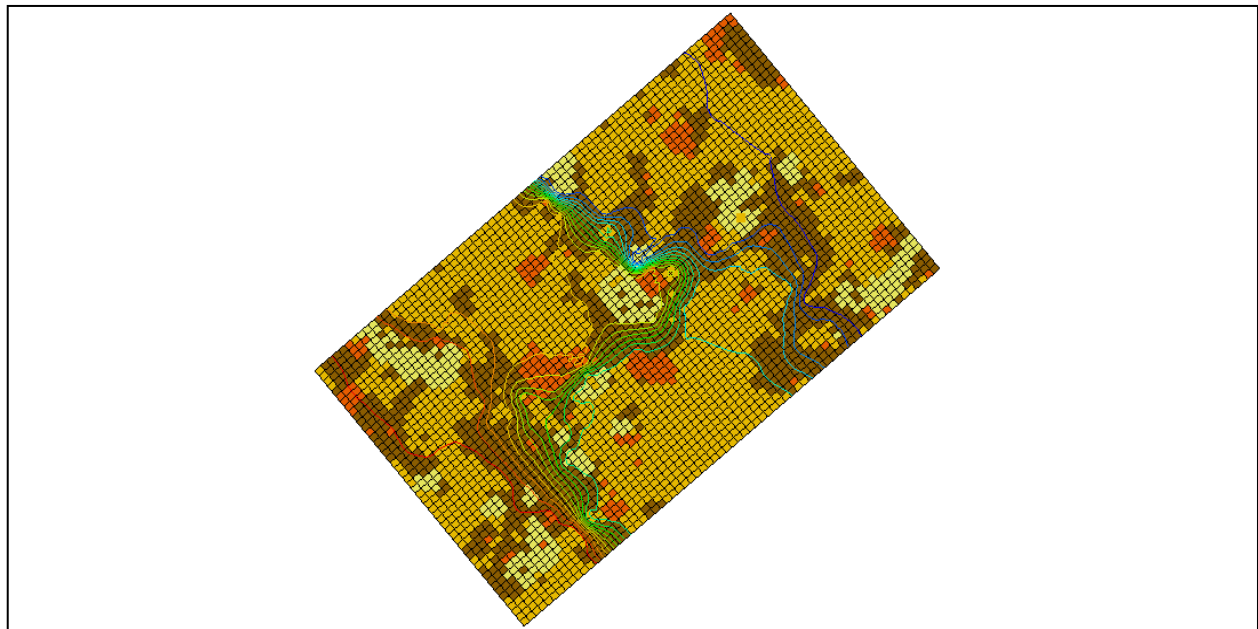


GMS 10.0 Tutorial

MODFLOW – Stochastic Modeling, Indicator Simulations

Use T-PROGS to create multiple material sets and run MODFLOW stochastically



Objectives

The indicator simulation approach with T-PROGS is described. With the indicator simulation approach, multiple equally probable “realizations” of the aquifer stratigraphy are generated. These realizations represent different distributions of material (indicator) zones within the aquifer. A set of aquifer properties is associated with the materials and the model is run once for each of the N realizations.

Prerequisite Tutorials

- MODFLOW – Stochastic Modeling, Parameter Randomization
- T-PROGS

Required Components

- Grid Module
- Map Module
- MODFLOW
- Stochastic Modeling

Time

- 35-50 minutes



1	Introduction	2
1.1	Outline.....	2
2	Description of Problem	2
3	Getting Started	3
4	Reading in the Project	4
5	The MODFLOW Model Data	4
6	Selecting the Stochastic Option	5
7	Saving the Project and Running MODFLOW	5
8	Reading in and Viewing the MODFLOW Solutions	6
9	Displaying Pathlines	6
10	Probabilistic Capture Zone Analysis	6
11	Conclusion	8

1 Introduction

GMS supports three methods for performing stochastic simulations: parameter randomization, indicator simulations, and PEST Null Space Monte Carlo. The indicator simulation approach is described in this tutorial. With the indicator simulation approach, multiple equally probable “realizations” of the aquifer stratigraphy are generated. These realizations represent different distributions of material (indicator) zones within the aquifer. A set of aquifer properties is associated with the materials, and the model is run once for each of the N realizations.

In GMS, the multiple realizations of the aquifer heterogeneity are typically generated using the T-PROGS software. T-PROGS can be used to generate two types of output: multiple material sets (arrays of material IDs), or multiple MODFLOW HUF input sets. In each case, GMS can launch MODFLOW in batch mode and generate a flow solution for each model instance. For this tutorial, the user will be using a pre-defined set of material sets generated by T-PROGS. The steps involved in running a T-PROGS simulation are described in the “T-PROGS” tutorial.

1.1 Outline

Here are the steps of the tutorial:

1. Open a MODFLOW model.
2. Edit parameters and run MODFLOW in stochastic mode.
3. Create pathlines.
4. Do a probabilistic capture zone analysis.

2 Description of Problem

The model for this tutorial is based on the Longhorn Army Ammunition Production (LHAAP) site in Texas used in the “T-PROGS” tutorial. While the user will be using the same site boundaries, the grid used in this simulation will have only one layer to

facilitate a shorter run time. The user will use a pre-defined T-PROGS simulation containing 30 material sets. The material sets will be used with the stresses and boundary conditions depicted in Figure 1. There is a small drinking water well on the right side of the model. This well averages $25 \text{ m}^3/\text{d}$. The regional ground water flow is from left to right. The user will use specified head boundaries at the left and right edges of the model at 180 m and 170 m, respectively.

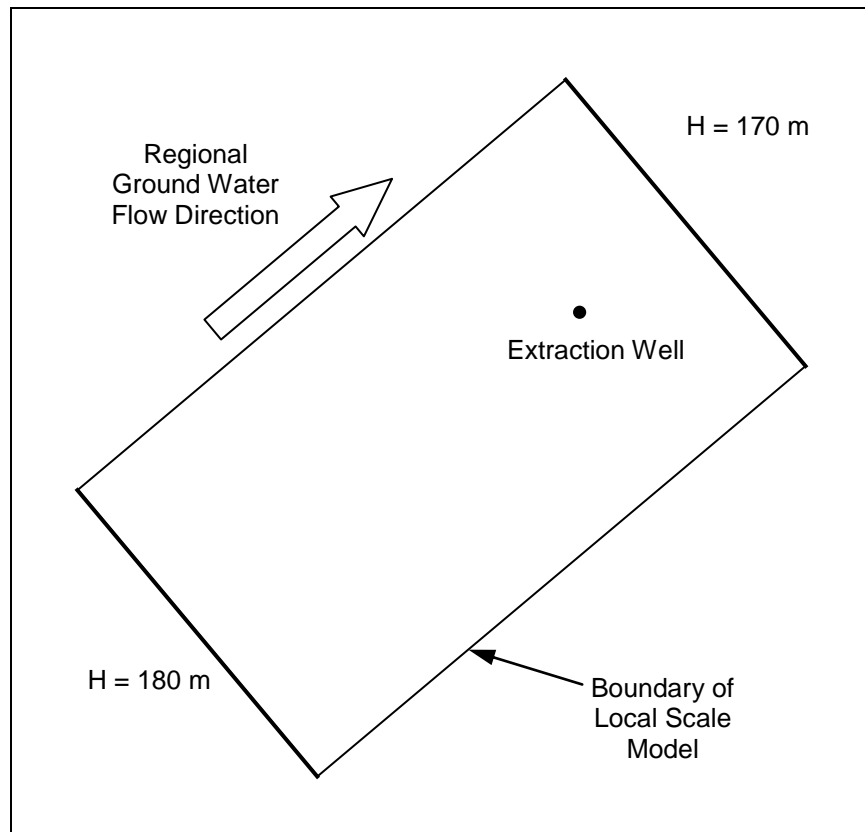


Figure 1 Conceptual representation of site to be modeled

Once the model is set up, the user will run MODFLOW in stochastic mode to generate 30 solutions. The user will then run the *Risk Analysis Wizard* to generate a data set representing probability of capture by the extraction well. The user will contour this dataset to illustrate capture zone boundaries for different probabilities of capture.


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 a project containing the MODFLOW model and the material sets generated by T-PROGS:

1. Select the **Open**  button.
2. Locate and open the *Tutorials\MODFLOW\sto_matset* directory.
3. Select the file entitled “lhaap.gpr.”
4. Click **Open**.

The user should see a one layer MODFLOW model rotated at a 40 degree angle showing a four-material distribution. The user will now view the different material sets generated by T-PROGS:

5. Expand the following items in the Project Explorer:
 - “3D Grid Data”  folder
 - ”grid (1)”  item
 - “Material Sets”  folder
 - “Simulation”  folder.
6. Click on any of the material sets in the “Simulation” folder. The user may wish to use the up and down arrow keys on the keyboard to cycle through the material sets.

5 The MODFLOW Model Data

Most of the MODFLOW data for this model (boundary conditions, well pumping rate, top and bottom elevations, etc.) has already been entered. However, the user will review the MODFLOW data that are somewhat more unique to this type of simulation.

1. Select the *MODFLOW | LPF - Layer Property Flow* command to open the *LPF Package* dialog.

At the top of the dialog, notice that the *Use material IDs* option is selected for the *Layer property entry method*. This means that the user will not enter an array of K (hydraulic conductivity) values as is normally the case with MODFLOW. Rather, the user will use material IDs to define the K values.

2. Select the **Material IDs** button to open the *Materials* dialog.

This dialog illustrates the material IDs assigned to cells. These material IDs are inherited from the active material set generated by T-PROGS.

3. Select the **OK** button to exit the *Materials* dialog.
4. Select the *Material Properties* button to open the *Materials* dialog.

This dialog is used to assign aquifer properties, including hydraulic conductivity, to each of the materials used by the model. Notice that a value has been assigned for Horizontal k and Vertical k for each material. Since this is a one layer model only the hydraulic conductivity value will be used. When the MODFLOW model is saved to disk, GMS uses the array of material IDs and the list of material properties to automatically generate the array of K values required by MODFLOW.

5. Select **OK** to exit the *Materials* dialog.
6. Select **OK** to exit the *LPF Package* dialog.

6 Selecting the Stochastic Option

Before running MODFLOW, it is necessary to turn on the appropriate stochastic simulation options. First, select the stochastic run option:

1. Select the *MODFLOW* | **Global Options** command to open the *MODFLOW Global/Basic Package* dialog.
2. In the *Run Options* section of the dialog, select the *Stochastic* option.
3. Choose **OK** to exit the dialog.

Next, the user needs to specify the use of the material set method (as opposed to parameter randomization) in this stochastic simulation. When the user chooses the material set option, the user must also specify which group (folder) of material sets he or she wishes to use. In this case, the user has only one group called “Simulation.”

4. Select the *MODFLOW* | **Stochastic** command to open the *Stochastic Options* dialog.
5. Select the *Material sets* method.
6. Verify that “Simulation” shows up in the combo box below the *Material sets* option.
7. Choose **OK** to exit the dialog.

7 Saving the Project and Running MODFLOW

Now save the project and run MODFLOW in stochastic mode.

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

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

MODFLOW is now running in stochastic mode. As each model run finishes, the spreadsheet at the top will indicate whether or not the run converged.

8 Reading in and Viewing the MODFLOW Solutions

Once all the MODFLOW runs are completed, the user can read in the solutions.

1. Make sure the *Read solution on exit* toggle is checked.
2. Select the **Close** button.
3. Select **OK** at the prompt to read in all converged solutions.

The user should see a new folder named “matsto (MODFLOW)(STO)” in the Project Explorer. The user may wish to expand this folder and view the individual solutions. The user will notice that the contours vary greatly depending on the distribution of materials. The head loss occurs primarily in the clay zones. As the user views a particular solution, notice that the material set is updated to correspond to the material set used to generate that particular solution.

9 Displaying Pathlines

Before performing the probabilistic capture zone analysis, the user will first view the capture zones corresponding to individual solutions.


1. Select the *MODPATH* / **Create Particles at Wells** command.
2. Select the **OK** button to accept the default options.

The user should see a set of pathlines appear. As the user clicks on different MODFLOW solutions in the Project Explorer, the pathlines will be automatically updated. Notice how dramatically the capture zone changes from one solution to the next.

10 Probabilistic Capture Zone Analysis

Now that the user has a MODFLOW stochastic solution set, it is possible to perform a probabilistic capture zone analysis for the wells in the model. A probabilistic capture zone analysis is performed by placing one or more particles in each cell and tracking the particles forward in time using MODPATH to see if they reach a well. If any of the particles from a cell reach the well, a counter for the cell is updated. After running


MODPATH on each of the MODFLOW solutions, the percentage of particles from a particular cell that are eventually captured by the well is computed and saved as the capture probability for the cell. The capture probability data set can then be contoured.

1. Right-click on “matsto (MODFLOW)(STO)”  folder in the Project Explorer.
2. Select the **Risk Analysis** command from the pop-up menu. This brings up the *Risk Analysis Wizard*.
3. Verify that *MODFLOW* is selected in the list box.
4. Select the *Probabilistic capture zone analysis* option.
5. Select the **Next** button.

The user should now see the next page in the *Risk Analysis Wizard*. This page can be used to specify options for the position and number of particles for each cell. In the *Particle starting locations* area, the user can choose to place particles on the water table or at the cell centers. The user will use the default and place particles at the water table. The user can also specify the tracking duration, although for this tutorial, the user will be tracking the particles until they terminate (“To end”).

6. Select the **Finish** button.

At this point, the wizard should go away and a progress bar should appear at the bottom. When the computations are finished, a new dataset, “well,” will be added to the “matsto (MODFLOW)(STO)” folder. This dataset contains the probability that any particles placed at the water table will reach the well. The best way to view this dataset is to turn on color filled contours.

7. Select **Contour Options**  from the main toolbar to open the *Dataset Contour Options – 3D Grid – well* dialog.
8. Verify that the *Legend* option is turned on.
9. Change the *Contour Method* to “Color Fill.”
10. Select the **Color Ramp** button to open the *Color Options* dialog.
11. Change the *Palette method* to “Hue ramp.”
12. Click the **Reverse** button to put red at the max and blue at the min.
13. Click **OK** to exit the *Color Options* dialog.
14. Click **OK** to exit the *Dataset Contour Options – 3D Grid – well* dialog.

The user should now see a zone of probability extending from the well. The user may want to uncheck the “Particle Sets” folder in the Project Explorer to make it easier to see the contours.

11 Conclusion

This concludes the “MODFLOW – Stochastic Modeling, Indicator Simulations” tutorial. Here are the key concepts in this tutorial:

- GMS supports two types of stochastic approaches: parameter randomization and indicator simulations
- The *Risk Analysis Wizard* can be used to do a probabilistic capture zone analysis.