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:

\[
\text{Case}(\text{Layer}, 1, 0, 0)
\]

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) and 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.
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.