_{gr}\,D\left(\frac{V}{V_*}\right)^n}{h}$$

The suggested applicability of this equation is for $D_{gr} \ge 1$ and for flows with Froude numbers less that 0.8.

The equations for n, A, M and c have recently been revised and both the original and revised equations are available in the model (sediment transport equations 2 and 3 respectively). The revised equations are:

For $1 \le D_{gr} \le 60$

$$M = \frac{6.83}{\sqrt{D_{gr}}} + 1.67$$

For $D_{gr} > 60$

- $M = 1.78$
- $c = 0.025$

Westrich-Jurashek (1985) Total Load Equation

The Westrich-Jurashek equation was developed from laboratory experiments on the transporting capacity of rigid boundary channels. Only fine sediments were used (0.026 to 0.11mm) in the experiments, which recorded the maximum transport without deposition on the channel bed. An energy balance equation was fitted to the results.

The equation is:

$$G = K\frac{0.0018 S V^2 W h}{(s-1)V_s}$$

where Vs = settling velocity of the sediment particles. (other variables are as defined above).

The applicability of the equation is limited to fine sediments and sediment transport over rigid boundaries.

Graded Sediments

Sediments can be divided into many size ranges so that the movement of graded sediments can be adequately represented. A representative size is input for each sediment fraction. The sediment transport rate for each fraction is computed by multiplying the predicted rate (from the sediment transport equation) by the proportion of the bed material which consists of that fraction. The bed material composition is either assumed to be constant and is set at input or it is predicted from the rates of deposition or erosion at each section. The former approach is termed the COMPOSITE algorithm, as a single composite grading represents the channel bed material. The latter approach is termed the SORTED algorithm because sediment sorting effects are modelled.

Under the SORTED algorithm, the composition of three layers is recorded: an active layer at the surface of the channel bed, a deposited layer below it and the parent bed material below the deposited layer. The quantity of sediment in each of these layers is recorded at each channel section for each sediment fraction. If there is net erosion at a section, then the deposited layer is not present. As deposition or erosion causes an exchange between the material in the active layer and the material in transport, the composition of the active layer is updated. For example, in the case of erosion from a widely graded channel bed by clear water, the finer sediments will be brought into transport more rapidly than the coarser sediments. So the proportion of the active layer consisting of the coarsest fraction will increase and the proportion of the finer fractions will decrease. If some fractions are not eroded at all, then the active layer will eventually stabilise to an armoured state for which only non-mobile sediments are present in the active layer. The active layer thickness is set at input and does not change so that for erosion the material eroded from the active layer is replaced by an equal quantity of material taken from the layer below (the deposited layer if present or the parent material otherwise). In the case of deposition, the material being added into the active layer is matched by an equal volume of material passed from that layer into the deposited layer.

If the sediments are fairly uniform, say $D_{75}/D_{25} < 2.0$, then a single sediment size can be used and no sorting effects need be modelled. In this case the COMPOSITE algorithm should be used. In other cases the SORTED algorithm is recommended. Significant errors can occur if the COMPOSITE algorithm is used in situations in which the bed composition would vary through time.

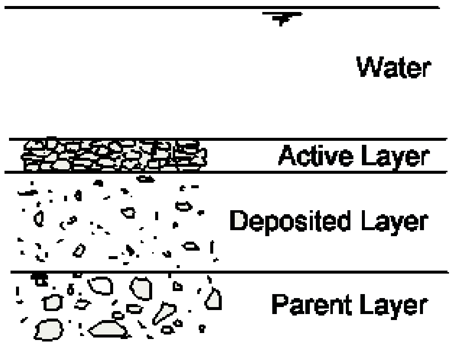


Figure B.1 - Bed Composition under the SORTED Algorithm

An important feature of sediment transport when the bed material is widely graded is the marked difference in sediment composition between the bed material and the material in transport. The material in transport will be considerably finer, in some cases the median size can be finer by several orders of magnitude. InfoWorks Sediment has input facilities that allow a difference in grading between the sediment inflow and the bed material at the inflow point.

The bed material input to the model can be made to vary from section to section when the COMPOSITE algorithm is used. If the SORTED algorithm is used these facilities enable the parent bed material to be varied from section to section (the parent bed material is also used to set the initial composition of the active layer).

Cohesive Sediments

Sand and coarser sediments are non-cohesive, while finer sediments can cohere to each other. Cohesive material is usually defined as material with diameter less than 0.063mm and consists of silt and clay sizes.

The transporting capacities for the non cohesive sediment sizes are calculated using the Engelund or Ackers & White sediment transport functions and are multiplied by their proportions in the bed material, as discussed above. The concept of a continuous interchange between suspended and bed material, which underlies sediment transport computations for non cohesive sizes, does not apply for finer sediments since cohesion prevents material present on the bed from being available for transport. Also large concentrations can be transported when there is no cohesive material in the bed. Thus a method for calculating silt transport is needed which does not depend on the bed material composition. Westrich and Jurasheks (1985) method, which was originally developed for fixed boundary channels, can be used. It has been adapted, as described in Atkinson (1992), to enable it to apply to cohesive sediment mixtures covering a wide range of settling velocities.

Erosion of cohesive material is often inhibited by its cohesive properties. This can be described by a threshold shear stress, below which no erosion can occur. The sediment properties, the rate at which it was deposited and the degree of consolidation that has occurred since deposition determine this threshold. These processes are not included in the model and so a threshold value is input for each cohesive sediment fraction, advice on determining the threshold is provided in the last section of this manual.

The settling velocities for the cohesive sizes are also given in the cohesive sediment input. They should usually be derived directly from field data.

When erosion of cohesive material is being modelled using the SORTED algorithm, which is recommended, then erosion of a fraction is prevented when there is none of that fraction in the active layer. (For non-cohesive sediments this feature is incorporated automatically by the direct use of bed material composition in the sediment transport computations).

Adaptation

The sediment transport functions predict the equilibrium sediment concentrations at each cross section. These methods are based on the assumption that sediment concentration in transport has adjusted to the local hydraulic conditions. The concentrations, particularly for deposition of finer sediments, however, may not be in equilibrium due to the time required for the sediment concentrations to adjust to a new equilibrium when settling velocities are low. The turbulent diffusion equation developed by Dobbins (1944) describes this effect, but its solution requires a two dimensional model. A simpler method is applied in InfoWorks Sediment, therefore, to simulate the adaptation effect: an equation was fitted to the results from a two-dimensional non-equilibrium sediment transport model based on the turbulent diffusion equation. The form of the equation is an exponential decay function that describes the adjustment of concentrations from the concentrations upstream. The rate of adjustment depends on shear velocities and settling velocities. The equation used in InfoWorks Sediment to calculate the adjusted downstream concentration, C_{k+1} is:

$$C_{k+1} = X_{k+1} + (C_k - X_{k+1})\, e^{-ay}$$

- X_{k+1} = downstream equilibrium transport concentration;
- C_k = sediment concentration at upstream section;
- A = max(1+3.7V_s / V*;1.25);
- V_s = settling velocity (for sand sizes this is computed from sediment diameters, using the relationship given by Soulsby, 1997);

- h = flow depth;
- V = mean flow velocity;
- V* = shear velocity.

While the equation can have a significant impact for cohesive sediments, it has little impact for medium sands or coarser material.

Updating Channel Geometry

A range of methods for updating the channel geometry are available:

- (0) no change in channel geometry (fully decoupled);
- (1) move all section points uniformly by the Δz calculated for the particular section;
- (2) move only those points at a section below water level uniformly by the Δz calculated for the particular section;
- (3) move those points below water level by a Δz distributed according to shear stress (scaling Δz at every data point).

Method (1) is faster as the existing channel section property arrays need only their elevation data changed, whereas methods (2) and (3) are slower as each step will probably necessitate a full recalculation of the section properties. However, it is important to note the possibility for method (1) to generate large errors, especially when significant proportions of the channel section are above water for most of the simulation period.

The channel geometry will be updated only after a user specified cumulative Δz value has been exceeded. For example the user can specify that the geometry will be recalculated after a net depth of 10mm of erosion or deposition has been achieved. This criteria is applied at each node independently.

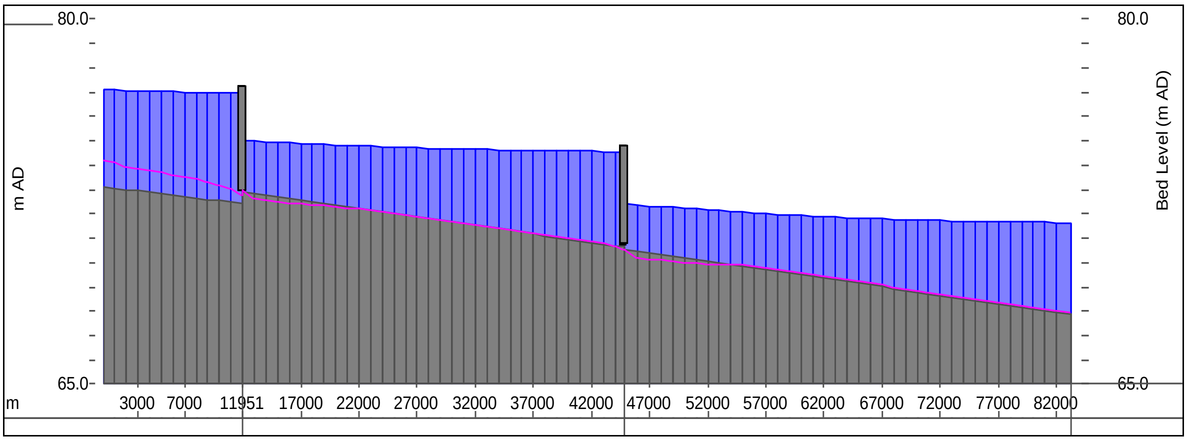


Figure B.2 -Longitudinal profile with indication of bottom variations.

B.5 2D module

Overview

InfoWorks2D (IW2D) was released by Wallingford Software in September 2007. This 2D hydrodynamic modelling software incorporates links with the existing 1D software for rivers (InfoWorks RS) and network systems (InfoWorks CS). The main characteristics of the 2D component are:

- finite volume formulation (weak solution of the shallow water equation);
- numerical scheme based on the Gudonov scheme and the Riemann solvers (Shockcapturing scheme);
- use of an unstructured mesh;
- full integration with the 1D existing engine.

The link between 2D cells and 1D stretches is made by the mean of lateral or in-line spills for 1D channels, and by the mean of manholes for 1D conduits. The discharge and momentum transfer between the 1D and 2D modules is similar to (Liang *et al.* 2007). The discharge is exchanged between the 1D module and the 2D module by relating Q_L and q_{1D}. For spills, the momentum is also passed by the 1D module to the 2D module with an assumption of critical flow to calculate u_{1D} and v_{1D}. The momentum is not passed by the 2D module to the 1D module, however the momentum corresponding to the amount of water transferred to the 1D module is dissipated to ensure conservation of momentum.

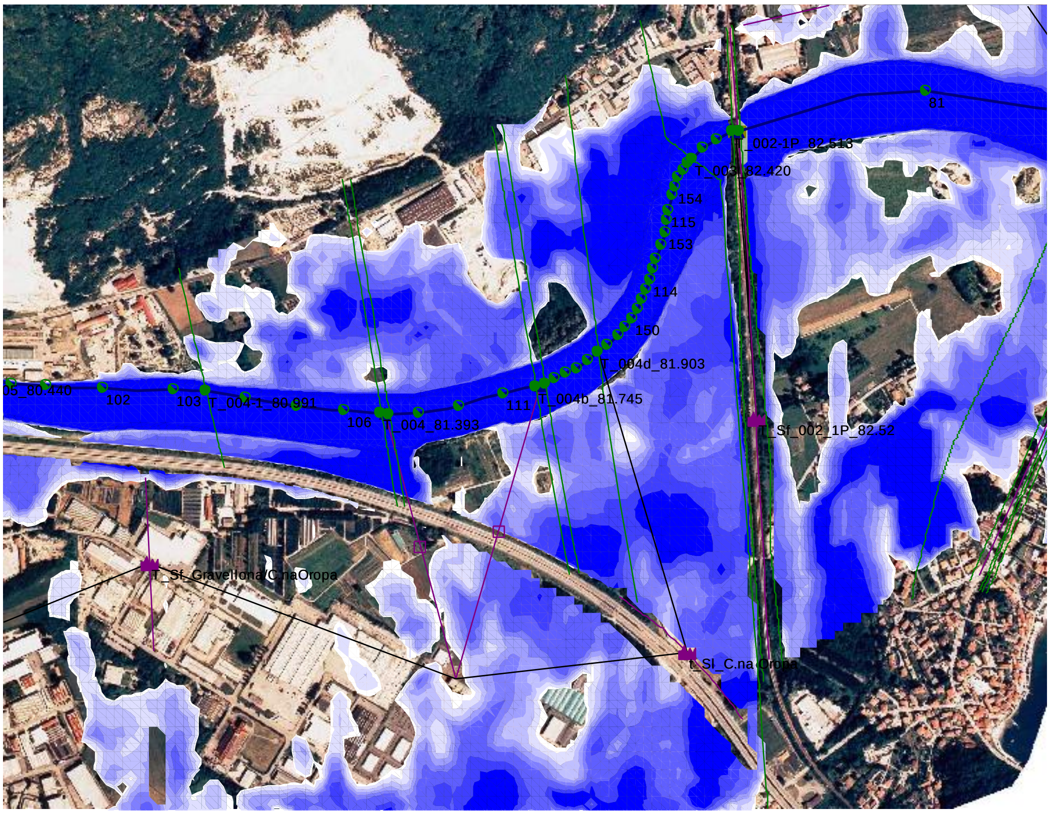


Figure B.3 - InfoWorks 1D-2D

The IW 2D engine is based on the 2D model MULFLOOD. The model used is based upon the Shallow Water Equations (SWE), that presently constitute the most common mathematical framework for this type of studies. The SWE represent so far a good compromise between physics complexity and solvability at the scales of real problems. More elaborate equations do not seem yet practical for routine use in real life problems because the length scale of the problem ranges in the order of kilometers and the required resolution is several orders of magnitude smaller.

Local description of the flow in urban regions is achieved without modifying the mathematical model which is still based upon the SWE. The presence of buildings is accounted for by introducing their effects on the shallow water model by representing them as solid walls, what only requires a precise meshing of the urban area (Figure B.4).

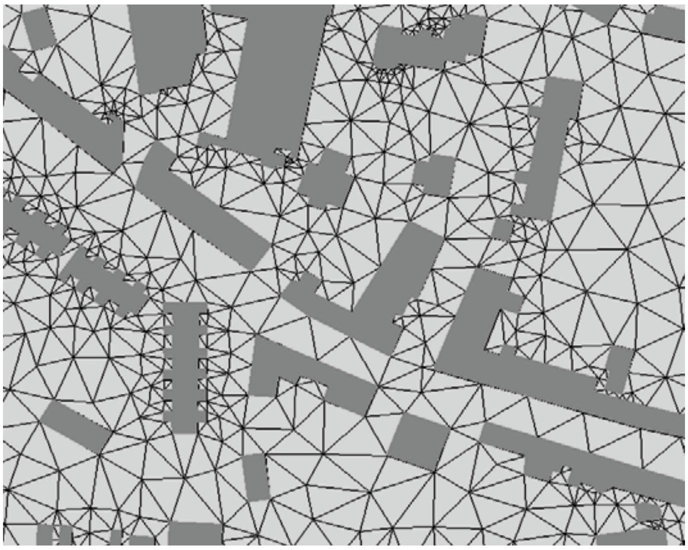


Figure B.4 - Representation of the building in the InfoWorks2D unstructured mesh.

Governing equations

The the Shallow Water Equations (SWE) represent an approximation to the free surface flow of a fluid in a horizontal plane over a prescribed bottom surface given by a bed function $z_B(x; y)$. Denoting the depth of the fluid layer by h and the depth averaged cartesian velocity components by u and v, the 2-D SWE can be written as:

$$\frac{\partial \mathbf{U}}{\partial t} + \nabla \cdot \mathbf{F} = \mathbf{S_b} + \mathbf{S_f}$$

where the flux tensor, **F** can be split in its two cartesian components:

$$\mathbf{F} = (\mathbf{E}, \mathbf{G})$$

the conserved variables and flux vectors **U**, **E**, **G**, are given by:

$$\mathbf{U} = \begin{bmatrix} h \\ hu \\ hv \end{bmatrix}, \quad \mathbf{E} = \begin{bmatrix} hu \\ hu^2 + gh^2/2 \\ huv \end{bmatrix}, \quad \mathbf{G} = \begin{bmatrix} hv \\ huv \\ hu^2 + gh^2/2 \end{bmatrix}$$

and the sources $\mathbf{S_b}$ and $\mathbf{S_f}$ by:

$$\mathbf{S_b} = -gh \begin{bmatrix} 0 \\ \partial z_B/\partial x \\ \partial z_B/\partial y \end{bmatrix}, \quad \mathbf{S_f} = -\frac{gn^2\sqrt{u^2+v^2}}{h^{1/3}} \begin{bmatrix} 0 \\ u \\ v \end{bmatrix}$$

where the empirical Manning's formula has been used to express the bed friction slope. In the form written above, the SWE embody the conservation of mass and momentum of the water layer. In the steady state, energy conservation along a streamline is also a property of the SWE system as long as there is no friction nor hydraulic jumps:

$$\frac{\partial H}{\partial t} + \mathbf{v} \cdot \mathbf{grad} H = g \frac{\partial h}{\partial t}$$

where H is the head:

$$H = \frac{1}{2} \mathbf{v} \cdot \mathbf{v} + g\zeta$$

and ζ the free surface:

$$\zeta = h + z_B$$

In 1-D the momentum and energy conservation equations are totally equivalent as long as the solution is smooth.

BIBLIOGRAFIA ALL'APPENDICE B:

ACKERS, P. (1993): *Sediment transport in open channels: Ackers and White update* Proc. Instn Civ. Engrs Wat. Marit. & Energy 101 Dec 247-249.

ACKERS, P. and WHITE, W.R. (1973): *Sediment transport: new approach and analysis.* J. Hydraul. Div., Am. Soc. Civ. Engrs., 99(HY11), 2041-2060.

AMIEN M. and FANG C.S. (1970): *Implicit Flood Routing in Natural Channels*. ASCE Journal of Hyd. Div.

ATKINSON, E. (1992): *The design of sluiced settling basins: a numerical modelling approach.* Report OD124, HR Wallingford, Oxon, UK

COURANT R. and LAX P. (1959): *On Non-Linear Partial Differential Equations with Two Independent Variables.* AEC (US) Research and Development Report No. AWL5990.

CUNGE J.A. HOLLY Jr F.M. and VERWEY A. (1980): *Practical Aspects of Computational River Hydraulics*. Pitman, London.

DOBBINS, W.E. (1944): *Effect of turbulence on sedimentation*. Trans ASCE Vol 109, Paper 2218, p629.

ENGELUND, F. and HANSEN, E. (1967): *A monograph on sediment transport in alluvial streams.* Teknisk Forlag, Copenhagen, 65pp.

ISAACSON E., STOKER J.J. and TROESCHI A. (1958): *Numerical Solution of Flow Problems in Rivers*. ASCE Journal of Hyd. Div.

MAHMOOD and YEVJEVICH (1975): *Unsteady Flow in Open Channels*. Fort Collins, Colorado.

PREISSMANN A. (1961): *Propagation des intumescences dans les canaux et rivieres.* Presented at the 1st Congress of the French Association for Computation, Grenoble, France, 1961

SOULSBY, R.L. (1997): *Dynamics of marine sands, a manual for practical applications.* HR Wallingford

Report SR466, HR Wallingford, Oxon, UK.

WESTRICH, B. and JURASHEK, M. (1985): *Flow transport capacity for suspended sediment*. Presented at the 21st Congress IAHR, Melbourne, Australia.