



iRIC Software

*Changing River Science*

---

***FaSTMECH Solver***  
***Manual***

# FaSTMECH Model Notes

**Jonathan M. Nelson**

U.S. Geological Survey, Box 25046, MS413, Lakewood, Colorado, USA 80225

## 1. FLOW CONSERVATION LAWS

### 1.1 Conservation of mass and momentum

In order to discuss modeling approaches and the approximations that lead to the formulations discussed in more detail below, the basic equations governing fluid flow and sediment motion are developed in this and the following section respectively. These equations are the fundamental building blocks of all the flow models, but various models use versions of the full equations that are reduced by neglecting certain terms, or even more commonly, by integrating over one or more dimensions to develop averaged equations. The most important thing to note in going through this exercise is the approximations that are required in order to develop certain methods; these will be explicitly noted in the text as will the physical meaning of the approximations. The first approximation to be used here is that, for all development, the flow will be assumed to be incompressible. This is a good assumption as long as the flow velocities are much less than the speed of sound, a condition that is well satisfied in channel flows. Using this assumption, conservation of mass and momentum for the flow are given by the following:

$$\nabla \cdot \bar{u} = 0 \quad (1)$$

$$\frac{\partial \bar{u}}{\partial t} + \bar{u} \cdot \nabla \bar{u} = \frac{1}{\rho} \nabla P + \bar{g} + \nu \nabla^2 \bar{u} \quad (2)$$

where  $\bar{u}$  is the vector velocity,  $\rho$  is the fluid density,  $P$  is pressure, and  $\nu$  is the fluid kinematic viscosity. These equations describe fluid motion in general; the only assumption made in deriving them is that the fluid is incompressible. In general, solving these equations in this full form in natural flows is difficult and impractical (although improvements in computational speed and algorithms are making direct solution of these equations progressively more attainable). Usually, the equations that are actually used to compute flow solutions are reduced forms of the above equations developed through temporal or spatial averaging, or by scaling the equations to discover which terms are most important and retaining only those terms in the numerical solution.

The primary reason that these equations are difficult to solve for most natural flows is the occurrence of turbulence. With the exception of flows characterized by appropriate combinations of low velocity, small scale, and/or high fluid viscosity (characterized by the Reynolds number, see Tennekes and Lumley, p. 1-26) flows are unstable to perturbations, and are characterized by three-dimensional variability across a wide range of time and length scales. For example, even if one creates a simple channel flow with a smooth bottom, simple rectilinear channel shape, and steady discharge, the velocity at any point in the flow will vary in time for typical length and time scales. In addition to adding substantially to the complexity of the flow, these variations in flow give rise to important momentum

fluxes, changing even the time-averaged character of the flow significantly. To avoid the necessity of computing the variations in flow associated with turbulence, by far the majority of computational flow models used for natural flows use the so-called Reynolds' equations. These equations are developed by splitting the vector velocity into a time mean part (or an ensemble-averaged part) and a time-varying part (or the variation about the ensemble average). For a detailed description of this procedure and the reasoning behind it, the reader is referred to Tennekes and Lumley (p.28-33), or any other beginning text on turbulence. In a cartesian coordinate system, the Reynolds' momentum equations for the  $x$ ,  $y$ , and  $z$  directions are given by:

$$\frac{\partial \bar{u}}{\partial t} + \bar{u} \frac{\partial \bar{u}}{\partial x} + \bar{v} \frac{\partial \bar{u}}{\partial y} + \bar{w} \frac{\partial \bar{u}}{\partial z} = -\frac{1}{\rho} \frac{\partial P}{\partial x} + \nu \nabla^2 \bar{u} - \frac{\partial \overline{u'^2}}{\partial x} - \frac{\partial \overline{u'v'}}{\partial y} - \frac{\partial \overline{u'w'}}{\partial z} \quad (3)$$

$$\frac{\partial \bar{v}}{\partial t} + \bar{u} \frac{\partial \bar{v}}{\partial x} + \bar{v} \frac{\partial \bar{v}}{\partial y} + \bar{w} \frac{\partial \bar{v}}{\partial z} = -\frac{1}{\rho} \frac{\partial P}{\partial y} + \nu \nabla^2 \bar{v} - \frac{\partial \overline{u'v'}}{\partial x} - \frac{\partial \overline{v'^2}}{\partial y} - \frac{\partial \overline{v'w'}}{\partial z} \quad (4)$$

$$\frac{\partial \bar{w}}{\partial t} + \bar{u} \frac{\partial \bar{w}}{\partial x} + \bar{v} \frac{\partial \bar{w}}{\partial y} + \bar{w} \frac{\partial \bar{w}}{\partial z} = -\frac{1}{\rho} \frac{\partial P}{\partial z} - g + \nu \nabla^2 \bar{w} - \frac{\partial \overline{u'w'}}{\partial x} - \frac{\partial \overline{v'w'}}{\partial y} - \frac{\partial \overline{w'^2}}{\partial z} \quad (5)$$

where  $u$ ,  $v$ , and  $w$  are the velocity components in the  $x$ ,  $y$ , and  $z$  directions, and where overbars present time (or ensemble) averages and primes represent deviations from that average (e.g.,  $u = \bar{u} + u'$ ). Strictly speaking, the process of time averaging would result in each of the first terms in the momentum equations being identically zero, but in practice, the time required to compute the average of a turbulent quantity is often less than the time scale associated with externally imposed unsteadiness. For example, in a channel flow with slowly varying discharge, it may be possible to construct a time average over the turbulence using an averaging time much, much smaller than the time over which discharge variations occur. For ensemble averages, where one averages over many realizations of the same flow, the inclusion of the unsteady term in the equations is not problematic. For example, if one makes measurements of velocity in a turbulent wave boundary layer, it is possible to average over many waves to determine the ensemble averaged behavior of the flow; the departure from that average over a specific wave or time series of waves yields the turbulent variability. The last three terms on the right-hand side of the above equations arise as a result of the momentum fluxes due to turbulent fluctuations. These terms are very important for transferring momentum within the flow, especially near boundaries or anywhere strong shears occur in the flow.

Proceeding with the same averaging procedure on the conservation of mass equation yields:

$$\frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} + \frac{\partial \bar{w}}{\partial z} = 0 \quad (6)$$

The original four equations expressing conservation of mass and momentum had four unknowns: the three components of velocity and the pressure. The number of unknowns matched the number of equations, so this was a well-posed problem. However, there are also four Reynolds' averaged mass and momentum equations, but there are more than four unknowns because of the appearance of the momentum fluxes associated with the turbulent fluctuations. This is the so-called closure problem of turbulence.

## 1.2 Reynolds stresses and turbulence closures

The quantities involving time or ensemble averages of products of time varying quantities shown in equations (3) through (5) are referred to as Reynolds' stresses. Although they are called stresses, it is important to remember that these terms arise due to advective transport of momentum. However, because they appear in the Reynolds' averaged momentum equations in a manner analogous to viscous stresses, they are referred to as stresses, and are often parameterized in terms of the mean flow using concepts developed for viscous stresses. Rewriting equations (3), (4) and (5) in terms of the components of the Reynolds stress tensor yields the following:

$$\frac{\partial \bar{u}}{\partial t} + \bar{u} \frac{\partial \bar{u}}{\partial x} + \bar{v} \frac{\partial \bar{u}}{\partial y} + \bar{w} \frac{\partial \bar{u}}{\partial z} = -\frac{1}{\rho} \frac{\partial P}{\partial x} + \nu \nabla^2 \bar{u} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} \quad (7)$$

$$\frac{\partial \bar{v}}{\partial t} + \bar{u} \frac{\partial \bar{v}}{\partial x} + \bar{v} \frac{\partial \bar{v}}{\partial y} + \bar{w} \frac{\partial \bar{v}}{\partial z} = -\frac{1}{\rho} \frac{\partial P}{\partial y} + \nu \nabla^2 \bar{v} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} \quad (8)$$

$$\frac{\partial \bar{w}}{\partial t} + \bar{u} \frac{\partial \bar{w}}{\partial x} + \bar{v} \frac{\partial \bar{w}}{\partial y} + \bar{w} \frac{\partial \bar{w}}{\partial z} = -\frac{1}{\rho} \frac{\partial P}{\partial z} - g + \nu \nabla^2 \bar{w} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} \quad (9)$$

where the Reynolds' stresses are defined as follows:

$$\begin{aligned} \tau_{xx} &= -\overline{\rho u'^2} & \tau_{xz} &= \tau_{zx} = -\overline{\rho u' w'} \\ \tau_{zz} &= -\overline{\rho w'^2} & \tau_{xy} &= \tau_{yx} = -\overline{\rho u' v'} \\ \tau_{yy} &= -\overline{\rho v'^2} & \tau_{yz} &= \tau_{zy} = -\overline{\rho v' w'} \end{aligned} \quad (10)$$

Generally, the Reynolds' stresses are much greater than viscous stresses in natural channel flows, and the viscous stresses are neglected in the momentum equations. Thus, the terms in the above equations involving  $\nu$ , the kinematic viscosity, are negligibly small and are omitted from the equations.

In order to solve the above equations, one must either rewrite the Reynolds' stresses in terms of the mean flow quantities or provide some other manner by which these terms may be evaluated using additional relations. The most common method in simulating natural flows is to relate the Reynolds stresses to the mean flow quantities by analogy with the relation between viscous stress and the rate of strain tensor. This leads to the concept of eddy viscosity, which assumes a proportionality between the Reynolds' stresses and the components of the rate of strain. While there is good justification for this kind of approach in situations where the flow is dominated by one length and velocity scale, as in a simple boundary layer, the concept is generally only a crude approximation for real, complex flows in nature. Nevertheless, many approaches are based on this concept, and there are a number of ways of estimating the spatial structure and values for eddy viscosity using simple dimensional arguments or more complex reasoning. For example,  $k-\varepsilon$  models use the eddy viscosity concept, but evaluate the local eddy viscosity using advection-diffusion equations for the turbulent kinetic energy and the length scale of the turbulence; this allows treatment of situations where the local flow parameters are not accurate predictors of local turbulence structure. There are also a variety of closure approaches that are not predicated on the existence of an eddy viscosity. For example, it is possible to manipulate the momentum equations to develop equations for each of the Reynolds' stresses, however, these equations introduce more unknowns

that must in turn be parameterized or estimated. A more complete discussion of turbulence closure techniques is beyond the scope of this chapter, but the reader is referred to the review by Rodi (1993) for an excellent discussion in simple terms.

If the existence of a scalar, isotropic eddy viscosity,  $K$ , is assumed, the Reynolds' stress terms in equations (7), (8) and (9) may be replaced by the following relations:

$$\begin{aligned}
 \tau_{xx} &\cong 2\rho K \frac{\partial \bar{u}}{\partial x} & \tau_{xz} = \tau_{zx} &\cong \rho K \left[ \frac{\partial \bar{u}}{\partial z} + \frac{\partial \bar{w}}{\partial x} \right] \\
 \tau_{zz} &\cong 2\rho K \frac{\partial \bar{w}}{\partial z} & \tau_{xy} = \tau_{yx} &\cong \rho K \left[ \frac{\partial \bar{u}}{\partial y} + \frac{\partial \bar{v}}{\partial x} \right] \\
 \tau_{yy} &\cong 2\rho K \frac{\partial \bar{v}}{\partial y} & \tau_{yz} = \tau_{zy} &\cong \rho K \left[ \frac{\partial \bar{v}}{\partial z} + \frac{\partial \bar{w}}{\partial y} \right]
 \end{aligned} \tag{11}$$

Substituting the above relations, equations (7), (8), and (9) once again become a closed set of equations, with unknowns consisting of the Reynolds' averaged velocities and pressure. However, in order to solve these equations, an eddy viscosity still needs to be determined. As noted above, there are many ways to do this, but one of the most common is based on extending the well-posed relations for simple steady, uniform boundary layers to more complex flows in channels. This extension is based on the observation that flows in unstratified channels are dominantly boundary-layer-like in character. In simple boundary layers, the local turbulence is well-described by the local boundary shear stress and distance from the boundary. Indeed, this result stems directly from simple dimensional analysis (e.g., Tennekes and Lumley, 1972). This result is complicated slightly when one considers the effect of finite depth, but not by much. Defining the shear velocity in terms of the local boundary shear stress and the fluid density as follows:

$$u_* = \left[ \frac{(\tau_{zx})_B}{\rho} \right]^{1/2} \tag{12}$$

dimensional analysis yields the result that the eddy viscosity,  $K$ , can be written in the following form:

$$K = k u_* h \kappa(\xi) \tag{13}$$

where  $k$  is an empirical constant of proportionality called von Karman's constant ( $\approx 0.407$ , see Long et al., 1993) and  $\kappa(\xi)$  is a shape function giving the vertical distribution of  $K$  between the bed and the water surface, using the definition  $\xi = z/h$ , and where  $h$  is the local flow depth and  $z$  is distance from the boundary. For the choice of a parabolic distribution of eddy viscosity, as given by

$$\kappa(\xi) = \xi(1-\xi) \tag{14}$$

the velocity profile in the boundary layer will be logarithmic, as follows:

$$\bar{u} = \frac{u_*}{k} \ln \frac{z}{z_0} \tag{15}$$

where  $z_0$ , the so-called roughness length, is a constant of integration that depends on the boundary shear stress, the fluid viscosity, and/or the size of the roughness elements on the bed (see Southard and Middleton, 1986, or any beginning text on wall-bounded shear flows for a discussion of roughness lengths). In practice, experimental evidence suggests that equation (14) is not the best choice, although it may be quite accurate close to the boundary. While there are several other possibilities suggested in the literature, there is not much evidence to suggest that more complicated structure functions are verifiably better than simply using equation (14) from the bed up to one fifth of the flow depth, and using a constant value above that level, i.e.

$$\begin{aligned} \kappa(\xi) &= \xi(1-\xi) & \xi < 0.2 \\ \kappa(\xi) &= 0.16 & \xi \geq 0.2 \end{aligned} \quad (16)$$

This choice for  $\kappa$  yields a logarithmic velocity profile near the bed, and a parabolic one well away from the bed.

In applying models that use the simple eddy viscosity closure described above, it is absolutely critical to note that this form of the eddy viscosity can only be strictly correct in a steady, uniform boundary layer. While natural rivers and streams are predominantly boundary layer-like in nature, they often contain components of both free shear layers and wakes. One immediate shortcoming of the model above is that it predicts zero flux of momentum due to turbulence in regions where the boundary shear stress is zero. In a simple shear layer bounding a separation zone in a river, this suggests that, as the boundary shear stress must change sign somewhere in the region of between upstream and downstream flow, there must be a surface across which no momentum is transferred by turbulence. Obviously, this is wrong; if these effects are important, a different closure must be employed. Nevertheless, these simple closures perform adequately in a wide variety of natural flows. The most important point here is that, when using a closure of a certain type, one must keep in mind the potential errors in that closure, and always bear in mind what physical processes are likely to be well treated and what processes are likely to be poorly treated. Understanding the detailed implications of the assumptions that go into building any flow model is critical for deciding how appropriate that model may be for a given physical situation.

### 1.3 Hydrostatic assumption

Up to this point, each of the three components of velocity have been treated equally and the terms in the momentum equations for  $u$ ,  $v$ , and  $w$  have been treated in the same manner. However, in many flows of interest, both vertical velocities and vertical accelerations are low, and the vertical equation of motion (5 or 9) can be accurately approximated by the retaining only the pressure gradient and stress terms:

$$-\frac{1}{\rho} \frac{\partial P}{\partial z} - g = 0 \quad (17)$$

This assumption is referred to as the hydrostatic assumption, as it results in the pressure being distributed hydrostatically in the vertical, meaning that the pressure is equivalent to the overlying weight of fluid per unit area at any point. This simplification is a good one provided vertical accelerations are small, meaning that bed slopes are relatively small along the direction of the flow. For flows with strong vertical acceleration produced by abrupt bed variations (as may be caused by bedrock, or man-made structures), this assumption may not be accurate.

In situations where equation (17) is a suitable approximation for equation (9), the pressure gradients in the horizontal equations of motion can be written in terms of the water surface elevation by integrating equation (17) in  $z$  and differentiating the result in each of the horizontal directions to obtain:

$$-\frac{1}{\rho} \frac{\partial P}{\partial x} = -g \frac{\partial E}{\partial x} \quad (18)$$

$$-\frac{1}{\rho} \frac{\partial P}{\partial y} = -g \frac{\partial E}{\partial y} \quad (19)$$

These relations simplify solution of the equations, because they reduce determining the pressure at each  $(x,y,z)$  location in the flow to only determining the water surface elevation at each horizontal  $(x,y)$  location.

## 1.4 Coordinate systems

All the equations above have been cast in a simple cartesian coordinate system. In practice, flow solutions are computed on a wide variety of coordinate systems, including cartesian, orthogonal curvilinear, and general coordinate systems for finite difference solutions and a variety of structured and unstructured grids for finite element solutions. The primary advantage of general or unstructured grids is that they allow the coordinate system to be

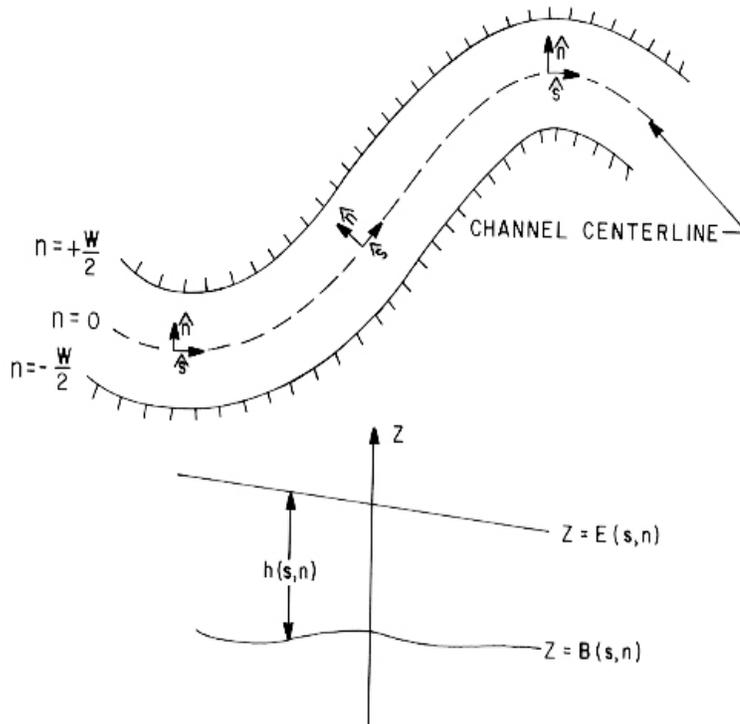


Figure 1. Schematic of the channel-fitted coordinate system. fitted precisely to the flow domain. The disadvantage is that they increase computational complexity considerably, and in cases where the bed and banks of the channel are evolving in time, the coordinate system must be recomputed at every time step, which is time consuming. In addition, most finite element

solutions conserve mass only in a global sense; they typically are poor at enforcing mass conservation locally (Oliveira et al., 2000). This problem can be mitigated by careful construction of the flow grid, but it is difficult to avoid entirely, especially in channels with strong spatial accelerations produced by topography or channel curvature. Oliveira et al. (2000) found errors in local mass conservation of up to 85% after only 3 days of simulation applying standard finite element methods to the Tagus Estuary. In channel flows, errors of this magnitude result in solutions that are not good representations of the real flow, and certainly could not be used to accurately compute the movement of sediment or other constituents within the flow.

Developing a variety of commonly used coordinate systems is not within the scope of this model description, but it is worth mentioning one specific orthogonal curvilinear system that has been widely used in modeling river flows. This coordinate system is essentially a generalization of a cylindrical coordinate system where the curvature of the coordinate system is allowed to vary in the streamwise direction. This so-called “channel-fitted” coordinate system has been used widely over the last 50 years or so, although most early applications involved only an incomplete set of equations. The system was formally derived and the full equations were published by Smith and McLean (1984). If the radius of curvature of the channel centerline is defined as  $R$  and  $s$ ,  $n$  and  $z$  are defined as the streamwise, cross-stream, and vertical coordinates, respectively, as shown in figure (1), the hydrostatic assumption is employed, and the viscous stresses are assumed to be negligibly small, the momentum equations in this coordinate system are given by the following:

$$\frac{1}{1-N} \frac{\partial u}{\partial s} - \frac{v}{(1-N)R} + \frac{\partial v}{\partial n} + \frac{\partial w}{\partial z} = 0 \quad (20)$$

$$\begin{aligned} \frac{\partial u}{\partial t} + \frac{u}{(1-N)} \frac{\partial u}{\partial s} + v \frac{\partial u}{\partial n} + w \frac{\partial u}{\partial z} - \frac{uv}{(1-N)R} = \frac{-g}{1-N} \frac{\partial E}{\partial s} + \\ \frac{1}{\rho} \left[ \frac{1}{1-N} \frac{\partial \tau_{ss}}{\partial s} + \frac{\partial \tau_{ns}}{\partial n} + \frac{\partial \tau_{zs}}{\partial z} - \frac{2\tau_{ns}}{(1-N)R} \right] \end{aligned} \quad (21)$$

$$\begin{aligned} \frac{\partial v}{\partial t} + \frac{u}{(1-N)} \frac{\partial v}{\partial s} + v \frac{\partial v}{\partial n} + w \frac{\partial v}{\partial z} + \frac{u^2}{(1-N)R} = \frac{-g}{1-N} \frac{\partial E}{\partial n} + \\ \frac{1}{\rho} \left[ \frac{1}{1-N} \frac{\partial \tau_{ns}}{\partial s} + \frac{\partial \tau_{nn}}{\partial n} + \frac{\partial \tau_{zn}}{\partial z} + \frac{\tau_{ss} - \tau_{nn}}{(1-N)R} \right] \end{aligned} \quad (22)$$

$$-\frac{1}{\rho} \frac{\partial P}{\partial z} - g = 0. \quad (23)$$

If the existence of a scalar, isotropic eddy viscosity is assumed, we can rewrite equation (11) in the channel-fitted coordinate system, resulting in the following expressions for the six independent components of the deviatoric Reynolds’ stress tensor:

$$\begin{aligned}
\tau_{ss} &= 2\rho K \left[ \frac{1}{1-N} \frac{\partial u}{\partial s} - \frac{v}{(1-N)R} \right] & \tau_{nn} &= 2\rho K \left[ \frac{\partial v}{\partial n} \right] \\
\tau_{ns} &= \rho K \left[ \frac{1}{1-N} \frac{\partial v}{\partial s} + \frac{u}{(1-N)R} + \frac{\partial u}{\partial n} \right] & \tau_{zn} &= \rho K \left[ \frac{\partial w}{\partial n} + \frac{\partial v}{\partial z} \right] \\
\tau_{zs} &= \rho K \left[ \frac{1}{1-N} \frac{\partial w}{\partial s} - \frac{\partial u}{\partial z} \right] & \tau_{zz} &= 2\rho K \left[ \frac{\partial w}{\partial z} \right]
\end{aligned} \tag{24}$$

where the overbars denoting Reynolds' averaging of the equations have been omitted for simplicity. Note that, if the radius of curvature of the channel centerline goes to infinity, meaning that the channel is straight, equations (20) through (24) revert back to the standard momentum equations with  $x$  and  $y$  oriented streamwise and cross-stream, respectively. However, if the channel is curved, the  $u$  and  $v$  velocity components in the  $s$ - $n$ - $z$  coordinate system still correspond to streamwise and cross-stream velocities, as the  $s$ -direction is always streamwise. Clearly, this would not be true if a cartesian system were used; the orientation of the  $x$  and  $y$  components of velocity with respect to the channel would change with position. Thus, the channel-fitted coordinate system is in some sense the natural one, as it divides local velocities vectors into streamwise and cross-stream components. This system is also the one typically used in analyzing field measurements in channels, because those measurements are frequently taken perpendicular to and parallel to sections that are themselves perpendicular to the channel centerline, or at least as close to perpendicular as practically possible.

The first, and perhaps most confusing, step in applying the channel-fitted coordinate system is determining the channel centerline and the radius of curvature of that centerline. This is not a purely mathematical process; it requires some consideration of what one is trying to capture in the channel-fitted coordinate system. As long as the numerics are correct and the full equations are used, the flow solution should be essentially independent of the coordinate system. Thus, one could use a cartesian coordinate system for a curved channel, or even a curved coordinate system for a straight channel. However, if one chooses a coordinate system that follows the path of the channel, at least approximately, two advantages arise: first, the number of grid points required is minimized and second, the convective accelerations associated with the curvature of the channel appear primarily in centripetal acceleration terms, rather than in differential terms in the governing equations. This latter consideration is the key to choosing the channel centerline for the coordinate system. Basically, one wants to find a centerline that captures the average curvature of the flow streamlines, which are approximately the same as the large-scale curvature of the banks. As the flow "averages" the effects of the local banks over a length scale comparable to width, one can digitize a centerline for the coordinate system (which need not correspond exactly to the channel centerline) in two ways. Either the centerline can be digitized with points that are closer together than the channel width, and then filter the resulting curve over distances of about a channel width, or one may simply choose a number of points, each about a channel width apart. In either case, the radius of curvature is easily found by noting that, if  $\theta$  is the angle between the down valley direction and the local tangent to the centerline, the radius of curvature is given by:

$$R = \left[ \frac{\partial \theta}{\partial s} \right]^{-1} \tag{25}$$

The important thing to keep in mind when generating a channel-fitted coordinate system is that the centerline defining the coordinate system should be drawn to approximate the average streamline curvature in the reach of interest as well as possible. It is not appropriate to take a precise channel centerline defined by a detailed (i.e., with spatial resolution much smaller than a channel width) survey of

the banks, as the resulting detailed centerline may have local curvature values that are very poor approximations to the average streamline curvature.

## 2. Two-Dimensional Model Equations

### 2.1 Spatial averaging

In many cases, solution of the full momentum equations is not warranted by either the nature of the questions to be addressed in a given study, or as a result of the kind and amount of data available. For example, applying a three-dimensional model to several hundred channel widths of a given river for a study of floodplain inundation when cross-sections of bathymetry are only available every 10 channel widths is not reasonable, because getting good results with a three dimensional model would require much more topographic data. Generally, more complete models that yield more precise results require much more input information in order to be applied relative to simpler models. In many cases, accurate results for a given purpose can be found using a simple model with relatively sparse topographic data. The two most common ways of developing simpler models are scaling analyses and spatial averaging. Scaling analyses simply refers to the concept of using the time and length scales of the flow to discover what terms in the momentum equations are likely to be most important, and to develop simpler equations by retaining only these important terms. This is a powerful tool for certain flows, but it generally results in a model that is specifically applicable only to a certain flow or class of flows. Spatial averaging is a method whereby one or more dimensions are removed from the model equations by integrating or averaging over those dimensions. For example, development of a 1-d flow model requires averaging the momentum equations over a channel cross-section, so that instead of solving for the velocity at every point in the channel, the model solves only for the cross-sectionally averaged velocity at each model cross-section. Note that while model simplicity is gained by spatial averaging, detail is lost.

The most common applications of spatial averaging result in one-dimensional models, two-dimensional models that treat the channel flow in planform (vertically averaged models), and two-dimensional models that treat the flow in the streamwise-vertical plane (cross-stream averaged models). While treating each of these in any detail is beyond the scope of this introduction to modeling flow and sediment transport, a single example is illustrative of some of the issues that arise in developing spatially averaged equations. Using  $\langle \rangle$  to represent vertical averaging, the vertical average of the  $u$  velocity component is defined as follows:

$$\langle u \rangle = \frac{1}{h} \int_B^E u \, dz \quad (26)$$

Applying this same operator to equations (20), (21), and (22), the following vertically averaged continuity and horizontal momentum equations arise in the channel-fitted coordinate system (again, note that the standard cartesian relations are easily found from the following by letting  $R$  go to infinity):

$$\frac{1}{1-N} \frac{\partial}{\partial s} (\langle u \rangle h) - \frac{\langle v \rangle h}{(1-N)R} + \frac{\partial}{\partial n} (\langle v \rangle h) = 0 \quad (27)$$

$$\frac{1}{1-N} \frac{\partial}{\partial s} (\langle u^2 \rangle h) + \frac{\partial}{\partial n} (\langle uv \rangle h) - \frac{2 \langle uv \rangle h}{(1-N)R} = -\frac{gh}{1-N} \frac{\partial E}{\partial s} +$$

$$\frac{1}{\rho} \left[ \frac{1}{1-N} \frac{\partial}{\partial s} (\langle \tau_{ss} \rangle h) + \frac{\partial}{\partial n} (\langle \tau_{ns} \rangle h) - \frac{2 \langle \tau_{ns} \rangle h}{(1-N)R} \right] + \quad (28)$$

$$\frac{1}{\rho} \left[ \frac{1}{1-N} (\tau_{ss})_B \frac{\partial B}{\partial s} + (\tau_{ns})_B \frac{\partial B}{\partial n} - (\tau_{zs})_B \right]$$

$$\frac{1}{1-N} \frac{\partial}{\partial s} (\langle uv \rangle h) + \frac{\partial}{\partial n} (\langle v^2 \rangle h) + \frac{(\langle u^2 \rangle - \langle v^2 \rangle)h}{(1-N)R} = -\frac{gh}{1-N} \frac{\partial E}{\partial n} +$$

$$\frac{1}{\rho} \left[ \frac{1}{1-N} \frac{\partial}{\partial s} (\langle \tau_{ns} \rangle h) + \frac{\partial}{\partial n} (\langle \tau_{nm} \rangle h) - \frac{\langle \tau_{ss} - \tau_{nm} \rangle h}{(1-N)R} \right] + \quad (29)$$

$$\frac{1}{\rho} \left[ \frac{1}{1-N} (\tau_{ns})_B \frac{\partial B}{\partial s} + (\tau_{nm})_B \frac{\partial B}{\partial n} - (\tau_{zn})_B \right]$$

These equations, which have been used in variety of models for flow and bed evolution (Smith and McLean, 1984; Nelson and Smith, 1989a,b; Shimizu et al.; 1991), introduce a new kind of closure problem that is analogous to the turbulence closure problem introduced by Reynolds' averaging. Terms that arise due to vertical correlations such as  $\langle uv \rangle$ ,  $\langle u^2 \rangle$  and  $\langle v^2 \rangle$  cannot be expressed in terms of simple vertically averaged variables like  $\langle u \rangle$  and  $\langle v \rangle$  except in the simple case where the velocities have no vertical structure whatsoever, so that  $\langle uv \rangle = \langle u \rangle \langle v \rangle$  and  $\langle u^2 \rangle = \langle u \rangle^2$ , and so forth. However, this is not generally true. For example, for a logarithmic velocity profile, the difference between  $\langle u^2 \rangle$  and  $\langle u \rangle^2$  depends on the ratio of the roughness length to the flow depth, and is typically on the order of five or ten percent. In almost all vertically averaged models, the correlations are neglected, and one assumes that the equalities that hold for the case of no vertical structure are accurate in cases with vertical structure. However, some important effects are excluded when this assumption is used. For example, in long meander bends with weak topography, the term  $\langle uv \rangle$  has been shown to be at least partially responsible for the movement of the high velocity region of the flow from the inner bank at the upstream part of the bend to the outer bank at the downstream part of the bend (Shimizu et al., 1991). This is because helical cross-stream flow moves high velocity fluid outward near the surface of the flow, and low velocity fluid inward near the bed, resulting in a net momentum flux toward the outer bank. This effect is overwhelmed by topographic steering of the flow in bends shorter bends with point bars, but it is potentially an important effect in some natural flows. Even though this effect is dependent on vertical structure, it can be treated to some degree in vertically averaged models using dispersion coefficients. Similarly, in any case where spatial averaging is carried out, spatial correlations between variables in the dimensions averaged over can be treated at least at some approximate level.

## 2.2 Dispersion coefficients

A general definition of a dispersion coefficient between two variables is given by the following

$$\alpha_{ab} = \frac{\langle ab \rangle}{\langle a \rangle \langle b \rangle} \quad (30)$$

where  $\langle \rangle$  may represent vertical averaging or some other spatial average (e.g., cross-sectional). Using this definition, equations (28) and (29) may be rewritten in terms of only  $\langle u \rangle$  and  $\langle v \rangle$  along with the dispersion coefficients  $\alpha_{uu}$ ,  $\alpha_{vv}$  and  $\alpha_{uv}$ . The values of these coefficients may be set from theoretical considerations, or through measurement. In either case, the coefficients allow at least approximate treatment of momentum fluxes that would otherwise be neglected. Another way to treat the correlation terms in averaged equations is to separate each variable into an averaged part and a deviation from that average, in parallel to the development of the Reynolds' momentum equations. For example, if we use primes to denote departures from the vertical average, such that we can write  $u(z) = \langle u \rangle + u'(z)$ , for example, we can rewrite equations (28) and (29) as follows:

$$\begin{aligned} \frac{1}{1-N} \frac{\partial}{\partial s} (\langle u \rangle^2 h) + \frac{\partial}{\partial n} (\langle u \rangle \langle v \rangle h) - \frac{2 \langle u \rangle \langle v \rangle h}{(1-N)R} + F' = -\frac{gh}{1-N} \frac{\partial E}{\partial s} + \\ \frac{1}{\rho} \left[ \frac{1}{1-N} \frac{\partial}{\partial s} (\langle \tau_{ss} \rangle h) + \frac{\partial}{\partial n} (\langle \tau_{ns} \rangle h) - \frac{2 \langle \tau_{ns} \rangle h}{(1-N)R} \right] + \\ \frac{1}{\rho} \left[ \frac{1}{1-N} (\tau_{ss})_B \frac{\partial B}{\partial s} + (\tau_{ns})_B \frac{\partial B}{\partial n} - (\tau_{zs})_B \right] \end{aligned} \quad (31)$$

$$\begin{aligned} \frac{1}{1-N} \frac{\partial}{\partial s} (\langle u \rangle \langle v \rangle h) + \frac{\partial}{\partial n} (\langle v \rangle^2 h) + \frac{(\langle u \rangle^2 - \langle v \rangle^2) h}{(1-N)R} + G' = -\frac{gh}{1-N} \frac{\partial E}{\partial n} + \\ \frac{1}{\rho} \left[ \frac{1}{1-N} \frac{\partial}{\partial s} (\langle \tau_{ns} \rangle h) + \frac{\partial}{\partial n} (\langle \tau_{mn} \rangle h) - \frac{\langle \tau_{ss} - \tau_{nn} \rangle h}{(1-N)R} \right] + \\ \frac{1}{\rho} \left[ \frac{1}{1-N} (\tau_{ns})_B \frac{\partial B}{\partial s} + (\tau_{mn})_B \frac{\partial B}{\partial n} - (\tau_{zn})_B \right] \end{aligned} \quad (32)$$

where the new terms are defined by

$$F' = \frac{1}{1-N} \frac{\partial}{\partial s} (\langle u'^2 \rangle h) + \frac{\partial}{\partial n} (\langle u'v' \rangle h) - \frac{2 \langle u'v' \rangle h}{(1-N)R} \quad (33)$$

and

$$G' = \frac{1}{1-N} \frac{\partial}{\partial s} (\langle u'v' \rangle h) + \frac{\partial}{\partial n} (\langle v'^2 \rangle h) + \frac{(\langle u'^2 \rangle - \langle v'^2 \rangle) h}{(1-N)R} \quad (34)$$

In cases where simple structure functions can be supplied for  $u$  and  $v$  based on measurements or theoretical arguments, these "extra" terms arising from correlations can be approximately evaluated. If these terms are set to zero, it is important to have an understanding of what kinds of processes are being

neglected in the formulation. Situations where spatial correlations are important can often be treated without solving the full equations.

### 2.3 Bed stress closure

Whenever the equations of motion are averaged in the direction perpendicular to a boundary, closures for stress terms at that boundary must be supplied. In the vertically averaged equations used as an example, the boundary shear stress terms that arise in the horizontal momentum equations must be expressed in terms of  $\langle u \rangle$  and  $\langle v \rangle$ . There are many ways to do this, including using Manning's or Chezy's closure, as discussed below, but the most common in multidimensional models is to use a drag coefficient closure:

$$\tau_B = \rho C_d (u^2 + v^2) \quad (35)$$

Splitting this into component parts yields:

$$(\tau_{zs})_B = \rho C_d \sqrt{\langle u \rangle^2 + \langle v \rangle^2} \langle u \rangle \quad (36)$$

and

$$(\tau_{zn})_B = \rho C_d \sqrt{\langle u \rangle^2 + \langle v \rangle^2} \langle v \rangle \quad (37)$$

There are many other choices of bottom stress closure, but most can be directly related to this one. For example, if the flow can be assumed to have a vertical structure as follows:

$$u = u_* f(z, z_0) \quad (38)$$

then the drag coefficient can be shown to be a function only of flow depth and  $z_0$ :

$$C_d = \left[ \frac{1}{h} \int_{z_0}^h f(z, z_0) dz \right]^{-2} \quad (39)$$

Closures for lateral shear stresses at banks can be handled in the same manner. Using this closure or others that are similar, the vertically averaged horizontal momentum equations and the continuity equation can be written entirely in terms of the vertically averaged  $u$  and  $v$  velocity components and the water surface elevation (if the flow is assumed hydrostatic). This is a well-posed system of equations and unknowns, so a solution is straightforward. In general, and despite the fact that these assumptions are often not explicitly stated, any model developed from spatial averaging the full equations requires specification of dispersion coefficients and closures for stresses at boundaries.

## 3. NUMERICAL METHODS

A full discussion of the various numerical methods used in computing flow, sediment transport, and bed evolution would be difficult to cover in a book, much less a short set of workshop notes. With the view in mind that the intent of this book is to provide an overview of the modeling interface, not tools in computational fluid mechanics, the subject of numerical techniques will be given short shrift herein. Nevertheless, this is an important part of constructing coupled models for predicting channel behavior, and particular care must be taken in choosing algorithms. There are two primary issues, somewhat related, that require special attention in choosing algorithms: stability and numerical dispersion.

Stability, or more precisely the lack of it, is easy to observe in model results. Poorly designed algorithms for computing flow and/or bed evolution will lead to unrealistic results that rapidly become more unrealistic as one iterates toward a steady solution or steps the model forward in time for unsteady solutions. Stability considerations for flow computations alone are generally outlined by the author of the flow computation method. Stability considerations for coupled flow/sediment/bed models are altogether more subtle and depend on a number of considerations. First, the time step of bed evolution must be chosen such that bed evolution is slow relative to the time scales associated with the flow field, as this is really the basic premise of the whole approach. If large changes in the bed and/or bank geometry occur within a single flow time step, the solution is almost certain to be unstable. Second, the numerical techniques must be chosen such that artificial phase lags between flow and sediment parameters are not introduced. This may seem complicated, but really it is simple common sense, especially if one considers the following example. If a simple one-dimensional model is used on a low-Froude number flow through a simple channel constriction, the cross-sectionally averaged velocity (which is all one computes in a true 1-D channel model) will be maximum at the constriction. If that velocity is used to compute bedload sediment transport, for example, the bedload transport will also be a maximum at the constriction, assuming normal relations between velocity, bed stress, and sediment flux. As the flux is maximum at the constriction, one expects that the spatial gradient in sediment flux is zero at that point. Since the spatial gradient of the flux is directly related to erosion and deposition, the constriction will neither expand or contract further. However, noting that the flux must be less than the value at the constriction both upstream and downstream of the constriction, a paradox arises. If the spatial gradient in the flux is computed at the crest using the value at the crest and the one immediately upstream, erosion is predicted to occur at the constriction, and if the value at the constriction and the value immediately downstream is used, deposition is predicted to occur at the constriction- both results are wrong, and will lead to runaway expansion or contraction of the constriction. This can be dealt with in a number of simple ways, but the example shows how phase lags introduced between the flow and sediment transport parameters can lead to instabilities in the bed that are not real. Numerical methods must be chosen to avoid artificial instability of the flow field as well as the coupled flow/bed/sediment system.

Excessive numerical dispersion is typically not as obvious to the user as stability problems are. One of the important physical elements of modeling flow and sediment transport is the treatment of the movement of mass and momentum due to true diffusion or advective processes that can be treated as diffusion-like (notably the transfer of momentum and mass by turbulence). Although a detailed mathematical discussion of this topic is clearly outside the scope of this chapter, one of the basic problems of treating continuous systems with discretized equations is that typically some artificial transfer of mass and/or momentum can occur as a result of the discretization process. This is typically referred to as numerical dispersion or numerical viscosity. The magnitude of these effects are strongly dependent on the numerical scheme chosen and the actual numerical grid. Ideally, one would like numerical dispersion to be vanishingly small relative to the real processes of dispersion that one is trying to treat in the numerical solution, thereby ensuring that the model results are consistent with real world observations. Unfortunately, numerical dispersion has an added benefit for models that tend to be unstable in that it effectively increases the stability of the model solutions. Accordingly, it is not unusual to see model results where the values of diffusivities are an order of magnitude or more larger than real world values. In fact, it is not uncommon to see diffusivities (especially lateral diffusivities in two or three-dimensional flow models) assigned unrealistically high values strictly to provide model stability. These models produce artificially smooth distributions of velocity and stress, and generally cannot provide accurate

predictions of sediment flux or bed morphology. The hallmarks of this kind of approach for two or three-dimensional models are separation eddies that are very short relative to real world values, rapid spreading of shear layers in the streamwise direction, and near-bank shears that are low relative to observations. Typically, models with very large values of numerical dispersion show insensitivity to the parameters of the model governing momentum exchange (e.g., drag coefficient, Manning's N, turbulent diffusivity). Models with unrealistically high values of diffusivity often demonstrate inability to produce stable solutions for realistic values.

Although the problems of stability issues and numerical dispersion are especially important in coupled models for flow, sediment transport, and bed evolution, there are many other considerations to be made in developing numerical techniques for such approaches. Fortunately, there are many excellent texts on this subject; for specific examples of different numerical solution techniques, the reader is referred to excellent texts by Patankar (1980) and others. Furthermore, for well-written algorithmic elements that are useful in a variety of different approaches (e.g., tridiagonal solvers, matrix inverters, alternating direction implicit solvers, mesh generators, etc), the reader is encouraged to explore Numerical Recipes (Press et al., 1986), IMSL, MatLab, etc.

## 4. TWO-DIMENSIONAL MODELS

In many cases, one-dimensional models may efficiently represent large-scale flow and sediment transport processes. However, if specific questions about at-a-point flow, sediment transport, and erosion and deposition must be answered, at least a two-dimensional model is required. For example, if the questions to be addressed are related to the position and amplitude of bars within the channel reach of interest, generally a two-dimensional model is necessary, as a one-dimensional model cannot predict the local flow and transport structure that gives rise to bar evolution. Generally, if the flow field of interest includes steering of the flow around islands or bars, or if there is significant cross-stream variability in the flow, at least a two-dimensional model should be applied to predict local sediment transport or changes in bed morphology. Notably, in some cases where one-dimensional models yield cross-sectionally averaged velocities that are incapable of entraining sediment, two-dimensional computations will show a high-velocity thalweg in the channel in which sediment is in motion. Obviously, in situations like this, a two-dimensional approach must be used to get accurate predictions of sediment flux and bed evolution.

### 4.1 Two-dimensional processes

While the preceding paragraph gives some idea as to what one gains in going to a two-dimensional approach, it is worth restating this in terms of specific physical processes. In going from a one-dimensional to a two-dimensional model, three critical improvements are gained. First, instead of predicting only the cross-sectionally averaged component of downstream velocity and bed stress, the model predicts the value of vertically averaged downstream velocity and bed stress at several points across the channel. This means that the model can explicitly treat situations with large cross-stream velocity gradients and even flow separation, even when computing sediment transport. Second, the model also predicts the cross-stream components of vertically averaged velocity and bed stress at each point in the computational grid. As already noted above, this means that a two-dimensional model can handle steering of the flow around bars and islands. This capability is critically important for prediction of the evolution and stability of bars in rivers, as the basic instability leading to these is often associated with the interaction of topographic steering of the flow and the sediment transport (Nelson and Smith, 1989b). Finally, two-dimensional models allow prediction of cross-stream structure in the water surface elevation, while one-dimensional approaches do not. In many cases, superelevation of the water surface due to channel curvature or bathymetric variability results in cross-stream gradients in water surface elevation

that are much larger than downstream components, so if accurate local water edge elevation are required, at least a two-dimensional model must be applied.

These three basic enhancements have several corollaries. Because it yields spatially localized quantities, a two-dimensional approach can be used to predict near-bank velocities, stresses, and sediment evacuation rates, which may be critical for predicting bank erosion. In addition, because these approaches detailed information in a planform sense, they are useful for evaluating fields other than simply sediment transport and bed evolution. For example, two-dimensional modeling approaches are currently becoming the standard for habitat modeling. Habitat evaluation for many riparian species typically requires physical variables including vertically averaged velocity, depth, substrate, and so forth. A one-dimensional model could only evaluate habitat on a cross-sectional basis, which is not sufficient in most cases, as streamwise variations are typically unimportant relative to lateral ones.

## 4.2 Two-dimensional flow models

There are a wide variety of steady and unsteady two-dimensional flow models available. As the equations expressing conservation of mass and momentum in a vertically averaged flow were developed in the discussion of spatial averaging in section 3, they will not be repeated here. The available models are more or less evenly split between finite difference and finite element solutions, with the advantages/disadvantages already discussed above. Readily available government models include FESWMS (available at [www.usgs.gov/software](http://www.usgs.gov/software)) and HEC2D (available at [www.hec.usace.army.mil](http://www.hec.usace.army.mil)) among others. There are also a variety of commercial two-dimensional modeling packages available, including SMS and certain versions of MIKE.

As noted above, many two-dimensional mobile bed models use steady flow solutions, but can handle hydrographs by varying the discharge in time without including the unsteady term in the momentum equations (this is called assuming the flow is “quasi-steady”). This is a reasonable assumption only if the unsteady term in the momentum equation can be shown to be small relative to the other terms in the equation. Although there are quite a few two-dimensional flow models available for use, there are still only a few coupled flow, sediment transport, and bed evolution models.

## 5.0 REFERENCES

Long, C.E., Wiberg, P.L., and Nowell, A.R.M., 1993, Evaluation of von Karman’s constant from integral flow parameters, *J. Hyd. Eng., ASCE*, 119(10), 1182-1190.

Middleton, G.V., and Southard, J.B., 1984, *Mechanics of Sediment Movement*, SEPM, Tulsa.

Nelson, J.M., and Smith, J.D., 1989a, Flow in meandering channels with natural topography. In: S. Ikeda and G. Parker (eds), *River Meandering*, AGU Water Resources Monograph 12, Washington, D.C., 69-102.

Nelson, J.M., and Smith, J.D., 1989b, Evolution and stability of erodible channel beds. In: S. Ikeda and G. Parker (eds), *River Meandering*, AGU Water Resources Monograph 12, Washington, D.C., 321-377.

Oliveira, A., Fortunato, A.B., and Baptista, A.M., Mass balance in Eulerian-Lagrangian transport simulations in estuaries, *J. Hyd. Eng., ASCE*, 126(8), 605-614.

Patankar, S.V., 1980, *Numerical Heat Transfer and Fluid Flow*, Hemisphere, Washington, D.C

Press, W.H., Flannery, B.P., Teukolsky, S.A., and Vetterling, W.T., 1986, Numerical Recipes: The Art of Scientific Computing, Cambridge Press, New York.

Rodi, W., 1993, Turbulence models and their application in hydraulics- A state of the art review, 3rd ed., International Association for Hydraulic Research, Delft, 104 pp.

Shimizu, Y., Yamaguchi, H., and Itakura, T., 1991, Three-dimensional computation of flow and bed deformation, J. Hydr. Eng., ASCE, 116(9), 1090-1108.

Smith, J.D., and McLean, S.R., 1984, A model for flow in meandering streams, Water Resources Res., 20(9), 1301-1315.

Tennekes, H., and Lumley, J.L., A First Course in Turbulence, 1972, MIT Press, Cambridge, MA, 300pp.

## 6. CALCULATION CONDITIONS

A number of boundary conditions, initial conditions, numeric parameters, and other variables must be specified before a simulation can be run using the iRIC application. Typically these are specified once the grid has been generated and elevation has been specified for all locations. To access the dialog to specify calculation conditions select Calculation Conditions→Setting from the Menu Bar.

In this section all calculation conditions required by the Fastmech solver will be described in the order they appear in the Setting dialog box in the iRIC interface (Fig a. showing Calculation Condition Setting dialog). The left window of the dialog displays the primary parameter groups that can be set using the FaSTMECH solver. Values for specific parameters are entered in the fields on the right side of the window. For each parameter group and each field we will describe what is required and provide guidelines for selecting appropriate values.

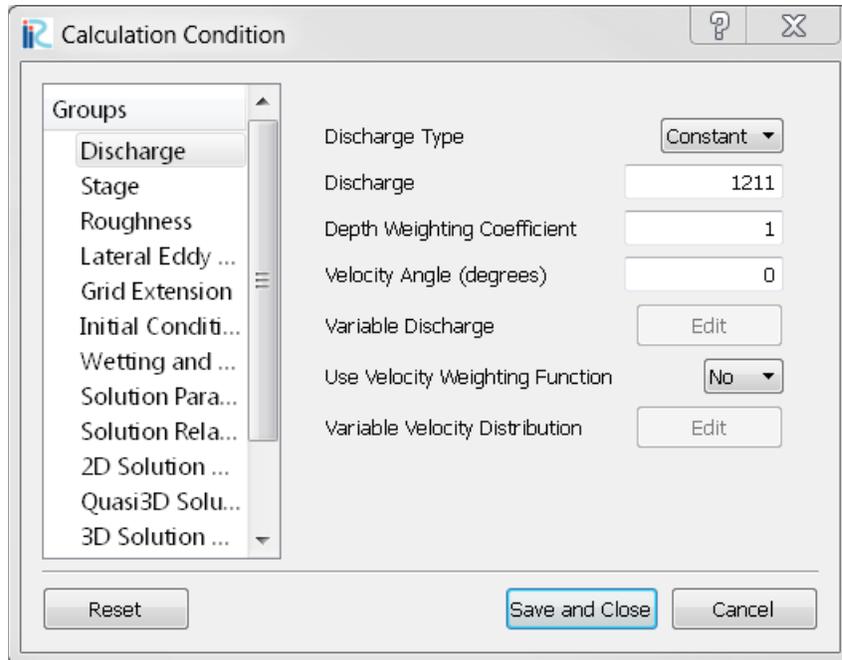


Figure A. Example of the Calculation Condition dialog.

## 6.1. Discharge

Figure A shows the fields associated with specifying the discharge entering the upstream boundary of the simulated reach. This discharge is applied to the areas of the grid that are wet based on the initial water-surface conditions specified in the Initial Conditions tab.

### Discharge Type

Select either Constant or Variable from the drop down menu. Constant applies the same discharge for each model iteration. Variable allows you to specify discharge through time. When variable is selected, the Variable Discharge Edit button becomes enabled allowing you to create a time-series or import a file.

### Discharge

Enter the river discharge in cubic meters per second,  $m^3/s$ .

### Depth Weighting Coefficient

The discharge is distributed by a power of the depth according to this coefficient. The upstream boundary condition at each node is calculated as a function of the weighted discharge.

### Velocity Angle

The velocity angle specifies a uniform angle for velocity at each node on the upstream boundary relative to the stream-wise direction defined by the grid orientation. The default value is 0 which directs velocity parallel to the stream-wise direction of the grid.

Ideally the grid is constructed such that no adjustment to the velocity angle is necessary, but this may not be possible in all cases. To adjust the angle, use the following:

$0 < \text{Velocity Angle} \leq 90$  directs velocity towards river left  
 $-90 \leq \text{Velocity Angle} < 0$  directs velocity towards river right.

### Variable Discharge

When Discharge Type is set to variable the edit button to the right of this field becomes enabled and opens a dialog box that allows you to create a discharge time-series, or import a time-series file (Figure B). Time is specified in seconds and discharge is  $\text{m}^3/\text{s}$ . Import files should contain the same information and be saved with the .txt or .csv extension. The information here would be stored in the file as

0,100  
30,500  
45,500  
62,500

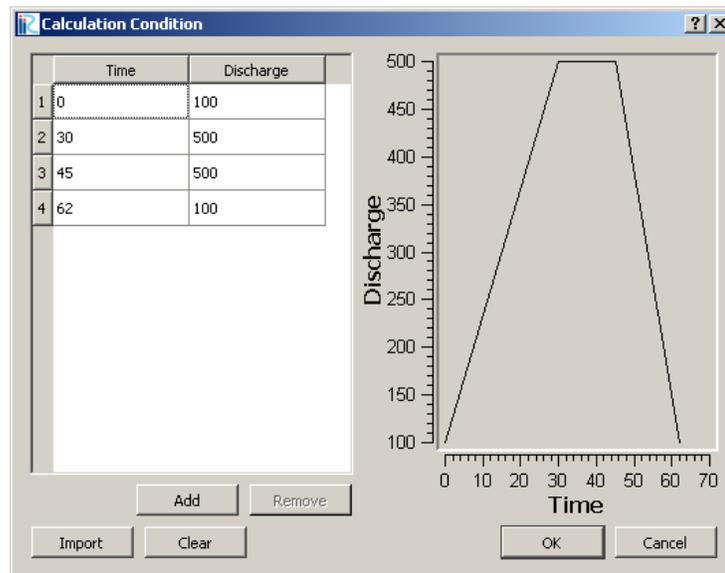


Figure B. Variable Discharge dialog.

### Use Velocity Weighting Function

This option allows you to specify the magnitude of velocity by distance along the upstream boundary. This data is normalized and velocity distribution is stretched to fit the wetted length of the upstream boundary in the model.

This feature can be especially helpful in short reaches or in situations where the upstream boundary is not sufficiently far enough upstream for the flow solution to be resolved in the primary area of interest.

When "No" is selected, the model estimates velocity based on the depth weighting coefficient.

When “Yes” is select the edit button next to Variable Discharge becomes enabled allowing you to specify a time-series velocity file.

### Variable Velocity Distribution

The edit button to the right of this field opens a dialog box that allows you to generate or import a normalized distance and velocity profile to distribute the velocity across the upstream boundary.

Normalized Distance is specified from the water’s edge at the right bank to the left bank (as viewed when looking downstream), in meters. The coordinate system is right-handed.

Normalized Velocity is the vertically averaged velocity magnitude in m/s.

Velocity Angle: this feature is not currently implemented.

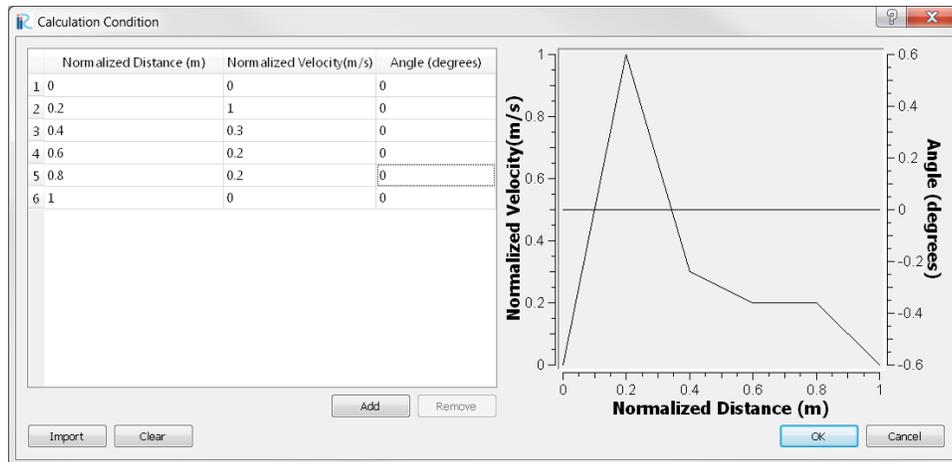


Figure C. Velocity weighting dialog.

### 6.2.Stage

Stage is the water-surface elevation at the downstream boundary for a given discharge. Figure D. shows the fields associated with specifying stage at the downstream boundary. All stage data should be entered as elevation in meters, based on the same datum as the topography dataset.

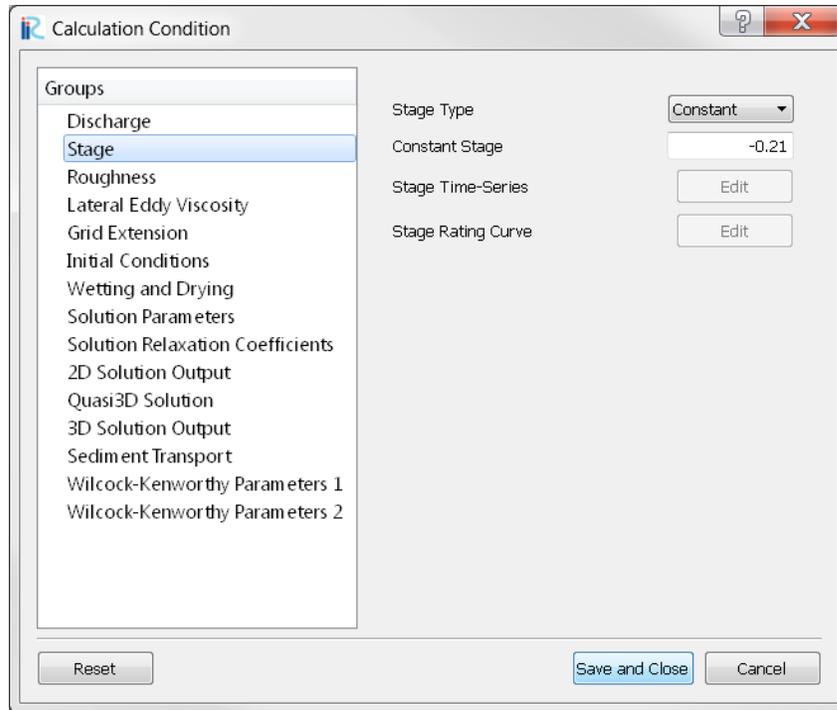


Figure D. Stage dialog.

### Stage Type

Allows you to select three different stage conditions; constant, time-series, and rating-curve from the pull down menu. Constant applies the same stage to the downstream boundary for the entire simulation. Time-series enables the Time-series button which allows you to enter or import a file that defines how stage changes through time. Rating-curve enables the Rating-curve button which opens a dialog that allows you to enter or import a file that defines how stage and discharge are related.

### Constant Stage

Enter the water-surface elevation at the downstream boundary in meters.

### Stage Time-Series

The edit button to the right of this field opens a dialog box that allows you to enter a discharge time-series, or import a time-series file for stage (Figure E). Time is specified in seconds and stage is meters. Import files should contain the same information and be saved with the .txt or .csv extension. The format of the file is time comma stage similar to that of discharge in Section 6.1.

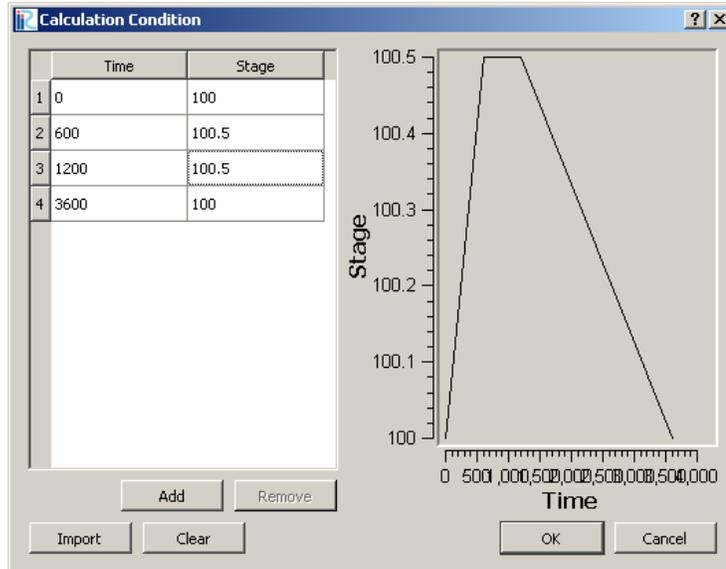


Figure E Time-series stage dialog.

### Stage Rating-Curve

The edit button to the right of this field opens a dialog box that allows you to generate stage-rating curve, or import a rating-curve file. Discharge is specified in  $\text{m}^3/\text{s}$  and stage is in meters. Import files should contain the same information and be saved with the .txt or .csv extension.

\*Note, if you use the extend grid feature, you will need to modify the stage to account for the extended grid.

## 6.3. Roughness

A hydraulic roughness parameter combines many potential sources of roughness into one value. Different sources of roughness include grain roughness, bed forms such as dunes and ripples, bank alignment, and vegetation, etc that may be difficult to measure in the field. Roughness is often determined through a calibration process that evaluates which roughness value produces the best fit with measured data such as water-surface elevations and velocity fields.

Roughness can be specified based on either drag coefficients or  $z_0$  parameters (Figure F). Roughness can be either a constant throughout the model reach or vary spatially. To use a spatially varying value you will need to create polygons with different roughness values that are then interpolated to the grid using tools in the Pre-Processor window.

### Roughness Type

Select either drag coefficient or  $z_0$  from the pull down menu.

#### *Drag coefficient*

Specify the drag coefficient. Typically the drag coefficient is determined by calibrating the value to measured water surface elevations through the reach. In the absence of measured water surface elevations you can estimate the drag coefficient

base on the mean depth through the reach and Mannings roughness coefficient (n) based on the following equation, where h is the mean flow depth and g is gravity.

$$C_d = \frac{n^2 g}{h^{1/3}}$$

$Z_0$

$Z_0$  is defined as the height above the bed where flow velocity equals zero. It can be related to bed grain size using  $Z_0 = C * \text{grain size}$ , where C is a coefficient typically set to 0.1. The  $Z_0$  roughness parameter can be a good choice when roughness is likely to be spatially variable due to depth dependence.

The FastMech Solver converts  $Z_0$  to  $C_d$  from an estimate of depth based on the initial water-surface elevation condition. This conversion can produce unrealistically low or high drag coefficient estimates. Any values that are lower than a user entered minimum value is reset to the minimum value and any values higher than the user entered maximum value is reset to the maximum drag coefficient.

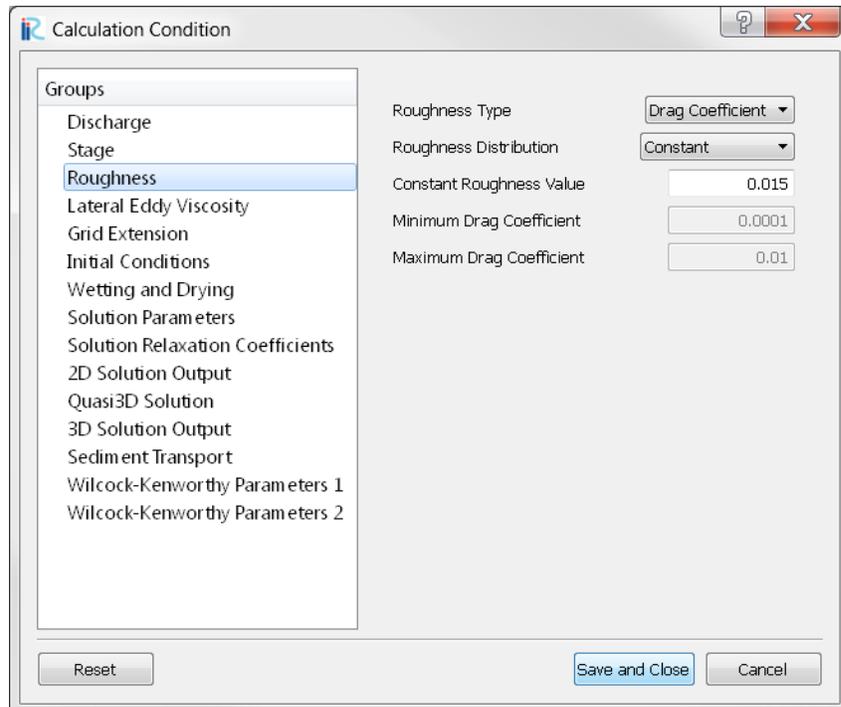


Figure F Roughness dialog.

## 6.4. Lateral Eddy Viscosity

The lateral eddy viscosity is a correction to the eddy viscosity used in the vertically averaged equations to treat lateral separation eddies. It can be estimated using:

$$LEV = 0.1 \times \text{average depth} \times \text{average velocity}$$

In some cases the model may be unstable for reasonable estimates of LEV. Typically this occurs when the grid centerline is perpendicular to the channel such that the initial velocities are directed towards one or the other bank. In that case it may be necessary

to start the solution at a higher LEV value and then decreases LEV to a reasonable estimate once the solution is partly converged. See Figure G for the dialog that controls LEV.

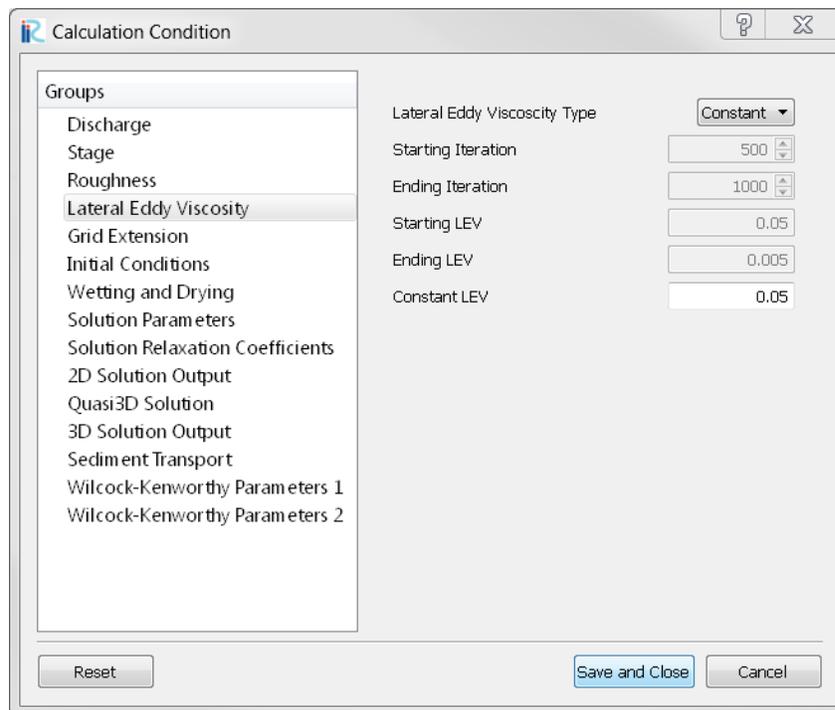


Figure G. Lateral Eddy Viscosity in the Calculation Condition dialog.

### Lateral Eddy Viscosity Type

Select Constant or Variable from the pull down menu. Constant uses the same LEV for each iteration. The variable option linearly reduces the LEV value between a specified beginning and ending value over a specified iteration interval. The fields to control the variable LEV features become enabled when the Variable option is selected.

### Starting Iteration

Enter the iteration at which the LEV will begin to decrease.

### Ending Iteration

Enter the iteration at which the LEV will stop decreasing

### Starting LEV

Enter the beginning LEV value

### Ending LEV

Enter the ending LEV value

### Constant LEV

Enter the LEV value

## 6.5. Grid Extension

It is always best to choose a downstream boundary where the flow is uniform. In practice this is often not the case. The model does not allow for upstream flow at the downstream boundary condition, which can occur when there is a recirculating eddy at the boundary. There are two ways to handle issues at the downstream boundary by either 1) extending the boundary with uniform channel geometry to try to prevent recirculating flow or 2) forcing downstream flow at the boundary.

Grid extension works by duplicating the topography in the last row of nodes by a specified number of streamwise nodes while reducing the elevation of each node based on a grid extension slope. For example, if the grid cell size in the stream-wise direction is 10 m, and you select 10 grid extension nodes, the total grid extension will be 100 m. It is important to note that the downstream boundary condition in the input file must account for this change in the model grid. In other words, you must decrease the water-surface elevation at the downstream boundary so that it corresponds with the extended topography.

Forcing the downstream velocity boundary to prevent recirculating flow often resolves issues related to recirculating flow. However it will affect the solution near the downstream boundary.

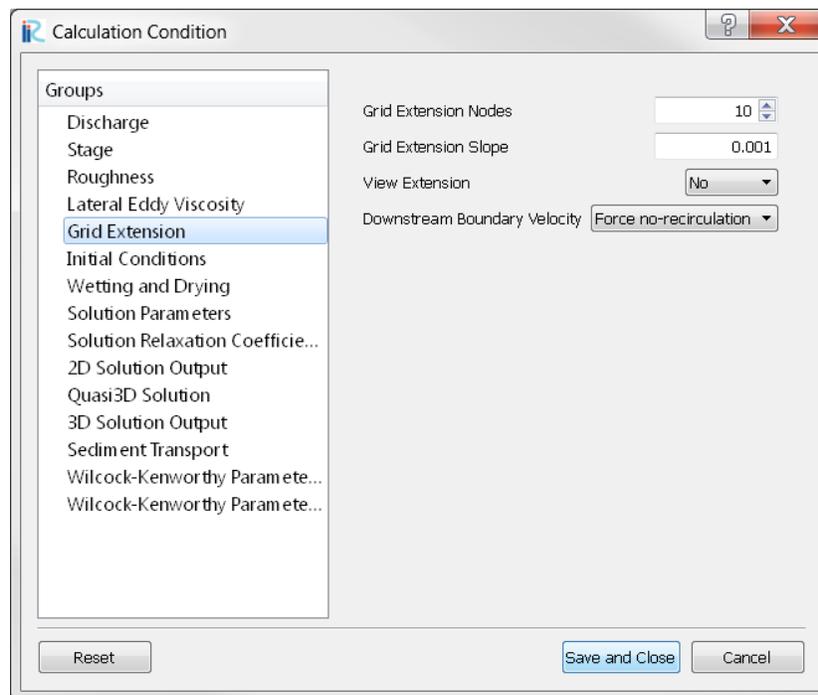


Figure H. Grid Extension in the Calculation Condition dialog.

### Grid Extension Nodes

Enter the number of nodes to extend the grid in the streamwise direction

### **Extension Slope**

Enter the slope to decrement the elevation of the extended grid.

### **View Extension**

Select yes or no from the pull down menu to display the extended topography in the Post-processing window.

### **Downstream Boundary Velocity**

Select No or Force no-recirculation from the pull down menu.

## **6.6. Initial Conditions**

The model requires an initial estimate or guess of the water-surface elevation for each node in the grid. This provides both a starting point in the numerical model and determines which nodes are wet and dry in the calculation. Better initial water-surface elevations lead to faster model convergence. There are a number of ways to specify the initial water-surface elevation; however the 1D step-backwater solution is recommended in most situations.

This input dialog also allows you to select a hot-start file to initialize the model. A hot-start file is simply the output from a previous model solution. This feature allows you to start the model using a converged solution, which can be particularly useful in simulations that simulate bed evolution. If you want to use the hot start option there are two requirements.

1. The solution to be used to hot start the current solution must have been saved using the File->Save Project command in iRIC.
2. The 2D Solution Output must have "Initial Velocity for Hot Start" selected.

The initially wet or active nodes in FaSTMECH are determined by the water-surface elevation initial guess. The initial water surface elevation is specified along the grid centerline. Wet and dry nodes are determined by extending the 1-dimensional water surface elevation along the grid centerline to each row or cross-section of nodes. If the water surface elevation is above the bed elevation the node is wet, otherwise it is dry. By default nodes will dry during the solution iteration process if the water surface elevation solution becomes less than the bed elevation.

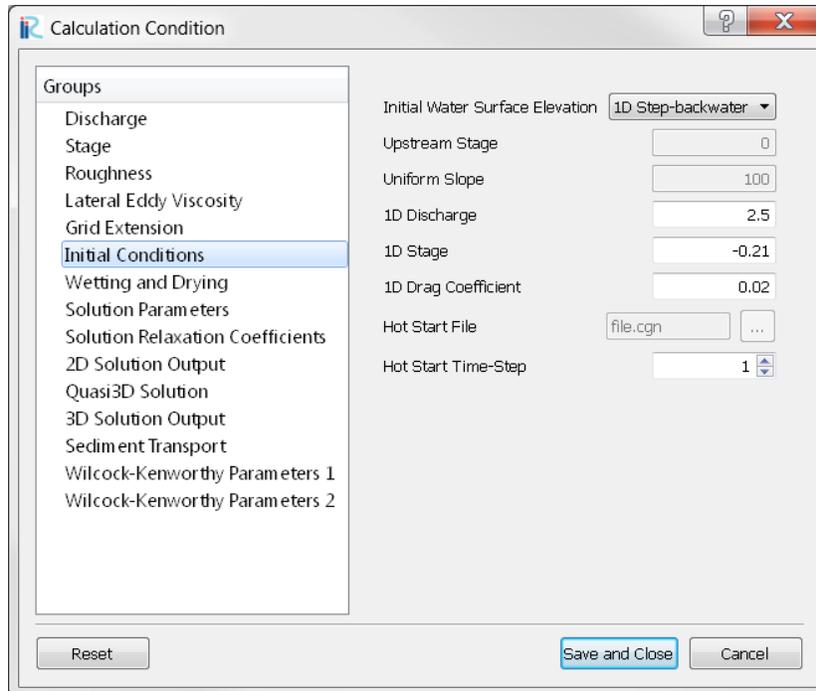


Figure I. Initial Conditions in the Calculation Condition dialog.

### Initial Water-Surface Elevation

Select Upstream Stage, Uniform Slope, 1D Step-backwater, or Hot Start from the pull down menu. The necessary fields are then enabled below.

### Upstream Stage

This option is enabled when Upstream Stage is selected from the Initial Water-Surface Elevation pull down menu. Enter the stage at the upstream boundary. The water-surface elevation is then linearly interpolated between the boundaries for each node in the grid.

### Uniform Slope

This option is enabled when Uniform Slope is selected from the Initial Water-Surface Elevation pull down menu. Enter a slope. The water-surface elevation is then projected upstream starting at the downstream boundary along the grid centerline.

### 1D Discharge

This option is enabled when 1D Step-backwater is selected from the Initial Water-Surface Elevation pull down menu. Enter the discharge you are interested in modeling.

### 1D Stage

This option is enabled when 1D Step-backwater is selected from the Initial Water-Surface Elevation pull down menu. Enter the stage at the downstream boundary.

## 1D Drag Coefficient

This option is enabled when 1D Step-backwater is selected from the Initial Water-Surface Elevation pull down menu. Select a roughness value. You may wish to select a higher roughness value to force the initial water-surface elevation to be somewhat higher.

## Hot Start File

Select the solution file you wish to use from the dialog window. The solution used for the hot start must have the requirements as specified in the [introduction to this section](#).

## Hot Start Time-step

Select the iteration or time-step from the solution file to initialize the model.

## 6.7. Wetting and Drying

The wetting and drying dialog controls how dry nodes are treated in the calculation, and provides settings to re-evaluate whether or not nodes are wet or dry during the simulation. The wet or dry state of a node can be visualized in the IBC solution scalar. A value of 0 is dry and a value of -1 is wet.

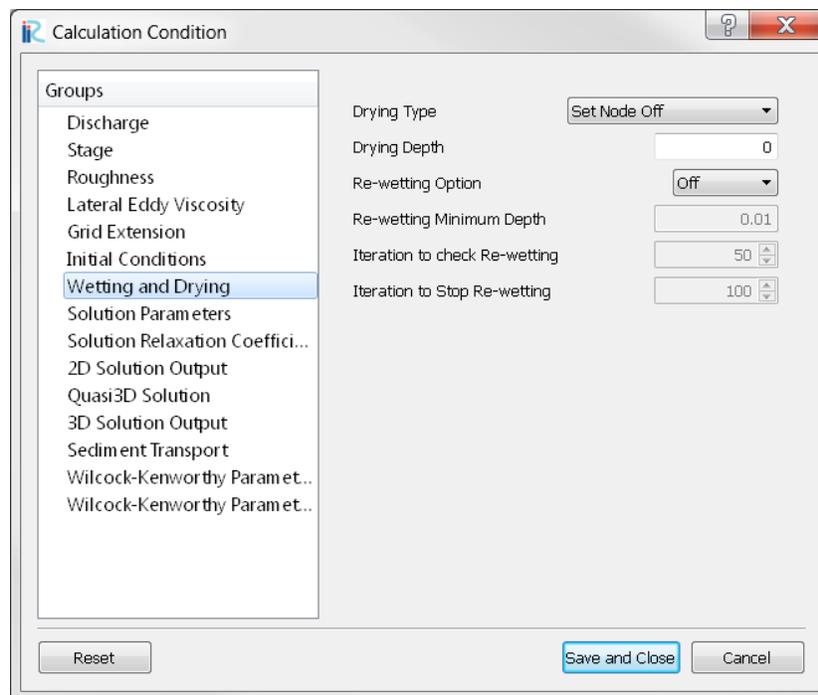


Figure J. Wetting and Drying in the Calculation Condition dialog.

## Drying Type

Select from Set Node Off and Set Node to Drying Depth in the pull down menu. The result of this option depends on the Drying Depth set below. Set Node Off turns the node off (or sets it to a dry state) if the depth drops below the minimum depth set below. Set Node to Drying Depth keeps the node in a wet state if it drops below the minimum depth but keeps the depth at the specified value. This can be useful when you have instabilities in the solution in very shallow locations. In general this latter option is rarely used..

### **Drying Depth**

Set the minimum depth at which a node becomes dry.

### **Re-Wetting Option**

Select from Off and On in the pull down menu. Off means that the wet status of a node is set by the initial water-surface condition and is not re-evaluated during the simulation. On allows the user to prescribe when the wet and dry status of a node is re-evaluated during the simulation. Using this option can make a stable solution difficult to acquire. If the channel has large areas of relatively shallow regions then it may be necessary to use this option to get a solution that predicts well these shallow area's.

### **Re-wetting Minimum Depth**

Enter the minimum depth at which a dry node becomes wet. This value should be larger than the Drying Depth set above. Generally good results are found by setting the drying depth to 0, and the re-wetting depth to 0.015.

### **Iteration to check Re-wetting**

Enter the iteration to check if the re-wetting condition has been met.

### **Iteration to stop check Re-wetting**

Entering the iteration to stop checking if the re-wetting condition has been met.

## **6.8.Solution Parameters**

The solution parameters dialog controls how many iterations are run using a constant discharge simulation and provides controls for time-stepping features for quasi steady discharge simulations.

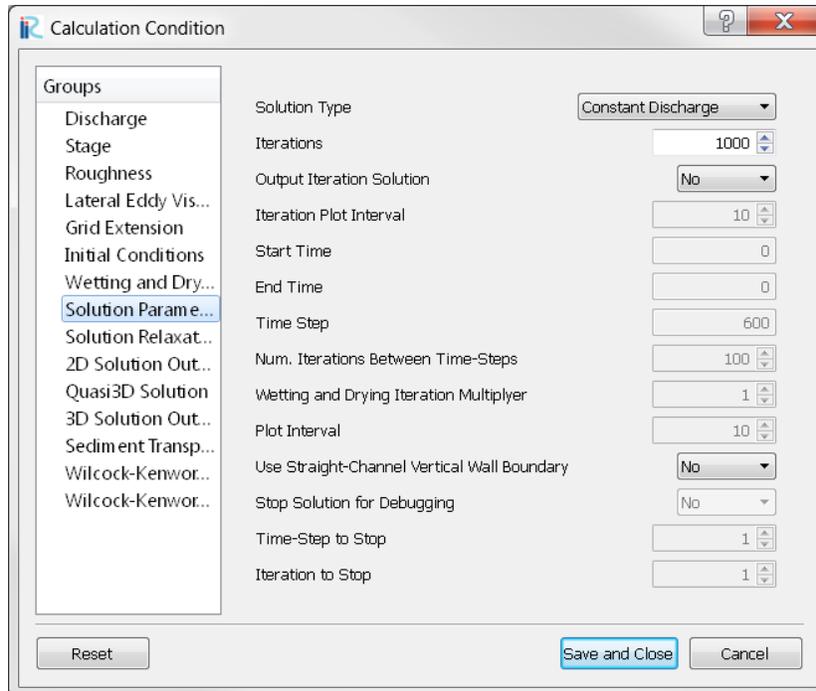


Figure K. Solution Parameters in the Calculation Condition dialog.

### Solution Type

Select Constant Discharge or Quasi-steady Discharge from the pull down menu. Constant Discharge is used when solving for a single discharge value. Quasi-Steady Discharge is when simulating a hydrograph or morphologic change over a period of time. The discharge may be constant or time-dependent.

### Iterations

Enter the number of iterations used to generate a solution. The number required to reach a converged solution varies. This field is set for both constant discharge and quasi-steady discharge options. In the quasi-steady option it specifies the number of iteration at the start of the calculation.

### Output Iteration Solution

Select No or Yes from the pull down menu. No will output solution results for the final iteration. Yes will output solution results for each iteration entered in the Iteration Plot Interval field. Generally this option is not used.

### Iteration Plot Interval

Enter the iteration interval to output solution results. For example, if your iteration plot interval is set to 50 and the iteration is set to 200, you will get output data at iteration 1, 50, 100, 150, and 200? enter 50 and have 200 This fields is enabled when the Output Iteration Solution is set to Yes.

### **Start Time**

Enter the time in seconds to start the simulation. The field is enabled when Quasi-steady Discharge is selected from the Solution Type pull down menu.

### **Time Step**

Enter the time step in seconds to evaluate the time-stepping solution.

### **Num. Iterations Between Time-steps**

This field specifies the number of iterations the solver completes before incrementing to the next time-step.

### **Wetting and Drying iteration Multiplier**

If the solution is solved for a variable discharge hydrograph then sometimes it is advantageous to increase the number of iteration between each time step when the solution either increases or decreases in the number of dry nodes. In the case where the number of wet or dry nodes changes then the num. iteration between time-steps increases by a multiple of the number specified in this field.

### **Plot Interval**

Select the interval, in seconds, to record solution results for Quasi-steady Discharge simulations.

### **Use Straight channel Wall Boundary**

This is in development and should be specified as no.

### **Stop Solution for debugging**

Select Yes or No from the pull down menu. Yes, allows you to output solution results at specific time-step and iteration within a time-step to better understand what causes a model to crash before completing the solution.

### **Time-step to stop**

Enter the time-step to stop the simulation and save a solution.

### **Iteration to stop**

Enter the iteration within the time-step to stop the simulation and save a solution.

## **6.9. Relaxation Coefficients**

Fastmech uses relaxation to iterate on a solution. Relaxation is essentially a way to speed up or slow down changes in a given parameter between one iteration and the next. For example, the ERelax controls the relaxation of the water-surface elevation. A value of 0.4, weights the current water-surface elevation solution at 0.4, and the previous solution at 0.6 to obtain the Water-surface used in the next iteration. In other words the current solution is made up of 60 percent of the previous solution and 40 percent of the current solution.

It is often necessary to hunt for the best parameters. The default values work in most cases but if the channel is complex, then the relaxation coefficient may need to be changed. Generally it is the ratio between ERelax and URelax which is important. Generally ARelax is set to the same value as URelax. In steep channels a values of 0.3, 0.6, 0.6 for each of the three values respectively are found to work well.

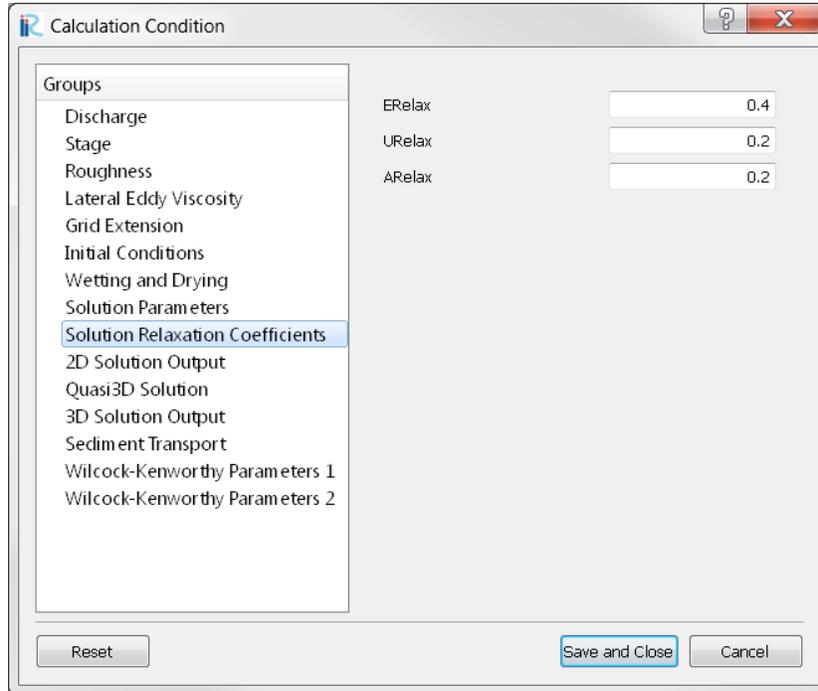


Figure L. Solution Relaxation Coefficients in the Calculation Condition dialog.

**ERelax**

Enter the relaxation coefficient for the water-surface elevation

**URelax**

Enter the relaxation coefficient for the velocity.

**ARelax**

Adjusts the global slope of the water surface at each iteration.

**6.10. 2D Solution Output**

This dialog allows you to calculate several additional parameters as part of the solution. These include:

Parameter	Description
Velocity SN	Outputs velocity vectors based on SN, or stream-wise and stream-normal coordinates.
Shear Stress XY	Outputs shear stress based on XY coordinates
Shear Stress XN	Outputs shear stress based on stream-

	wise and stream normal coordinates.
Shear Stress Divergence	Outputs shear stress divergence
Drag Coefficient	Outputs the drag coefficient used by the similar.
Initial Velocity for Hotstart	Saves the initial velocity to use as a Hot Start

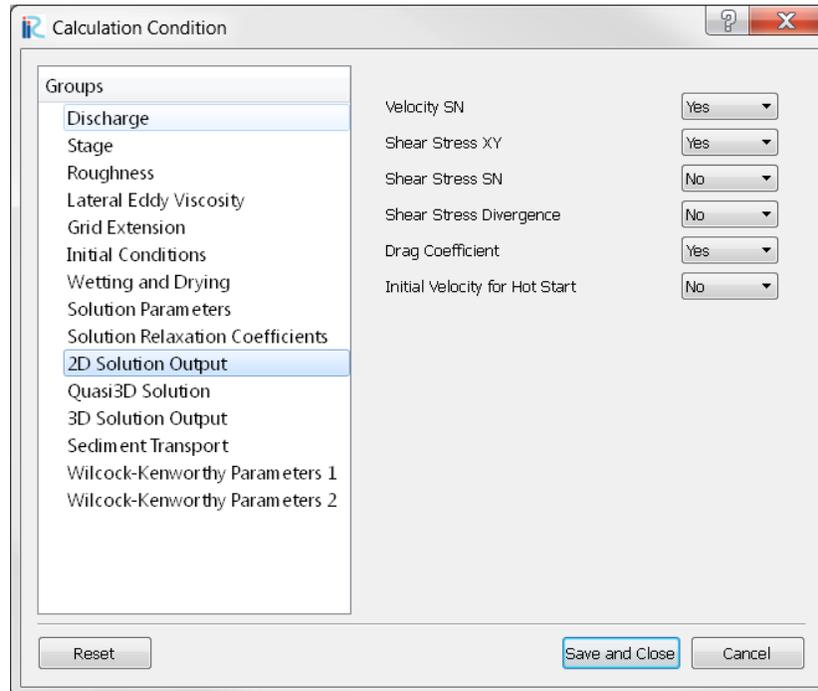


Figure M. 2D Solution Output in the Calculation Condition dialog.

## 6.11. Quasi-3D Solution

The vertical structure of the flow can be developed using the Quasi-3D option.

### Calculate Quasi-3D Solution

Select Yes or No from the pull down menu. Yes enables the Quasi-3D solution.

### Number of Vertical Nodes

Enter the number of vertical nodes to include in the calculation

### Use Streamline-curvature

Select Yes or No from the pull down menu. This option uses the streamline curvature in addition to the grid curvature to develop the secondary flow.

### Minimum Streamline Curvature

This value should not be set less than approximately 5 times the mean depth.

## 6.12. 3D Solution Output

Select whether or not to include aspects of 3D flow in the solution calculation. The 3D Scalar and Vector option can make file sizes very large.

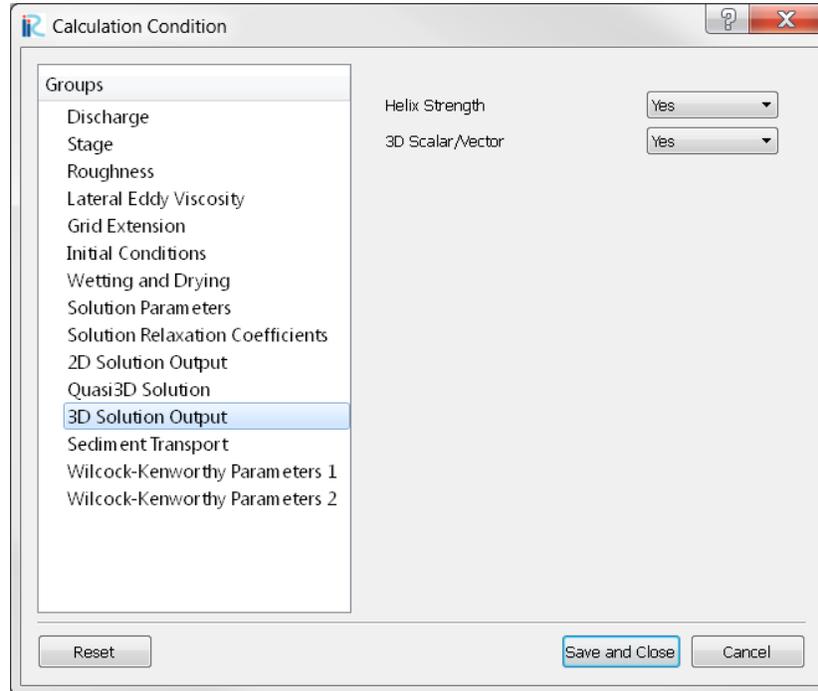


Figure N

Parameter	Description
Helix Strength	The difference in angle between the surface and near bed velocities
3D Scalar/Vector	Outputs the 3D Scalar and Vector quantities

## 6.13. Sediment Transport

The sediment Transport dialog controls many functions of sediment transport and bed evolution simulations.

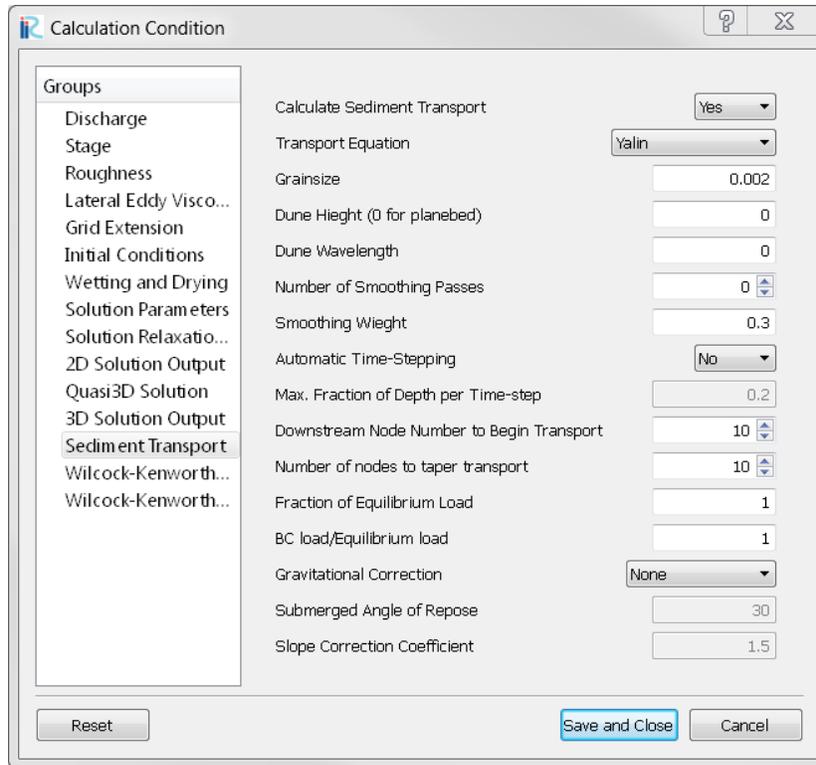


Figure O

## Calculate Sediment Transport

Choose yes or no to calculate sediment transport.

### Transport Equation

- Yalin (bedload)
- Engelund-Hansen (total load)
- Wilcock-Kenworthy (two-fraction model, uses fine fraction only)

### Grainsize

Mean grainsize to use in sediment transport calculations.

### Dune Height

If dune height and dune wavelength are set boundary shear stress will be reduced in the sediment transport calculations. Set the average dune height in meters

### Dune Wavelength

Set the average dune wavelength in meters.

### Number of smoothing passes

Smooth the sediment transport rate using a box-car smoothing algorithm.

### Smoothing Weight

Specify the weight of the adjacent nodes in the smoothing algorithm.

### **Automatic Time-Stepping**

When this value is set to yes the time step for each new calculation is determined from the previous time step based on the Max. Fraction of Depth per Time-step set below. Based on the current sediment flux and depths, the time step is determined such that no where in the solution will the change in elevation of the bed result in a change in depth greater than the value specified below.

### **Max. Fraction of Depth per Time-step**

The time step is constrained by this fraction. A value of 0.1 means that the time-step calculated from the previous solution will be set such that the change in elevation will not exceed 10 percent of the depth anywhere in the solution domain.

### **Downstream Node Number to Begin Transport.**

The model can become unstable with the transport is started near the upstream boundary. This sets the node number in the stream-wise direction to begin the transport calculation.

### **Number of Nodes to Taper Transport**

Over a long morphologic change simulation there can be a large change in elevation upstream and downstream of the location where transport begins. This parameter tapers the magnitude of transport over specified number of nodes in the stream-wise direction to try and eliminate any large changes in bed elevation resulting from the transport at the upstream boundary.

### **Fraction of Equilibrium Load**

The transport rate is based on the specified transport equation. You can increase or decrease the predicted transport everywhere by setting this value less than or greater than 1 respectively.

### **BC load/Equilibrium load**

Over or under supply the load at the upstream boundary by specifying this value to be greater than or less than 1 respectively.

### **Gravitational Correction**

Specify the gravitational correction to use.