Types of Boundary Conditions for 1D SWEs

All CFD problems are defined in terms of their boundary (BC) and initial conditions (IC). With similar governing equations, what differentiates two distinct problems is their BCs and ICs. It is important to understand the boundary conditions and specify them in the algorithm correctly. If you are dealing with a transient unsteady problem, the ICs specify all parameters at $t = 0s$ and BCs have to be specified at all time steps.

Note that there are special nodes/values that are located right at the boundary, for instance $i = 1$ and $i = I_{max+1}$ for q and u. But also there are internal face values that require special treatment, for instance, $Fuuh(1)$ and $Fuuh(I_{max})$ because of their vicinity to the boundary.

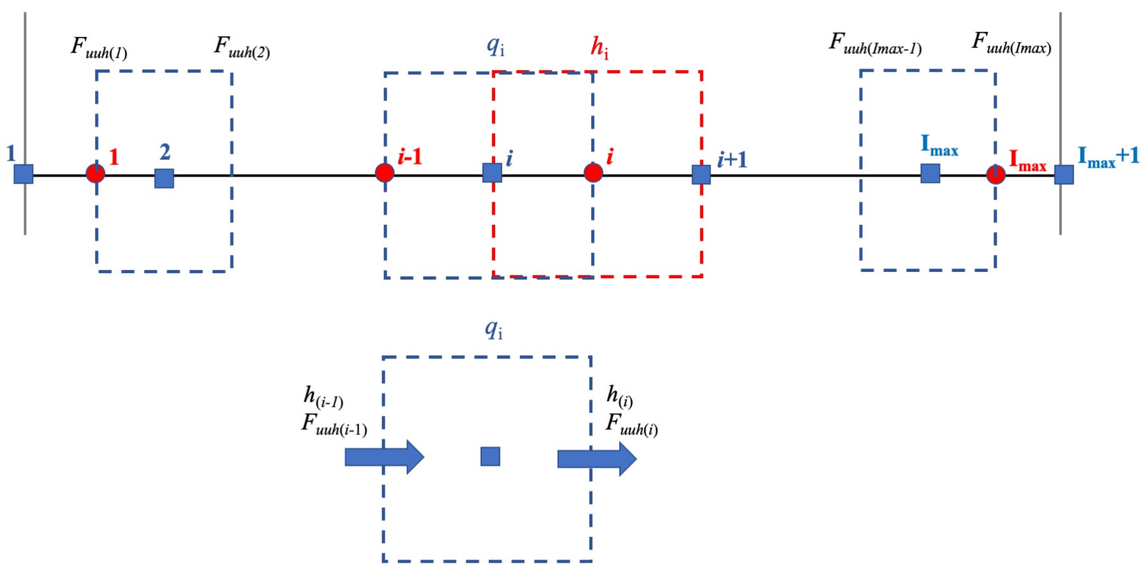

Figure 1: Orientation of a 1D SW staggered grid along the boundaries.

Some of the most common boundary conditions are:

- **Inlet**

- **Outlet**
- **Wall**
- **Prescribed depth/pressure**
- **Periodic**

In the construction of your grid, you may have to setup additional nodes/grids surrounding the physical boundary.

- **Inlet Boundary condition:** In this boundary the distribution of flow variables need to be specified at the boundaries. In a staggered grid system, the x-component of the velocity is located right at the physical boundary, making it convenient to define u or q_x at the boundary at $i = 1$.
  Additionally, you need to estimate the flux in the proximity of the inlet boundary.

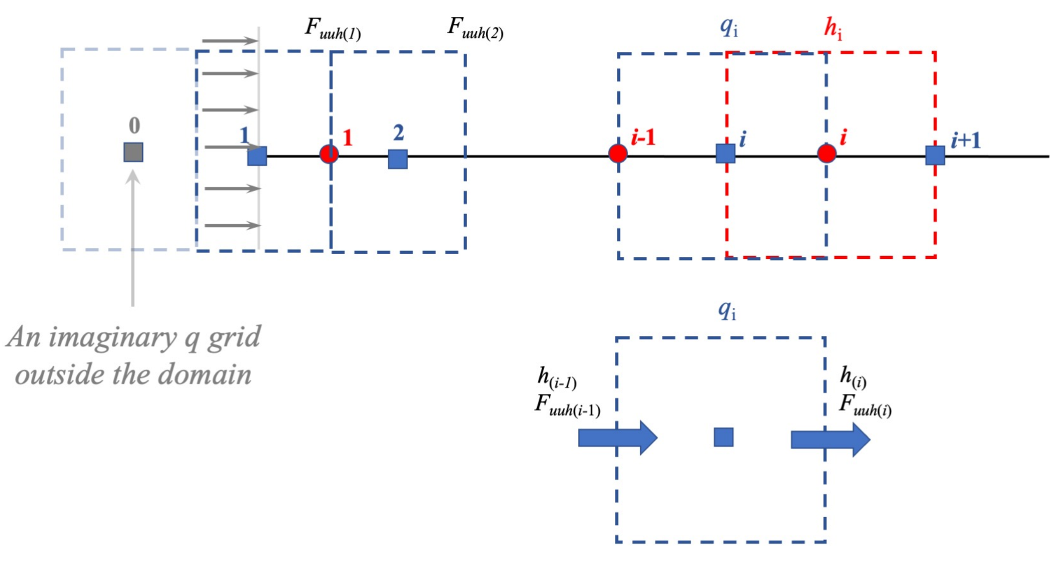

Figure 2: Inlet BC and orientation of a 1D SW staggered grid and the imaginary node.

For instance, lets say that you are using a First order Upwind (FOU) method. To approximate $Fuuh(1)$ you will only need $u(1)$,$q(1)$ or $u(2)$,$q(2)$. However, if you are using a QUICK scheme, for $Fuuh(1)$ you will need two points upstream, one of which is not available in the domain. Therefore, you can create an imaginary node on the left hand side of the inlet boundary. Since you have a constant inlet flow, you can assume that this imaginary node, demonstrated as **0** in Figure 2, also have the velocity and discharge of the inlet.

> **Note#1**: Depending on the **order** of the spatial discretization scheme, you may need additional imaginary nodes upstream.

- **Wall Boundary condition:** In this boundary all flow variables need to be specified at the boundary as 0. Similar to the Inlet condition, on the east and west boundaries, u and q_x can be easily set to 0.
  Near boundary fluxes also require special treatment. Two common options are: a)

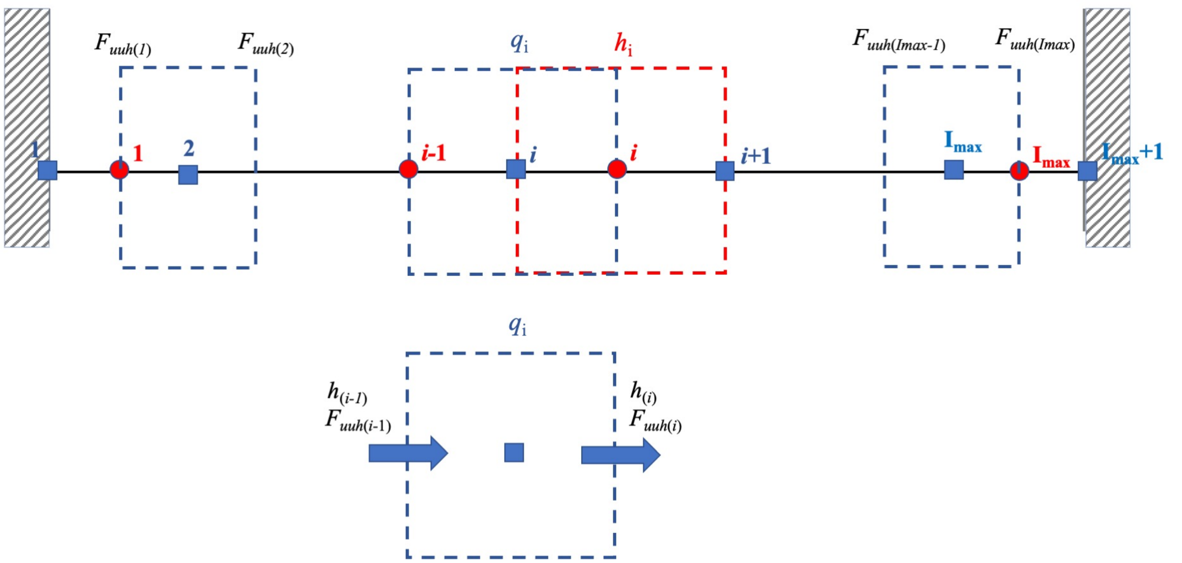

Figure 3: Wall BC and orientation of a 1D SW staggered grid.

you can either assume imaginary nodes with 0 velocity outside the domain; b) you can drop the order of spatial integration in the vicinity of the boundary. For instance, if you are using a Ultra-QUICK scheme in the domain, you can drop it to the FOU for $Fuuh(1)$ and $Fuuh(I_{max})$.

- **Outlet Boundary condition:** This often can be used in conjunction with Inlet boundary. This is the opposite of the Wall condition. In the wall condition, waves and any flow disturbance propagating towards the wall reflects back into the domain. This can be disruptive if we want to simulate an infinitely large domain. Implementation of an outlet boundary can be complex, specially in non-linear flow. In linear flow, the boundary values can be estimated as:

$$u'(x = 0) = -\sqrt{g/d} * z$$

$$u'(x = L) = +\sqrt{g/d} * z$$

where $u = \overline{u} + u'$. These values are derived based on the linearized SWEs and their characteristic lines (not discussed in the course).

- **Depth Boundary Condition:** This is where the pressure, water depth condition is exactly specified at the boundary. This is a typical boundary condition where we have free-surface tidal flow problems. As seen in A3, this can be implemented as an imaginary cell approach.

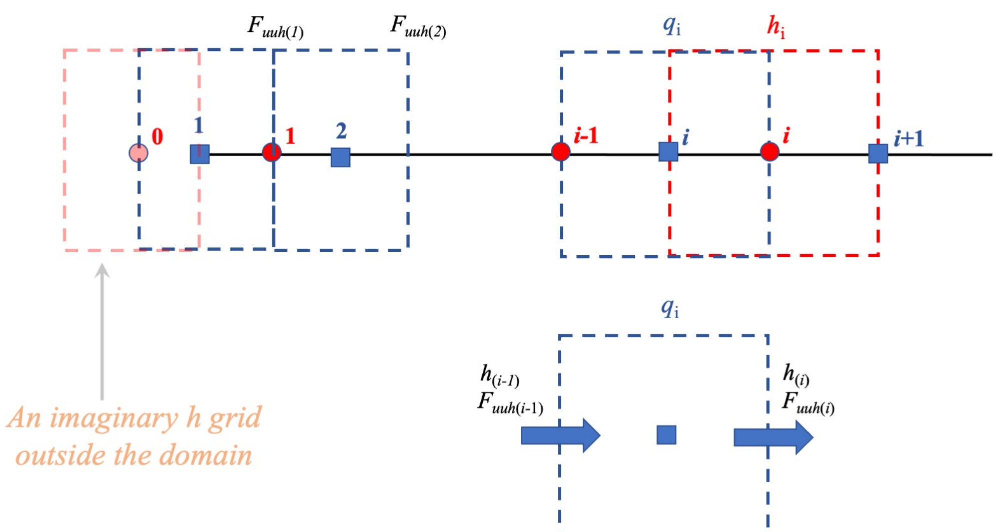

Figure 4: Pressure BC at the inlet and orientation of a 1D SW staggered grid.

- **Periodic Boundary Condition:** This boundary condition arise due to the symmetry in a problem. is where the pressure, water depth condition is exactly specified at the boundary. This for instance is applicable where you want to simulate a long river training transverse groyne system. **To be discussed in 2D flow**.