

Two Dimensional flow in confined aquifer and transport of a conservative contaminant

December 31, 2016

This example demonstrates how to create a model of a two dimensional groundwater flow system in a confined aquifer. The groundwater system is composed on three no-flow boundary conditions, one fixed head boundary, and two wells. The aquifer has a depth of 10m, a hydraulic conductivity of 1m/day, and a porosity of 0.35. Refer to figure 1 for a representation of the modeled system.

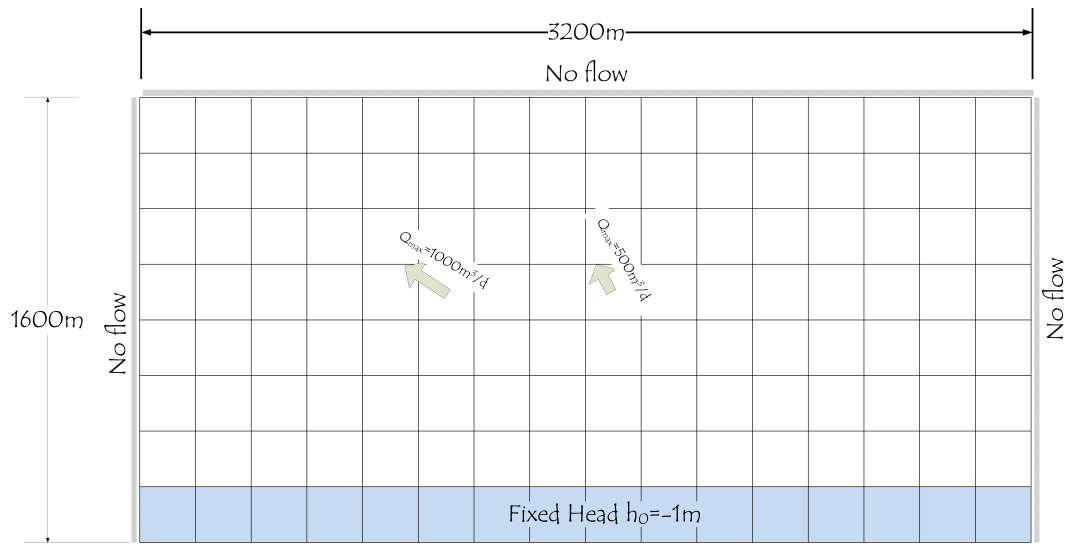



Figure 1: The schematic of the 2-D confined aquifer system

- start GIFMod
- **Create a single Darcy block:**

Click on the Darcy icon  on the top ribbon. Set the following properties:

- Bottom area: $40000m^2$.
- Initial moisture content: 0.35 (this assumes that the initial hydraulic head is zero.)
- Saturated moisture content: 0.35
- Saturated hydraulic conductivity: $1m/day$
- Precipitation: *Yes* (This allows introducing recharge using the precipitation feature.)
- Storage coefficient: $0.0001m^{-1}$

- Bottom elevation: $-11m$ (this sets the datum on the interface between the confined aquifer and the top confining layer.)
- Depth: $10m$
- Width: $200m$
- Length: $200m$

Leave the rest of the properties unchanged. Default values will be used.

- **Create an array of blocks:**

In this step, we create an array of the Darcy block created in the previous step. The array will be composed of 8 rows and 16 columns.

- Right-click on the Darcy block created in the previous step and choose **Make array of blocks** from the drop-down menu.
- Choose the **Horizontal 2D array** option and enter the number 16 in the text box labeled **Number of columns** and 8 in the text box labeled **Number of rows**.
- For the **Horizontal distance between cell grids**, enter 200m.
- For the **Vertical distance between cell grids**, enter 200m.
- Click on **Ok** button.

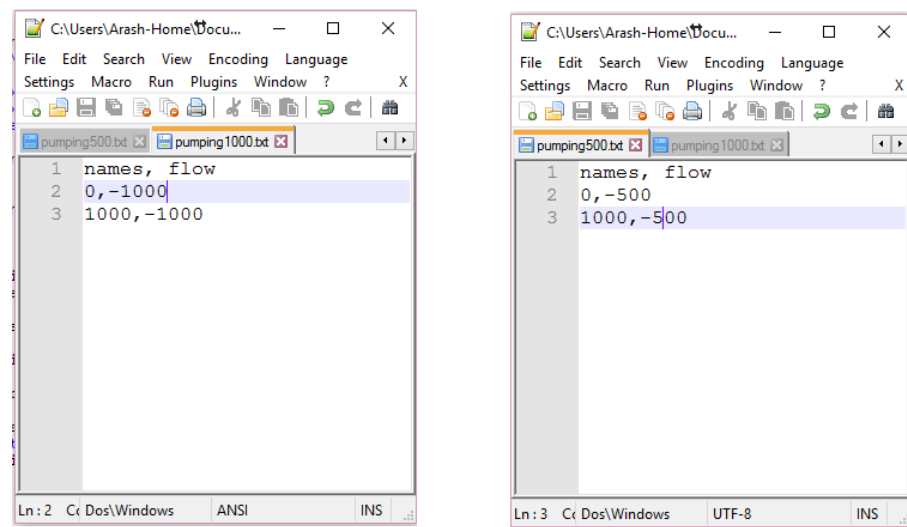
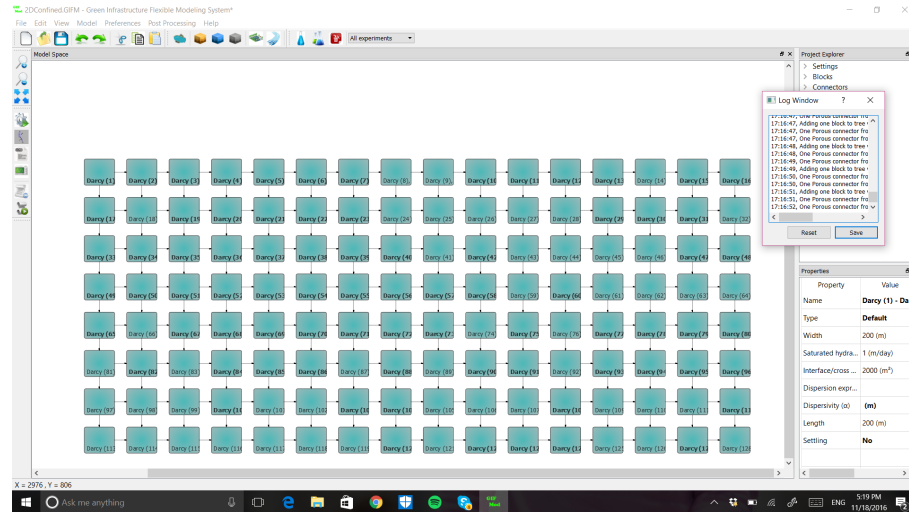
Once you have created the array, your screen should look like Figure ??.

- **Imposition the fixed-head boundary condition:** To impose the fixed head boundary condition at $h_0 = -1m$ select the storage blocks on the bottom row.

- Select the Darcy block labeled "Darcy (113)" and type "-1" in the property **Head-storage relationship**.
- Repeat the previous step for all block in the lowest row (Darcy (114)-Darcy (128)).

- **Introducing the pumping wells:**

At the time we consider a constant pumping from storage block *Darcy (54)* at $1000m^3/day$ and storage block *Darcy (57)* at $500m^3/day$ over a 1000days period. The inflow time-series files should look like Figure 3. Create the files and save them respectively as




"pumping1000.txt" and "pumping500.txt". Select the block labeled *Storage(54)* and from the properties window find the property called **Inflow time series** and choose *pumping1000.txt*. Repeat the previous task for *Darcy (57)* block and select *pumping500.txt*.

- **Setting the duration of the simulation:**

The duration of the simulation is from day zero to day 1000. From the **Project Explorer** select **Setting**→**Project settings**. From the property window fine right-click in the label **Simulation end time** and click on **Input Number**. Enter 1000 in the input box that appear.

- **Changing the initial time-step:** The final results will be stored at intervals equal to the initial time step size. So having the initial time step at 0.01 day (default value) while the simulation time is 1000 days will make the size of the results to be stored in the memory for post-processing too large. From the **Settings**→**Solver Settings** choose **Initial time step size** and change the value to 1 day.

- Save the project.

- **Running the model:** The model is now ready for running. From the left hand ribbon click on the run button  and wait until the simulation ends.

- **Inspecting the results:**

- Right-click on a block of your choice and select **Plot Hydraulic Results**→**Plot Storage** from the drop-down menu that appears. You may copy and paste the results on one graph to another one for comparison. For example figure 4 shows the storage in blocks *Darcy (53)* to *Darcy (57)*. As it can be seen the drop in the hydraulic head at the wells are quite significant. This is due to the high pumping rates relative to the hydraulic conductivities of the aquifer.
- Right-click on connectors of your choice and select **Velocity** from the drop-down menu. Figure 5 shows the flow rate in connectors connecting *Storage(54)* to *Darcy (57)*. This shows the Darcy flux in the connectors.

- **Examining the spatio-temporal variation of state variables:**

From the top menu bar click on post-processing and then select **Blocks**→**Head**.

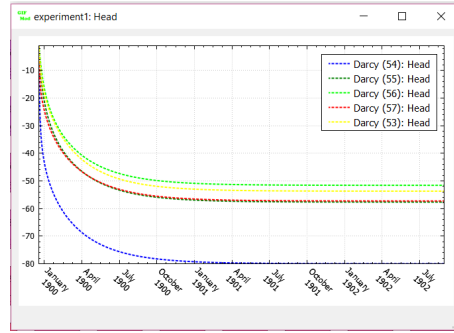


Figure 4: Hydraulic head in select blocks

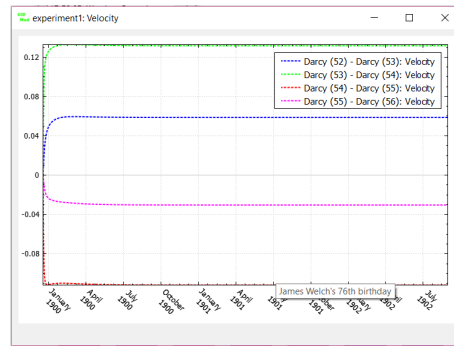


Figure 5: Darcy flux in select blocks

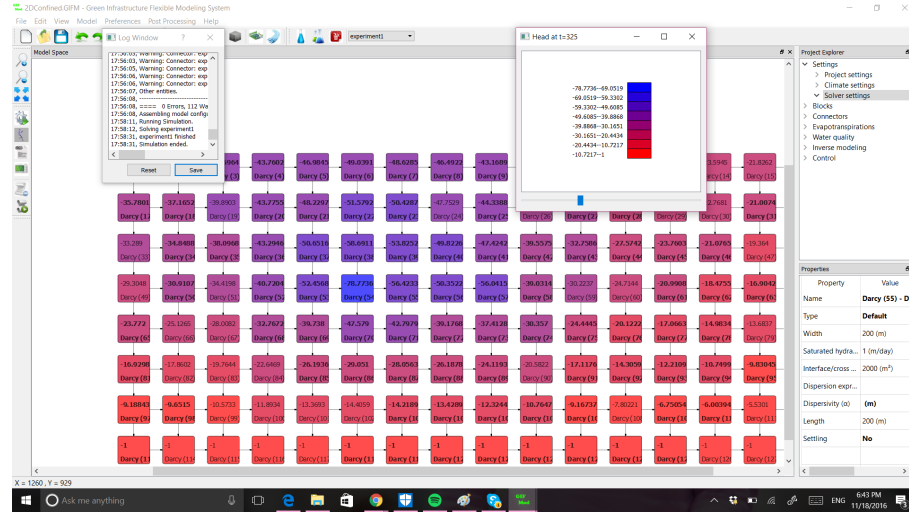


Figure 6: Spatial distribution of hydraulic head at $t=325$

Using the scroll bar in the legend box that will appear, change the time at which you would like to see the spatial distribution of hydraulic head over the domain (Figure 6).

0.1 Revising the example: recovery as a result of reduced pumping after 200 days

Here we are going to modify the previous example by reducing the pumping rate by a factor of five after 200 days.

- Make a copy of the pumping files and modify them as shown in figure 7. Save the newly created pumping files with a new names.
- Select the revised inflow time-series files as **Inflow time series** for blocks *Storage(54)* and *Storage (57)* respectively.
- Rerun the program.
- Select desired blocks and connectors and check the new variation of state variables over time.

In this example we will add transport of a conservative contaminant to the example done in the previous chapter ??.

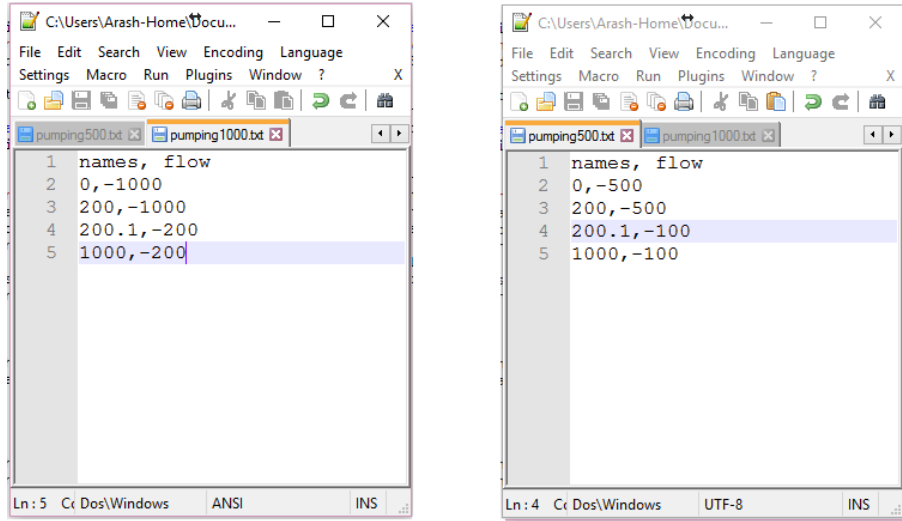


Figure 7: Revised pumping time-series files

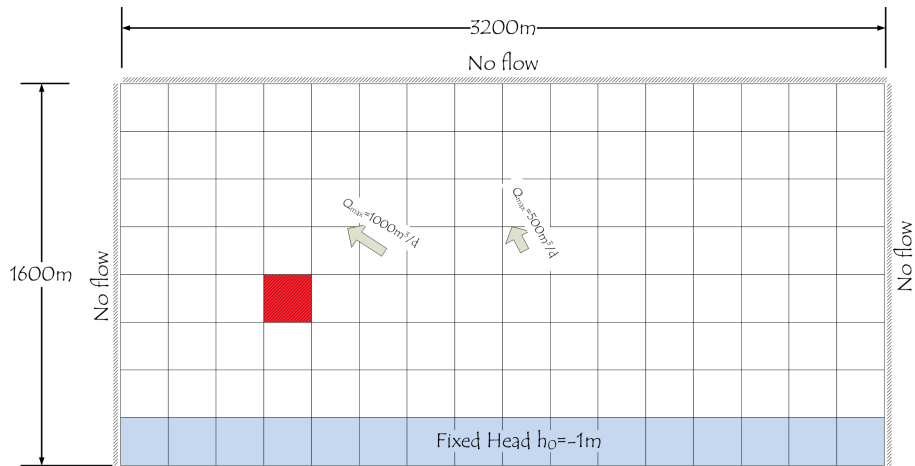


Figure 8: The initial location of the spill

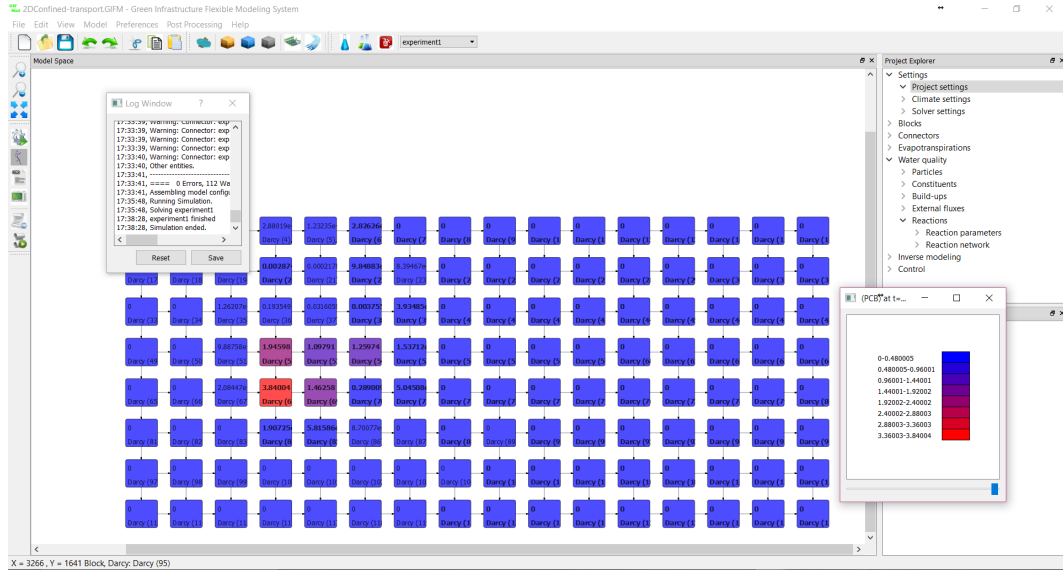


Figure 9: The spread of PCB plume after 1000 days

The initial location of a pollutant spill is shown in figure 9
Below are the steps to introduce transport:

- **Introducing a new water quality constituent:** Right click on **Project Explorer**→**Water Quality**→**Constituents** and select **Add Constituent**. Change the name of the new constituent to PDB. Leave the rest of the properties unchanged. This means that the pollutant will behave without any interactions with the solid matrix and will not be affected by gravity and that the molecular diffusion and dispersion are zero.
- **Setting the initial condition:** Select the block labeled as *Darcy (68)* and from the properties window click on **Initial Constituent Concentration** property. In the window that will show up, assign a concentration of 10 to *PCB*.
- **Run the model**
- **Inspecting the results:** From the **Post-processing** menu choose **Water Quality**→*PCB* and use the scroll button to see the evaluation of PCD concentration in the aquifer (Figure ??).