AquaDyn

Scientific Reference

Flow Variables

AquaDyn offers solutions to a wide range of problems involved in the hydrodynamic analysis of open channel flow. The St. Venant equations on which AquaDyn is based are described in Chapter 2.

The solution of hydrodynamic problems consists in evaluating the spatio-temporal variation of the velocities U and V and the level h of the water under various conditions. Specifically:

$U(x, y, t)$, $V(x, y, t)$ = depth-averaged velocity components

$h(x, y, t)$ = instantaneous depth (m)

where x, y, t are the spatial and time coordinates.

Various forces, such as gravitational, inertial, friction and viscous forces, act on open channel flow, each to a different extent. In order to compare their relative importance and assess their influence on the flow for classification purposes, we define dimensionless numbers. The Reynolds and Froude numbers are the most commonly used.

The Reynolds number is expressed as follows:

$$\mathrm{Re} = h\,\frac{\sqrt{U^2 + V^2}}{n} \qquad \left(\frac{\text{inertial forces}}{\text{viscous forces}}\right) \tag{1.01}$$

where h, U and V are the hydrodynamic variables defined above.

An increasing Reynolds number implies a decrease in the influence of the viscous forces on the flow.

The Froude number, the most important of all the dimensionless numbers studied in open channel flow, is expressed as:

$$Fr = \sqrt{\frac{U^2 + V^2}{gh}} \qquad \left(\frac{\text{inertial forces}}{\text{gravitational forces}}\right) \tag{1.02}$$

where h, U and V are the hydrodynamic variables defined above and g is the acceleration due to gravity. The meaning of this number is given in the next section. Note that $\sqrt{gh}$ represents the celerity of a long wave in shallow water.

This number plays the same role as the Mach number (Ma) used in aerodynamics for compressible-fluid flow:

$$Ma = \frac{U_a}{\sqrt{\frac{k_c}{r}}} \tag{1.03}$$

where K_c is the compressibility (N / m^2), r the density (kg / m^3), U_a the speed of air and $\sqrt{\frac{k_c}{r}}$ the speed of sound in air.

Another parameter affecting the flow is the Manning coefficient n (in m$^{-1/3}$.s), which is an empirical measure of the effect of roughness as a function of the nature of the wall surface and the bottom or bed configuration. The roughness creates friction and resistance to the natural flow of the fluid. Appendix A presents the Manning coefficient in greater detail.

Another two coefficients affecting flow are the fluid viscosity ν and a turbulence coefficient γ, which are empirical measures of energy dissipation. The former represents the energy loss due to internal molecular collisions between the water molecules, while the latter characterizes the energy loss due to shear flow, rapid elevation and channel expansion or contraction, which occurs at scales smaller than the triangular decomposition of the domain.

Steady Flow

AquaDyn is used to study problems related to steady flow as found in rivers and lakes, i.e. a flow which is independent of time. Any change made to a water course, such as the construction of a dam, the piers of a bridge or dikes, for example, can be studied and optimized with a view to minimizing its environmental impact.

This type of flow may be characterized by the Froude number: when the Froude number is less than 1, i.e. when the fluid velocity is less than the celerity of a surface wave, the flow is termed *subcritical* or *fluvial*. Under these conditions, small waves may propagate upstream and downstream.

When the Froude number is greater than 1, i.e. when the fluid velocity exceeds the celerity of the surface wave, the flow is described as being *supercritical* or *torrential*. Under these conditions, an upstream disturbance will propagate rapidly downstream and may cause shocks and instabilities, but the downstream conditions will not have any perceptible effect on the upstream flow behavior.

A transition from a torrential to a fluvial flow involves a hydraulic jump, which is a discontinuity in the water level. A transition from a fluvial to a torrential flow will occur without discontinuity.

AquaDyn represents an excellent tool for simulating a wide range of steady flow under fluvial conditions. Torrential conditions are more unstable however, and a finer mesh and a lower value for the relaxation parameter ω are required to improve convergence.

See chapter three on how to improve convergence.

Unsteady Flow

AquaDyn also allows the user to study problems related to unsteady flow as in estuaries and ports. An unsteady flow is a flow which varies in time. Simulations of unsteady flow are often characterized by time-dependent boundary conditions. Examples of unsteady flow include cyclical propagation of tidal waves and flood wave propagation.

Modeling tidal waves offers a means of studying the hydrodynamics of an estuary or a port. Phenomena such as current reversal in straits or a rise in the high-tide level in estuaries can be modeled in detail.

Modeling flood propagation along the entire length of a river provides a means of determining the high- and low-risk areas. An illustrative example is the flow increase due to opening of flood gates at a specific time.

Governing Equations

St. Venant Equations

The governing equations for hydrodynamic flow are the St. Venant equations; which can be written

$$\frac{\partial U}{\partial t}+U\frac{\partial U}{\partial x}+V\frac{\partial U}{\partial y}-fV+g\frac{\partial H}{\partial x}+gU\frac{\sqrt{U^2+V^2}}{C^2h}-\frac{\partial}{\partial x}\left(\nu_t\frac{\partial U}{\partial x}\right)-\frac{\partial}{\partial y}\left(\nu_t\frac{\partial U}{\partial y}\right)=F_x \qquad (2.01)$$

$$\frac{\partial V}{\partial t}+U\frac{\partial V}{\partial x}+V\frac{\partial V}{\partial y}+fU+g\frac{\partial H}{\partial y}+gV\frac{\sqrt{U^2+V^2}}{C^2h}-\frac{\partial}{\partial x}\left(\nu_t\frac{\partial V}{\partial x}\right)-\frac{\partial}{\partial y}\left(\nu_t\frac{\partial V}{\partial y}\right)=F_y \qquad (2.02)$$

$$\frac{\partial h}{\partial t}+\frac{\partial hU}{\partial x}+\frac{\partial hV}{\partial y}=0 \qquad (2.03)$$

where the variables are defined as follows:

H = water level

h = water depth ($h = H - Z$)

Z = bed elevation or bathymetry

U, V = two horizontal components of the depth-averaged velocity

C = Chézy coefficient

f = Coriolis coefficient

ν_t = total kinematic viscosity

The Coriolis coefficient is given by the relation

$$f = 2\Omega \sin\phi \qquad (2.04)$$

where Ω is the earth's rotation speed (rad/s) and ϕ the angle of latitude (degrees).

The Chézy coefficient is given by the Manning equation

$$C = \frac{1}{n} h^{1/6} \qquad (2.05)$$

where n is the Manning coefficient ($m^{-1/3}$.s).

The Manning coefficient depends on the surface characteristics of the water bed. Appendix C presents the Chézy and Manning coefficients in greater detail.

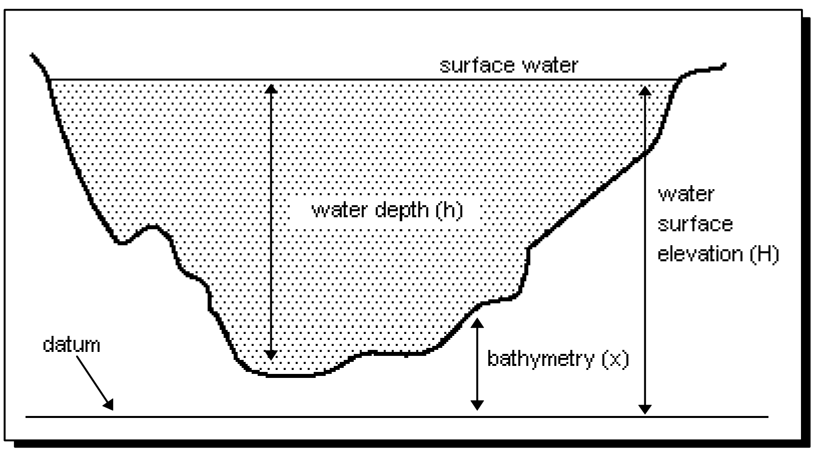

Figure 2.1 General schematic of water flow

The total kinematic viscosity ν_t is the sum of the fluid viscosity and the turbulent viscosity, which is controlled by the turbulence coefficient γ:

$$\nu_t = \nu + \gamma A \sqrt{2\left(\frac{\partial U}{\partial x}\right)^2 + 2\left(\frac{\partial V}{\partial y}\right)^2 + \left(\frac{\partial U}{\partial y} + \frac{\partial V}{\partial x}\right)^2} \qquad (2.06)$$

where ν is the fluid viscosity, A the area of a triangular element and γ the turbulence coefficient. The turbulence coefficient γ allows the model to be calibrated in terms of the observed energy loss due to channel contraction or expansion, or variations in the bed elevation.

The external force (F_x, F_y) represents a wind-imposed surface stress and is expressed as

$$F_x = D\tilde{U}\,\frac{\sqrt{\tilde{U}^2 + \tilde{V}^2}}{h} \qquad (2.07)$$

$$F_y = D\tilde{V}\,\frac{\sqrt{\tilde{U}^2 + \tilde{V}^2}}{h} \qquad (2.08)$$

where $\tilde{U}$ and $\tilde{V}$ represent the horizontal wind velocities (m / s) and h, the water depth. The friction coefficient D is an empirical coefficient depending on the height above the water surface at which the wind velocities are measured. Details are given in Appendix A.

The St. Venant equations are obtained from the Navier-Stokes equations by averaging the horizontal velocities along the depth of the water body. Moreover, the pressure variable is replaced by the water height using the hydrostatic pressure distribution assumption (see, for instance, Pinder or Gray).

Boundary Conditions

AquaDyn allows the user to impose different types of conditions at the boundaries of the domain under study. For each boundary node or segment, the user can impose:

- closed boundaries with no slip conditions: $U = V = 0$
- closed boundaries with free slip conditions: $Un = 0$
- discharge rate, water level or normal velocity of any constant or variable value at nodes or along cross-sections.

Here, Un is the normal component of water velocity.

By default, the finite-element method forces the derivative normal to the boundary of the velocities to be zero, unless a wall boundary, normal velocity or tangential velocity is imposed.

Note that the third type of boundary conditions takes priority over the first three.

Initial Conditions

To start a numerical simulation, the user must specify the initial velocity values and water level for all nodes. In the case of steady flow, the initial state is used as the first approximate solution of the iterative solver. Good initialization may significantly reduce the time the solver takes to converge to the solution at the desired accuracy. Sometimes, convergence can only be achieved if the initial state is well chosen. In the case of unsteady flow, the simulation will begin using the imposed initial state and the calculation will evolve as a function of time.

Numerical Method and Convergence

Numerical Integration Method

Galerkin's method is used to numerically solve the St. Venant equations. The domain to be simulated is decomposed into triangular elements. Each triangle contains 15 degrees of freedom, as shown in Figure 3.1. The field variables U and V are quadratically interpolated from the six node values U_n and V_n on the triangle respectively. The field variables H, h, Z and n are linearly interpolated from the three node values to obtain H_n, h_n, Z_n and n_n on the triangle respectively:

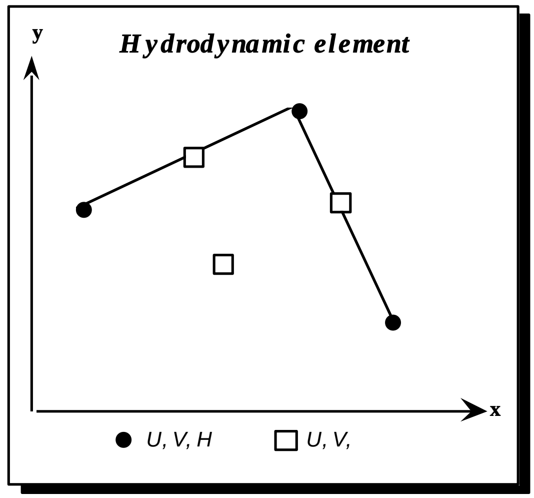

Figure 3.1 Numerical integration method

$$U(x,y) = \sum_{n=1}^{6} \varphi_U^n(x,y)\, U_n \tag{3.01}$$

$$V(x,y) = \sum_{n=1}^{6} \varphi_V^n(x,y)\, V_n \tag{3.02}$$

$$H(x,y) = \sum_{n=1}^{3} \varphi_H^n(x,y)\, H_n \tag{3.03}$$

The interpolating function for the variables h, n and Z is φ_H^n, as for H.

The interpolating functions φ_U^n, φ_V^n and φ_H^n are the standard Lagrangian interpolating functions [Reddy]. Note that $\varphi_U^n = \varphi_V^n$, as there is no preferred direction.

The horizontal wind velocities $\tilde{U}$ and $\tilde{V}$ and the wind coefficient D have a constant value over each element of the domain under study but they can vary from one element to another. The parameters f, g, γ and ν are constant throughout the domain.

In order to obtain a complete set of discretized equations, the finite-element method requires that Eqs. 3.01, 3.02 and 3.03 be substituted into the St. Venant equations, that the equation for U,

V and H be multiplied by the weight functions φ_U^n, φ_V^n and φ_H^n respectively, and that each equation be integrated over each triangular element e. We obtain the following set of equations:

$$\iint_e \Big[\varphi_U\big(U_{t+\Delta t} - U_t + \alpha\, B_1(U_{t+\Delta t}, V_{t+\Delta t}, H_{t+\Delta t}) + (1-\alpha)\, B_1(U_t, V_t, H_t)\big) + \alpha\, A_1(U_{t+\Delta t}, V_{t+\Delta t}, H_{t+\Delta t}) + (1-\alpha)\, A_1(U_t, V_t, H_t)\Big]\, dx\, dy = 0 \quad (3.04)$$

$$\iint_e \Big[\varphi_V\big(V_{t+\Delta t} - V_t + \alpha\, B_2(U_{t+\Delta t}, V_{t+\Delta t}, H_{t+\Delta t}) + (1-\alpha)\, B_2(U_t, V_t, H_t)\big) + \alpha\, A_2(U_{t+\Delta t}, V_{t+\Delta t}, H_{t+\Delta t}) + (1-\alpha)\, A_2(U_t, V_t, H_t)\Big]\, dx\, dy = 0 \quad (3.05)$$

$$\iint_e \varphi_H\big(H_{t+\Delta t} - H_t + \alpha\, B_3(U_{t+\Delta t}, V_{t+\Delta t}, H_{t+\Delta t}) + (1-\alpha)\, B_3(U_t, V_t, H_t)\big)\, dxdy = 0 \quad (3.06)$$

where α is the weight in the weighted Crank-Nicholson time discretization and

$$A_1(U,V,H) = \nu_t \left(\frac{\partial \varphi_U}{\partial x}\frac{\partial U}{\partial x} + \frac{\partial \varphi_U}{\partial y}\frac{\partial U}{\partial y} \right) \quad (3.07)$$

$$B_1(U,V,H) = U\frac{\partial U}{\partial x} + V\frac{\partial U}{\partial y} - fV + g\frac{\partial H}{\partial x} + gn^2 U\frac{\sqrt{U^2+V^2}}{h^{4/3}} - D\tilde{U}\frac{\sqrt{\tilde{U}^2+\tilde{V}^2}}{h} \quad (3.08)$$

$$A_2(U,V,H) = \nu_t \left(\frac{\partial \varphi_V}{\partial x}\frac{\partial V}{\partial x} + \frac{\partial \varphi_V}{\partial y}\frac{\partial V}{\partial y} \right) \quad (3.09)$$

$$B_2(U,V,H) = U\frac{\partial V}{\partial x} + V\frac{\partial V}{\partial y} + fU + g\frac{\partial H}{\partial y} + gn^2 V\frac{\sqrt{U^2+V^2}}{h^{4/3}} - D\tilde{V}\frac{\sqrt{\tilde{U}^2+\tilde{V}^2}}{h} \quad ($$

$H_{t+\Delta t,n}$ to be solved for. For the sake of clarity in what follows, we will drop the indices on the vector *W*, *F* and matrix *K*.

The system of equations is resolved iteratively using the Newton-Raphson method:

$$K_t^{(i)} \, \Delta W^{(i+1)} = R^{(i)} = F^{(i)} - K^{(i)} \, W^{(i)} \tag{3.14}$$

where *R* is the residual vector, *W* the unknowns, *i* the iteration number and K_t the tangent matrix defined as follows:

$$K_t^{(i)} = \left. \frac{\partial R}{\partial W} \right|_{W = W^{(i)}} = K^{(i)} + \left. \left[\frac{\partial K(W)}{\partial W} \right] \right|_{W = W^{(i)}} W^{(i)} \tag{3.15}$$

Therefore, the approximate solution after the i-th iteration is given by:

$$W^{(i+1)} = W^{(i)} + \omega \, \Delta W^{(i+1)} \tag{3.16}$$

where

$$\Delta W^{(i+1)} = \left[\, K_t^{(i)} \, \right]^{-1} R^{(i)} \tag{3.17}$$

and ω is a relaxation parameter (between 0 and 2), which improves the convergence of the iterative process.

The inverse of the matrix is calculated using the Thomas algorithm (*LU* decomposition).

In order to reduce the memory requirement, the unknowns are numbered such that the non-zero value of the matrix *K* lies as close as possible to the diagonal. This procedure is referred to as bandwidth reduction of the matrix. Only the non-zero values are stored in memory.

The calculation comprises the following steps:

1. initialization of vector $W^{(0)}$ at iteration zero and $(i = 0)$
2. calculation of $R^{(i)}$ using Eq. 3.14
3. calculation of $K_t^{(i)}$ using Eq. 3.15
4. calculation of $\Delta W^{(i+1)}$ using Eq. 3.17
5. calculation of $W^{(i+1)}$ using Eq. 3.16
6. $i \leftarrow i + 1$
7. verification of convergence using Eqs. 3.18, 3.19 and 3.20
8. repetition of steps 2 to 8 until the desired accuracy is reached.

The following criterion is used to evaluate the desired accuracy:

$$\max_k \left| \Delta U_k^{(i+1)} \right| < \varepsilon_U \tag{3.18}$$

$$\max_k \left| \Delta V_k^{(i+1)} \right| < \varepsilon_V \tag{3.19}$$

$$\max_{k} \left| \Delta H_k^{(i+1)} \right| \langle \; e_H \qquad (3.20)$$

The initial values specified for W, the unknowns, are determining factors for accelerating the computation and ensuring convergence. A special effort must therefore be made to specify the initial state that would most closely approximate the desired solution. To this end, the Automatic function in the Initial State menu estimates a solution by taking into account the characteristics of the boundary conditions and the values of the fluid and environment parameters.

Convergence

It is possible that convergence toward a solution is difficult to achieve for some simulations. This depends on the case under study. Nevertheless, to improve convergence, the user has a number of options:

1. Try a better guess for the initial state.
2. Modify the relaxation parameter w in the **Convergence** submenu of the **Solution** menu by first giving it a lower value. A value of 0.01 may be needed if some areas have a shallow depth of water. Once convergence is smooth, the relaxation parameter can be gradually increased in order to accelerate the process. Note that the optimum value of the relaxation parameter is 1, and this is the value you should use unless divergence occurs.
3. Reduce the Peclet number by refining the mesh or increasing the viscosity. For a Peclet number greater than 2, some oscillations in the velocity variables may occur if the solution is showing signs of rapid spatial variations. These oscillations are well known in finite-element methods and make convergence more difficult to achieve. They can be removed by an upwind differentiation technique, which should be available in the next version of this software.
4. Change the boundary conditions gradually to approach the desired values and use the results of the previous simulation as the initial values for the next one, etc. Running a series of simulations with different boundary conditions will bring the solution from a smooth, fluvial regime to the desired regime with the desired boundary conditions.
5. Set the conversion parameter to zero or a value smaller than one to reduce the non linear effect.

To isolate the wave propagation from other variations over time, the initial state should be the steady-state solution corresponding to the boundary conditions at the initial time. This will help with convergence when simulating waves.