

# 1 General background

The code was created to simulate the motion of shallow water in an excited domain. The Riemann solver of the fluid solver is written based on the Riemann solver presented in [1] and some other routines were written based on the lumped mass code by Manderbacka[2].

The code is written so that it should allow an arbitrary amount of square shaped tank domains in arbitrary locations. However, as of August 25, 2015, the testing of the code has been restricted to two tanks located beside each other with the interface of the tanks located in  $X, Y, Z = 0, 0, 0$  of the hull coordinate system. Additionally, there is a small bug in the initialization routine when other than horizontal initial water level is set that causes a wrong amount of mass added to the tank.

# 2 Structure of the code

The code consists of 7 different modules and a driver main program that calls all the modules. Figure 1 shows a diagram of the module structure. All the modules are written in separate files and 3 of the 7 modules (ARRAYS,CONSTANTS,WORKSPACE) have no routines and are used only as placeholders for the simulation variables. The module ARRAYS consists of the arrays, which are solved during the simulation. The module CONSTANTS has all the variables, which are either taken from the input files or calculated directly from them at the beginning of the simulation. The module WORKSPACE has the variables which are not solved in the simulation but have a changing value, for example time.

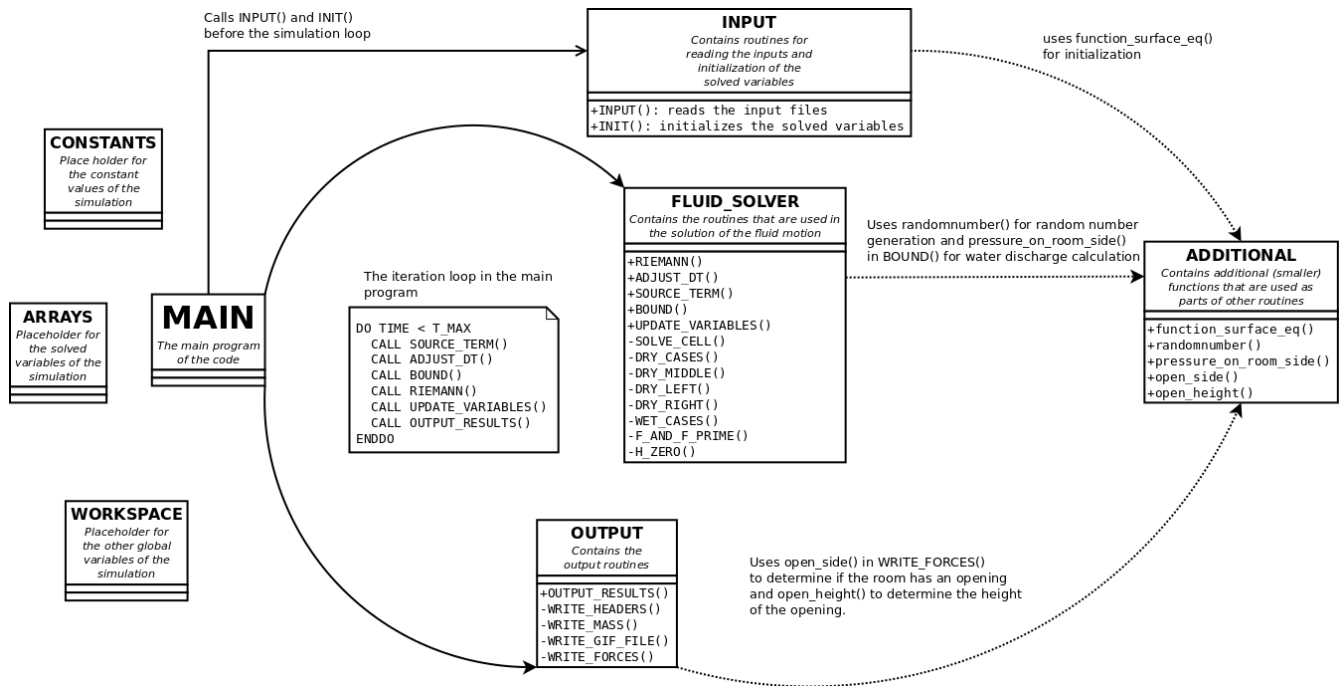


Figure 1: A diagram of the code structure.

The main program begins with reading in the input files and initialization of the solved variables. These routines are called from the INPUT module before the simulation loop begins. The actual simulation loop begins with calculation of the source terms using the routine SOURCE\_TERM() from the FLUID\_SOLVER module. The purpose of the SOURCE\_TERM() routine is to calculate the additional body forces ( $A_Z$  and  $F_Y$ ) at time  $t$  due to motion of the domain.

Next the routine ADJUST\_DT is called. This routine first finds the maximum wave speed from all the fluid domains and sets the time step so that that the Courant number gets the input value CFL\_MAX. Since the stability limit for the method is  $CFL < 0.5$  and the velocities are adjusted in the operational splitting phase, the CFL\_MAX value should be well below 0.5. It should be noted that setting the CFL\_MAX to very small values may result in unsatisfied mass balance and the mass balance should be checked after every simulation.  $CFL\_MAX \approx 0.4$  has been found to be the optimal for multiple simulations.

After the time step has been adjusted, the boundary conditions are set in the subroutine BOUND() from the FLUID\_SOLVER module. The boundary conditions are taken into account by additional ghost nodes located at both ends of all fluid domains (rooms). If the room has no openings, the domain ends are treated as solid walls, if the room has an opening from which the water of another room can flow in, the water exchange is handled through Bernoulli's equation between the corresponding domain nodes (not ghost nodes). Using the notation in 2, the volume flow rate from left room to right room can be calculated from

$$v = \text{sgn}(p_L - p_R) \sqrt{\frac{2}{\rho} |p_L - p_R|} \quad (1)$$

$$\dot{Q} = C_d v h_{hole},$$

where  $C_d$  is the discharge coefficient taken as an input and  $p_{L/R}$  are the hydrostatic pressures at the domain nodes closest to the to the interface of the two rooms. Since the solved variables are the water height and the velocity, the amount of mass coming or leaving the room must be converted to either an increase or decrease of water height. The changes of water heights are

$$\Delta h_L = \frac{\dot{Q} \Delta t}{\Delta y_L}, \quad \Delta h_R = \frac{\dot{Q} \Delta t}{\Delta y_R}. \quad (2)$$

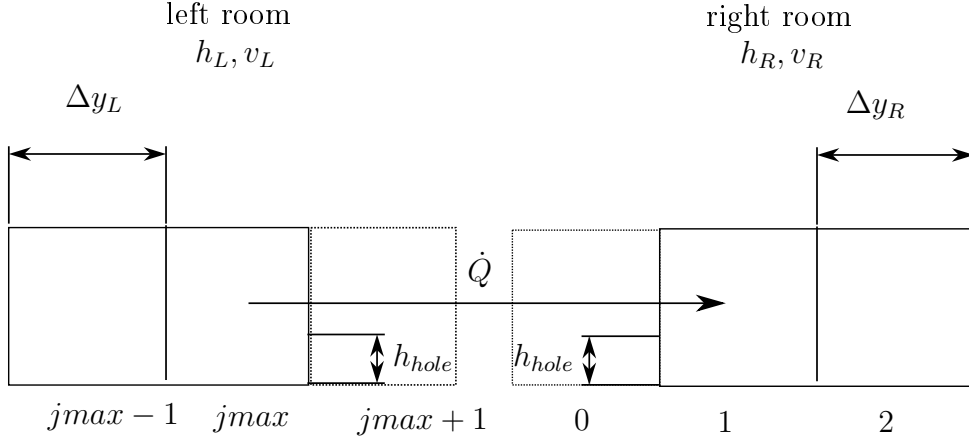


Figure 2: A sketch of the water exchange between two rooms. The  $jmax + 1$  node in the left room and the 0 node in the right room denote the ghost cells and the lower indices  $L$  and  $R$  denote the parameter values of the left and right rooms respectively.

The increase or decrease of water height is added to or subtracted from the closest corresponding nodal values of the room interface. In this case

$$\begin{aligned} h_L^{jmax} &= h_L^{jmax} - \Delta h_L \\ h_R^1 &= h_R^1 + \Delta h_R. \end{aligned} \quad (3)$$

The momentum exchange between the two rooms is handled either as an increase or decrease of the body force array  $F\_Y$ .

$$\Delta f_{mom} = \rho \dot{Q} v h_{hole} \quad (4)$$

Again, the change of momentum is added to the first domain nodes corresponding to the room interface

$$\begin{aligned} F\_Y_L^{jmax} &= F\_Y_L^{jmax} + \Delta f_{mom} \\ F\_Y_R^1 &= F\_Y_R^1 - \Delta f_{mom}. \end{aligned} \quad (5)$$

When the time step has been adjusted and boundary conditions set, the main program calls the RIEMANN() routine from the FLUID\_SOLVER module. This routine uses multiple other routines (denoted with - in figure 1) located in the same module and calculates the new values for the velocities and water heights. The details of the fluid solver are discussed in more detail in the longer report that comes with the code. If there are no additional body forces, the velocities and water heights calculated by the RIEMANN() routine are the final values of the iteration round. If not, the velocities are adjusted in the subroutine UPDATE\_VARIABLES() using operational splitting and Euler time integration

$$v^{n+1} = v_{rcm} + \Delta t F\_Y. \quad (6)$$

Without domain motion the last term on the RHS in the above equation vanishes.

The simulation loop ends with the call of routine OUTPUT\_RESULTS() from the OUTPUT module. Although this routine is called at every time level, the results are output only at intervals taken as an input. Thus subroutine OUTPUT\_RESULTS() first checks whether it is time to output and outputs if necessary.

## References

- [1] E. F. Toro, *Shock-capturing methods for free-surface shallow flows*. John Wiley, 2001.
- [2] T. Manderbacka, J. Kulovesi, M. A. C. Celis, J. E. Matusiak, and M. A. S. Neves, “Model tests on the impact of the opening location on the water motions in a flooded tank with two compartments,” *Ocean Engineering*, 2014.