Chapter 6

Structures and discharges model

The theoretical background behind the implemented model units is described in this chapter. This is the result of a detailed review of algorithms and schemes of similar models, such as Delft3D (2009), SIMONA (2009) and of the technical literature on hydraulic structures. This review shows a resemblance of the state of art in numerical modeling for the schematization of hydraulic structures.

6.1 Dry cells

Dry cells are defined as grid cells that are taken as permanently dry during the simulation, irrespective of the local water depth. In practice, this means that the program sets the bathymetric depth to the flag (land) value depmean_flag. Moreover, no flow exchange nor transport of scalars are allowed between dry cells and the adjacent grid cells. An example of a dry cell is given in Figure 6.1.

This functionality is used to define areas in the computational domain that will be permanently dry and excluded from computations. The goal of this functionality is to schematize or simulate the effect of some types of hydraulic structures which cannot be submerged. Dry cells can be defined as "isolated" units or groups to schematize the presence of structures in a study area. An example of groups of dry cells is illustrated in Figure 6.2.

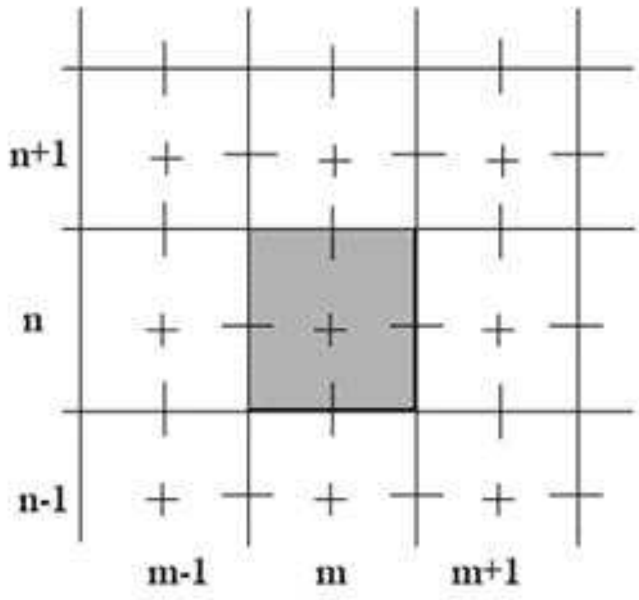

Figure 6.1: Dry cell defined at grid location (m,n).

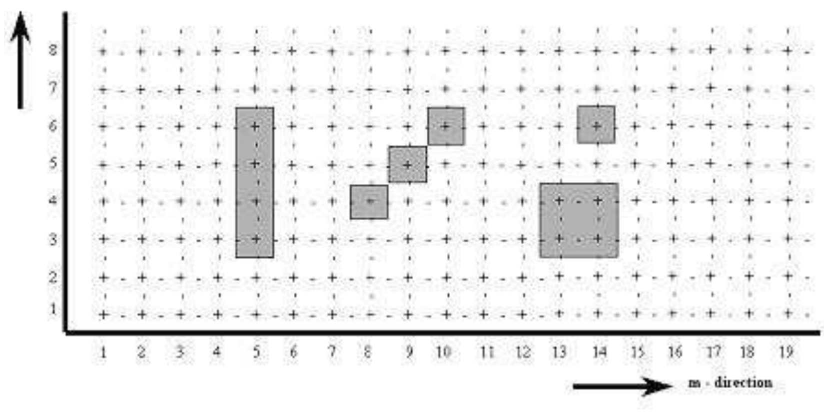

Figure 6.2: Groups of dry cells defined in a computational domain.

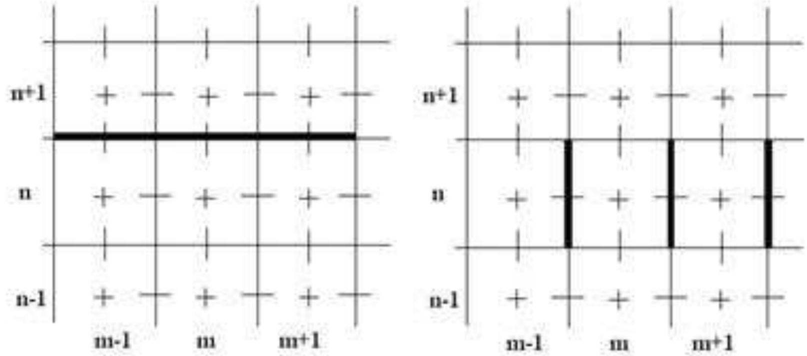

Figure 6.3: Definition of thin dams at V- and U-nodes.

6.2 Thin dams

Thin dams are defined as infinitely thin vertical walls. They are located at velocity nodes and prohibit flow exchange and fluxes of scalars between the two adjacent computational grid cells without reducing the total wet surface and the volume of the model. This functionality can be used both in 2-D or 3-D mode applications. Note that, in 3-D mode, thin dams will block the flow and scalar exchange at all vertical layers.

The purpose of a thin dam is to represent small obstacles (e.g. breakwaters, dams) in the model which have sub-grid dimensions, but still large enough so that they have an impact on the local flow pattern. They can be defined at the velocity (U- or V-)nodes, either as single elements or as a line of thin dams. An example is depicted in Figure 6.3.

COHERENS defines the bathymetry at the centre of the grid cells (C-nodes). In this way it is possible to apply a thin dam while having a different bathymetry on both sides of the dam.

Moreover, some restrictions regarding application of thin dams are identified:

- Thin dams can only be specified along lines parallel to one of the numerical grid axes.

- No thin dams can (obviously) be defined at open boundaries or at the edges of the computational grid.

- Thin dams perpendicular to open boundaries are allowed.

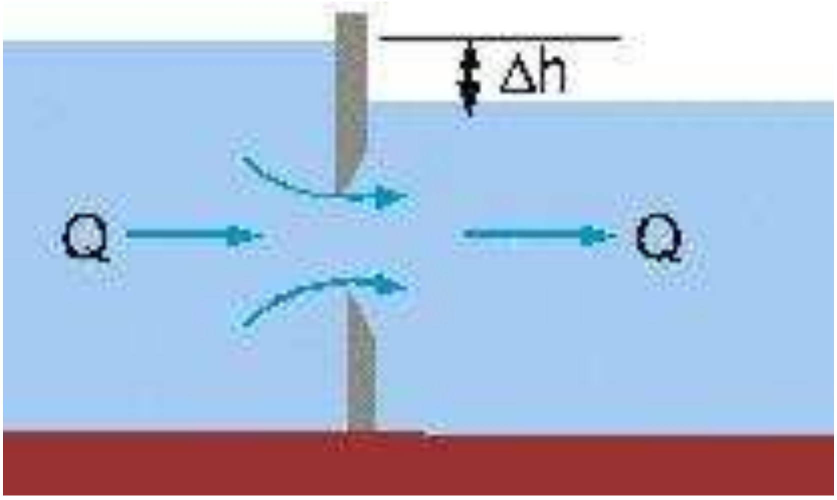

Figure 6.4: Scheme of a barrier (submerged structure).

6.3 Weirs and barriers

The aim of this model unit is twofold, the first aim is the schematization of structures that can be submerged like "levees" or "dikes". These structures are denoted as"'weirs". Weirs are fixed non-movable constructions that generate energy losses to the flow due to contraction and expansion of the flow (see Figures 6.7 and 6.8). The second aim is a particular case of "weirs", further denoted as "barriers", oriented to schematize structures that can be submerged and present an opening close to the bottom, denoted as orifices (e.g. current deflecting walls), generating energy losses due to contraction and expansion of the flow (see Figure 6.4). Both energy losses are calculated using similar equations.

Upstream of the structure the flow is accelerated due to contraction and downstream the flow is decelerated due to expansion. This expansion introduces an important energy loss due to turbulent friction, and this loss needs to be calculated. This energy loss is added as an opposing force in the momentum equation by adding an extra term to the momentum equation.

The energy loss generated by the structure is not computed directly by the convective terms in the momentum equations, but is parameterized and added in the momentum equations by adding an extra term sink term on the right hand side of the 2-D and 3-D momentum equations, given by

$$\mathcal{S}_{wb}(U) = -g\frac{\Delta\eta}{h_1}\frac{U}{|\overline{u}|}, \quad \mathcal{S}_{wb}(V) = -g\frac{\Delta\eta}{h_2}\frac{V}{|\overline{v}|} \tag{6.1}$$

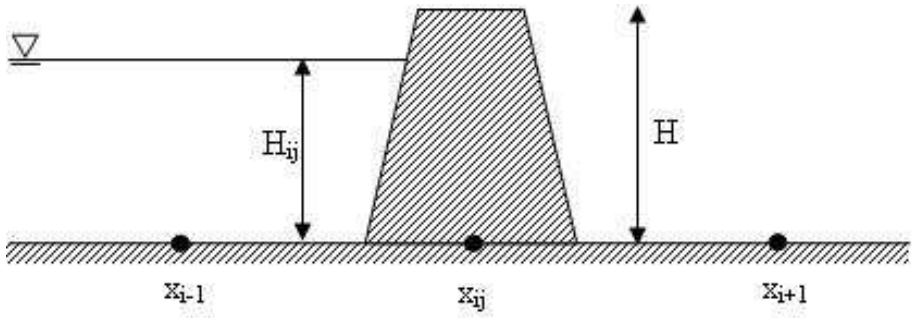

Figure 6.5: Blocking of flow when the water depth is less that the crest level.

in the 2-D case, and

$$\mathcal{S}_{wb}(u) = -g\frac{\Delta\eta}{h_1}\frac{u}{|\overline{u}|}\,, \quad \mathcal{S}_{wb}(v) = -g\frac{\Delta\eta}{h_2}\frac{v}{|\overline{v}|} \tag{6.2}$$

in the 3-D momentum equations, where η is the so-called "energy level" defined as $\eta = E/g$ so that the energy loss is given by $\Delta E = g\Delta\eta$.

6.3.1 Weirs

This type of structure is similar to a thin dam, except that a weir can be inundated, in which case an energy loss is generated. This structure will work as a thin dam in cases where the total water depth upstream of the structure is less than the crest level of the structure. In this case a blocking of flow exchange is imposed by the module (see Figure 6.5). The blocking process is present in 2-D as well as in 3-D mode simulations. Depending on the simulation mode, the blocking works differently. In the 2-D case, there is no necessity to define additional parameters since the complete water column is blocked. However for 3-D simulations, some additional considerations should be taken.

For the 2-D mode, the implementation of the model unit follows a certain criteria. In case of a U-node weir perpendicular to the X-axis, a "blocking" of the flow is imposed when the total water depth at the upstream side is less than the height of the crest of the weir, whereas a loss of energy is imposed when the total water depth is greater than the height of the crest. Therefore, the following scheme is used, where the verification of the water depth is performed at the C-nodes upstream (H_{ij} if $U > 0$ or $H_{i+1,j}$ if $U < 0$) and downstream ($H_{i+1,j}$ if $U > 0$ or $H_{i+1,j}$ if $U < 0$) of the location of the weir:

$H_{i,j} > H_{weir}$ and $H_{i+1,j} > H_{weir} \rightarrow$ loss of energy

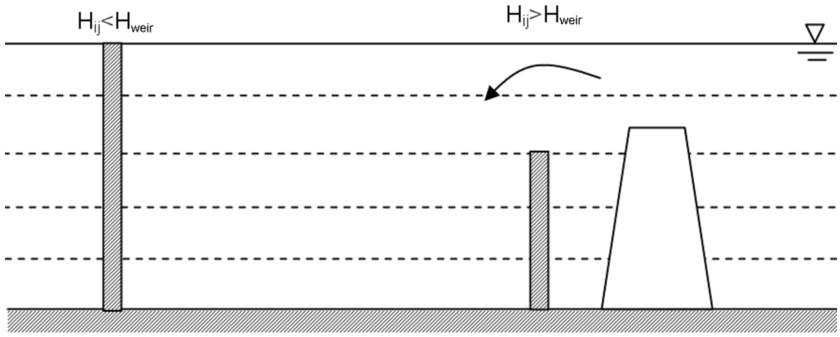

Figure 6.6: Scheme of a "weir" in 3-D mode for non-submerged and submerged conditions (compared with the height of a structure).

$H_{i,j} > H_{weir}$ and $H_{i+1,j} < H_{weir} \rightarrow$ loss of energy

$H_{i,j} > H_{weir}$ and $H_{i+1,j} < H_{weir} \rightarrow$ loss of energy

$H_{i,j} < H_{weir}$ and $H_{i+1,j} < H_{weir} \rightarrow$ blocking

For 3-D simulations, it is necessary to make some additional considerations. When the total water depth is less than the height of the crest of the structure, a "blocking" will be imposed. Then, all the layers will be blocked for flow exchange (see Figure 6.6 at the left side). When the total water depth is greater than the crest of the structure, a partial blocking process is imposed. A limited number of vertical grid layers will be blocked allowing flow over the structure. However, it is necessary to consider that the height that corresponds to the number of blocked layers will not be equal to the height of the structure (see Figure 6.6 at the right side). Hence, it is assumed that the layer that corresponds to the level of the crest will be blocked if less than 50% of this layer is above the crest and taken as open otherwise.

The loss of energy is calculated from the discharge rate defined for a weir. Two conditions are defined: "free flow" where the water depth downstream of the structure has no influence on the structure (see Figure 6.7), and "submerged flow", where the downstream water depth has an influence on the structure (see Figure 6.8). The conditions for free and submerged flow are determined from the value of the modular limit m, see Table 6.1.

In table 6.1, it is assumed that $U > 0$ so that H_{i-1} and H_{i+1} are the water depths upstream, respectively downstream of the structure.

Table 6.1: Conditions for the determination of free or submerged

Flow condition	First direction →	Second direction ←
Free flow	$H_{i+1}/H_{i-1} \leq m$	$H_{i-1}/H_{i+1} \leq m$
Submerged flow	$H_{i+1}/H_{i-1} > m$	$H_{i-1}/H_{i+1} > m$

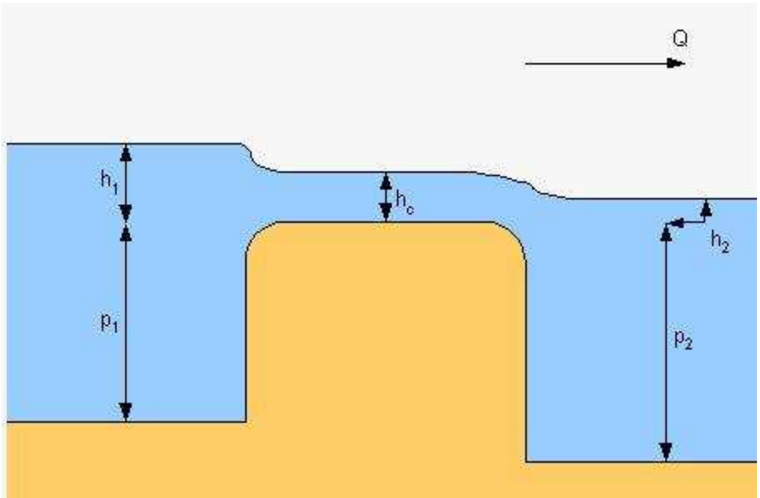

Figure 6.7: Scheme of free flow over a weir.

6.3.1.1 Free flow

In the case of a free flow, the flow rate is defined by:

$$Q = C_{st} b \left(h_1\right)^{3/2} \tag{6.3}$$

where Q denotes the discharge, b the width of the approaching channel, h_1 the water depth upstream of the structure (related to the energy head) and C_{st} is a coefficient that includes the shape of the approach channel and the geometry. Generally this coefficient is considered as a calibration coefficient and should be provided by the user. This coefficient was originally designed for open channel flow. Technical literature suggests different values for this coefficient (mostly around unity). However, preliminary tests of flow under tidal conditions showed that lower values are required to guarantee numerical stability. Hence, it is suggested to use values between 0 and 1.

For a weir defined at a U-node, one has $Q = bU$ where U is the depth-integated current. Equation (6.3) can then be solved for h_1:

$$h_1 = \left(\frac{U}{C_{st}}\right)^{2/3} \tag{6.4}$$

In free flows, it is assumed that the loss of energy equals the difference of

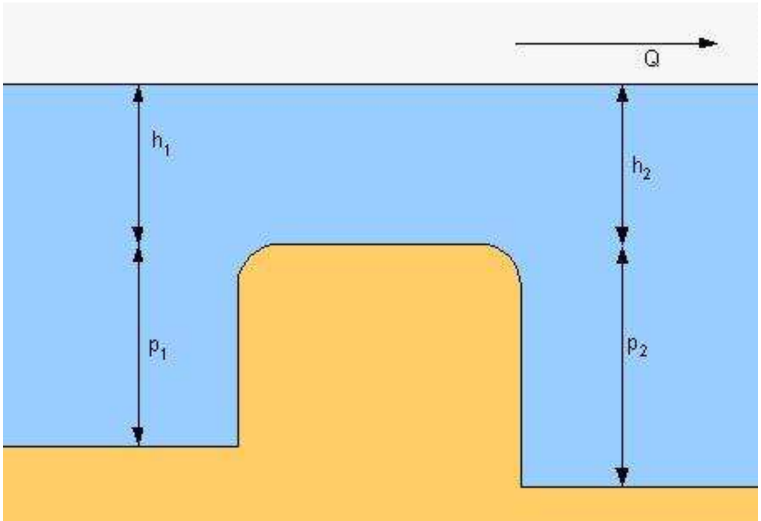

Figure 6.8: Scheme of submerged flow over a weir.

water depths upstream of the weir so that

$$\Delta\eta = H_1 - (h_{cr} + h_1) \tag{6.5}$$

where H_1 is the (total) water depth upstream of the weir and h_{cr} the height of the weir crest.

6.3.1.2 Submerged flow

For submerged flow the water depth downstream of the weir influences the water level upstream (see Figure 6.8). Its flow rate is defined by:

$$Q = C_{st} b h_1 \left(\frac{h_1 - h_2}{1 - m} \right)^{1/2} \tag{6.6}$$

where Q denotes the discharge, b the width of the approaching channel, h_1, h_2 are the water depths (relative to the energy head) upstream, respectively downstream of the structure, C_{st} is a coefficient that includes the shape of the approach channel and the geometry and m denotes the modular limit (Bos, 1998). The modular limit, which is a user-defined parameter, relates the downstream energy level to the energy level over the weir crest, presenting values between 0.6 and 1.0.

The energy loss is calculated differently in submerged flows, since the downstream conditions are affecting the flow over the weir, and equals the difference of water depths upstream and downstream the structure

$$\Delta\eta = h_1 - h_2 \tag{6.7}$$

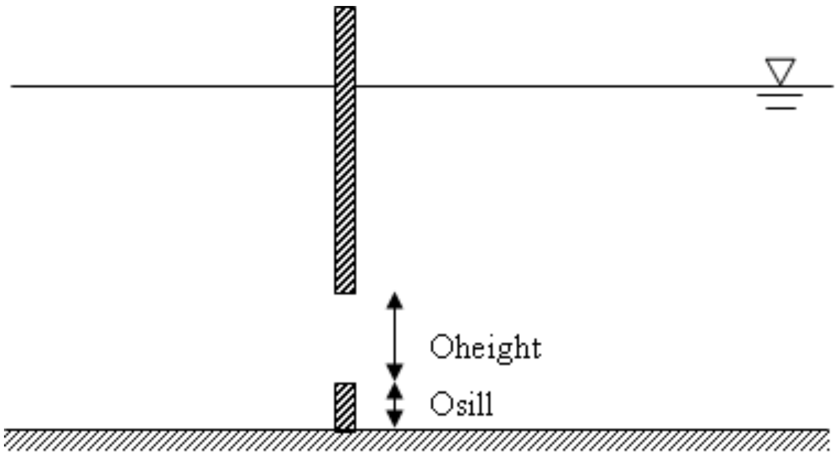

Figure 6.9: Sketch of a barrier.

Elimiminating h_2 from (6.6) and (6.7) one has

$$\Delta\eta = (1-m)\Big(\frac{Q}{C_{st}bh_1}\Big)^2 \tag{6.8}$$

Since $Q = bU$ for a weir at a U-node, this gives

$$\Delta\eta = (1-m)\Big(\frac{U}{C_{st}h_1}\Big)^2 \tag{6.9}$$

In summary, the following parameters are needed to define a weir: weir crest h_{cr}, discharge coefficient C_{st} and modular limit m.

6.3.2 Barriers

For structures of the barrier type, it is considered that there is an opening close to the bottom, where users can define the width of the opening and the height of the sill above the sea bed (see Figure 6.9). Since this structure defines a partial blocking of vertical layers, it is more oriented to be applied in 3-D mode simulations. The blocking process is similar to the process described for weirs. This functionality can alternate between weirs and barriers by defining the dimension of the opening close to the bottom. In case the opening is closed, a "weir" is defined. If the opening is different from zero, a "barrier" is defined.

The loss of energy is generated due to the contraction and expansion of the flow using the definition of flow in a submerged orifice (also denoted as a "current deflecting wall"), a schematic presentation of this type of structure is shown in Figure 6.10.

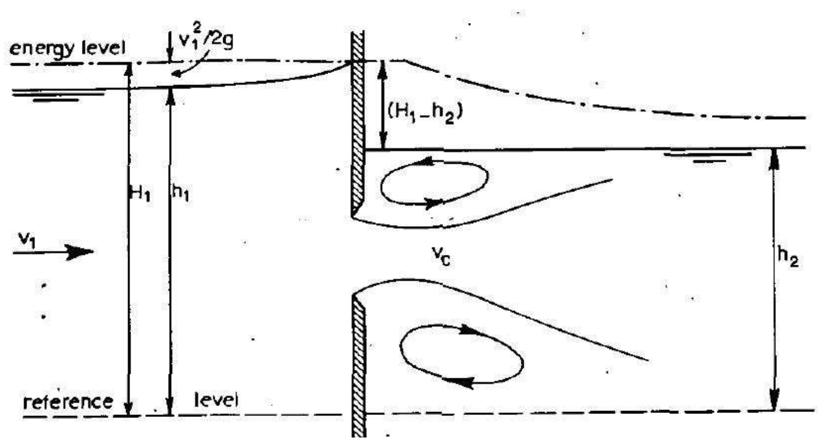

Figure 6.10: Scheme of a submerged orifice flow.

The flow rate is defined by

$$Q = C_e A \left(2g\left(h_1 - h_2\right)\right)^{1/2} \tag{6.10}$$

where Q denotes the discharge, A the area of the opening, h_1, h_2 the water depths (relative to the energy head) upstream, respectively downstream of the structure and C_e is a coefficient, defined by the user, that includes the shape of the approach channel and the geometry. Similar to the C_{st} coefficient in the weir case, the C_e coefficient should take values between 0 and 1.

The calculation of the energy loss is similar to the weir case, since the downstream conditions are affecting the flow over the structure so that the loss equals the difference of the water depths upstream and downstream of the barrier

$$\Delta\eta = h_1 - h_2 \tag{6.11}$$

From (6.10) one obtains

$$\Delta\eta = \frac{1}{2g}\left(\frac{Q}{C_e A}\right)^2 \tag{6.12}$$

or, using $Q = bu$ at U-nodes,

$$\Delta\eta = \frac{1}{2g}\left(\frac{U}{C_e O_w}\right)^2 \tag{6.13}$$

where O_w is the width of the opening.

In summary, the following parameters are needed to define a barrier: discharge coefficient C_e, orifice width O_w and height of the sill above the sea bed O_h.

6.3.3 Numerical implementation

The sink terms in the momentum equations are discretised quasi-implicitly in time using the Patankar (1980) scheme, described in Section 5.6. For the sink terms at U-nodes, one then obtains from (6.1)–(6.2):

$$\mathcal{S}_{wb}(U) = -g\frac{\Delta\eta^*}{h_1}\frac{U^{n+1}}{|\overline{u}^n|}\,, \quad \mathcal{S}_{wb}(u) = -g\frac{\Delta\eta^*}{h_1}\frac{u^{n+1}}{|\overline{u}^n|} \tag{6.14}$$

with similar expressions at V-nodes.

To prevent oscillations, a relaxation parameter θ_{wrl} has been introduced for the energy loss $\Delta\eta$ and which can be set to a value between 0 and 1 (the default value is 1). The relaxation procedure is defined as follows

$$\Delta\eta^* = (1-\theta_{wrl})\Delta\eta^{n-1} + \theta_{wrl}\Delta\eta^n \tag{6.15}$$

Water depths at weirs are determined using an upwind scheme, as described in SIMONA (2009). At U-nodes, this gives

$$\begin{array}{rcl} H^u_{ij} & = & h^u_{ij} + \zeta_{ij} \quad \text{if} \quad U_{ij} > 0 \\ H^u_{ij} & = & h^u_{ij} + \zeta_{i+1,j} \quad \text{if} \quad U_{ij} < 0 \\ H^u_{ij} & = & \max(\zeta_{ij}, \zeta_{i+1,j}) \quad \text{if} \quad U_{ij} = 0 \end{array} \tag{6.16}$$

where h_{ij} is the mean water depth above the crest. In case of a barrier, the following value is added

$$\min(H_{ij} - h_{cr}, O_h + O_w) - O_h \tag{6.17}$$

To prevent negative depths, a minimum water depth is taken, given by the same parameter d_{min} used in the inundation schemes.

6.4 Discharges

Discharges are represented as sources or sinks in the continuity, momentum or scalar equations supplied at specified (fixed or moving) locations at the surface, bottom or within the water column (e.g. discharge structures, pumping stations, discharge from moving ships . . .) by adding water to the water column with a specified salinity, temperature or contimant concentration. The discharge may or may not have a preferential direction. Figure 6.11 illustrates the schematization of a discharge at the bottom of the sea defined in a 3-D simulation.

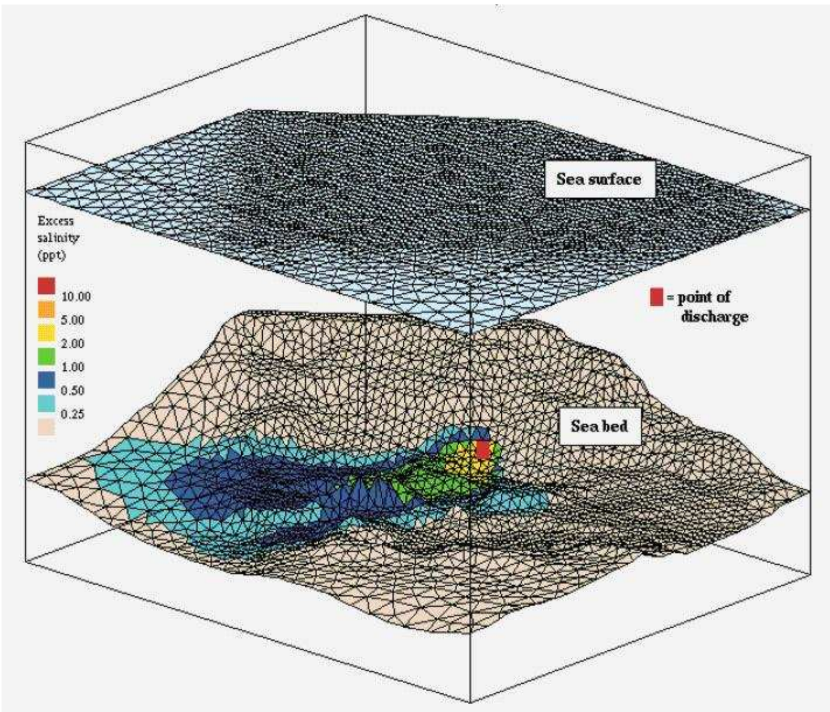

Figure 6.11: Schematization of a discharge in a 3-D simulation

Discharges can be applied in 2-D as well 3-D mode. Discharge points are defined at specified C-nodes. In the 3-D case the discharge location can be specified at a vertical grid point or distributed uniformly over all layers. Figure 6.12 depicts the definition of a discharge point in a grid cell.

Considering the direction of the discharge, two types can be defined:

1. Normal: the discharge is defined without a specific discharge direction.
2. Momentum: the momentum of the discharge is taken into account, both discharge area and direction have to be provided.

As a result, depending on the simulation mode and the type of discharge, six types may defined in COHERENS

1. 2-D normal discharge (Figure 6.13)
2. 2-D momentum discharge (Figure 6.14)
3. 3-D normal discharge distributed uniformly over all layers (Figure 6.15)
4. 3-D normal discharge defined at a specific height (Figure 6.16)

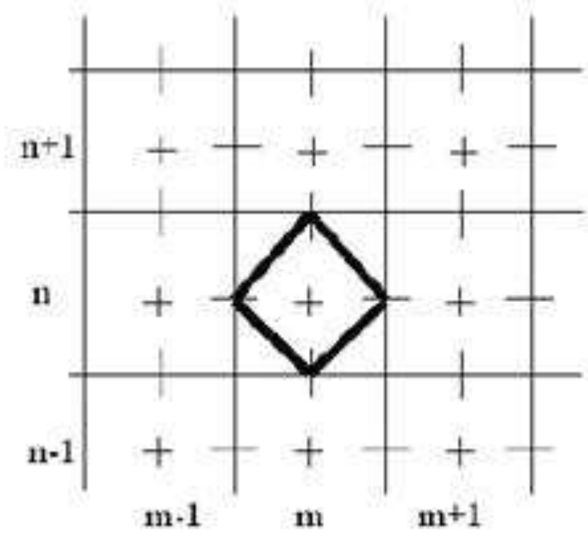

Figure 6.12: Location of a discharge point in a grid cell.

5. 3-D momentum discharge distributed uniformly over all layers (Figure 6.17)

6. 3-D momentum discharge defined at a specific height (Figure 6.18)

The schematization of discharges requires

- a modification of the continuity equation by the addition of a source term

- an additional source or sink term in the 2-D and 3-D momentum equation, in case of a directional (momentum) discharge

- an (eventually) addditional source term in the scalar transport equations. If no scalar discharge is provided by the user, the added water volume is assumed to have the same temperature, salinity or contaminant (sediment) concentration as the one at the discharge location.

The discharge locations are defined at specific heights or uniformly over the vertical and can be placed in several locations allowing the users to use several options for modelling of discharges. The following vertical locations can be used:

1. Discharge, uniformly distributed over the vertical

2. Discharge located at the bottom

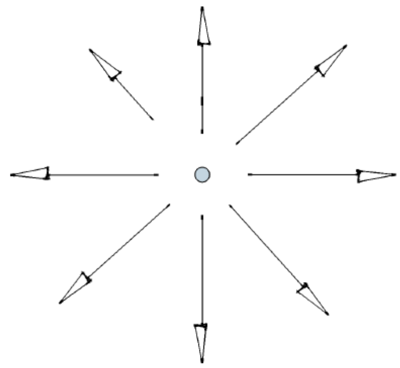

Figure 6.13: 2-D normal discharge

Figure 6.14: 2-D momentum discharge

3. Discharge located at the surface

4. Discharge located at a given distance from the sea bed

5. Discharge located at a given distance below the surface

The locations are either be fixed or moving in time.

6.4.1 Continuity equation

A source term term is added to the right hand side of the 2-D continuity equation (4.85), given by

$$q_{dis} = \frac{Q_{dis}}{h_1 h_2} \tag{6.18}$$

where Q_{dis} represents the volume discharge in m^3/s and q_{dis} denotes the "discharge speed" (m/s).

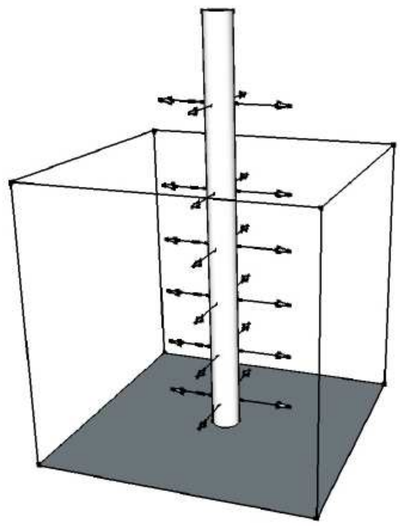

Figure 6.15: 3-D normal discharge distributed uniformly over all layers

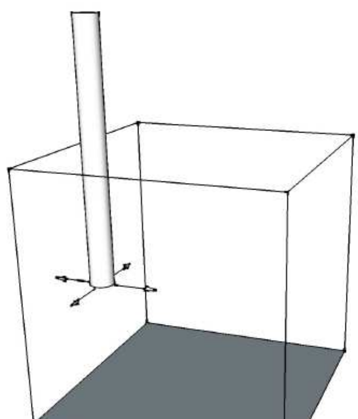

Figure 6.16: 3-D normal discharge defined at a specific height

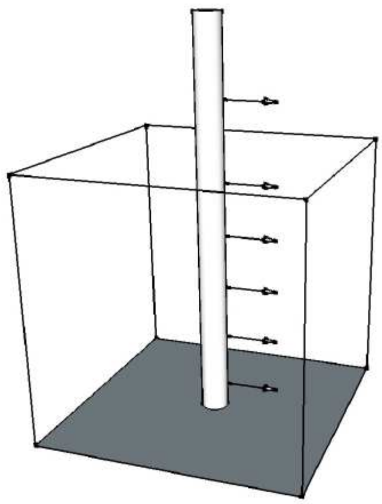

Figure 6.17: 3-D momentum discharge distributed uniformly over all layers

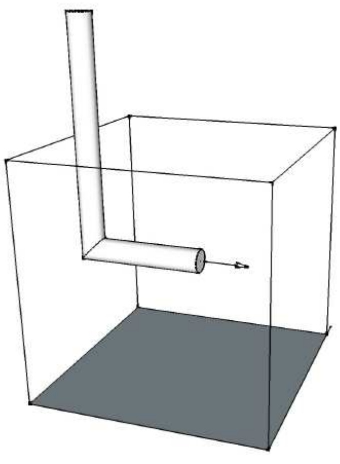

Figure 6.18: 3-D momentum discharge defined at a specific height

6.4.2 Momentum equations

A source (or sink) term is added to the right hand side of the momentum equations in case a directional discharge is specified. This is given by

$$\mathcal{Q}_m(U) = q_{dis}\Big(\frac{Q_{dis}}{A_{dis}}\cos\theta_{dis} - \overline{u}\Big)\,, \quad \mathcal{Q}_m(V) = q_{dis}\Big(\frac{Q_{dis}}{A_{dis}}\sin\theta_{dis} - \overline{v}\Big) \quad (6.19)$$

in the 2-D momentum equation (4.86–4.87), and

$$\mathcal{Q}_m(u) = \frac{q_{dis}}{\Delta_{dis}}\Big(\frac{Q_{dis}}{A_{dis}}\cos\theta_{dis} - u\Big)\,, \quad \mathcal{Q}_m(v) = \frac{q_{dis}}{\Delta_{dis}}\Big(\frac{Q_{dis}}{A_{dis}}\sin\theta_{dis} - v\Big) \quad (6.20)$$

in the 3-D momentum equations (4.61–4.62) where θ_{dis} is the discharge direction with respect to the reference axis (Eastward direction in the spherical case), A_{dis} the discharge area (in m^2) and Δ_{dis} the vertical distance over which the discharge takes place, given by H for a uniform or $h_3 = \Delta z$ for a non-uniform discharge.

6.4.2.1 Discharge of scalars

Two additional types of discharge can be defined for scalars. In the case of a "wet" discharge, water with a specified temperature, salinity or other concentration is added to the specified cells. The following source term is added to the right hand side of the scalar transport equation

$$\mathcal{P}(\psi) = \frac{q_{dis}}{\Delta_{dis}}\psi_{dis} \quad (6.21)$$

where ψ_{dis} is the user-specified temperature, salinity or concentration of the discharged water.

In the case of a "dry" discharge, no water is added. The source term takes the simple form

$$\mathcal{P}(\psi) = \psi_{dis} \quad (6.22)$$

where ψ_{dis} now represents the amount of the concentration added per unit of time. Note that a dry discharge is not allowed for temperature or salinity.

6.4.2.2 Numerical implementation

The numerical discretisation of the source terms in the continuity and scalar equations is straightforward. The discretisation of the discharge term in the momentum equations requires some case since the discharge is defined at C-nodes whereas the components of the depth integrated and 3-D current are taken at the velocity (U- or V-) nodes. The discharge term is therefore taken as an average between adjacent velocity nodes. For the U- and u-momentum equations this gives

$$\begin{aligned}
\mathcal{Q}_m(\overline{u})_{ij} &= q_{dis}\Big(\frac{Q_{dis}}{A_{dis}}\cos\theta_{dis} - (m_{i-1,j}+m_{ij})\overline{u}_{ij}\Big) \\
\mathcal{Q}_m(u)_{ijk} &= \frac{q_{dis}}{\Delta_{dis;ijk}}\Big(\frac{Q_{dis}}{A_{dis}}\cos\theta_{dis} - (m_{i-1,j}+m_{ij})u_{ijk}\Big)
\end{aligned} \tag{6.23}$$

where $\Delta_{dis;ijk}$ equals H_{ij} for uniform and $h^u_{3;ijk}$ for non-uniform discharge. The weight factor is defined as follows

$$\begin{aligned}
m_{ij} &= \frac{1}{2} \quad \text{if} \quad \text{(i-1,j) and (i,j) are interior wet velocity points} \\
m_{ij} &= 1 \quad \text{if} \quad \text{one of the velocity nodes (i-1,j) and (i,j) is interior wet and the other not} \\
m_{ij} &= 0 \quad \text{if} \quad \text{none of the velocity nodes (i-1,j) and (i,j) is interior wet}
\end{aligned} \tag{6.24}$$

The formulations at V-nodes are similar.