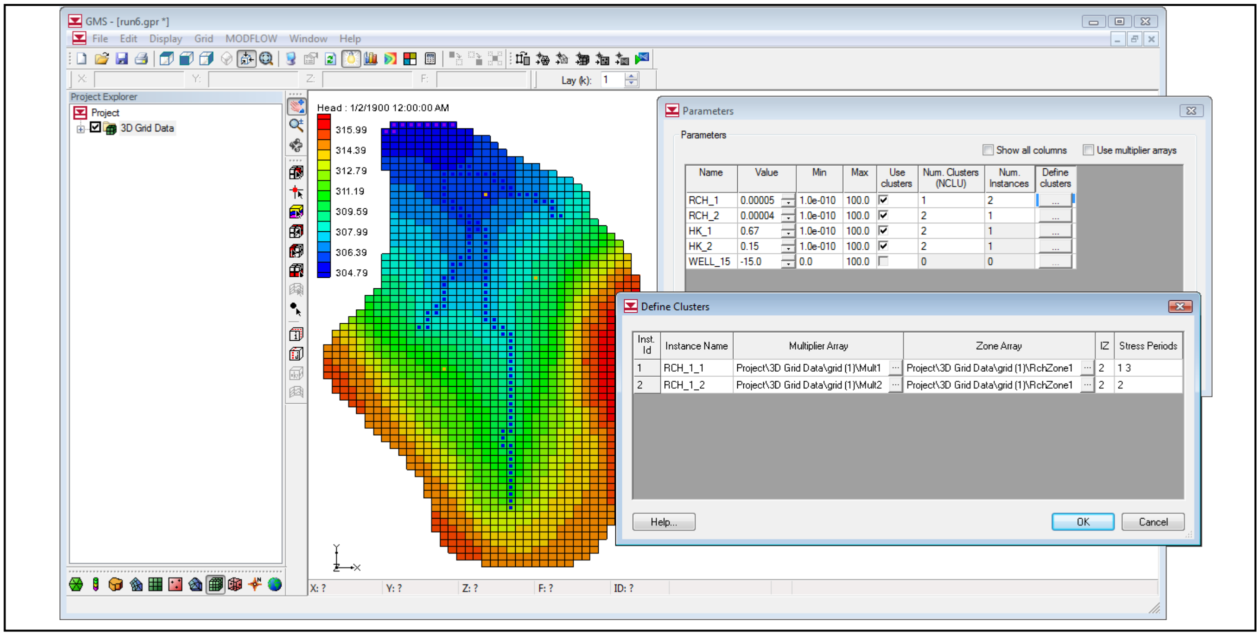
GMS

v. 10.0

GMS 10.0 Tutorial

MODFLOW – Advanced Parameter Options

GMS support for native MODFLOW parameters, instances, and clusters



Objectives

Learn how to use some of the advanced features associated with MODFLOW parameters including instances and clusters. Define array-based parameters using clusters. Learn about parameter factors.

Prerequisite Tutorials

- MODFLOW – Automated Parameter Estimation

Required Components

- Grid Module
- Inverse Modeling
- Map Module
- MODFLOW

Time

- 25-40 minutes

AQUAVEO™

1 Introduction

This tutorial illustrates how to use some of the advanced features associated with MODFLOW parameters. Since this tutorial assumes that the user understands how to use the parameter estimation tools available in GMS, the user should complete the "MODFLOW –Automated Parameter Estimation" tutorial before starting this tutorial. The user should also understand how to define array-based parameters using clusters and the concept of parameter instances. Learn more about these concepts by reading the MODFLOW documentation.

1.1 Outline

Here are the steps of this tutorial:

1. Open a parameterized MODFLOW model and solution.
2. Define RCH parameters using clusters.
3. Run MODFLOW.
4. Change the model to be transient.
5. Change the parameters to use instances.
6. Run MODFLOW.
7. Define LPF parameters using clusters.

8. Run MODFLOW.

9. Adjust parameter factors.

10. Run MODFLOW.

2 Description of Problem

The model in this tutorial is the same model featured in the “MODFLOW – Automated Parameter Estimation” tutorial. For most cases, array-based parameters can be defined using the key value approach used in the “MODFLOW – Automated Parameter Estimation” tutorial. However, this approach is not completely compatible with the native MODFLOW parameter formats. MODFLOW array-based parameters are defined using clusters. A cluster is a multiplier array, a zone array, and an IZ value (key values associated with the zone array). The key value approach used in GMS would be like creating an array-based parameter that is defined by a single cluster. MODFLOW supports parameters that are defined by multiple clusters and GMS supports this feature. In the *Parameters* dialog, if the user toggles on the *Show all columns* option, it is possible to turn on the option to define a parameter using clusters. This allows the user to select a multiplier array and a zone array and to specify IZ.

In addition to clusters, MODFLOW also supports defining parameter instances for parameters associated with a transient MODFLOW model. This feature allows a user to create a single parameter and then associate multiple sets of clusters with instances of that parameter. Then the user can specify which stress period each instance is used with.

Since this can be quite confusing, the user will follow an example where RCH parameters are first defined using clusters in a steady state model. Then the user will change the model to be transient and create more than one instance for one of the recharge parameters.

3 Getting Started

Do the following to get started:

1. If necessary, launch GMS.

2. If GMS is already running, select the *File* | **New** command to ensure that the program settings are restored to their default state.

4 Reading in the Project

First, read in the modeling project:

1. Select the **Open** button.

2. Locate and open the *Tutorials\MODFLOW\advparam* directory.

3. Select the file entitled “start.gpr.”

4. Click **Open.**

The user should see a MODFLOW model with a solution and a set of map coverages. In the Project Explorer, expand the “3D Grid Data” folder and the “grid” item. The user should see seven data sets below the 3D Grid named as follows: “HKZone1,” “HKZone2,” “Mult1,” “Mult2,” “RchZone1,” “RchZone2,” and “RchZone3.” These datasets will be used to define the RCH and LPF parameters.

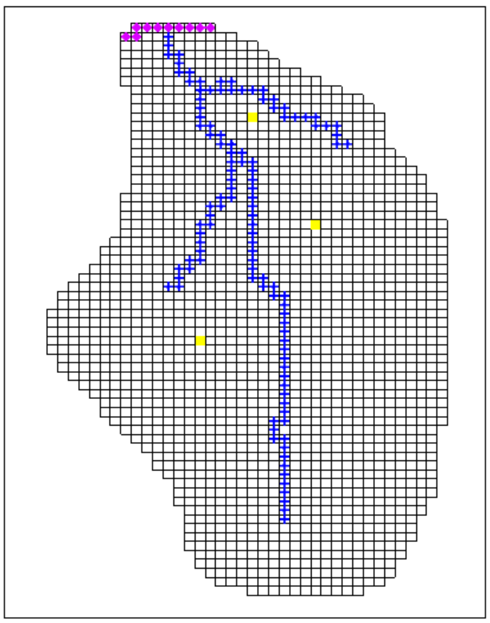


Figure 1 The model

5 Creating a Recharge Parameter with a Cluster

First the user will create a recharge parameter that is defined by a single cluster. Then the user will run MODFLOW.

1. Select the *MODFLOW* | **Parameters** command to open the *Parameters* dialog.

2. Select the *Show all columns* toggle at the top of the dialog.

3. Select the *New* button near the bottom left of the dialog. This will create a new parameter.

4. Change the following values for the newly created parameter:

   - *Name* of “RCH_1”

- *Type* of "RCH"
- *Value* of "0.00005."

5. Turn on the *Use clusters* toggle for the parameter.
6. Enter "1" for the *Num. Clusters.*
7. Enter "1" for *Num. Instances*.
8. Select the **Define Clusters** button to open the *Define Clusters* dialog.

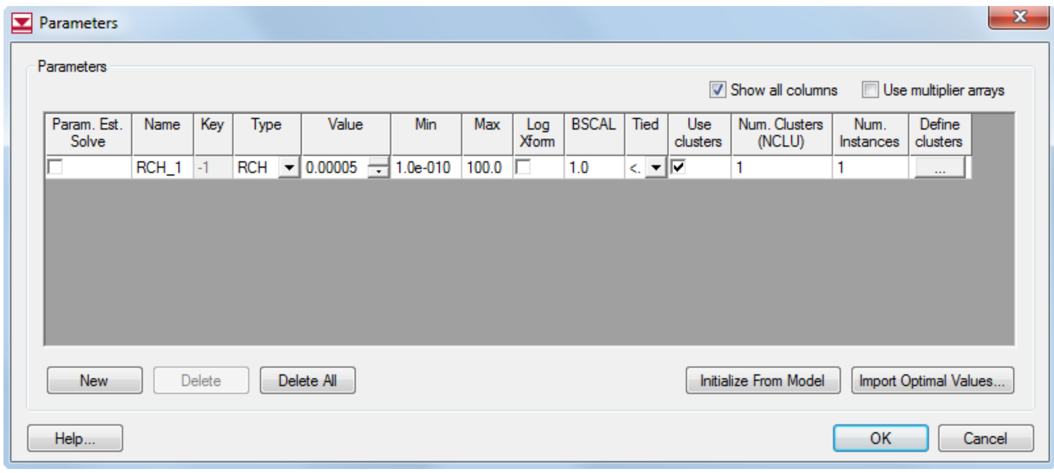


Figure 2 Parameter Instances dialog

5.1 The Define Clusters Dialog

Clusters are defined in the *Define Clusters* dialog. Since this is a steady state model, the user will only have one instance. In this dialog, the user can define parameter instances and clusters. Currently, the example has only 1 instance and 1 cluster, so only one row is available in the dialog.

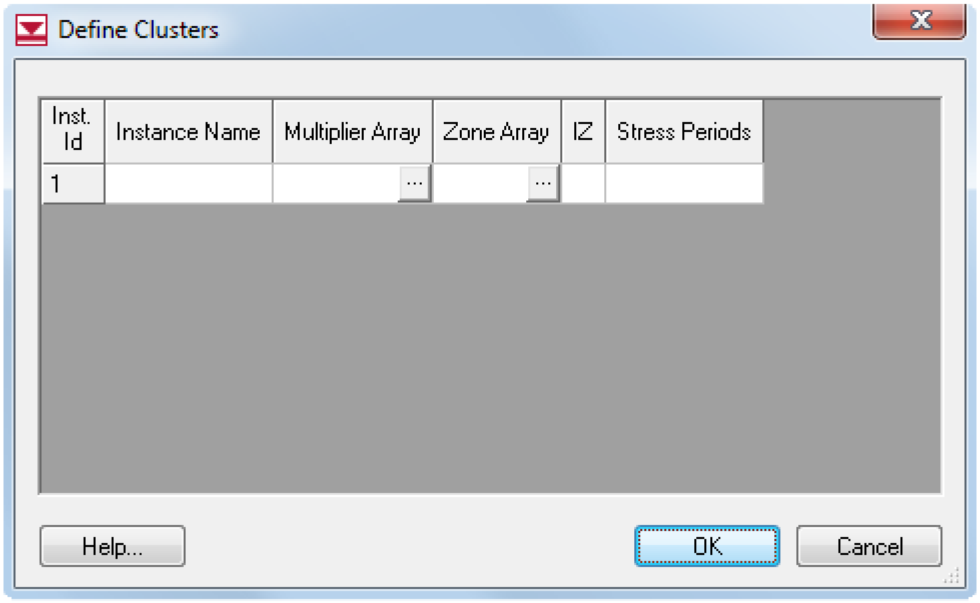


Figure 3 Define Clusters dialog

1. Select the **Cancel** button.
2. Change the *Num. Clusters* to "2."
3. Select the **Define Clusters** button to open the *Define Clusters* dialog.

Notice that now the user has two rows in the spreadsheet. The user would have to define two multiplier arrays and two zone arrays as well as IZ values to successfully define the clusters.

4. Select the **Cancel** button.
5. Change the *Num. Instances* to be "2."
6. Select the **Define Clusters** button to open the *Define Clusters* dialog.

Notice that now the user has four rows in the spreadsheet. The user would have to define two clusters for each instance. Notice also that the user must specify an instance name as well as define the stress periods where the instance is used. So for a transient model with 3 stress periods, one of the instances could be used for stress periods 1 and 2, and the other instance could be used with stress period 3. If the user defines a RCH or EVT parameter using clusters and instances, it is necessary to make sure that at least one parameter instance is assigned to every stress period in the model.

7. Select the **Cancel** button.
8. Change the *Num. Clusters* to be "1."
9. Change the *Num. Instances* to be "1."
10. Select the **Define Clusters** button.

11. Enter “RCH_1_1” for the *Instance Name*. Note: The user should limit the length of an instance name to 10 characters; this is a MODFLOW limitation.

12. Select the button in the *Multiplier Array* column. This brings up the *Select Data Set* dialog.

13. Select “Mult1.”

14. Select **OK**.

15. Select the button in the *Zone Array* column. This opens the *Select Data Set* dialog.

16. Select “RchZone1.”

17. Then select **OK**.

18. Enter “2” in the *IZ* field.

The IZ field is used to specify where the parameter is active in the MODFLOW grid. RchZone1 has values of 0 and 2. The user specifies 2 as the IZ, and then, when MODFLOW runs, the parameter will only be used where the zone array has a value of 2.

19. Enter “1” in the *Stress Periods* field.

20. Click **OK** to exit the *Define Clusters* dialog.

21. Click **OK** to exit the *Parameters* dialog.

The user has now defined a parameter using clusters, and the user will run MODFLOW.

6 Running MODFLOW

Now save the project and run MODFLOW.

1. Select the *File* | **Save As** command.

2. Change the project name to “run1.gpr.”

3. Click the **Save** button.

4. Select the *MODFLOW* | **Run MODFLOW** command.

Once MODFLOW is done running, the user can read in the solution.

5. Make sure that the *Read solution on exit* toggle is checked.

6. Select the **Close** button.

The user should now see the head contours from the MODFLOW solution.

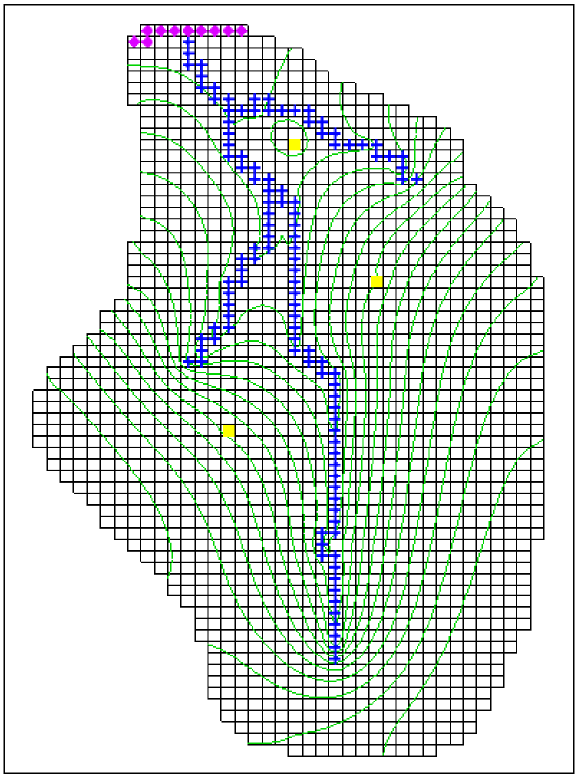


Figure 4 Model with solution contours

7 Creating a Recharge Parameter with Multiple Clusters

Now create another recharge parameter that is defined by two clusters before running MODFLOW again.

1. Select the *MODFLOW* | **Parameters** command to open the *Parameters* dialog.
2. Make sure the *Show all columns* toggle at the top of the dialog is checked.
3. Select the **New** button near the bottom left of the dialog
4. Change the values for the newly created parameter as follows:
    - *Name* of “RCH_2”
    - *Type* of “RCH”
    - *Value* of “0.00004.”
5. Turn on the *Use clusters* toggle for the parameter.
6. Enter “2” for the *Num. Clusters*.
7. Enter “1” for *Num. Instances*.

8. Select the **Define Clusters** button to open the *Define Clusters* dialog.

9. Enter the values for the clusters as shown in the table below. Only the dataset names are shown for the Multiplier and Zone arrays below, but it is necessary to use the button to select the datasets after which the full path to the datasets will be displayed. The IZ values should be separated by spaces.

Instance Name	Multiplier Array	Zone Array	IZ	Stress Per.
RCH_2_1	Mult2	RchZone2	4 5 3	1
	Mult2	RchZone3	6 7	

10. When finished, click **OK** to exit the *Define Clusters* dialog.

11. Click **OK** to exit the *Parameters* dialog.

Since two RCH parameters are defined for stress period 1, the final value of the recharge rate will be the sum of the two parameters. MODFLOW computes this internally.

8 Running MODFLOW

Now save the project and run MODFLOW.

1. Select the *File* | **Save As** command.

2. Change the project name to "run2.gpr."

3. Click the **Save** button.

4. Select the *MODFLOW* | **Run MODFLOW** command.

Once MODFLOW is done running, the user can read in the solution.

5. Make sure that the *Read solution on exit* toggle is checked.

6. Select the **Close** button.

9 Creating a Transient Model

In addition to defining parameters using clusters, the user can also define parameter instances for transient models. Now the user will change the model to be transient and use parameter instances.

9.1 Setting up the Stress Periods

The user will change the model to be transient and make 3 stress periods.

1. Select the *MODFLOW* | **Global Options** command to open the *MODFLOW Global/Basic Package* dialog.

2. In the *Model type* section of the dialog, change the option to *Transient*.

3. Select the **Stress Periods** button to open the *Stress Periods* dialog.

4. Set the *Number of stress periods* to "3."

5. Set the *Num Time Steps* for each stress period to be "1."

6. Select **OK** to exit the *Stress Periods* dialog.

7. Select **OK** to exit the *MODFLOW Global/Basic Package* dialog.

9.2 Editing the Specific Yield

Since this is an unconfined model, the user needs to edit the specific yield value in the LPF package.

1. Choose the **Select Cells**  tool.

2. Select the *Edit* | **Select All** command.

3. Select the *Edit* | **Properties** command to open the *3D Grid Cell Properties* dialog.

4. Enter "0.01" for *Specific yield.*

5. Select **OK** to exit the dialog.

9.3 Editing the RCH Parameters

Now it is necessary to update the parameters to be compatible with the stress periods.

1. Select the *MODFLOW* | **Parameters** command.

2. Select the **Define Clusters** button for the "RCH_1" parameter.

3. In the *Define Clusters* dialog, change the *Stress Periods* field to have "1 2 3" as the value (numbers separated by spaces).

4. Click **OK** to exit the *Define Clusters* dialog.

Now this parameter will be used for each of the stress periods.

5. Repeat these steps for the "RCH_2" parameter.

6. Click **OK** to exit the *Parameters* dialog.

10 Running MODFLOW

Now save the project and run MODFLOW.

1. Select the *File* | **Save As** command.
2. Change the project name to "run3.gpr."
3. Click **Save.**
4. Select the *MODFLOW* | **Run MODFLOW** command.

Once MODFLOW is done running, the user can read in the solution.

5. Make sure that the *Read solution on exit* toggle is checked.
6. Select the **Close** button.

11 Using Parameter Instances

Now change one of the recharge parameters to use instances.

1. Select the *MODFLOW* | **Parameters** command to open the *Parameters* dialog.
2. For the "RCH_1" parameter, change the *Num. Instances* to be "2."
3. Select the *Define Clusters* button for the "RCH_1" parameter to open the *Define Clusters* dialog.
4. For row 1 ("RCH_1_1" in the first row):
   - Change the *Stress Periods* field to be "1 3."
5. For row 2:
   - Enter "RCH_1_2" for the *Instance Name.*
   - Set the *Multiplier Array* to be the "Mult2" data set.
   - Set the *Zone array* to be "RchZone1**.**"
   - Set the *IZ* value to "2."
   - Set the *Stress Periods* field to be "2."
6. Select **OK** to exit the *Define Clusters* dialog.
7. Select **OK** to exit the *Parameters* dialog.

12 Running MODFLOW

Now save the project and run MODFLOW.

1. Select the *File* | **Save As** command.
2. Change the project name to "run4.gpr."
3. Select the **Save** button.
4. Select the *MODFLOW* | **Run MODFLOW** command.

Once MODFLOW is done running, the user can read in the solution.

5. Make sure that the *Read solution on exit* toggle is checked.
6. Select the **Close** button.

13 LPF Parameters

GMS supports parameters with the LPF and HUF packages. In this tutorial, the user is using the LPF package, and the user will change horizontal conductivity to use parameters. When entering parameters for the LPF package, a layer number is used in place of an instance name.

1. Select the *MODFLOW* | **Parameters** command to open the *Parameters* dialog.
2. Make sure the *Show all columns* toggle at the top of the dialog is checked.
3. Select the **New** button near the bottom left of the dialog.
4. Change the values for the newly created parameter as follows:
    - *Name* of "HK_1"
    - *Type* of "HK"
    - *Value* of "0.67"
5. Turn on the *Use clusters* toggle for the parameter.
6. Enter "2" for the *Num. Clusters*.
7. Select the **Define Clusters** button to open the *Define Clusters* dialog.
8. Enter the values for the cluster shown in the table below.

Layer	Multiplier Array	Zone Array	IZ
1	Mult1	HKZone1	1 2
2	Mult1	HKZone1	1 2

9. Repeat steps 3–7, creating a new parameter with the following values:
    - *Name* of “HK_2”
    - *Type* of “HK”
    - *Value* of “0.15”
10. Enter the values for the cluster shown in the table below.

Layer	Multiplier Array	Zone Array	IZ
1	Mult1	HKZone2	3 4
2	Mult1	HKZone2	3 4

11. When finished, click **OK** to exit the *Define Clusters* dialog.
12. Click **OK** to exit the *Parameters* dialog.

14 Running MODFLOW

Now save the project and run MODFLOW.

1. Select the *File* | **Save As** command.
2. Change the project name to “run5.gpr.”
3. Click **Save.**
4. Select the *MODFLOW* | **Run MODFLOW** command.

Once MODFLOW is done running, the user can read in the solution.

5. Make sure that the *Read solution on exit* toggle is checked.
6. Select the **Close** button.

15 Parameter Factors

The last item that this tutorial will cover is parameter factors. The MODFLOW list based packages (WEL, DRN, GHB, RIV, CHD, STR) allow parameter factors to be defined along with the parameters. For example, in GMS, when creating a river boundary condition using the map module, GMS will store the length of the arc in the grid cell that is associated with each river boundary condition. Then if the user wishes to parameterize river conductance, the parameter can be conductance per unit length, and the parameter factor will be the length of the arc. The final conductance for a particular river boundary condition will be the product of the parameter value and the parameter factor.

In MODFLOW, this parameter factor can actually vary per stress period. This feature has been added to GMS in order to fully support MODFLOW parameters. The user will now create a parameter in the WEL package and then change the parameter factor on one of the stress periods.

1. Select the *MODFLOW* | **Parameters** command.
2. Select the **New** button near the bottom left of the dialog. This will create a new parameter.
3. Change the values for the newly created parameter as follows:
    - *Key* of "-15"
    - *Type* of "WELL"
    - *Value* of "-15.0"
4. Select **OK** to exit the dialog.

15.1 Editing Parameter Factors

By default, the parameter factors are 1.0 (which will have no effect on the final value that the parameter represents). The user will change the parameter factor for the second stress period for the parameter that was just created.

1. Select the *MODFLOW* | *Optional Packages* | **WEL - Well** command to open the *MODFLOW Well Package* dialog.
2. Change the *Stress period* to "2."
3. Turn off the *Use previous* toggle.

In the spreadsheet, notice the *Q factor* column. This is the parameter factor.

4. For the well in cell ID 1613, change the *Q factor* to "10.0."

This means that in the second stress period the pumping rate for this well will be 10.0 times greater or -150.0 instead of -15.0. Note that the Q factor is only used by parameters.

5. Select **OK** to exit the dialog.

16 Running MODFLOW

Now save the project and run MODFLOW.

1. Select the *File* | **Save As** command.

2. Change the project name to “run6.gpr.”

3. Click **Save.**

4. Select the *MODFLOW* | **Run MODFLOW** command.

Once MODFLOW is done running, the user can read in the solution.

5. Make sure that the *Read solution on exit* toggle is checked.

6. Select the **Close** button.

Notice the changes in the contours around the well on the left side of the model when switching to the second time step on the MODFLOW solution.

17 Conclusion

This concludes the “MODFLOW – Advanced Parameters Options” tutorial. Here are the key concepts in this tutorial:

- Array-based parameters can be defined using clusters.
- Parameter instances can be defined for array-based parameters for transient models.
- Parameter factors can be defined for list-based packages in GMS.