

Simple pond with a contaminant undergoing first order decay

December 31, 2016

This example is intended to serve as a quick start guide by showing the steps needed to create a simple model of a pond with a simple water quality process consisting of a single hypothetical contaminant (named PCB) undergoing first order decay. The pond is assumed to be divided into two compartments where one of the compartments receive contaminants from a stream and the second compartment discharges into an imaginary adjacent receiving water body (conceptually represented in Figure 1a). Figure 1b shows the temporal variation of water flow rate and PCB concentration in the inflow. The outflow rate is assumed to be controlled by a weir with a given rating curve to be discussed later.

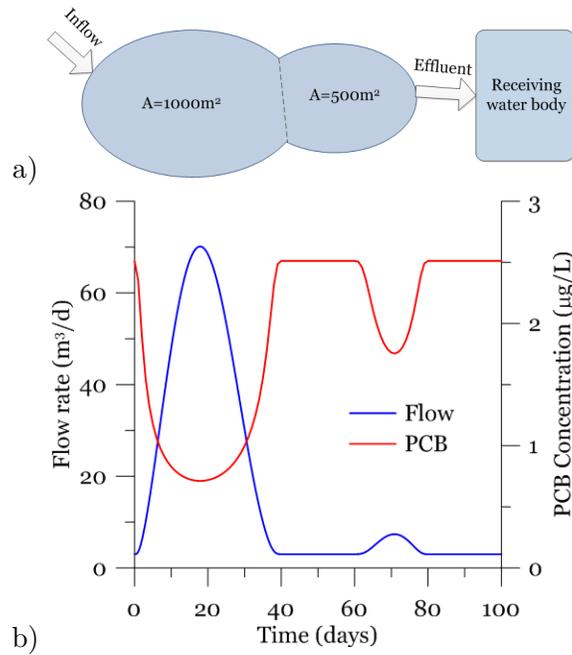


Figure 1: a) Schematic of the single two compartment pond model, b) Temporal variation of the hypothetical water flow and contaminant concentration into the pond

Below are the steps to set up the model:

1. **Launching the program and creating a project:** Launch GIF-Mod program through the start menu by first choosing the GIFMod

folder and clicking on GIFMod icon or the icon on the desktop.

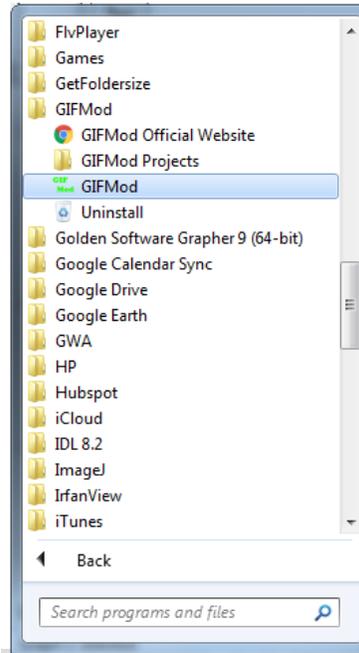


Figure 2: GIFMod icon in the start menu

2. Setting the duration of the project:

- From the **Project Explorer** window on the right hand side of the screen select **Project Settings**→**Settings**
- Scroll down the **Properties** window on the bottom right side of the screen until you see the item labeled **Simulation End Time**, and pick 4/9/1900. This essentially makes the duration of simulation to be 100 days starting from day zero and ending at day 100.

3. **Add a pond to the model:** Add a pond by clicking on the pond icon  on the top toolbox (Figure 3).

4. **Assigning the properties of the pond:** Click on the pond object added to the project so it's properties appear in the **Properties** window on the lower right side of the screen.

- From the **Properties** window change the name of the object from the default "Pond (1)" to "Main Pond".
- Change the bottom area to $1000m^2$.
Note: Most physical properties can be entered using multiple units provided in the drop-down menu adjacent to the text boxes where they are entered.

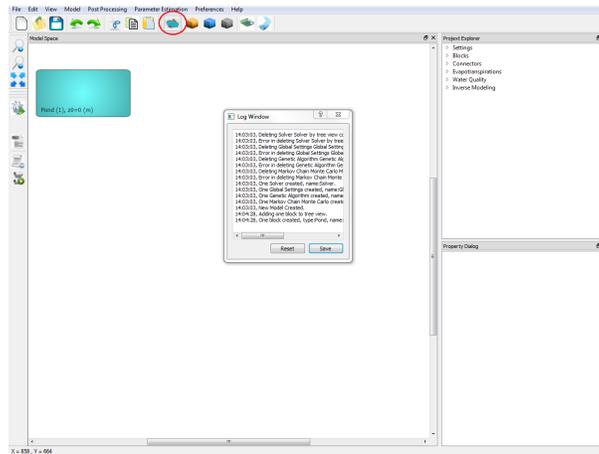


Figure 3: Adding a pond

- In the **Properties** window, set "Initial Water Depth" to 2m. You can now run the model. Click on the "forward run" icon  on the left side of the screen.
 - The run window should appear and you can see the simulation as it progresses. The graph in the run window shows the time step sizes being used during the simulation. Because the model in this case is very simple, the time step size keeps linearly increasing.
 - To see the results right click on the "Main Pond" block and select "Water Depth" from the "Plot Hydraulic Results" menu. As it is expected the water depth remains constant at 2m since no inflows are entering the pond and now outflows are leaving it.
5. **Adding the second pond:** The second pond can be added similarly to the first pond but another feature of the program called "Grid" can also be used to create it. Let's explore that here.

- Right click on the "Main Pond" and select "Make Array of Blocks" from the drop-down menu.

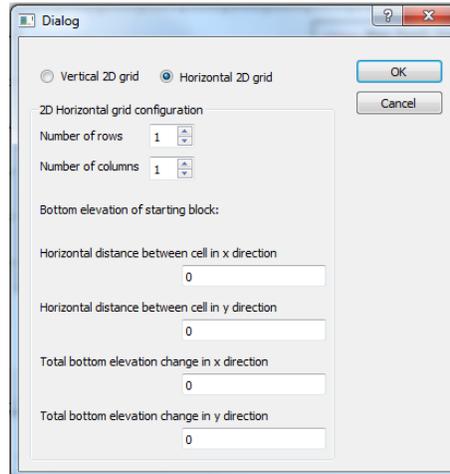


Figure 4: Grid Dialog Box

- Click "Horizontal 2D array" and change the "Number of Columns" to 2.
- Type 50 in the text box labeled as "Horizontal distance between cells in x direction". This distance is the presumed distance between the centroids of each pond that will be used to calculate the hydraulic gradient for flow and concentration gradients for diffusive/dispersive flux. Press "OK" to generate the grid. The grid feature automatically connects the blocks that are generated together. It is assumed that the bottom elevation of the two compartments are equal.

Notes:

- The newly created block also will have the same properties as the original block.
- The connector connecting two blocks will by default gets its name as the combination of the blocks being connected.
- Now we need to set up the properties of the connector and the newly generated block. Click on the newly generated block and from the **Properties** window, change the name to **Embayment**.
 - Click to select the connector connecting the two blocks. Because

the name of the second block at the time of creation has been **Main Pond (1)**, the name of the connector will be assigned as **Main Pond - Main Pond (1)**, however the name can now be changed to anything else. Let's call it **Main-Embayment**.

- From the **Properties** dialog change:
 - **Manning Roughness coefficient:** 0.11
 - **Width:** 10m

The width represent the cross sectional width of the pond at the interface between the two compartments.

6. **Adding the receiving water body:** Adding a dummy block to represent the receiving water body is useful from several points of view. First it makes imposing the downstream boundary easy and secondly it can help to obtain information about the flow volume and total contaminant loading into the receiving water body.

- Add another pond:
 - Change the name of the pond to "**Receiving water**".
 - **Bottom Area:** 10000 m². Note: This is not important as we will fix the head of the receiving water body later to make sure there will be no back-flow from it into the ponds.
 - **Head-Storage Relationship:** -1. The "Head-Storage relationships" box provides a way for the user to specify a user defined Head-Storage relationship to replace the default hydrostatic relationship. By typing the -1 value the hydraulic head of the receiving water body will be -1 regardless of its storage.
- Connect the **Embayment** block to the receiving water block by clicking on the edge of the first and dragging to the second when the cursor sign turns to a cross.
- Set the length of the newly added connector to 1.0. The value entered here does not affect the result of the simulation because we

7. **Setting the connector to the "receiving water" to behave as a weir:** Click on the **Embayment-Receiving water** connector. From the **Properties** window change the **Type** to "Rating curve". Then click anywhere in the **Properties** window. The following equation is used for rating curves: $Q = Coefficient(h - Datum)^{Exponent}$.

Set:

- **Exponent:** 2.0

- **Coefficient: 200**

- Datum to 1.8m. This means the flow will occur only if the water depth is above 1.8m.

8. **Setting the inflow time-series:** The inflow files are provided as text files with .csv format. Flow rate and concentration of constituents or particles are provided in a single file for each inflow. Two columns for each constituent is needed, the first one indicating time and the second, the value of the constituent. This allows introducing flow and each constituent with it's own time increment. The time increments in the inflow file do not need to be uniform as the program interpolate the values at the computational time step. When the length of time series for different constituents are different, the terminated time-series should be filled with ", ,". The inflow files should have a heading line specifying the nature of each pair of columns. Figure 5 shows an example inflow file structure. The "names" keyword in the heading indicates that the names of the constituents in the inflow are indicated in that line. The inflow file used in this tutorial is provided in the example folder. Here we include PCB concentration in the inflow although as long as the constituent PCB has not been indicated in the model, the entries related to PCB will be ignored. As it is seen in Figure 5 the PCB concentration time-series in the inflow is shorter than the flow rate and the empty cells corresponding PCB are left blank.

- To add the inflow to the main pond first select the **Main Pond** and from the **Properties** window click on the box labeled "Inflow Time Series".
- Select the file called "inflow.txt" from **User/Documents/GIFMod/Projects/Examples/2 ponds pcb** using the file dialog box and click OK.
- Right-click on the **Main Pond** block and choose **Plot Inflow Properties** and then click on the "Flow" item. A graph window will appear where the flow rate is shown. The hydraulic component of the model is ready and now the model can be run.

9. Running the hydraulic module:

- Click on the run icon  on the left hand side tool bar. And wait until the simulation is over. You can also check the warnings show in the "Log Window".

```

1 names, flow, PCB
2 0,3,0,2.511886432
3 0.1,3.000581287,1,2.35314198
4 0.2,3.004646578,2,1.898929719
5 0.3,3.015650297,3,1.552610261
6 0.4,3.03695544,4,1.327888892
7 0.5,3.071738548,5,1.174766492
8 0.6,3.12287333,6,1.06459884
9 0.7,3.192811614,7,0.982051435
10 0.8,3.283483928,8,0.918414512
11 0.9,3.396239792,9,0.868413151
12 1,3.531839223,10,0.828675787
13 1.1,3.690494874,11,0.79695321
14 1.2,3.871953278,12,0.771694824
15 1.3,4.075597673,13,0.751805909
16 1.4,4.300554751,14,0.736502113
17 1.5,4.545791878,15,0.72521899
18 1.6,4.810197166,16,0.71755422
19 1.7,5.092639979,17,0.713230136
20 1.8,5.392013072,18,0.71206946
21 1.9,5.707259395,19,0.713980098
22 2,6.037387154,20,0.718946583

87 8.5,39.92694591,85,2.511886432
88 8.6,40.48388749,86,2.511886432
89 8.7,41.03857999,87,2.511886432
90 8.8,41.5909013,88,2.511886432
91 8.9,42.14073045,89,2.511886432
92 9,42.68794771,90,2.511886432
93 9.1,43.2324345,91,2.511886432
94 9.2,43.77407347,92,2.511886432
95 9.3,44.31274847,93,2.511886432
96 9.4,44.84834454,94,2.511886432
97 9.5,45.38074797,95,2.511886432
98 9.6,45.90984626,96,2.511886432
99 9.7,46.43552814,97,2.511886432
100 9.8,46.95768359,98,2.511886432
101 9.9,47.47620385,99,2.511886432
102 10,47.99098139,100,2.511886432
103 10.1,48.50190999,,
104 10.2,49.00888466,,
105 10.3,49.51180175,,
106 10.4,50.01055886,,
107 10.5,50.50505491,,
108 10.6,50.99519016,,
109 10.7,51.48086617,,

```

Figure 5: A sample inflow input file

- When the simulation is done, right-click on the **Main Pond** and choose "Plot Hydraulic Results" and the click on "Water Depth". A graph showing the water depth variation in the **Main Pond** over the course of simulation will appear.
- Right click on the connector connecting the **Embayment** to the **Receiving Water** and choose "Plot Flow". A graph showing the discharge flow rate variation vs. time will appear. You can inspect other results available for other blocks or connectors sim-

ilarly.

10. Adding a water quality constituent:

- Right-click on the "Constituents" from **Project Explorer**→**Water Quality** and click on "Add Constituent".
- Click on the "Constituent" that was just added.
- Change the name of the constituent to "PCB" from the **Properties** window.
- Leave the other properties unchanged. At this time we will ignore diffusive exchange of PCB between the blocks.

11. Adding a reaction:

- Right click on "Reaction Parameters" item from **Project Explorer** → **Water Quality** → **Reactions** → **Reaction Parameters** and click on "Add Reaction Parameters".
- Using the **Properties** window, change the name of the parameter to "kd" (decay rate) and change the value to 0.02 meaning 0.02 day^{-1} .
Note: The time dimension for all reaction parameters is day.
- Right-click on "Reaction network" under **Water Quality**→**Reactions**. The window shown in Figure 11 will appear. This window allows entering Peterson's matrix representing the reactions between the constituents.

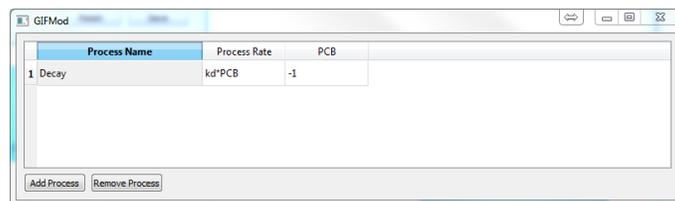


Figure 6: Reactions Window

- Enter "Decay" under "Process Name".
- Enter "kd*PCB" under "Process Rate" heading. This means that PCB is undergoing a first-order decay with a rate constant of "kd".

Note: Any other expression including numerical values can be used both for process rates and stoichiometries. So in this case typing $0.02 * PCB$ as the process rate would have the same effect.

- Enter -1 under the column with heading "PCB". This indicates the stoichiometric coefficient of PCB in this process. The negative sign indicates that PCB is being consumed as the reaction proceeds and the value of 1 means that for each unit of process proceeding, one unit concentration of PCB is being consumed.
- Close the reactions window.
- In the **Project Explorer** window click on **Settings**→**Project Settings** and make sure that the property "Perform Water Quality Simulation" is set to "Yes".
- Run the model.
- After the simulation is done, right click on the **Main Pond** block and click on "PCB" under "Water Quality Results" item. Similarly you can inspect the temporal variation of PCB concentration in any other blocks.