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**".

🐉 ModelMuse		_		×
Geo Reference and Model Description				
Simulation starting date Projection type 1/ 1/2000 epsg proj4 	Length unit Meters	Time unit Seconds	-	
Simulation starting time Projection				
00:00:00				
		1		
		? <u>H</u> elp	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.

🐉 MODFL	OW Time							- 🗆	\times
			Length	Max first time step length	Multiplier	Steady State/ Transient			
Stress period	Starting time	Ending time	Length	Max first time step length	Multiplier	Steady State/ Transient	Drawdown reference	Number of steps (calculated)	
1	-1	0	1	1	1	Steady state		1	
1 Number of stress periods seconds (1) Time unit (ITMUNI) Delete Insert 									
Conve	ert time un	its				? <u>H</u> e	lp 🗸 C	OK X Ca	ancel

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:

🥵 Data Sets		- 🗆 X	
 ⊖ Optional ⇒ Required ⇒ Hydrology → Active → Cell_Type → IDOMAIN → Kx → Ky → Kz → Modflow_Initial_Head → Modflow_Specified_Head → XT3D_Angle_1 → XT3D_Angle_3 ⊞ Layer Definition 	Basic Comment Name Name Kx Type Real • Evaluated At Cells • Interpolation none • Default formula Case(Layer, 30., 0.	Orientation 3D Units Anisotropy 1 Edit formula 001, 100.)	
Add Delete	? <u>H</u> elp	Apply <u>I C</u> lose	

"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)

🔀 Data Sets	- 🗆 ×
 Optional Required Hydrology Active Cell_Type IDOMAIN Kx Ky Kz Modflow_Initial_Head Modflow_Specified_Head XT3D_Angle_1 XT3D_Angle_2 XT3D_Angle_3 E Layer Definition 	Basic Comment Name Name Kz Type Orientation Real 3D Evaluated At Units Cells Interpolation Anisotropy none 1 Default formula Edit formula Case(Layer, 5., 0.001, 10.)
Add Delete	<u>? H</u> elp ✓ Apply <u>I C</u> lose

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)

😕 Data Sets		_		\times
 Optional Required Hydrology Active Cell Type IDOMAIN Kx Ky Kz Modflow_Initial_Head Modflow_Specified_Head XT3D_Angle_1 XT3D_Angle_2 XT3D_Angle_3 E Layer Definition 	Basic Comment Name Cell_Type Type Integer ✓ Evaluated At Cells ✓ Interpolation none ✓ Default formula Case(Layer, 1, 0, 0)	Orientation 3D Units Anisotropy 1 Edit form	nula	
Add Delete	<u>? H</u> elp ↓	Apply	<u>i</u> <u>C</u> los	se

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

🐉 Data Sets		– 🗆 X					
Optional	Basic Comment						
⊞ Required	Name						
	Modflow_Initial_Head						
	Туре	Orientation					
	Real	3D -					
	Evaluated At	Units					
	Cells -						
	Interpolation	Anisotropy					
	none 👻	1					
	Default formula	Edit formula					
	150						
	150						
Add Delete	? <u>H</u> elp	Apply					

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**.

MODFLOW Packages and Pr	ograms	-		×
Flow Packages Flow Packages Boundary conditions Specified head □ Fire CHD: Time-V Specified flux □ WEL Welp & WEL Welp & Weth Welp & Wet	VEL: Well package IE-6 Cell adjustment fraction (PHIRAMP - MODFLOW-NV		1.1)	< >
	0	×	Delete	
	? ∐еір 🗸 ОК		Canc	el
	· 2.9	-		

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:

Object Properties		-	- 🗆 🗙
Properties Data Sets MO	DFLOW Features Vertices Comments/Ca	ptions	
Evaluated at Cells CCe	bil comers	Object information (not e	ditable)
Name CHD		Object length 892.857142857143	
Duplicate cells allowed	0 Quadtree refinement	Object area	
Use to set grid cell size		0	
Grid cell size	100	Object order	
Color object line	Set object line color	1	
Color object interior	Set object fill color		
Set values of enclosed of			
 Set values of intersected Oct values of cells by intersected 	0		
Number of Z formulas	r Two		
Z-coordinate	(Model_Top + Upper_Aquifer_Bottom) / 2.		Edit F()
Higher Z-coordinate	Model_Top		Edit F()
Lower Z-coordinate	Upper_Aquifer_Bottom		Edit F()

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.

🔀 Object Properties					_		<
Properties Data Sets MC	DFLOW Features	Vertices	Comments/C	aptions			
CHD: Time-Variant		CHD: Tim	ne-Variant Spe	cified-Head p	ackage		
WEL: Well package							
		Formula					
	Starting Ending time	Starting head	Ending head				
•	-1 0		30 F()				
		6 King a a					1
	1 🔮 Number o				• ■ Insert	× <u>D</u> elete	
< >>	Time-series interp	olation	LINEAR-END	T			
Convert	time units			? <u>H</u> elp	🗸 ОК	X Cance	el

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**

🔀 Object Properties		-		×		
Properties Data Sets M	ODFLOW Features Vertices Comments/Captions					
Evaluated at Collis Collis	Cell corners Position locked Object information (not e Object length	ditat	ole)			
Duplicate cells allowed Use to set grid cell size Grid cell size	Object area					
Color object line	Set object line color 2					
Color object interior	Set object fill color					
 Set values of enclosed Set values of intersecter Set values of cells by in 	ed cells					
Number of Z formulas ↑ Zero ↑ One ↑ Two						
Z-coordinate	(Model_Top + Upper_Aquifer_Bottom) / 2.		Edit F	()		
Higher Z-coordinate	Upper_Aquifer_Bottom	[Edit F	()		
Lower Z-coordinate	Middle_Aquifer_Bottom		Edit F	()		
	<u>? H</u> elp ✓ OK		🗙 Car	ncel		

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 $\ensuremath{\mathsf{OK}}$ to continue with the next boundary condition.

	Modelinduse Cr(osels/Computer/Documents/Glaanas/Course_GroundwaterinduelingMODPLOW/We
	File Edit Grid Data Object Navigation View Customize Model Model Selection Help
🐉 Object Properties — 🗆 🛛 🗡	🗋 😂 📕 🕨 - 🖻 💷 ڬ 🖉 📾 🏙 📟 🗙 K ව 🏹 🏷 । 🛤 🦉 🖉 🦉 🔍
Properties Data Sets MODFLOW Features Vertices Comments/Captions	× ᡧ Ⅰ - 苒 図 凾 漢 ╋ ┃ ・ ጏ ひ ノ ロ ※ 彡 冬 2 ▼ ⑦ ⑦ ❸ ∂ ● @ ■
CHD: Time-Variant WEL: Well package	
Formula	
Starting Ending Pumping rate	88
1 0 0.005	
I € Number of times	0,
Pumping rate interpretation Direct	
Time-series interpolation STEPWISE Tabfile	
Convert time units ? Help 🗸 OK X Cancel	•

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**"

🗋 🤌 🛛	• •	🔽 îs	5	đ	Ж
★ ⊀⊪>	Run	MODFL	OW 6 (C	trl+E) •

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 UN	IIT 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 RAT	ES FOR THIS TIME STEP	L**3/T	PACKAGE NAME					
IN:		IN:							
	0.0000		0.0000						
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"

<u>F</u> ile	<u>E</u> dit	G	rid	<u>D</u> ata	<u>O</u> bject	<u>N</u> aviga	ation	<u>V</u> iew	<u>C</u> ust	omize
	8	Ы		- 🔽	isi 🛛	5	Ж	Þa 🛍	Luud	
X	← →	I	-	₩	+ _s `≓ mport an	d displa	i y mo	del resul	ts (Ct	rl+M)

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.

