COURSE: GROUNDWATER MODELLING USING MODFLOW

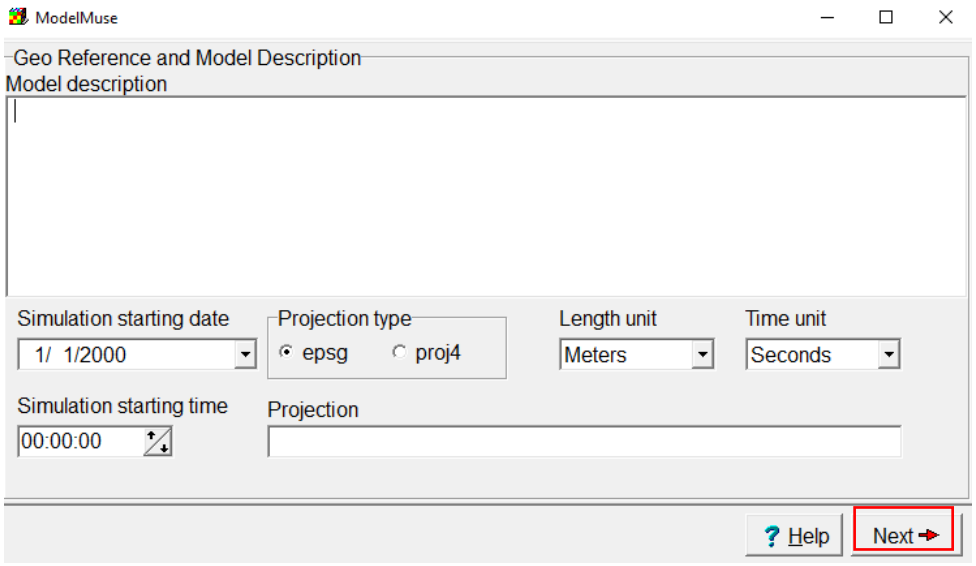
Session 7: Three-dimensional steady-state simulation with recharge, constant-head, a river and wells.

Objective:

The objective of this session is learning how to define the aquifer type, establish hydraulic parameters, select and unselect objects, define time values, assign boundary conditions, run the model, analyze the global water balance, and import the hydraulic heads.

Setting up model parameters

Open ModelMuse.exe and choose the option Create New MODFLOW Model. In the "Geo Reference and model Description" window change the length units to "meters" and time units to "seconds", after that click on "Next".



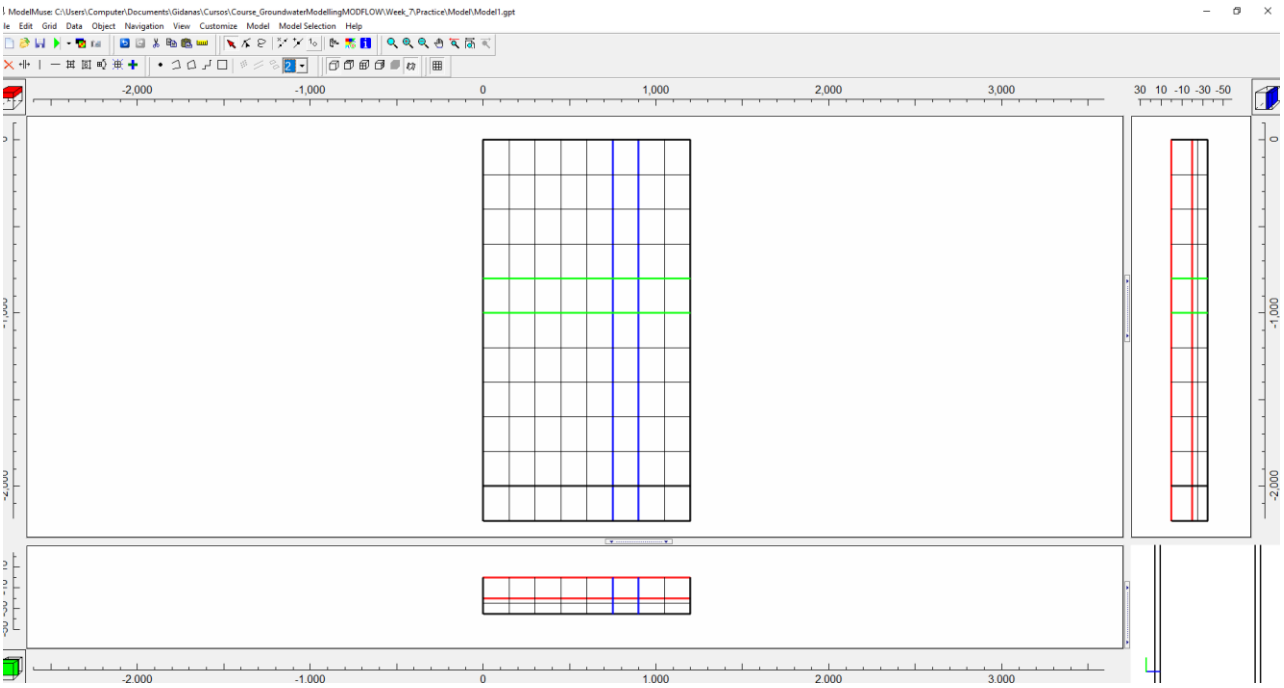
Defining model grid properties

The configuration of the "**Specify initial grid"** options is as follow:

- The number of rows is equal to 11 and the number of columns is equal to 8.
- The **number of layers** is equal to **3**.
- The column width is equal to 150 and row width is 200.
- The **model top** is set as **0**

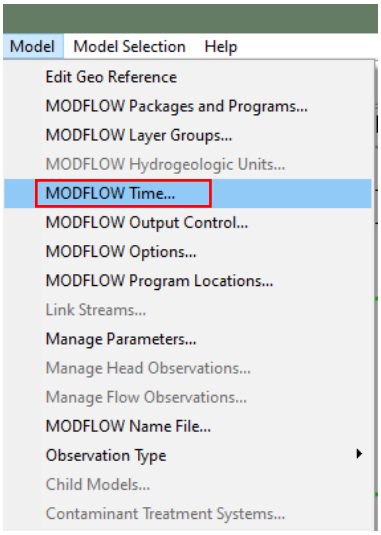
- The **layer group name** is set as **AqX**, where "X" is the layer number.
- The bottom elevations are -20, -25 and -35 meters.

  After making these configurations click on "**Finish".**

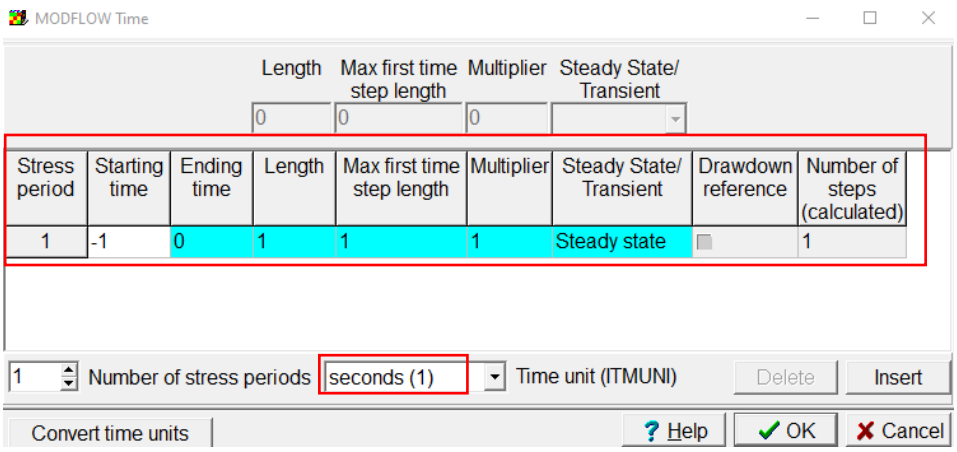


Defining time properties

To define the time properties, go to "**Model / MODFLOW Time…"**



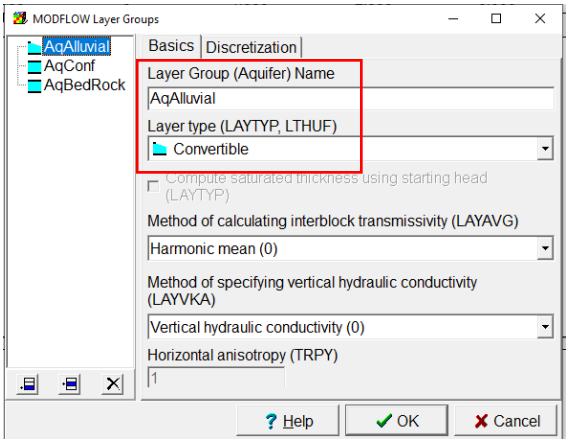
The time units are left as default, a steady-state simulation of 1 stress-period of 1 day.

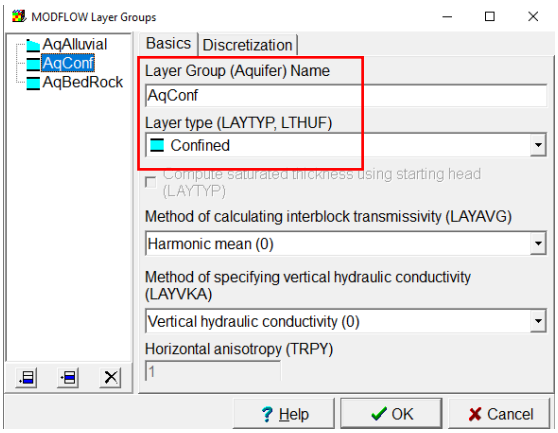


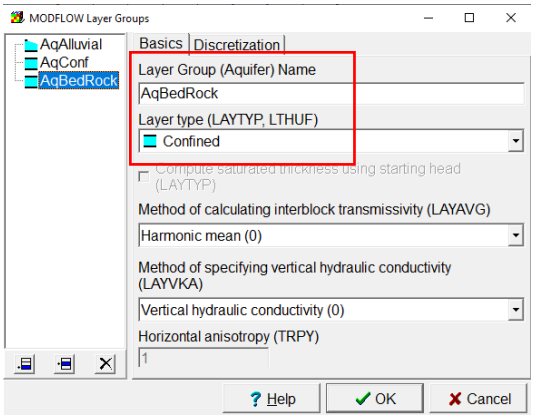
Discretization model

Then go to **Model /Modflow Layer Group**. Then will appear a dialog box and the layers in the model and change the Layer type:

- Unconfined aquifer - Convertible
- Confined - Confined Layer
- Confined – Confined Layer





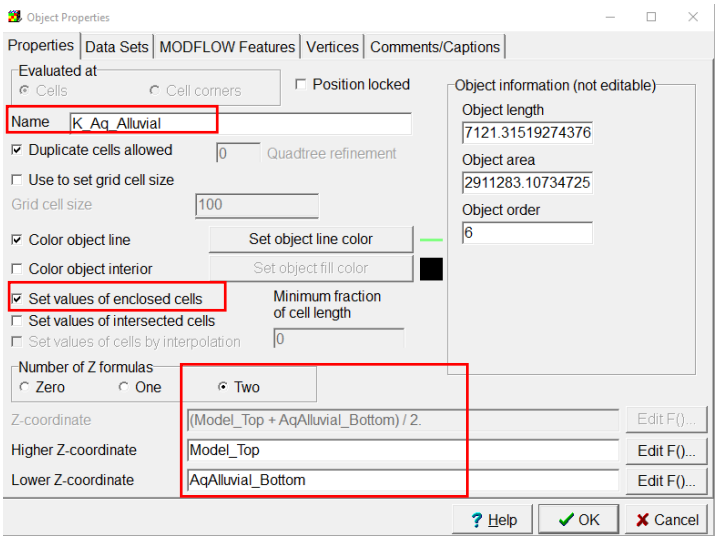


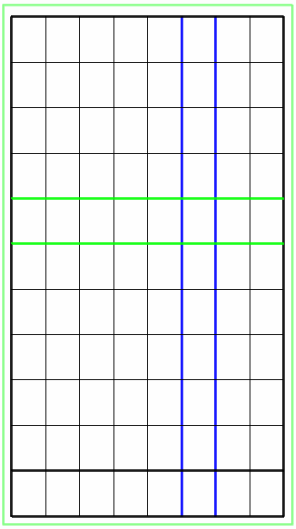
Defining hydraulic conductivities

According to the characteristics of the aquifer, the hydraulic conductivities for each layer have been defined, introducing the following values in m²/seconds.

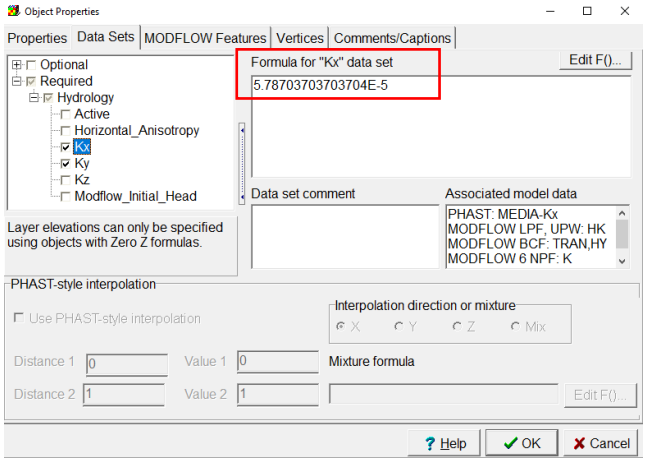
Layer	Kh(m/d)	Kh(m/s)	Kv (m/s)
AqAlluvial	5	5.78704E-05	1.15741E-05
AqConf	0.1	1.15741E-06	5.78704E-07
AqBedRock	10	0.000115741	1.15741E-05

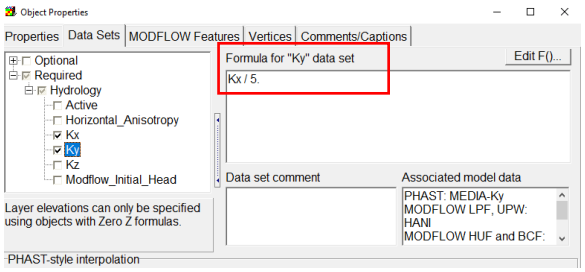
Then create a rectangle object with **Create rectangle object** as the imagen. In the properties of this object change the following:



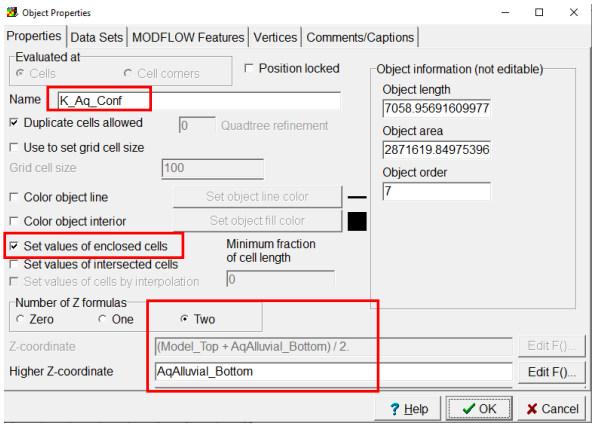


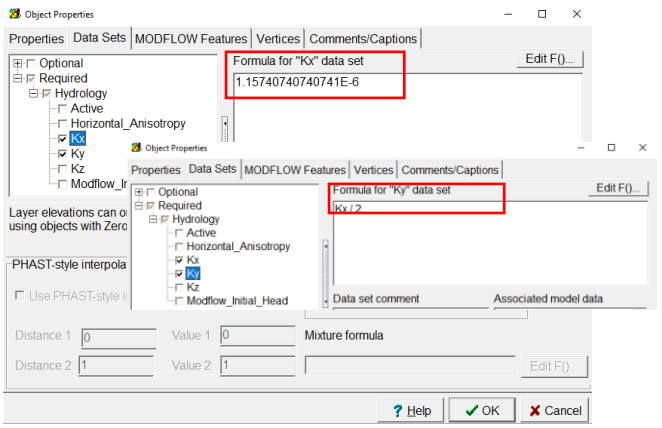
Select **Data Sets** and go **Required/ Hydrology / Kx**, here we can define the hydraulic conductivities mentioned (Kx and Ky). Click on **OK.**



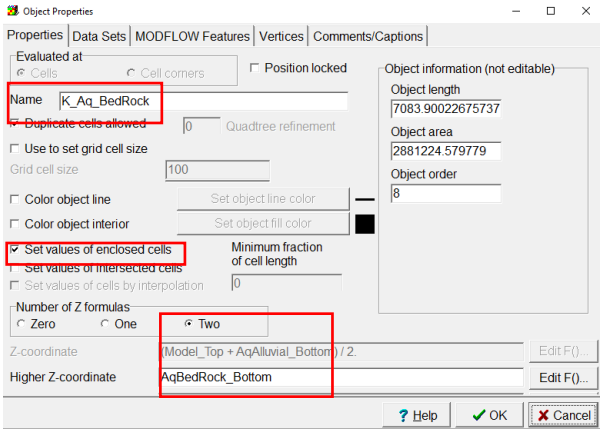


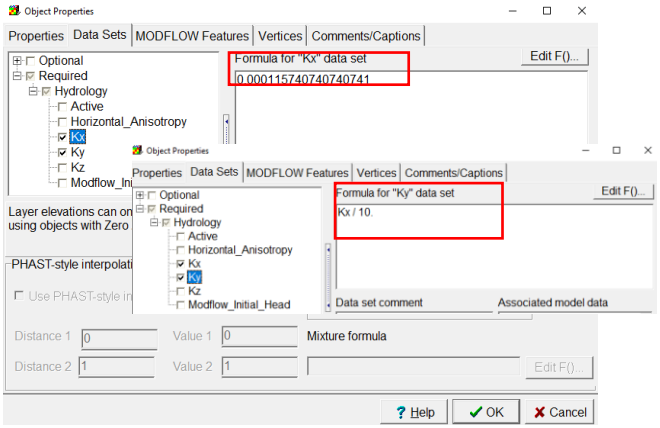
Then generate a new object to define the hydraulic conductivities for AqConf:





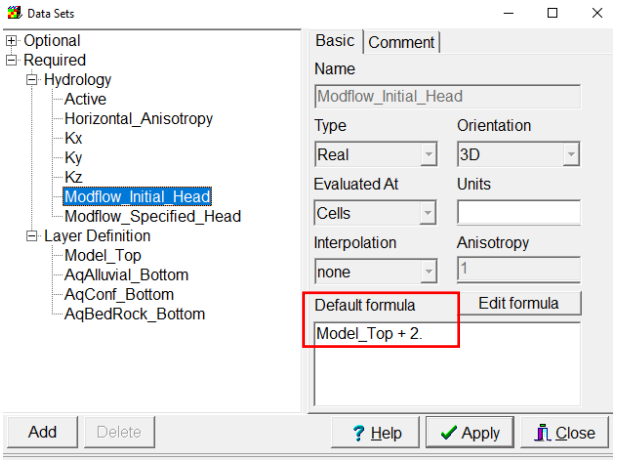
Then generate a new object to define the hydraulic conductivities for AqBedRock:





Defining initial condition

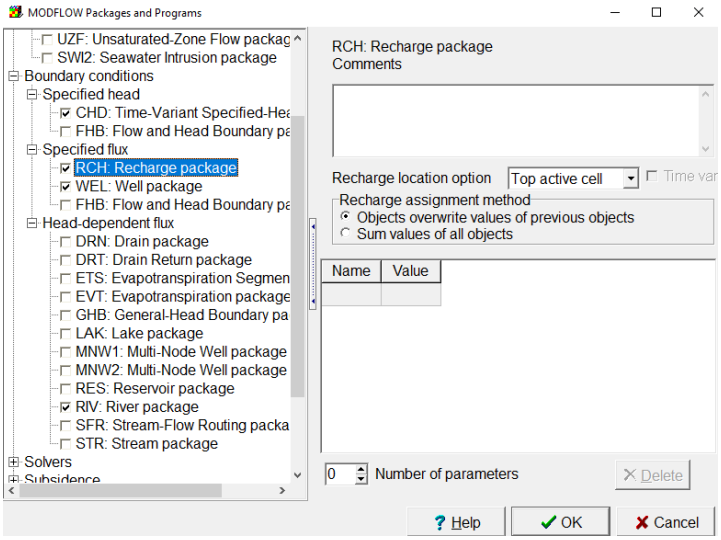
Let´s define the initial condition to do this go to **Edit/Edit data sets...** and choose the **Required/Hydrology** options, here we can define the **Modflow_Initial_Head** mentioned.



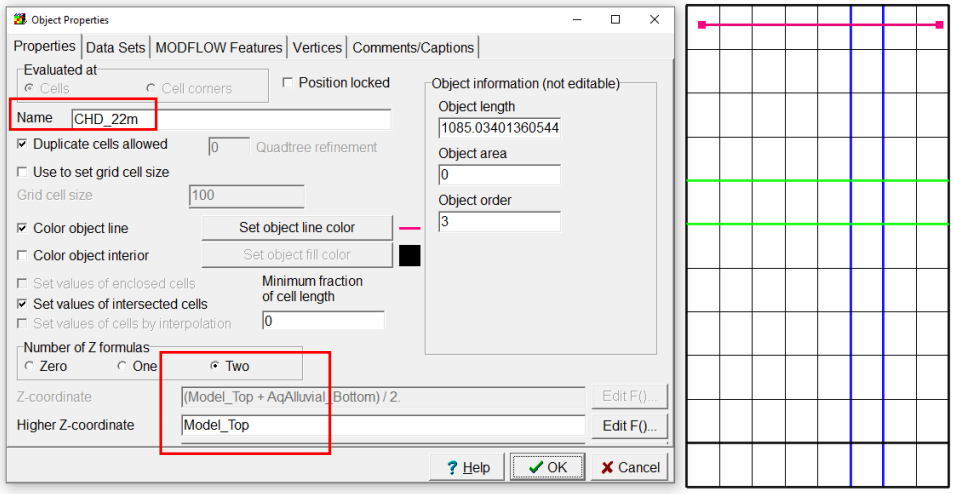
Defining boundary conditions

In this exercise, we add 4 boundary conditions, Recharge (RCH), the Constant head (CHD), well (WEL) and River (RIV).To activate the packages go to **Model/MODFLOW Packages and Programs….**

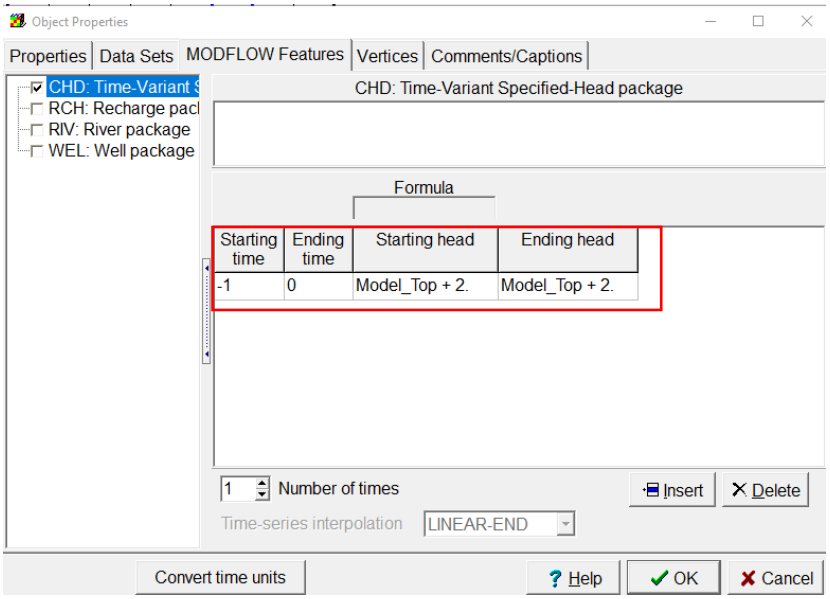
Once you have checked the three boundary conditions click on **Ok.**



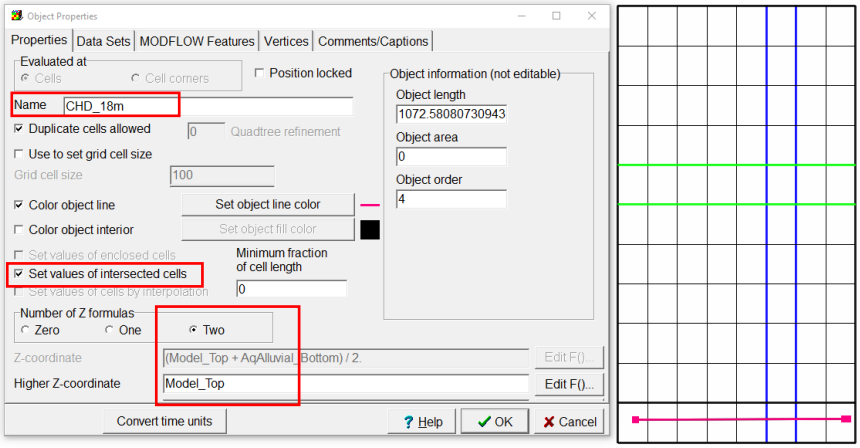
First, to define the **CHD** Object, create a polyline with **Create polyline object** as the imagen. In the properties of this object change the following:



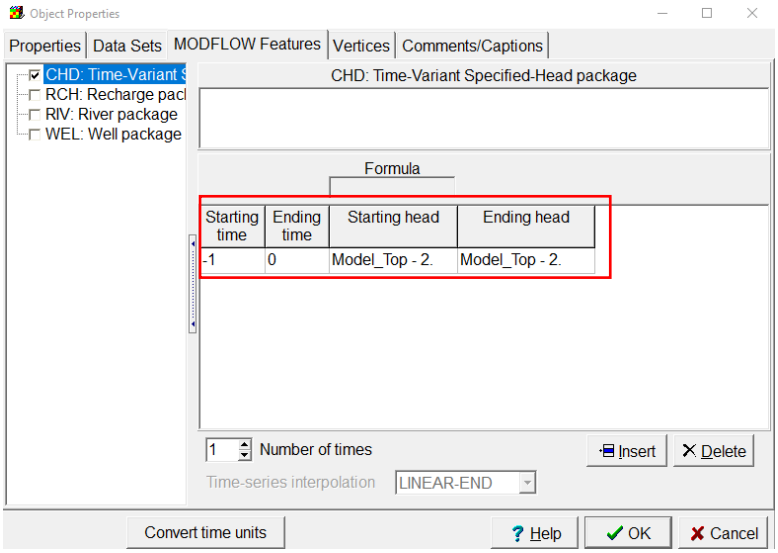
Now go to the "**MODFLOW Features**" tab and check the "**CHD**" option, the "Starting and ending times" are set from "-1 to 0" and the Starting and ending head is set equal to ModelTop+2. Click on **OK** to proceed with the next boundary condition.



Then create another CHD Object with the following properties:



Now go to the "**MODFLOW Features**" tab and check the "**CHD**" option, the "Starting and ending times" are set from "-1 to 0" and the Starting and ending head is set equal to ModelTop-2. Click on **OK** to proceed with the next boundary condition.



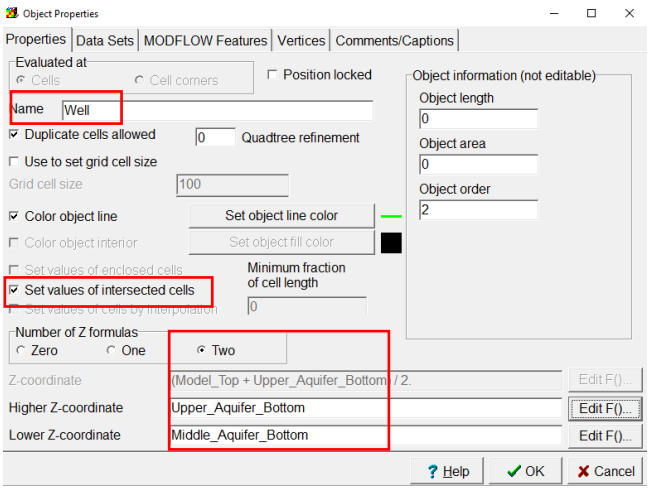
The "WEL" package is located in a grid cell, use the **Create point object** tool. Draw a point located in the layer 3, row 5 and column 6

The properties of the point are as follows:

- Name it **WEL**
- Give it a **Color line** to differentiate the well.
- Set the **number of Z formulas** to **Two** with the following characteristics:

  Higher Z-coordinate = **Upper_Bottom**

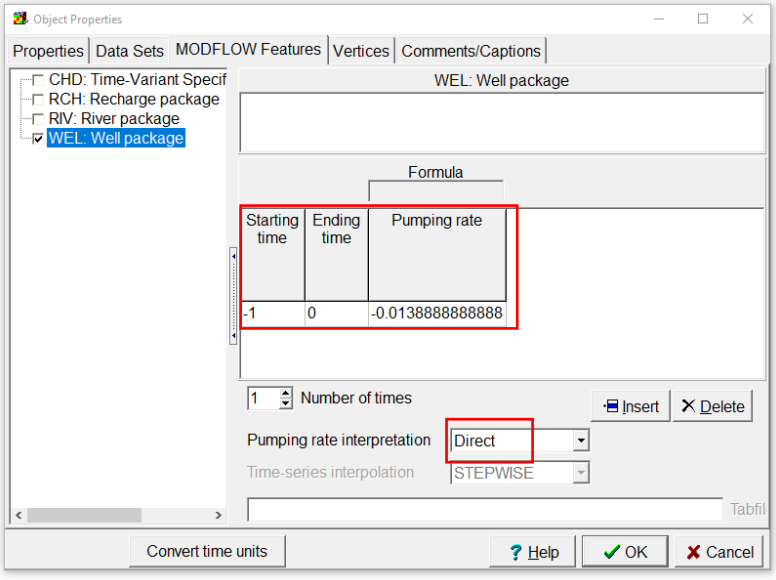
  Lower Z-coordinate = **Middle_Bottom**

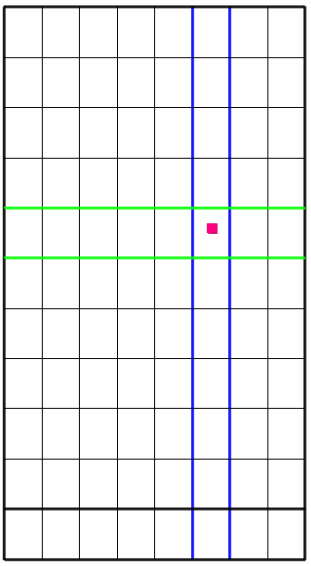


In the **"MODFLOW Features"** tab, **check** the WEL package and do the following settings:

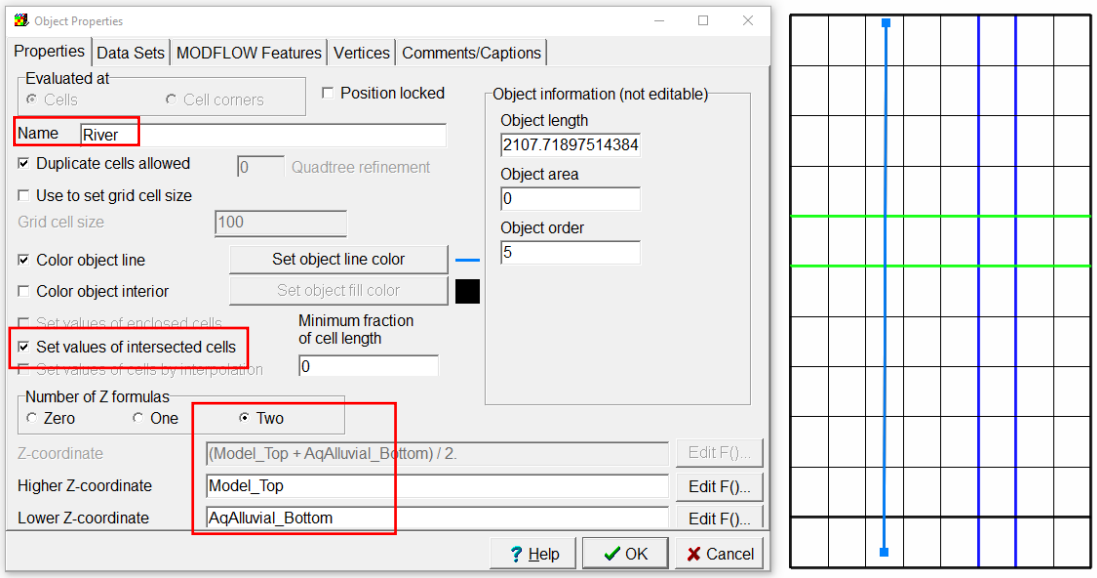
- Set the Pumping rate to -0.00138 m3/s.
- Be sure to select Direct in the Pumping rate interpretation options.

Click on **OK** to continue with the next boundary condition.





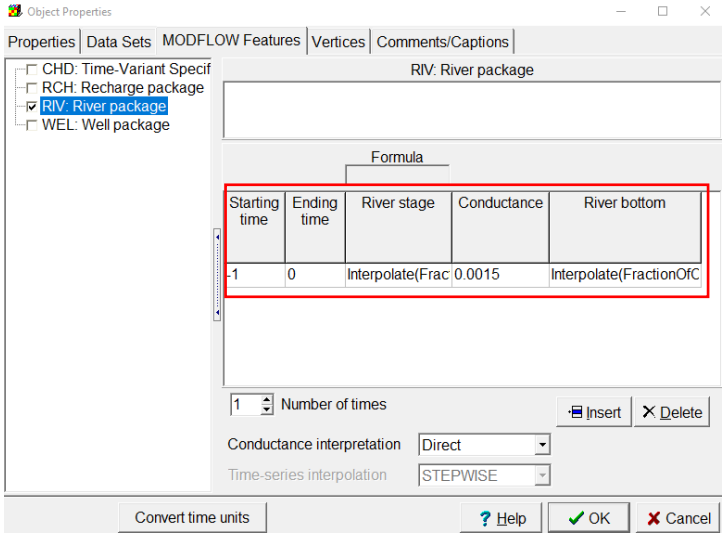
The “RIV” package is located in a grid cell, use the **Create point polyline** tool and draw the polyline as the image:



In the “MODFLOW Features” tab, check the RIV package. the “Starting and ending times” are set from “-1 to 0” and the characters of the river are:

River stage: Interpolate(FractionOfObjectLength, (Model_Top + 2.), 0.0125, (Model_Top - 2.), 1.)

River bottom: Interpolate(FractionOfObjectLength, Model_Top, 0.0125, (Model_Top - 4.), 1.)

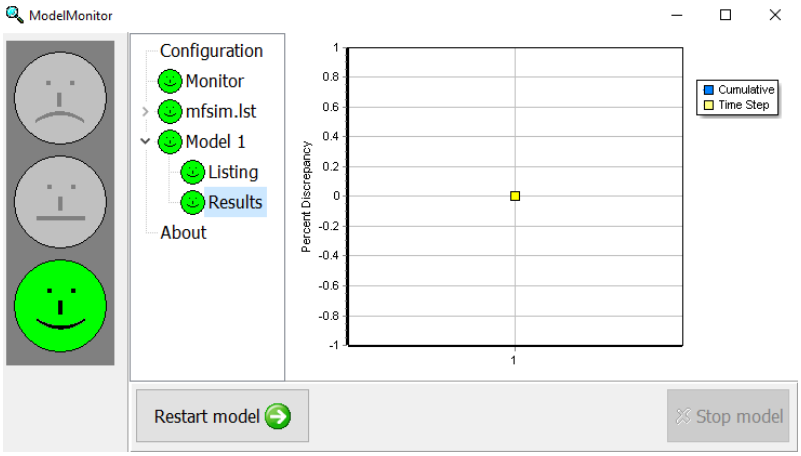


Click on **OK**.

Running the model

First, save the model **File/Save As**, save the simulation with the name **Model1.gpt.** Now we can run our model by clicking in the green arrow located in the upper left corner called "**Run**"

Save the model files in the same folder as the **".gpt**", this folder appears as default, and click on **Save.** The model starts to run and when it finishes it shows a yellow square in the center of the window with green faces, which means the model has run successfully.



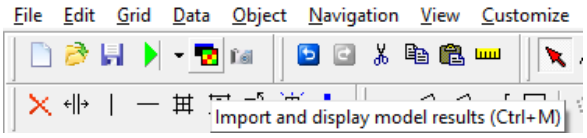
At the end of the water budget file, you can see the water balance and the importance of choosing the **"interpretation"** as **"Direct"** to have the exact value for the pumping and conductance in the boundary conditions.

```
 VOLUMETRIC BUDGET FOR ENTIRE MODEL AT END OF TIME STEP    1, STRESS PERIOD   1
 ----------------------------------------------------------------------------

    CUMULATIVE VOLUMES      L**3       RATES FOR THIS TIME STEP      L**3/T
    ------------------                 ------------------------

         IN:                                   IN:
         ---                                   ---
           STORAGE =          0.0000             STORAGE =          0.0000
    CONSTANT HEAD =       1.1822E-02      CONSTANT HEAD =       1.1822E-02
           WELLS =          0.0000             WELLS =          0.0000
    RIVER LEAKAGE =       1.0098E-03      RIVER LEAKAGE =       1.0098E-03
         RECHARGE =       1.3699E-02           RECHARGE =       1.3699E-02

          TOTAL IN =       2.6531E-02            TOTAL IN =       2.6531E-02

        OUT:                                  OUT:
        ----                                  ----
           STORAGE =          0.0000             STORAGE =          0.0000
    CONSTANT HEAD =       9.3190E-03      CONSTANT HEAD =       9.3190E-03
           WELLS =       1.3889E-02             WELLS =       1.3889E-02
    RIVER LEAKAGE =       3.3228E-03      RIVER LEAKAGE =       3.3228E-03
         RECHARGE =          0.0000           RECHARGE =          0.0000

          TOTAL OUT =       2.6531E-02            TOTAL OUT =       2.6531E-02

           IN - OUT =      -1.8626E-09             IN - OUT =      -1.8626E-09

PERCENT DISCREPANCY =          -0.00     PERCENT DISCREPANCY =          -0.00
```

Close the water budget files and then we can continue to visualize the outputs.

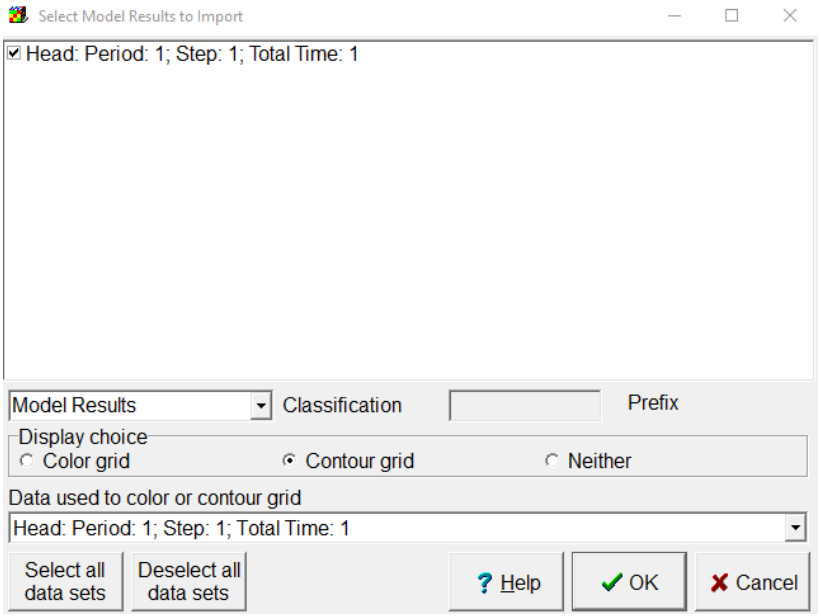
Importing results

We can import the simulated heads going to "**Import and display model results**"

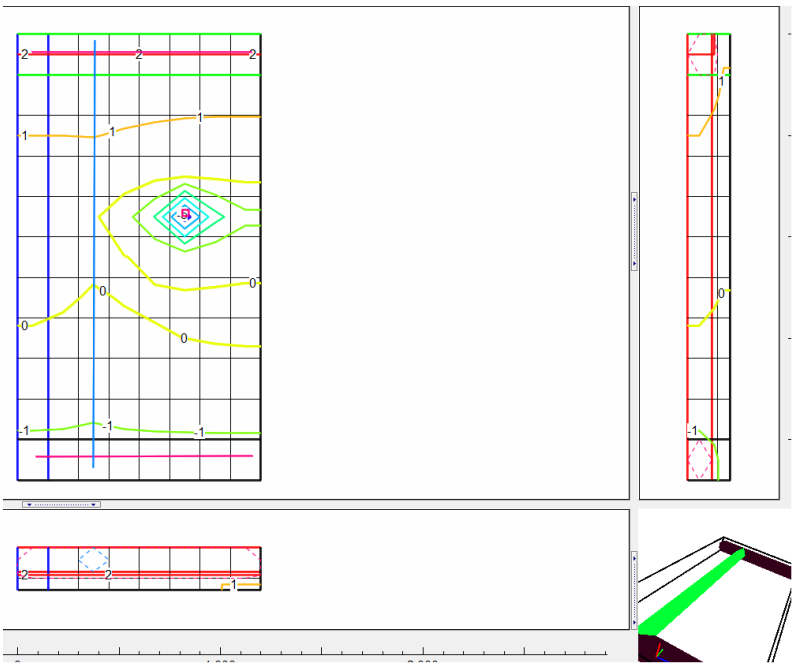


Select the **Model1.fhd** file and click in **Open**

A new window appears indicating the period to be imported and the "**Display choices**", import only period we have as "**contour grid"**



And you get the distribution of the model, in which you can see the distribution of the water table.


2
2
2
1
1
1
5
0
0
0
-1
-1
-1
1
0
-1
2
2
1