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 C epsg C proj4	Length unit Meters	Time unit Seconds	•	
Simulation starting time Projection	,	,		
00:00:00				
		? <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	Numb ste (calcu	per of ps lated)	
1	-1	0	1	1	1	Steady state		1		
1	Number o	of stress p	eriods s	seconds (1)	▼ Time	e unit (ITMUNI)	Dele	te	Inse	rt
Conve	rt time un	its				? <u>H</u> e	lp 🗸 O	K	🗙 Ca	ncel

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		_		\times
 Poptional Required Hydrology Active Cell_Type IDOMAIN Kx Ky Kz Modflow_Initial_Head Modflow_Specified_Head XT3D_Angle_1 XT3D_Angle_3 E 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 formu 001, 100.)		
Add Delete	? <u>H</u> elp	Apply	<u>Clos</u>	e

"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	– 🗆 X
 Optional Required Hydrology Active Cell_Type IDOMAIN Kx Ky Kz Modflow_Initial_Head Modflow_Specified_Head XT3D_Angle_1 XT3D_Angle_3 E Layer Definition 	Basic Comment Name Cell_Type Type Orientation Integer 3D Evaluated At Units Cells Interpolation Anisotropy none Default formula Edit formula Case(Layer, 1, 0, 0)
Add Delete	<u>? H</u> elp

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					
)	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 P	rograms	-		\times
Flow Packages Boundary conditions Specified head □ CHD: Time-V Specified flux □ CH: Rechar □ WEL Welp or ○ Wet Welp or ○ Nores ○ Observations ○ Post processors ⊕ MT3DMS or MT3D-U	VEL: Well package TE-6 Cell adjustment fraction (PHIRAMP - MODFLOW-NV Use tabfiles to define well pumpage (TABFILES) (MODFLOW- comments	VT) NWT ·	1.1)	< >
	0 👙 Number of parameters	×	<u>D</u> elete]
	<u>? H</u> elp ✓ OK	;	Canc	el

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:

Properties Data Sets M	ODFLOW Features Vertices Comments/C	Captions			
Cells CC	ell corners	Object information (not ec	litable)		
Name CHD		Object length			
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 object fill color			
Set values of enclosed	cells Minimum fraction				
 Set values of intersecte Oct values of cells by in 	d cells of cell length				
-Number of ∠ formulas ○ Zero ○ One	○ Two				
C Zero C One	Two (Model_Top + Upper_Aquifer_Bottom) / 2	2	Edit F()		
Zero One Zero One Z-coordinate Higher Z-coordinate	© Two (Model_Top + Upper_Aquifer_Bottom) / 2 Model_Top	2	Edit F()		
Zero One Zero One Zero One Lower Z-coordinate Lower Z-coordinate	© Two (Model_Top + Upper_Aquifer_Bottom) / 2 Model_Top Upper_Aquifer_Bottom	2.	Edit F() Edit F() 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	Starting	Ending				
•	-1 0	30	30 F()				
		6 King a a					1
		i umes			•∎ <u>I</u> nsert	× <u>D</u> elete	
< >>	Time-series interp	olation	JLINEAR-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	Cell corners Position locked Object information (not e Object length	ditat	ole)		
Duplicate cells allowed Use to set grid cell size Grid cell size	0 Quadtree refinement 0				
Color object line	Set object line color				
Color object interior	Set object fill color				
 Set values of enclosed Set values of intersecter Set values of cells by in 	cells Minimum fraction of cell length terpolation				
Clark Construction Constr					
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	1	🗙 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.

	wodelinuse c.(osers(computer)b)cuments(oluanas(cursos(course_oroundwaterinodelininginobretow(we — — — — — — — — — — — — — — — — — —
	File Edit Grid Data Object Navigation View Customize Model Model Selection Help
💈 Object Properties — 🗆 🗙	🗋 👌 🖬 🕨 - 🔞 📾 ڬ 🔤 📽 📾 🗮 📗 🗮 🗙 K 🖻 🗡 🏏 🕨 🦉 🔍 🔍 🔍 🖓 🗮
Properties Data Sets MODFLOW Features Vertices Comments/Captions	× 小 - 井 図 較 漢 ┿ ・ ユ ひ ノ ロ ※ 彡 ※ 2 ・ ① ⑦ 舠 ⑦ ● ☞ 冊
CHD: Time-Variant § WEL: Well package WEL-Well package	
Formula	
1	
Time-series interpolation STEPWISE Tabfile Tabfile Convert time units ? Help VOK 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**"

	8	H	•	7	Êđ		ъ	đ	Ж
×	← →	I	Run	MC	DFL	DW	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 C	DN UNIT 1014 AT ENE	OF TIME STEP 1, ST	RESS PERIOD 1								
VOLUME BUDGET FOR E	NTIRE MODEL AT END	OF TIME STEP 1, ST	RESS PERIOD 1								
CUMULATIVE VOLUM	1E L**3	RATES FOR THIS TIME ST	EP L**3/T	PACKAGE NAME							
IN:		IN:									
WFI	= 0.0000	 WFI	= 0.0000	WEL-1							
CHD	= 32.9966	CHD	= 32.9960	CHD-1							
TOTAL IN	= 32.9966	TOTAL IN	= 32.9960								
OUT:		OUT:									
WEL	= 5.0000E-03	WEL	= 5.0000E-03	WEL-1							
CHD	= 32.9916	CHD CHD	= 32.9910	CHD-1							
TOTAL OUT	= 32.9966	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	t <u>G</u>	rid	<u>D</u> at	a <u>O</u>	bject	N	aviga	tion	Vi	ew	<u>C</u> us	tomi	ze
	8	5		-	2	1	5	e	¥	Ē7	ĉ	шш		R I
×	¢∥→		_	挕	 Imp	ort ar	⊷ nd di	isplay	i y mo	ر del i	esul	v r Its (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.

