COURSE: GROUNDWATER MODELLING USING MODFLOW

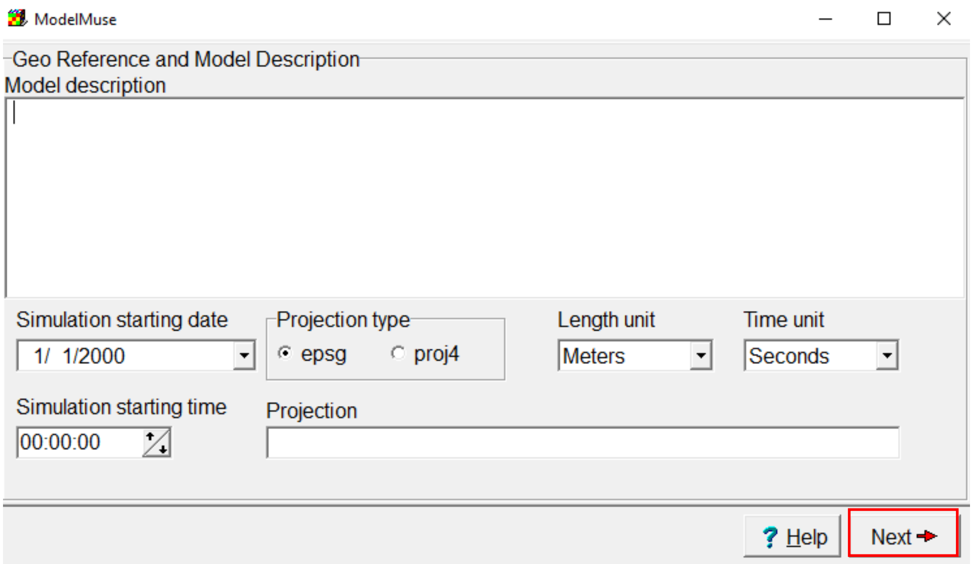
Session 6: Developing a multilayer groundwater model

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".**



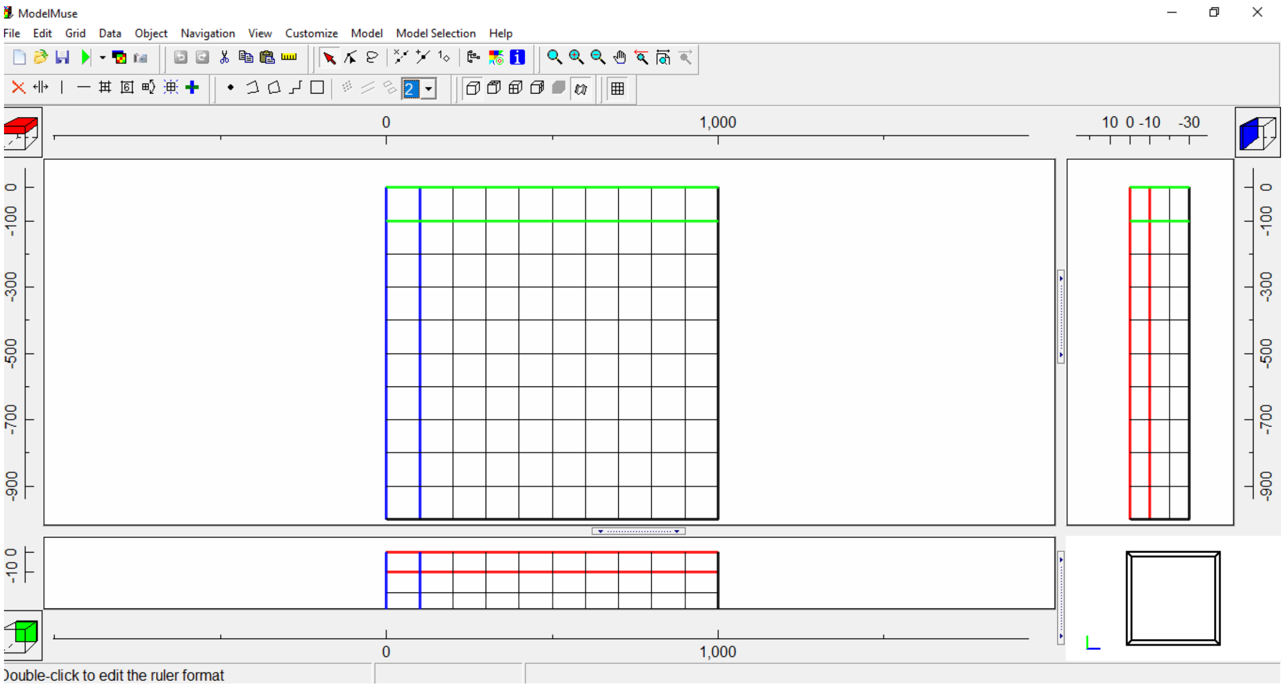
Defining model grid properties

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

- The number of rows is equal to 10 and the number of columns.
- The **number of layers** is equal to **3**.
- The column and row widths are equal to **100**.
- The **model top** is set as **0**

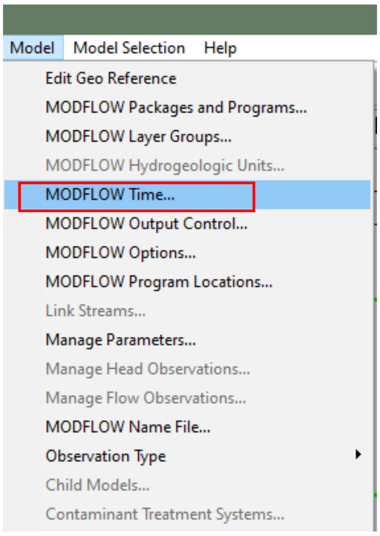
- The **layer group name** is set as **AqX**, where "X" is the layer number.
- The bottom elevations are -10, -20 and -30 meters.

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

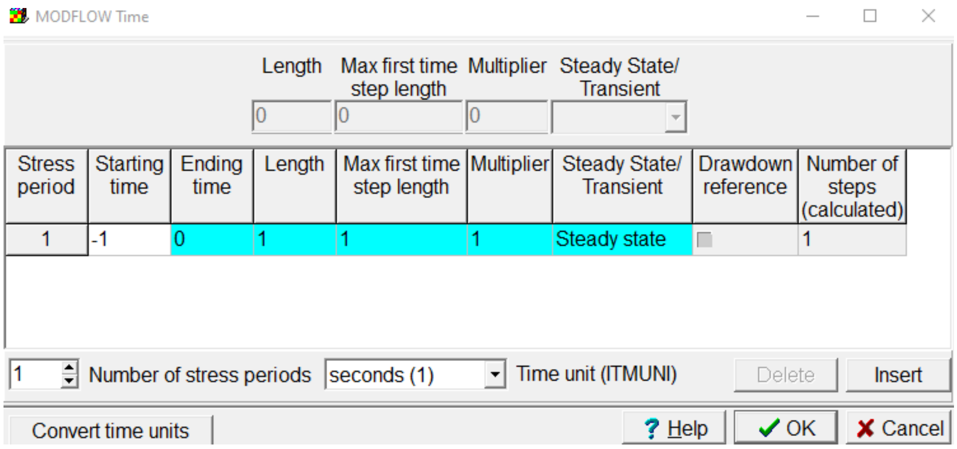


Defining time properties

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.

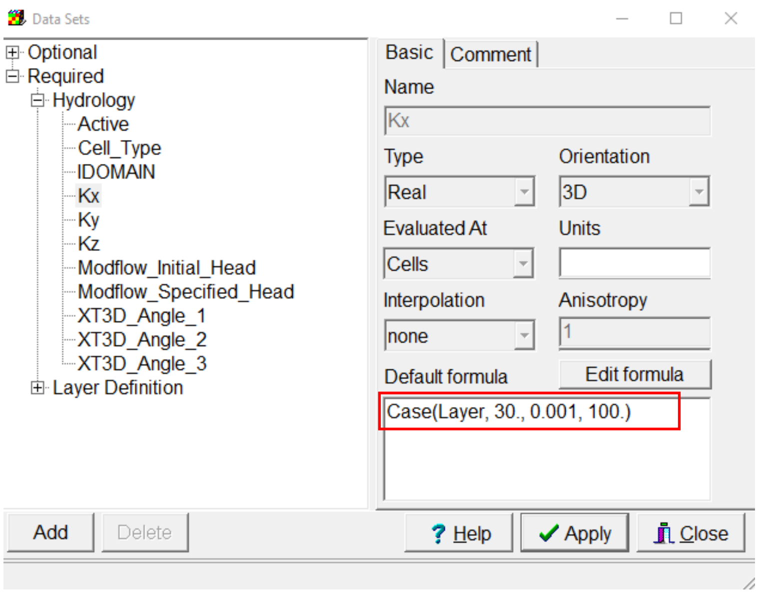


Defining model properties

Let´s define the horizontal and vertical hydraulic conductivities, to do this go to **Edit/Edit data sets…** and choose the **Required/Hydrology** options, here we can define the hydraulic conductivities mentioned.

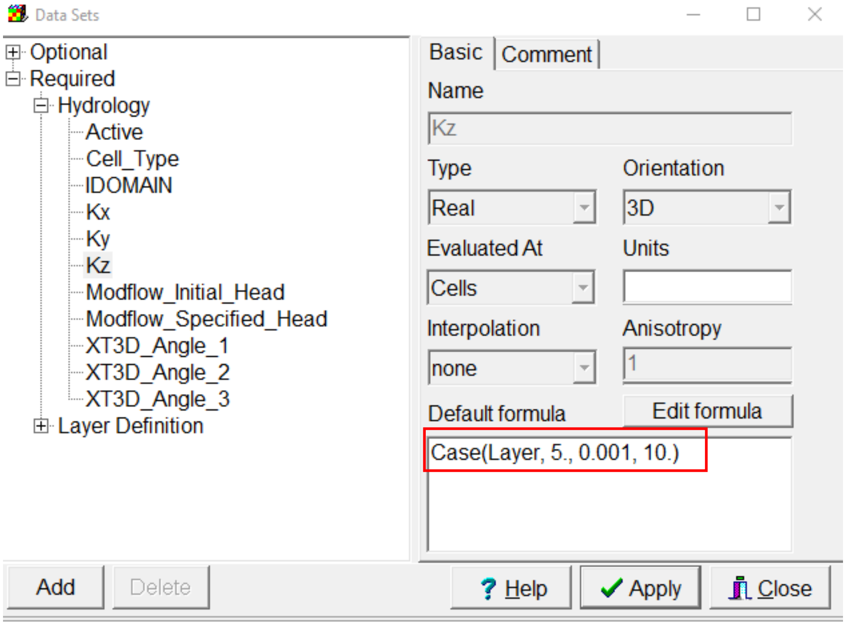
First, the horizontal hydraulic conductivity is under the **"Kx"** option, we set a value for each layer with the following formula:

"Case(Layer, 30., 0.001, 100.)"



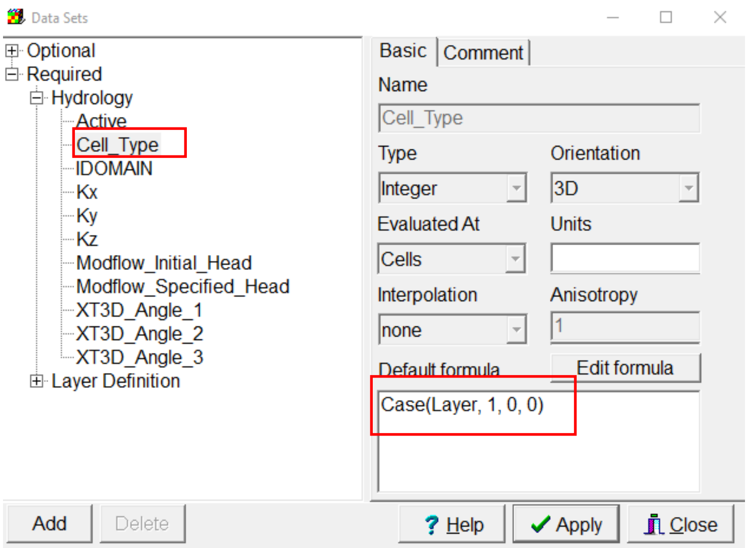
Secondly, the vertical hydraulic conductivity, this one is under the "**Kz"** option, as for the **"Kx"**, set a value for each layer with the following formula:

Case(Layer, 5, 0.001, 10)

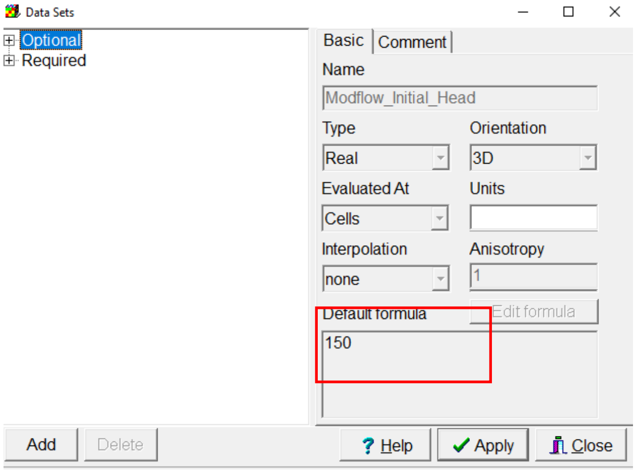


In ModelMuse all the cells are confined as default, we need to change the first layer to convertible, to do this select the **"Cell_Type"** option in **"Data Sets"** and place the following formula:

Case(Layer, 1, 0, 0)



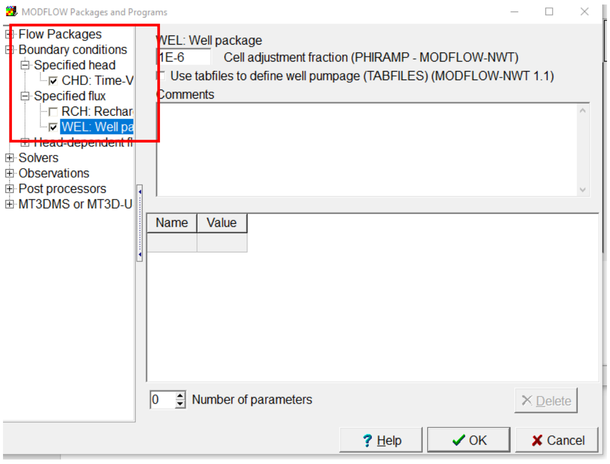
The last parameter you need to modify in the "**Data Sets"** window is the **"Modflow_Initial_head"**, change this value to **150.**



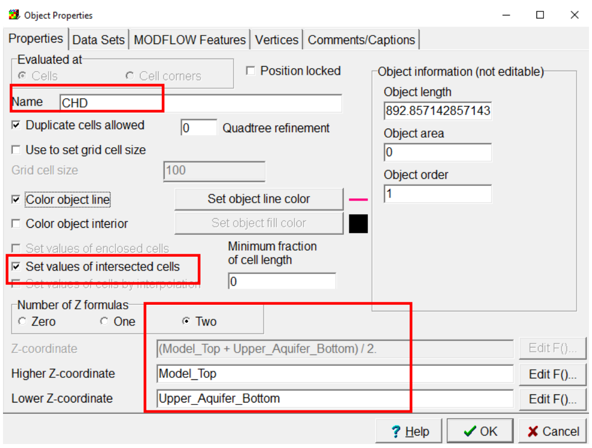
Defining boundary conditions

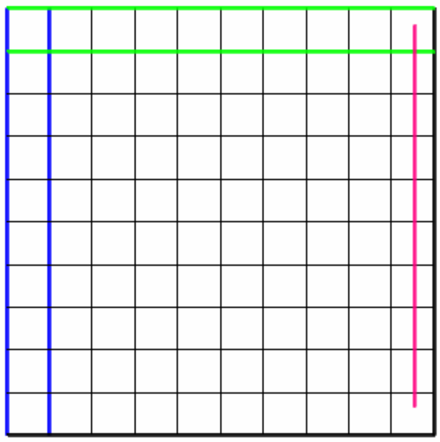
In this exercise, we add 2 boundary conditions, the **Constant head (CHD) a**nd well (WEL). To activate the packages go to **Model/MODFLOW Packages and Programs….**

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

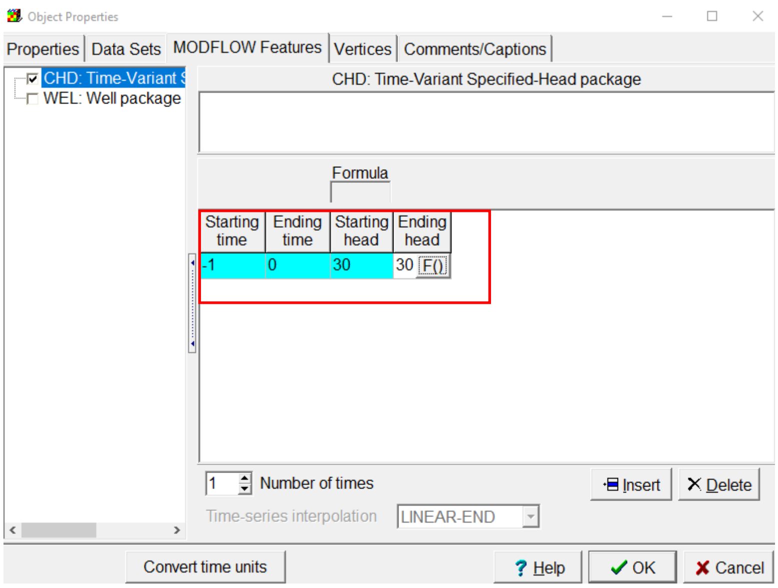


First, to define the **CHD** Object, create a polyline with **Create polyline object** as the imagen. In the properties of this object change the following:





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 30. Click on **OK** to proceed with the next boundary condition.



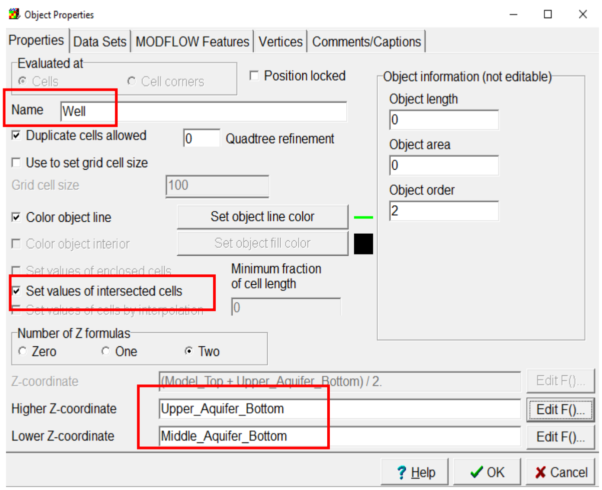
The "WEL" package is located in a grid cell, use the **Create point object** tool. Draw a point located in the layer 3, row 5 and column 6

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_aquifer_bottom**

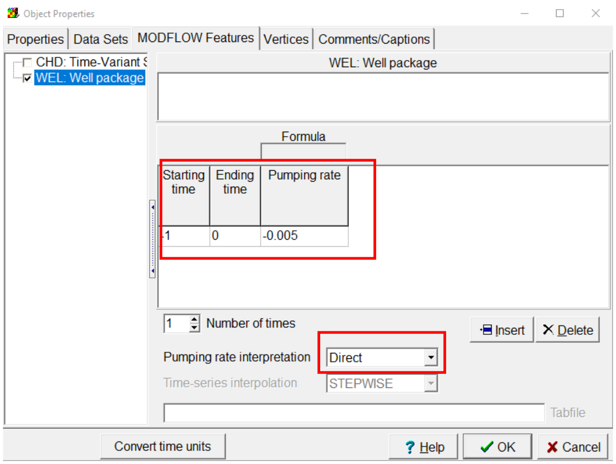
Lower Z-coordinate = **Middle_aquifer_bottom**

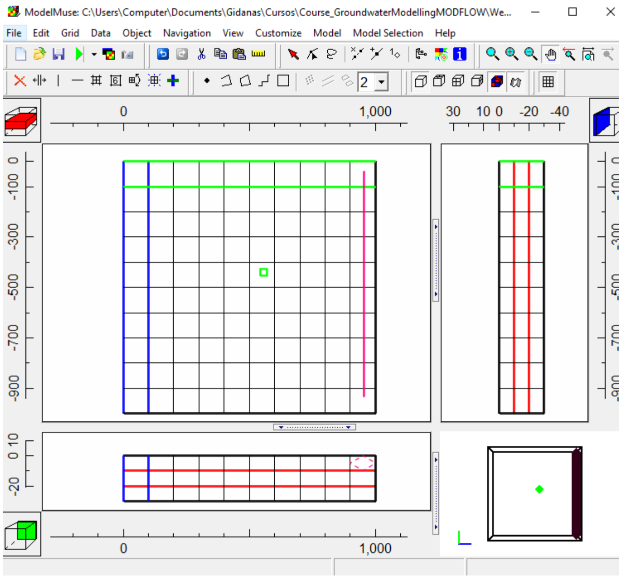


In **"MODFLOW Features"**, activate the option WEL package and do the following settings:

- Set the Pumping rate to -0.005 m3/s.
- Be sure to select Direct in the Pumping rate interpretation options.

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

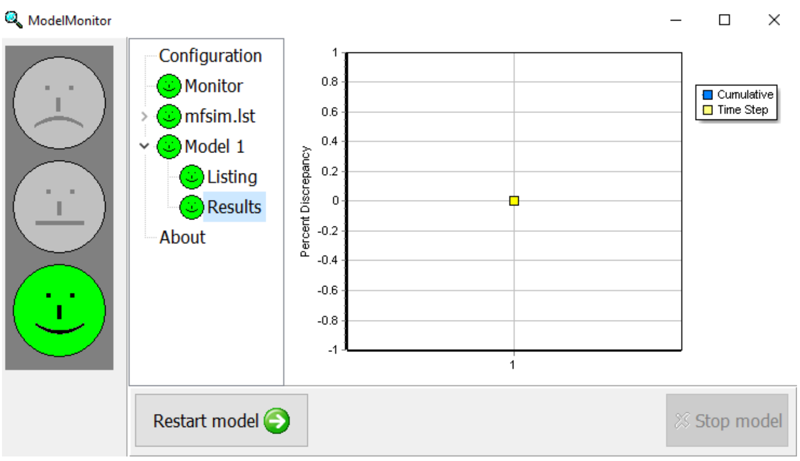




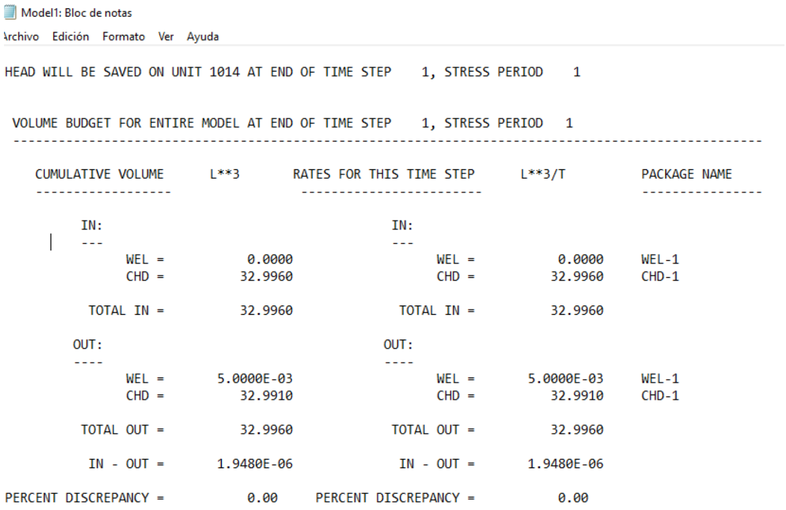
Running the model

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 MODFLOW 6**"

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.



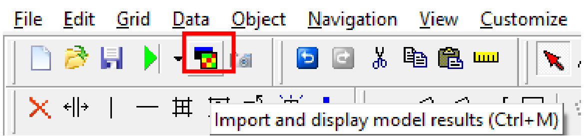
Model1: Bloc de notas
Archivo Edición Formato Ver Ayuda

```
HEAD WILL BE SAVED ON UNIT 1014 AT END OF TIME STEP    1, STRESS PERIOD    1

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

    CUMULATIVE VOLUME      L**3       RATES FOR THIS TIME STEP     L**3/T          PACKAGE NAME
    ------------------                 ------------------------                   ----------------

          IN:                                   IN:
          ---                                   ---
                WEL =           0.0000                WEL =           0.0000     WEL-1
                CHD =          32.9960                CHD =          32.9960     CHD-1

           TOTAL IN =          32.9960             TOTAL IN =          32.9960

         OUT:                                  OUT:
         ----                                  ----
                WEL =       5.0000E-03                WEL =       5.0000E-03     WEL-1
                CHD =          32.9910                CHD =          32.9910     CHD-1

          TOTAL OUT =          32.9960            TOTAL OUT =          32.9960

           IN - OUT =       1.9480E-06             IN - OUT =       1.9480E-06

PERCENT DISCREPANCY =             0.00     PERCENT DISCREPANCY =             0.00
```

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

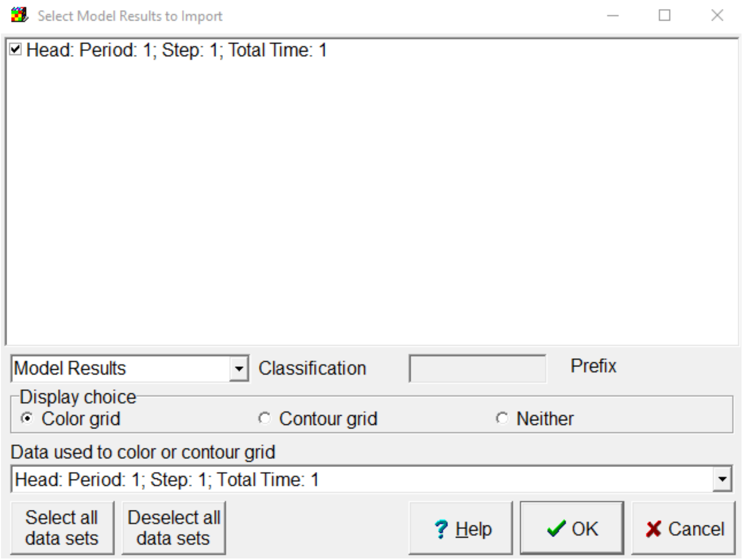
Importing results

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



Select the **Model1.bhd** 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 "**color grid"**



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