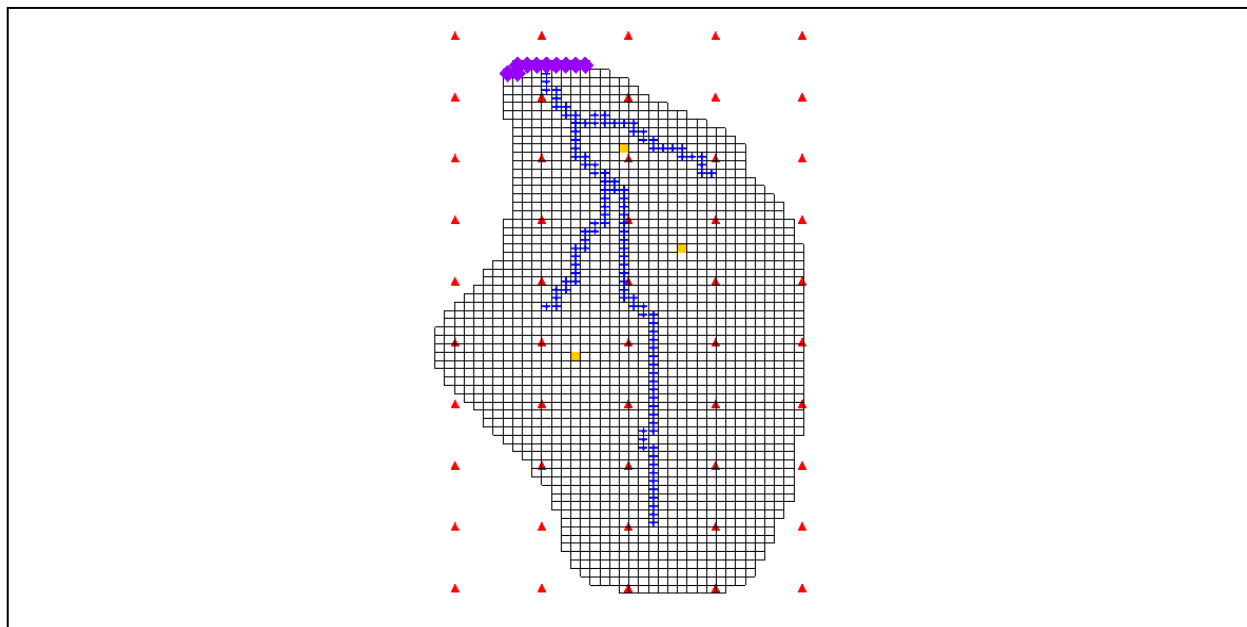


GMS 9.2 Tutorial

MODFLOW – PEST Pilot Points

Use pilot points with PEST to automatically calibrate a MODFLOW model



Objectives

Learn the features and options related to pilot points when used with PEST. Use fixed value pilot points, regularization, and multiple parameters.

Prerequisite Tutorials

- MODFLOW - Model Calibration
- MODFLOW - Automated Parameter Estimation

Required Components

- Grid
- Geostatistics
- Map
- MODFLOW
- Inverse Modeling

Time

- 30-60 minutes



1 Contents

1	Contents	2
2	Introduction.....	2
2.1	Outline.....	3
3	Description of Problem	3
4	Getting Started	4
5	Reading in the Project	4
6	Creating Pilot Points.....	4
7	Editing the HK parameter.....	6
7.1	Editing the Parameters.....	6
7.2	Limiting the Number of Parameter Estimation Runs.....	7
8	Saving the Project and Running PEST	7
9	Viewing the Solution	8
10	Viewing the Final Hydraulic Conductivity	8
11	Adding Fixed Value Pilot Points.....	9
12	Saving the Project and Running PEST	11
13	Changing the Regularization Option.....	11
14	Saving the Project	11
14.1	Prior Information equations.....	11
15	Running PEST.....	12
16	Multiple Parameters using Pilot Points.....	12
16.1	Creating a Second Set of Pilot Points.....	13
16.2	Creating the New HK Parameter	13
16.3	Using Pilot Points with RCH Parameter.....	14
17	Saving the Project and Running PEST	15
18	A Note on Highly Parameterized Models.....	15
19	Conclusion.....	15

2 Introduction

Pilot points can be thought of as a 2D scatter point set. Instead of creating a zone and having the inverse model estimate one value for the entire zone, the value of the parameter within the zone is interpolated from the pilot points. Then the inverse model estimates the values at the pilot points. Using pilot points, values will vary from cell to cell. When the inverse model runs, the values at the pilot points are adjusted and reinterpolated to the grid cells until the objective function is minimized.

PEST provides an option for the pilot point method called “regularization”. Regularization imposes an additional measure of “stiffness” to the parameter being interpolated. This “stiffness” is imposed by providing PEST with additional information about the parameter in the form of prior information equations. This constraint makes the inversion process much more stable and makes it possible to violate one of the typical constraints associated with parameter estimation: namely, the requirement that the number of parameters must be less than the number of observations. With regularization, the number of parameters can greatly exceed the number of observations. As a result, complex hydraulic conductivity distributions can be defined, resulting in extremely low residual error. The pilot point method with regularization is an incredibly powerful feature of PEST.

There are two methods available in GMS for defining the prior information equations for PEST. The two methods can be used simultaneously but usually they are used separately. The first method is **Preferred homogeneous regularization**. When this option is selected the prior information equations written to the PEST control file relate the pilot points to one another. These equations indicate to PEST that in the absence of any strong influence from the PEST objective function, pilot points that are near to one another should have about the same value.

The second method of regularization available in GMS is **Preferred value regularization**. When this option is selected prior information equations written to the PEST control file relate the pilot points to their starting value. These equations indicate to PEST that in the absence of any strong influence from the PEST objective function, the pilot point values should be equal to their starting value. Depending on the particular problem that you are trying to solve, one method may be preferable over the other.

2.1 Outline

This is what you will do:

1. Open a MODFLOW model and solution.
2. Create pilot points and run PEST.
3. Load optimal parameter values and view the resulting HK field.
4. Include pilot points with fixed values and run PEST.
5. Change regularization options and run PEST.
6. Use pilot points on different zones and different parameter types.

3 Description of Problem

The model we will be calibrating in this tutorial is the same model featured in the *MODFLOW-Model Calibration* tutorial. The model includes observed flow data for the stream and observed heads at a set of scattered observation wells. The conceptual model

for the site consists of a set of recharge and hydraulic conductivity zones. These zones will be marked as parameters and an inverse model will be used to find a set of recharge and hydraulic conductivity values that minimize the calibration error.

4 Getting Started

Let's get started.

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

5 Reading in the Project

First, we will read in the modeling project:

1. Select the *Open* button .
2. Locate and open the **Tutorials\MODFLOW\pilotpoints** directory.
3. Open the file entitled **start.gpr**.

You should see a MODFLOW model with a solution and a set of map coverages. Three of the coverages are the source/sink, recharge, and hydraulic conductivity coverages used to define the conceptual model. The active coverage contains a set of observed head values from observation wells. If you switch to the source/sink coverage, you will notice that an observed flow value has been assigned to the stream network.


This model has already been parameterized so we just need to create the pilot points and then run PEST.

6 Creating Pilot Points

We will create the pilot points that define the hydraulic conductivity distribution for our model. The pilot points are defined as 2D scatter point sets in GMS. It should be noted that there are many different ways in GMS to create 2D scatter points. Scatter points can be created by converting a 2D Grid, 2D Mesh, TIN, Borehole contacts, or Map data to a scatter point set. The easiest way to create scatter points is to create a new scatter point set and then click out the points in the graphics window.

In this example we will create a 2D grid and then convert it to scatter points. We want to create about 50 pilot points so we will make a cell-centered 2D Grid with 50 cells.



1. In the *Project Explorer* right-click on the empty space and then, from the pop-up menu, select the *New | Grid Frame* command.

2. In the *Project Explorer* right-click on the *Grid Frame*  and select the *Fit to Active Coverage* command.




Now that the coverages and the Grid Frame are created, we are now ready to create the grid.

1. Select the *Feature Objects | Map → 2D Grid* command.
2. Enter **5** for the *Number Cells in the X-Dimension*.
3. Enter **10** for the *Number Cells in the Y-Direction*.
4. Select the *OK* button to exit the *Grid Frame* dialog.

Now, we have a 2D grid with 50 cells. We can convert it to 2D Scatter Points.

5. Right-click on the 2D Grid  and select the *Convert to 2D Scatter Points* command.
6. Enter **HK** for the name of the new scatter set and select *OK*.
7. Right-click on the 2D Grid  and select the *Delete* command.

We want our default starting value of hydraulic conductivity to be 0.5 m/d so we'll change the data values at the scatter points to this value.

8. If necessary, expand the *2D Scatter Data* folder  in the *Project Explorer*.
9. Select the **HK** scatter point set  from the *Project Explorer*.
10. Unlock the scatter points by selecting the *Scatter Points | Lock All Scatter Points* menu command.
11. Select the *Select Scatter Points* tool .
12. Type CTRL+A to select all the points (or click the *Edit | Select All* menu command).
13. In the *Edit XYZF* toolbar, for the *F value* enter **0.5** and hit *Enter*.

Your graphics window should look similar to the following figure.

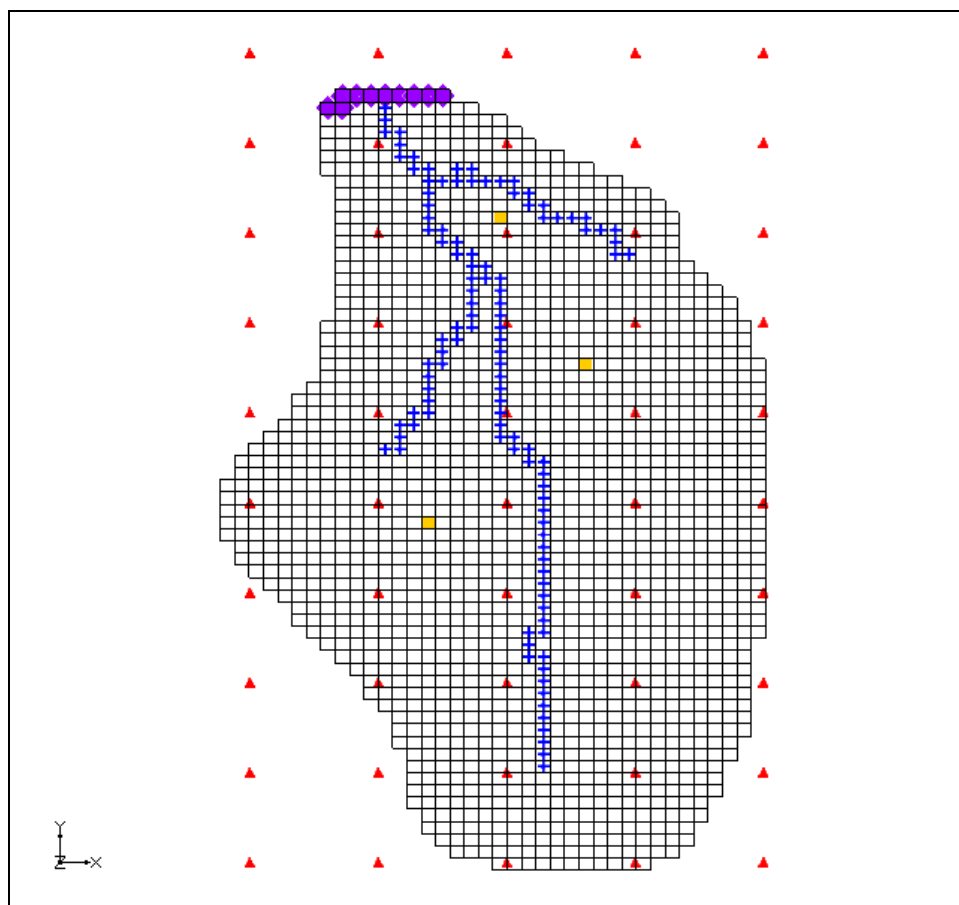



Figure 1. Placement of scatter points.

7 Editing the HK parameter


In the *MODFLOW – Automated Parameter Estimation* example we had four hydraulic conductivity parameters. For this problem we are estimating the hydraulic conductivity for the entire layer with the pilot points. Therefore, we only need one parameter for hydraulic conductivity. This parameter will then be linked to the pilot points using the *Parameters* dialog.

7.1 Editing the Parameters

Now we will edit the parameters that we currently have defined for MODFLOW.

1. Select the *MODFLOW | Parameters* command.
2. Turn on the *Pilot Points* option for parameter *HK_30* by selecting the drop down arrow  in the *Value* column. Then select **<Pilot points>** from the drop down list.

The interpolation options associated with the pilot points can be changed by clicking on the small button above the drop down arrow in the *Value* column.

3. Click on the button  above the drop down arrow in the *Value* column for parameter *HK_30*. This brings up the *2D Interpolation Options* dialog. Here you can select the scatter point set and data set used with your parameter as well as the interpolation scheme.
4. The defaults are appropriate in this case so we won't change anything. Select the *OK* button twice to exit the dialogs.

7.2 Limiting the Number of Parameter Estimation Runs

In the interest of time we will limit the number of iterations that PEST does for our problem

1. Select the *MODFLOW | Parameter Estimation* command.
2. Change the *NOPTMAX. Max number of iterations* to **5**.


Notice the *Tikhonov regularization* section of the dialog. In this section the user can specify the type of regularization to use with PEST. By default *Preferred homogeneous regularization* is turned on.

The *Prior information power factor* is used to change the weight applied to the prior information equations for the pilot points. When the prior information equations are created, GMS will compute an inverse distance weight between each pilot point and all other pilot points for a given parameter. This weight is then raised to the power of the *Prior information power factor* and assigned to the equation for a given pair of points. So to increase the homogeneity constraint (assign a higher weight to the prior information equation), the user should decrease this value. To decrease the homogeneity constraint (assign a lower weight to the prior information equation), the user should increase this value.

3. Select *OK* to exit the dialog.

8 Saving the Project and Running PEST

We are now ready to save and run PEST.

1. Select the *File | Save As* command.
2. Save the project with the name **mfpest_pilot.gpr**.
3. Select the  *Run MODFLOW* button.

PEST is now running. The error and parameter values are shown in the spreadsheet in the upper right side of the dialog and the plot on the left shows the error. In this case you

may notice some strange parameter names like sc1v1. These names were automatically generated and assigned to the pilot points.





PEST will take several minutes to run. You should see the residual error decrease. When PEST is finished, you will see a message in the text portion of the window and the *Abort* button will change to *Close*.

9 Viewing the Solution

Once PEST is finished, you can read in the solution.

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

The contours currently shown on the 3D grid are the heads from the MODFLOW run with the optimum parameter values. We will now look at the observation targets in the map model and the error associated with this model run.

2. In the *Project Explorer*, turn **off** the 3D grid in order to see the coverage data better.
3. Select the *Sources & Sinks* coverage  from the *Project Explorer*. Notice that the observation target on the arc group almost exactly matches.
4. Select the *Select Arc Group* tool .
5. Select the arc group by clicking on the river arc. Notice in the edit strip at the bottom of the graphics window the computed and observed flow is reported.
6. Select the *Observation Wells* coverage  from the *Project Explorer* to see how closely the computed heads match the field observations.
7. Right-click on the *mfpst_pilot (MODFLOW)* solution  in the *Project Explorer* and select the *Properties* command.

This command brings up a spreadsheet showing the error from this model run. The spreadsheet shows the error from the head observations, the flow observations, and the combined weighted observations. Note that these values are lower than the values we obtained using the zonal parameters approach.

8. When you finish viewing the properties select *Done*.



10 Viewing the Final Hydraulic Conductivity

When PEST ran a new conductivity value was estimated at each of the scatter points used with the **HK_30** parameter. Now we will read in the optimal parameter values as

determined by PEST. Reading in the optimal parameter values will create a new data set for our scatter point set. Then we can see our final hydraulic conductivity field.

1. In the *Project Explorer*, turn **on** the 3D grid.
2. Select the *MODFLOW | Parameters* command.
3. Click the *Import Optimal Values* button.
4. Select the **mfpest_pilot.par** file and select the *Open* button. Notice that the starting values for the parameters have changed.
5. View the pilot point options by selecting the button above the drop down arrow in the *Value* column for parameter *HK_30*.

Notice the **HK_30 (mfpest_pilot)** data set is listed as the one being used by the pilot points. This data set was imported when you imported the optimal values and represents the optimal values at each pilot point as determined by PEST.

6. Select *OK* to exit both dialogs.
7. Expand the *MODFLOW* item  in the *Project Explorer*. Under the *LPF* package  select the *HK* data set.



You should now see the final hydraulic conductivity values for your MODFLOW simulation.

11 Adding Fixed Value Pilot Points

Pilot points can be supplemented by field measured hydraulic conductivity (such as pump test data) values. We will now add additional pilot points to our model and fix the HK value at those points. First we will change the display so we can see our grid.

1. Select the  *Display Options* button.
2. Turn **off** the *Contours* option and select *OK*.

Notice that there are 3 pumping wells in our model. For purposes of illustration, we will assume that a pump test was performed at each of these wells and HK estimated. We will create a pilot point at each of these wells.

3. Select the *2D Scatter Data* folder  in the *Project Explorer*.
4. Select the *Create Scatter Points* tool .
5. Create a scatter point at each of the pumping wells as shown in the following figure.

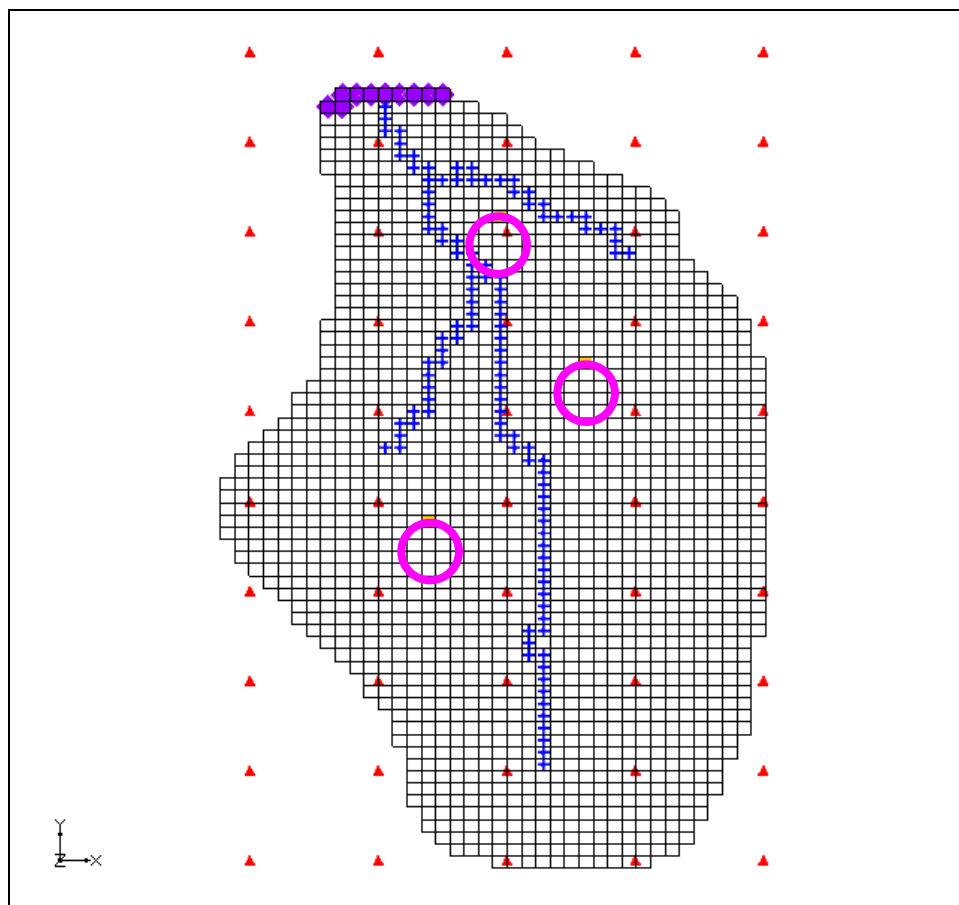




Figure 2. Location of pumping wells.

6. Unlock the scatter points by selecting the *Scatter Points | Lock All Scatter Points* menu command.
7. Select the *Select Scatter Points* tool .
8. Select the new scatter point for the well closest to the top of the model. You may want to turn off the 3D Grid in the *Project Explorer* to more easily see the new scatter points.
9. In the *F:* field near the top of the GMS window enter **1.0**.
10. Now, select the scatter point at the well on the east (right) side of the model and enter **0.1**.
11. Next, select the scatter point at the well on the west (left) side of the model and enter **5.0**.
12. Finally, select all three of the new scatter points (hold the shift key down to multiselect) and then select the *Edit | Properties* command.
13. Toggle on the *Fixed Pilot Point* field for each of the points and select *OK*.

12 Saving the Project and Running PEST

We are now ready to save and run PEST.

1. Select the *File | Save As* command.
2. Save the project with the name **mfpest_pilot_fixed.gpr**.
3. Select the  *Run MODFLOW* button.

PEST will take several minutes to run. When PEST is finished, you will see a message in the text portion of the window and the *Abort* button will change to *Close*.

Once PEST is finished, you can read in the solution.

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

You may wish to view the new HK field by importing the optimal values and examining the contours like we did previously.

13 Changing the Regularization Option

We will now change our regularization option to use the *Preferred value regularization* method.

1. Select the *MODFLOW | Parameter Estimation* command.
2. Turn **off** the *Preferred homogeneous regularization* toggle.
3. Turn **on** the *Preferred value regularization* toggle and select *OK*.

14 Saving the Project

We are now ready to save and run PEST.

1. Select the *File | Save As* command.
2. Save the project with the name **mfpest_pilot_pref_val.gpr**.

14.1 Prior Information equations

You can compare the different prior information equations written from GMS by looking at the *.rpf files in the MODFLOW directories. The prior information equations for *Preferred homogenous regularization*, found in mfpest_pilot.rpf should look like the following:

```
pi0 1.0 * log(sc1v1) - 1.0 * log(sc1v2) = 0.0 0.0018556829581 regul_1
```

```

pi1 1.0 * log(sc1v1) - 1.0 * log(sc1v3) = 0.0 0.0003314571401 regul_1
pi2 1.0 * log(sc1v1) - 1.0 * log(sc1v4) = 0.0 0.0000983152822 regul_1

```

Notice that these equations define a relationship between the different pilot points. In addition, the weight applied to these equations changes as shown by the last number written to each line. In contrast, the prior information equations for *Preferred value regularization*, found in `mfpilot_pref_val.rpf` shown below, have the same weight applied and define a preferred value for each point.

```

pi0 1.0 * log(sc1v1) = -0.877359449863 1.0 regul_1
pi1 1.0 * log(sc1v2) = -0.87738275528 1.0 regul_1
pi2 1.0 * log(sc1v3) = -0.764383137226 1.0 regul_1

```

15 Running PEST

We will now run PEST.

1. Select the  *Run MODFLOW* button.

PEST will take several minutes to run. When PEST is finished, you will see a message in the text portion of the window and the *Abort* button will change to *Close*.




Once PEST is finished, you can read in the solution.

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

You may wish to view the new HK field by importing the optimal values and examining the contours like we did previously.

16 Multiple Parameters using Pilot Points

In GMS, pilot points can be used with HK and RCH parameters. Also, you can have multiple HK (or RCH) parameters that use the same or different pilot points. We will now create a second HK parameter and create a new set of pilot points.

1. In the *Project Explorer*, turn **off** the 3D grid.
2. Select the *Hydraulic Conductivity* coverage  from the *Project Explorer*.
3. Select the *Select Polygons* tool  and double-click the polygon surrounding the river in the middle of the model.
4. Enter a value of **-60.0** for the *Horizontal K* and select *OK*.
5. Right-click on the *Hydraulic Conductivity* coverage  and select the *Map To MODFLOW/MODPATH* command.

16.1 Creating a Second Set of Pilot Points

Now we will create another scatter point set for the new HK parameter. We will again create a 2D grid and convert it to a scatter point set.

1. In the *Project Explorer* right-click on the empty space and then, from the pop-up menu, select the *New | 2D Grid* command.
2. Enter the values shown in the following figure to create the 2D Grid. Make sure to enter **0.5** for the Origin in the Z-Dimension section as this will be the value assigned to the scatter points created from the grid. Also, make sure that the *Grid type* near the bottom of the dialog is set to *Cell centered*.

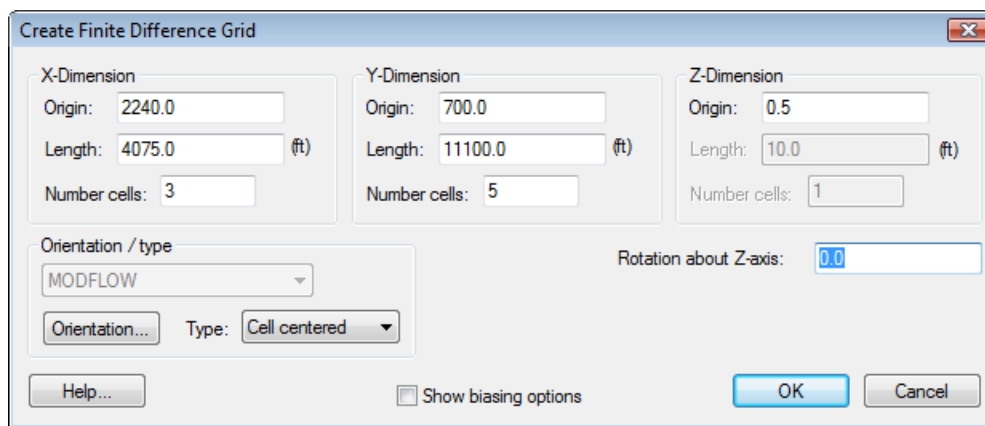




Figure 3. Create 2D Grid dialog.

3. Right-click on the 2D Grid and select the *Convert to 2D Scatter Points* command.
4. Enter **HK_60** for the name of the new scatter set.
5. Right-click on the 2D Grid and select the *Delete* command

16.2 Creating the New HK Parameter



Now we will create a new parameter for the pilot points that we created.

1. Select the *MODFLOW | Parameters* command.
2. Select the *Initialize From Model* button.
3. Turn on the toggle in the *Param. Est. Solve* column and the *Log Xform* column for the **HK_60** parameter.
4. Turn on the *Pilot Points* option for parameter **HK_60** by selecting the drop down arrow  in the *Value* column. Then select **<Pilot points>** from the drop down list.

5. Click on the button  above the drop down arrow in the *Value* column for parameter **HK_60**. Make sure that the scatter point set selected in the dialog is **HK_60** and select *OK* to exit the dialog.


16.3 Using Pilot Points with RCH Parameter

We will also use pilot points to estimate recharge with our model. For the RCH parameter we will use the same set of pilot points that the HK_30 parameter uses but we will create a new data set with starting values for the RCH parameter.

1. Right-click the *Recharge* coverage  from the *Project Explorer* and select the *Attribute Table* command.
2. Change the *Feature type* to **Polygons** at the top of the dialog.
3. In the *All* row change the value of *Recharge rate* to **-150.0** and select *OK*.
4. Right-click on the *Recharge* coverage  and select the *Map To MODFLOW/MODPATH* command.


Creating New Starting Values for the RCH Parameter


We will create a new data set on the HK scatter set to provide the starting values for the RCH parameter.

5. Select the **HK** scatter point set  from the *Project Explorer*.
6. Select the *Edit | Data Set Calculator* command.
7. Enter **1e-5** in the *Expression* field and enter **RCH** in the *Result* field.
8. Select the *Compute* button to create a new data set with all values equal to 1.0e-5.
9. Select *Done* to exit the dialog.

Editing the RCH Parameters

Now we will edit the RCH parameters to use pilot points.

1. Select the *MODFLOW | Parameters* command.
2. Delete the **RCH_180** and **RCH_210** parameters by selecting the row and then the *Delete* button.
3. Turn on the *Pilot Points* option for parameter **RCH_150** by selecting the drop down arrow  in the *Value* column. Then select **<Pilot points>** from the drop down list.

4. Click on the button  above the drop down arrow in the *Value* column for parameter **RCH_150**. Make sure that the scatter point set selected in the dialog is **HK** and that the data set is **RCH**.
5. Select *OK* twice to exit both dialogs.

17 Saving the Project and Running PEST

We are now ready to save and run PEST.

1. Select the *File | Save As* command.
2. Save the project with the name **mfpest_pilot_2zones.gpr**.

When pilot points are assigned to both HK and RCH parameters, the prior information equations for the HK and RCH parameters are assigned to different regularization groups. According to John Doherty, this helps PEST to differentiate weighting amongst pertinent prior information equations (read as: PEST works better with this option).

3. Select the  *Run MODFLOW* button.

PEST will take several minutes to run. When PEST is finished, you will see a message in the text portion of the window and the *Abort* button will change to *Close*. Once PEST is finished, you can read in the solution.

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


You may wish to view the new HK field by importing the optimal values and examining the contours like we did previously.

18 A Note on Highly Parameterized Models

The model that we have just created has over 100 parameters. This is a fairly simple MODFLOW model that converges rather quickly; most real world problems take longer to run. Thus, it may not be practical to run MODFLOW over 100 times for each PEST iteration. However, PEST supports a very innovative method known as SVD-Assist which can dramatically reduce the number of model runs required for each PEST iteration. When you combine SVD-Assist with Parallel PEST it becomes practical to use PEST with models containing hundreds or even thousands of parameters. You can learn more about these methods in the tutorial: *MODFLOW – Advanced PEST*.

19 Conclusion

This concludes the *MODFLOW – Pilot Points* tutorial. Here are the things that you should have learned in this tutorial:

- When using pilot points you may have more parameters than observations because regularization is included in the parameter estimation.
- When you bring up the *MODFLOW Parameters* dialog, you can change a parameter to use pilot points by using the  button.
- You use 2D scatter points to create pilot points in GMS.