COURSE: GROUNDWATER MODELLING USING MODFLOW

Session 9: Simulation a groundwater model

Objective:

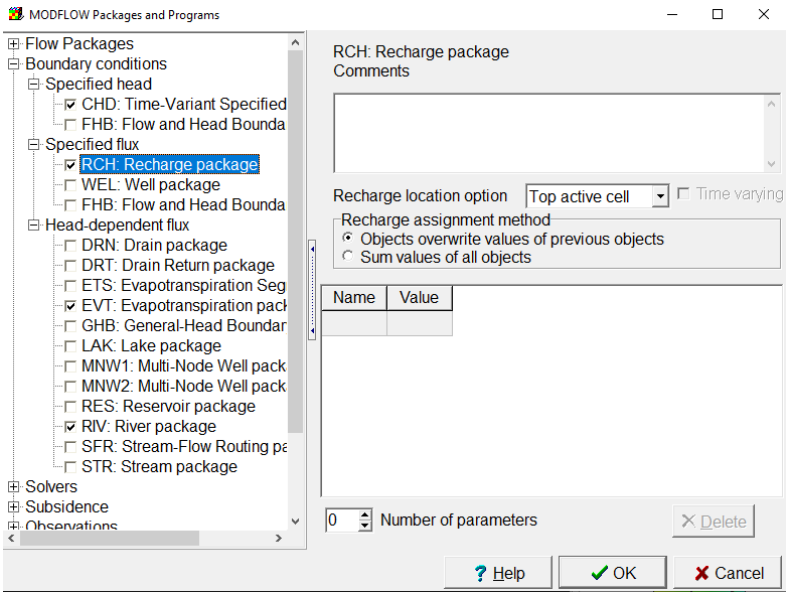
The objective of this session is define package and programs, assign the boundary conditions CHD, RCH, EVT and RIV, run the model, analyze the water balance, import the results and represent the water table in a profile view.

Go to
Course_GroundwaterModellingMODFLOW\Week_9\Practice\Models

Then open **Model1_a.gpt.**

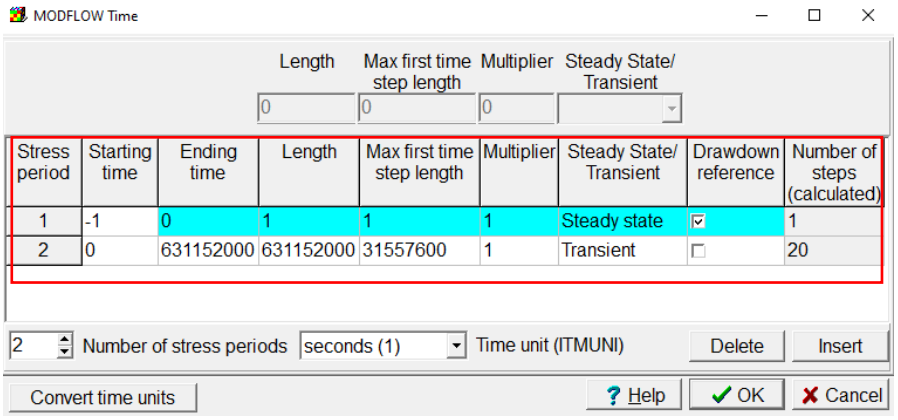
Defining package and program

To define the boundary conditions it is necessary that they are active. Go to **Model/ MODFLOW Packages and Programs** and activate **MODFLOW-NWT**. Then, go to **RCH** and change the Recharge Location Option to **Top active cell.**



Defining MODFLOW Time

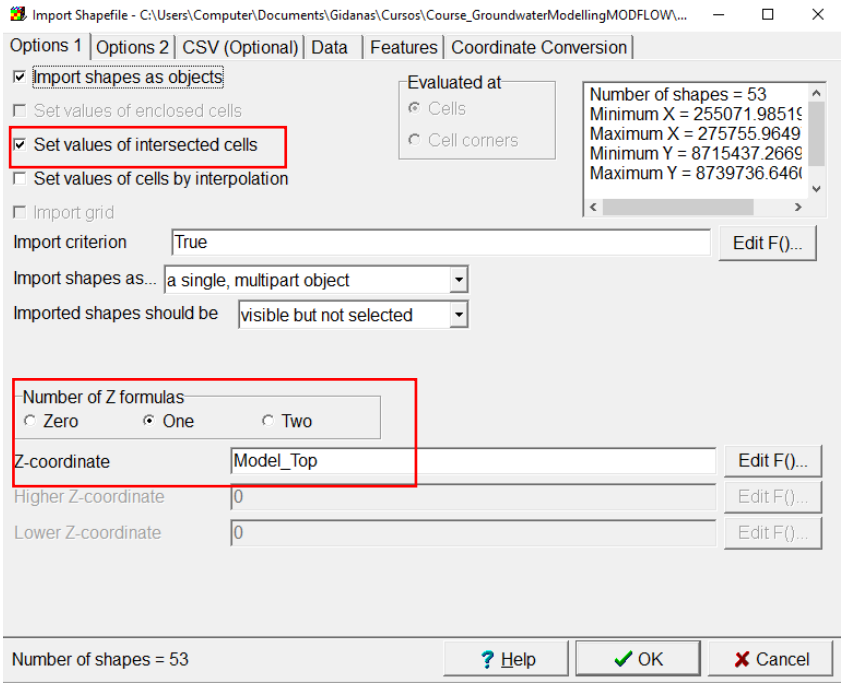
Verify the time values in **Model/ MODFLOW time**. The values should be the ones shown below.



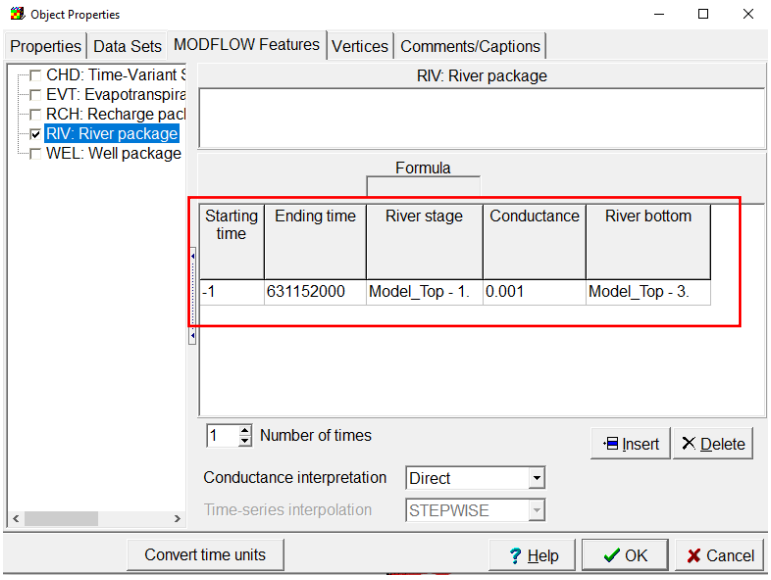
Defining boundary conditions

To import the boundary conditions, go to File/ Import/ Shapefile. The file is found in:

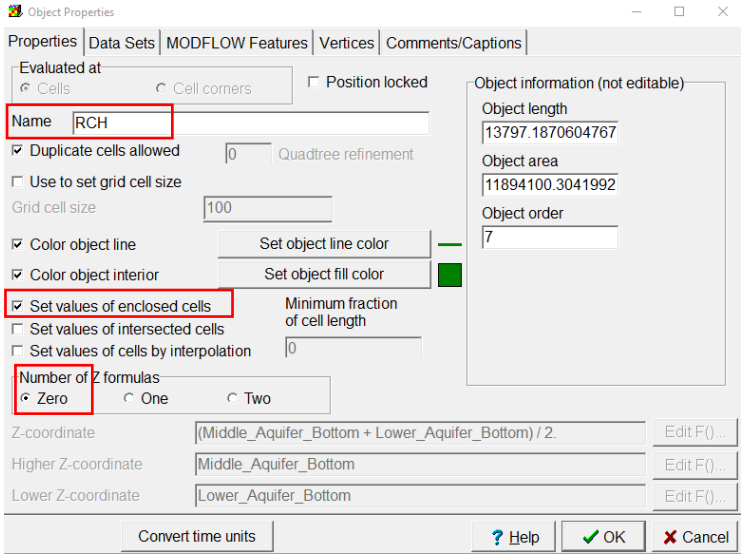
Course_GroundwaterModellingMODFLOW\Week_9\Practice\Data, and it is called **river.shp.** Click OK



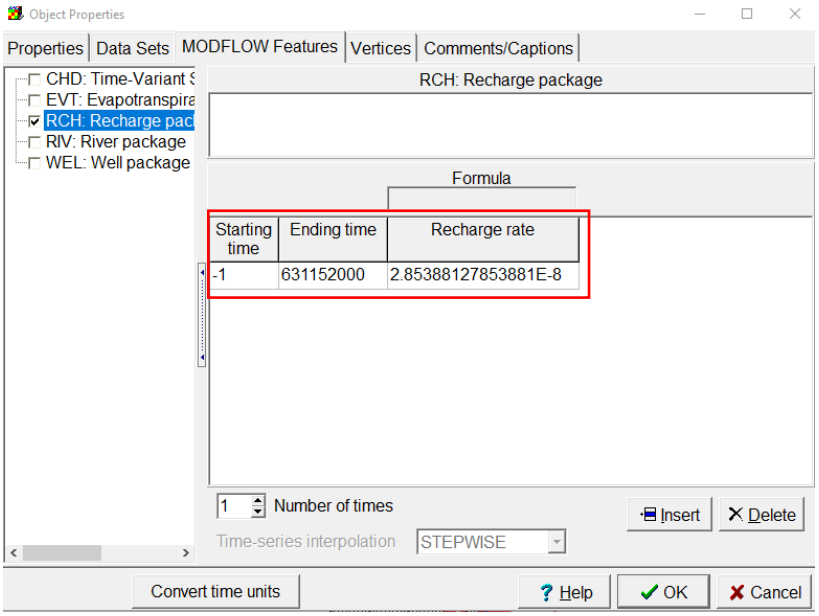
Then double click on the river and go to **MODFLOW Features**, select **RIV** and fill the values as in the image.



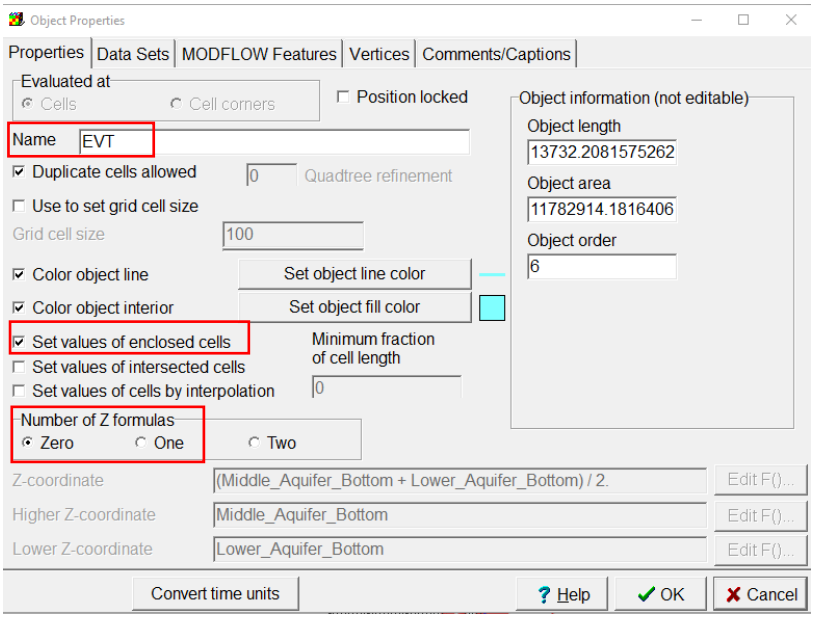
To generate the recharge boundary condition, create a rectangle that encompasses the whole basin area, double click and consider the same values shown in the below image.



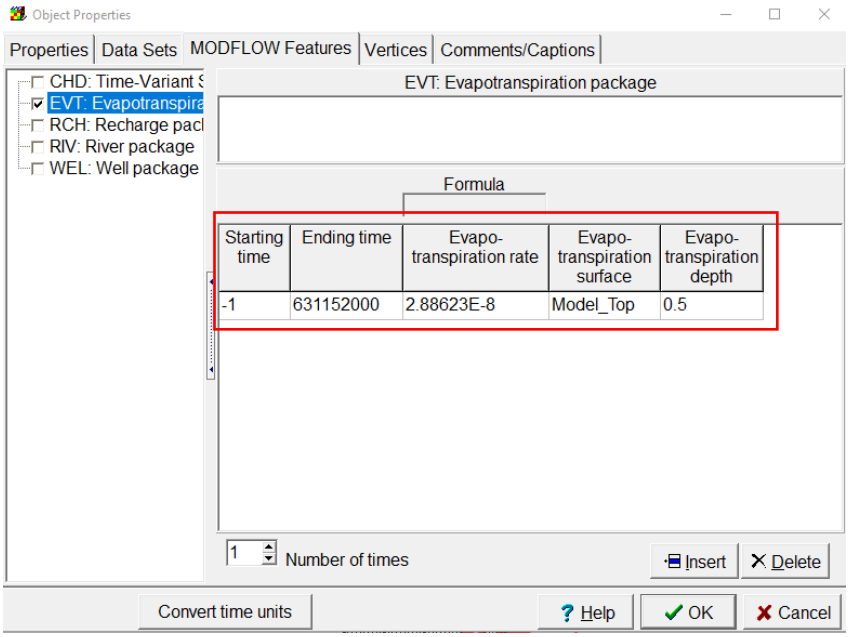
Go to **MODFLOW Features**, select **RCH** and fill the values as in the image.



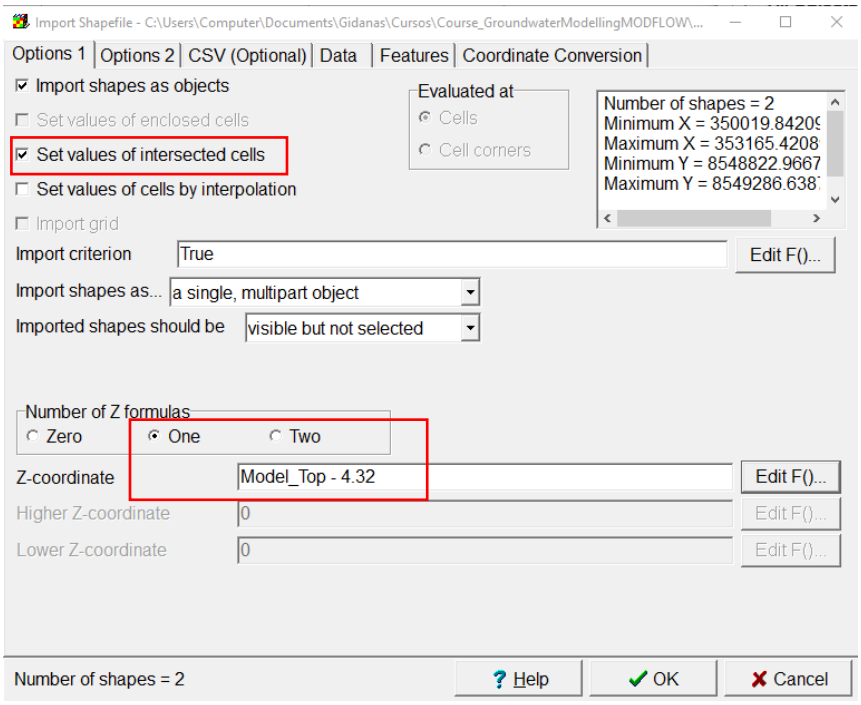
Then generate the evapotranspiration boundary condition, create a rectangle that encompasses the whole basin area, double click and consider the same values shown in the below image.



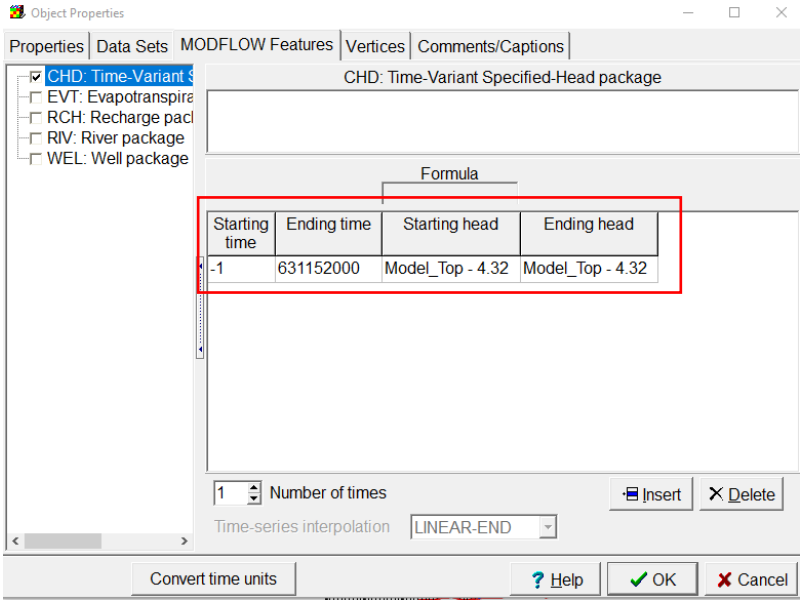
Select the **ETP** option and insert the values according to the emerging window. Click **OK.**



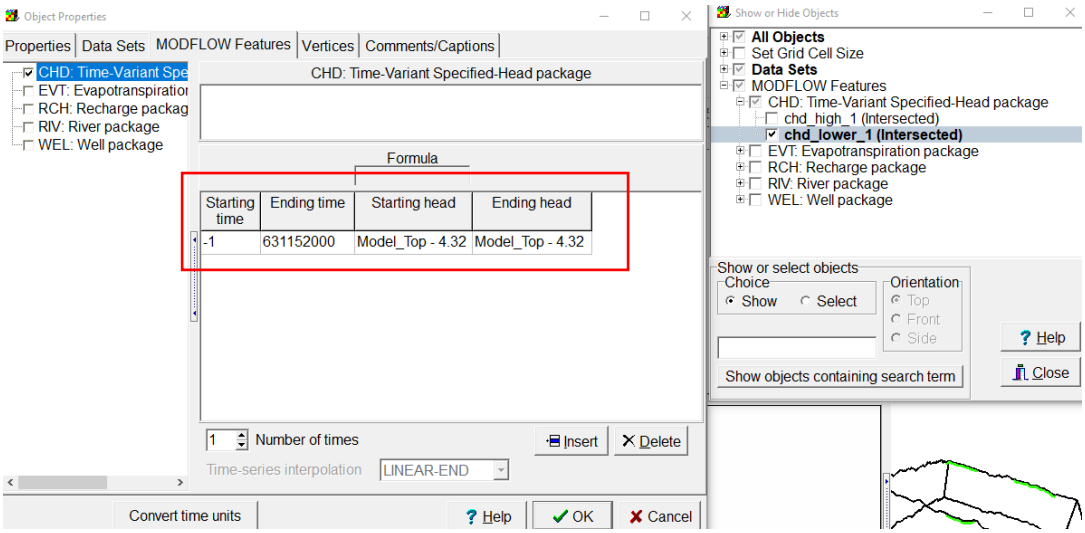
To define the perimetric Constant_Head, go to **File/ Import/ Shapefile**. The file is found in: **Course_GroundwaterModellingMODFLOW\Week_9\Practice\Data**, and it is called **chd_high.shp.** Click **OK**



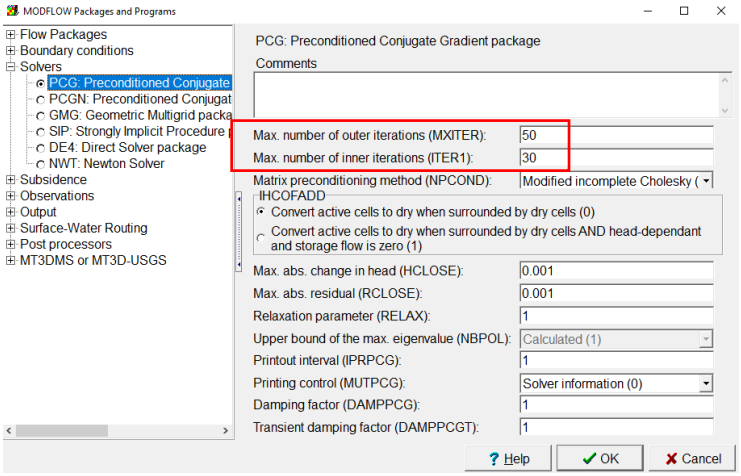
Double click the object. Insert the data of the **MODFLOW Features**. Click OK.



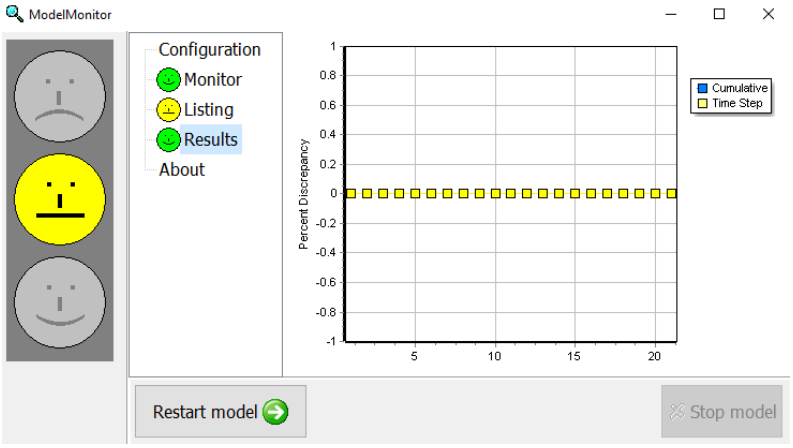
Define the second constant head with the same instructions of **chd_high.shp** for **chd_lower.shp**



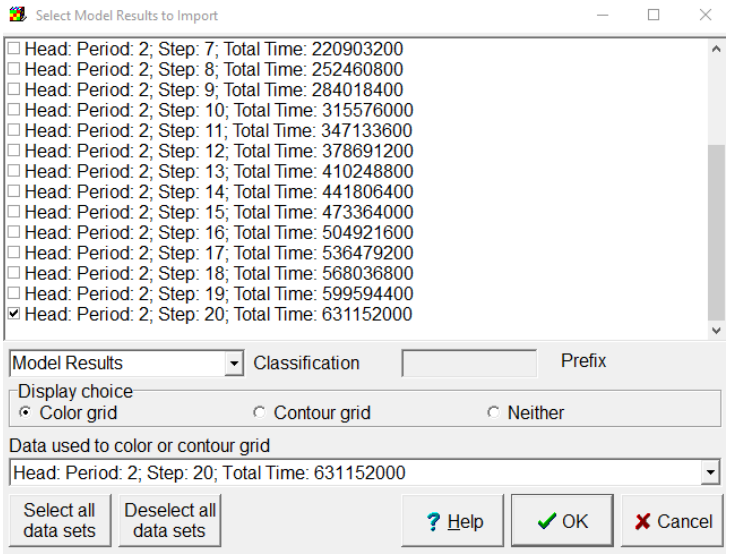
To increase the model iterations, go to **Model/Packages and Programs/Solvers** and activate **PCG**. Change the maximum number of outer iterations to 50 (MIXITER) and ITER 1 as 30.



Run the flow model. The happy face indicates that the convergence criteria has been reached. That means that all that all the fluxes that enter the model are equal to the ones exiting (water balance).



Import the hydraulic heads to the model (.fhd file). Click to **Open.**



The result should look like the image: