In the previous problem set, you got an idea on how to analyze the effect of pumping wells on the hydrologic flow pattern. While doing the analysis, you assumed a fixed head boundary condition for the river. However, in reality, the river stage changes with time, and so does the hydrologic flow pattern. [**Extra Credit**: in addition, village ponds continuously receive organic waste (that may trigger microbial respiration and hence the mobilization of arsenic from oxide surfaces), and thus can be considered as continuous sources of organic contaminants]. In this problem set, you will be working with transient boundary condition [and tracing solute transport at the areas of interest].

In order to assist you in your analysis, we are introducing two new features of Feflow, namely "Postprocessor" and "Reference Point".

Postprocessor

The *Postprocessor* option allows post-run investigation and evaluating computational results (such as analyzing the temporal fluxes to and from the river). The option can be selected from the main menu bar on the top. It reads *.dac files which need to be produced during the simulation using the *Control Output* option. After setting up the model BUT prior to execution, select the *Control Output* option from the *Simulator Run* menu and the following window will show up:
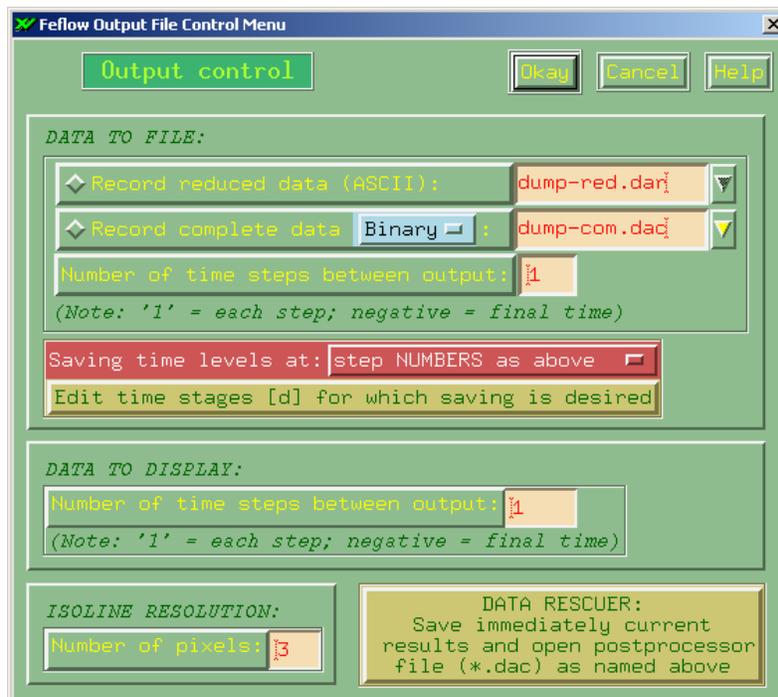
This option controls the saving modes for the computation results and the display of the current results during the simulation. Choose the *Record complete data (Binary)* with a file name of *dac* extension as needed by the *Postprocessor*. Then, you can either choose *Time steps between run* or *Edit time stages* as you desire. With the later option, you can mention the specific time steps for which you want to analyze the model data. The *data to display* section controls the update frequency for the visualization of results during the simulation by specifying the number of time steps between subsequent redrawings (this is particularly important when you are working with a larger model; for your case, you can keep it to the default value of 1). After setting up the *Output Control*, run the model and exit to master menu. Choose *Postprocessor* from the top menu, load the saved *dac* file and the simulation from the last time step will show up. The *Postprocessor* displays the same diagram windows as the *Simulator*. Using the *Browse file* option, you can view the results of selected time steps and analyze the results in the same way you did before from the *Simulator* menu.

Reference Point

You can insert and edit reference data (such as observation points) for the evaluation of the simulated results. Choose *Reference Data* from the *Problem editor* menu, and the *Reference Data Editor* with four options will show up. Among these options, the *Single Observation Points* can be inserted where contaminant breakthrough or hydraulic-head values are to be monitored during the simulation. The number of observation points is unlimited. Observation points can either be set on nodes or arbitrary locations within the model domain. A special *Point property* dialog box allows you to define additional properties for the observation points. While you are welcome to explore the other three options under *reference data editor* menu, you are expected to utilize at least the *Observation single points* option in your model.

Finally, before running the model for specific cases, remember to save the FEM files for revisiting as well as for submission (as email attachment) along with your write-ups. And, similar to Problem set # 1, you may work on this problem set in pairs. However, each person must hand in their own solutions including FEM files, tables, plots, and/or figures to illustrate your work. Please indicate your partner's name on your write-up.

Now, let's get down to business!

Transient Boundary Condition

From the *Problem Editor* menu, click *Temporal and control data,* and choose *edit time varying power functions* from the pop-up menu. The *Power function window* appears where time-related functions can be interactively edited. The power functions are (x,y) value pairs, where the x-values describe the time in days while the y-values define the corresponding function value. The time varying functions can be applied to pumping rates or time varying water levels in rivers and lakes. In the *Power function* window, click on *New ID,* put an identifying name for the function in the *Comment* box, and click on *edit curve data* where you can input the time and function values. Using this feature,

create a time-varying function for the river boundary condition with the following river stage data (you will need the aquifer head and pond level data to calibrate your model):

Date	River stage (m)	Aquifer head (m)	Pond level (m)
12/5/2003	123.739	124.563	125.59
12/14/2003	123.669	124.466	125.374
12/16/2003	123.549		
12/20/2003	123.589		
12/22/2003	123.739		
12/25/2003	123.809	124.409	124.973
12/29/2003	123.519	124.341	125.142
1/27/2004	123.479	123.744	124.865
2/20/2004	122.849	123.058	124.756
3/11/2004	122.429	122.848	124.654
3/25/2004	123.649	122.401	124.515
4/9/2004	124.029	123.126	124.690
4/24/2004	124.329	123.626	124.722
5/15/2004	124.239	123.296	124.315
5/28/2004	125.069	124.271	124.500
6/10/2004	125.089	124.821	124.896
6/24/2004	125.499	125.261	125.499

Assume December 1, 2003 as the first day for the modeling period. However, keep the irrigation pumping period same as in previous problem set [you may want to create another power function ID for the pumping schedule].

In order to set up the transient river head boundary condition along the western border of the model, go to *flow boundaries*, make sure *assign* option is on, choose *border,* below *time varying function ID option* click the corresponding ID for the transient river head, click *Head* and assign the head along the river border. The corresponding function ID number will appear in blue while you are assigning the transient head boundary. Use initial heads of 124.6m and 125.65m for the aquifer and pond areas, respectively (being surrounded by villages, the ponds are separated from the rice fields and can retain water levels higher than the ground surface elevation of the field areas).

Consider the river depth is 3m and so, apply the transient boundary condition for the river only to the top-most layer. Use the river stage data as the specified head, and the aquifer and pond water level data for model calibration. You need to adjust the conductivity of the clay layer (the second layer) below the pond and river area so that you get reasonable fit between the observed and modeled data for the aquifer and the pond. For this purpose, adjust the conductivity value(s) of the second layer only below the pond area and of the triangular elements adjacent to the western boundary.

The monthly total rainfall and evaporation data for the region during the modeling period is as given below:

|	Dec	Jan	Feb	Mar	Apr	May	June
Rainfall (mm)	11	6	21	40	127	283	380
ET (mm)	66	70	87	136	159	150	119

You need to create a time-varying power-function ID to incorporate the data in the model. First, input the net recharge [(Rainfall – ET) for non-irrigated areas and (irrigated return water + Rainfall – ET) for rice field areas] under a new ID in the power-function window (as you did for the river and pumping data). Then from the *Flow Material* menu, check *assign*, *database*, and click on *In/Out flow from Top* (make sure you are on the top-most slice while assigning the net recharge). The following *Data Regionalization* window will show up:
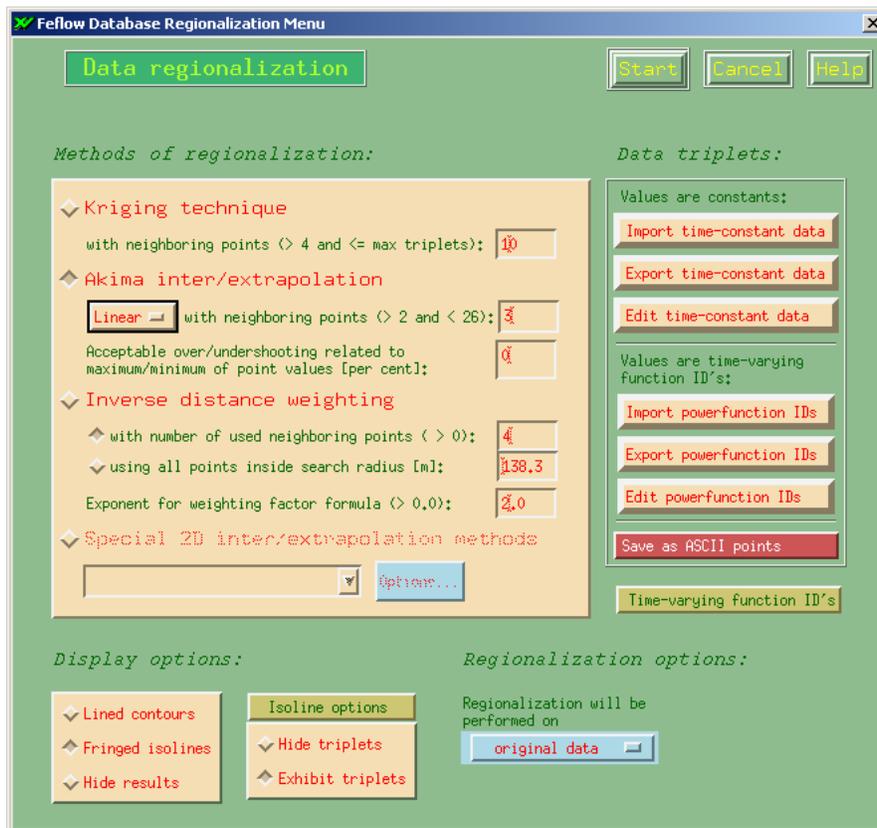
Select *Akima inter/extrapolation, Linear*, and put values 3 and 0 corresponding to neighboring points and acceptable over/undershooting, respectively (as shown in the snap-shot above). Click on the button *Edit powerfunction IDs,* and the following window will pop up. Input at least four X, Y coordinates (corresponding to the four corners of your map) and the function ID (corresponding to the time varying power function for the net recharge), and click *Okay*.
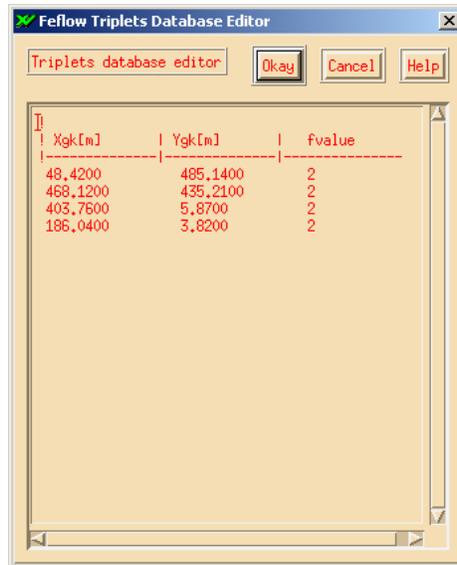
Click *Start*, and the power-function ID will show up in your model domain (see the snap-shot below):
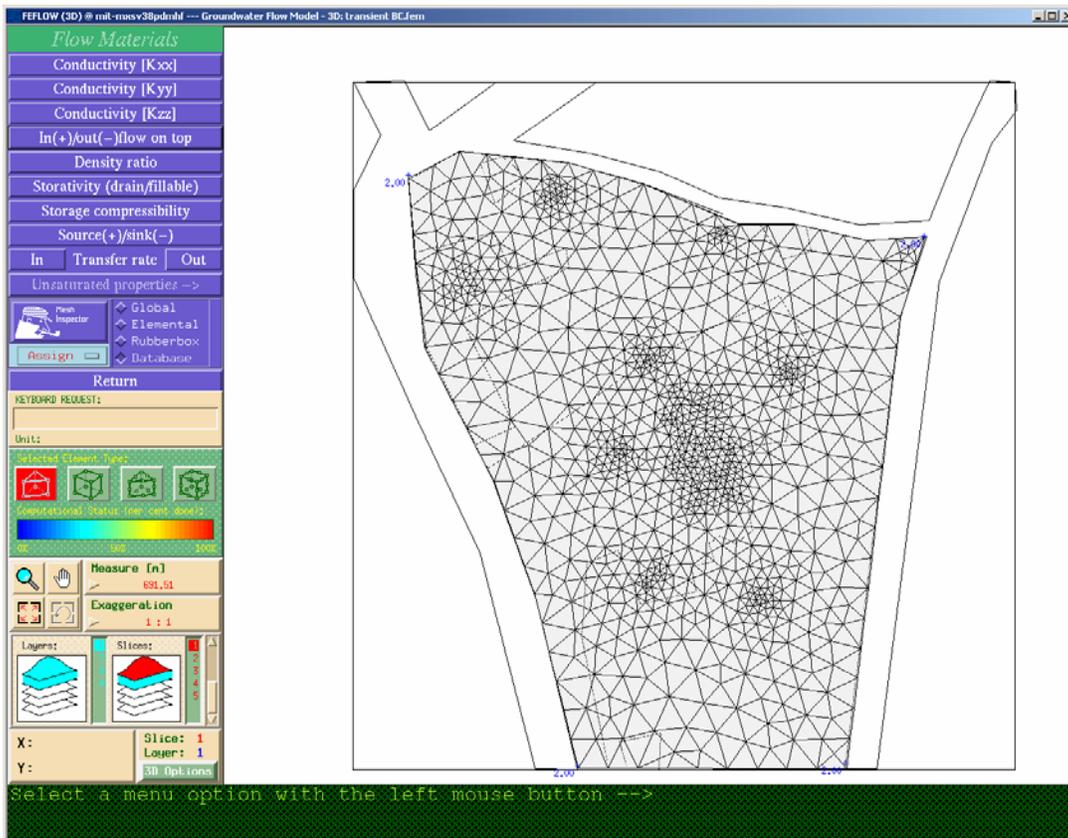
Keep all other flow and material properties same as in the last part of the previous problem set.

Save the FEM file before running the model (for submission; you can name it as 'transient BC')

1) Before running, make sure you have set up the model parameters correctly

2) What values of hydraulic conductivities you obtain for the pond and river bottoms after calibrating your model to the aquifer and pond water level data?

3) How does the flow pattern look different with the transient boundary condition?

4) When (time period of the year) is the groundwater discharging into the river? When is the river recharging the aquifer?

5) How do the flux values look like for both recharge and discharge scenarios? Give explanation of any differences in fluxes you find

Extra Credit: Contaminant Mass Transport

Let's assume that the pond at the center of the modeled region is receiving an organic waste flux of 50 mg/L m^2/d, and to avoid any contamination in the drinking water wells, the dissolved organic carbon (DOC) concentration should be less than 3 mg/L. Also assume the initial DOC concentration in the modeled region to be 0 mg/L, and the inflow from the river does not contain any organic waste either.

Applying the initial and boundary conditions for the mass transport is very similar to that for flow data. However, we want to apply a freshwater boundary condition for the rivers in such a way that the inflow from the river is fresh but the outflow from the aquifer to the river is not necessarily fresh. In order to achieve a freshwater condition, assign a value of 0 mg/L of mass along the river boundary. This will make both the inflow and outflow across the boundary to be fresh, and a contaminant plume cannot leave the model freely. To ensure the outflow across the boundary not to be necessarily fresh, we need to apply a constraint as *Min mass flux* of 0 mg/L m^3/d. This will guarantee that the first-kind boundary condition is applied only when water enters the model. On the other hand, during an outflow situation across such a constrained boundary, the first-kind boundary condition is automatically switched off and the contaminant mass can freely flow out. [To apply the constraint condition, click on the arrow sign on the right of the *Mass (1st kind)* button]

For the *Mass transport material,* use appropriate values of porosity, longitudinal and transverse dispersivities for different layers (don't forget to site the reference of your data), and keep the default values for other properties. **Save the FEM file before running the model** (for submission; you can name it as 'mass transport').

6) Before running, make sure you have set up the model parameters correctly

7) Are any of the drinking water wells getting contaminated at any time of the year? Is there any contaminant outflow to the river?