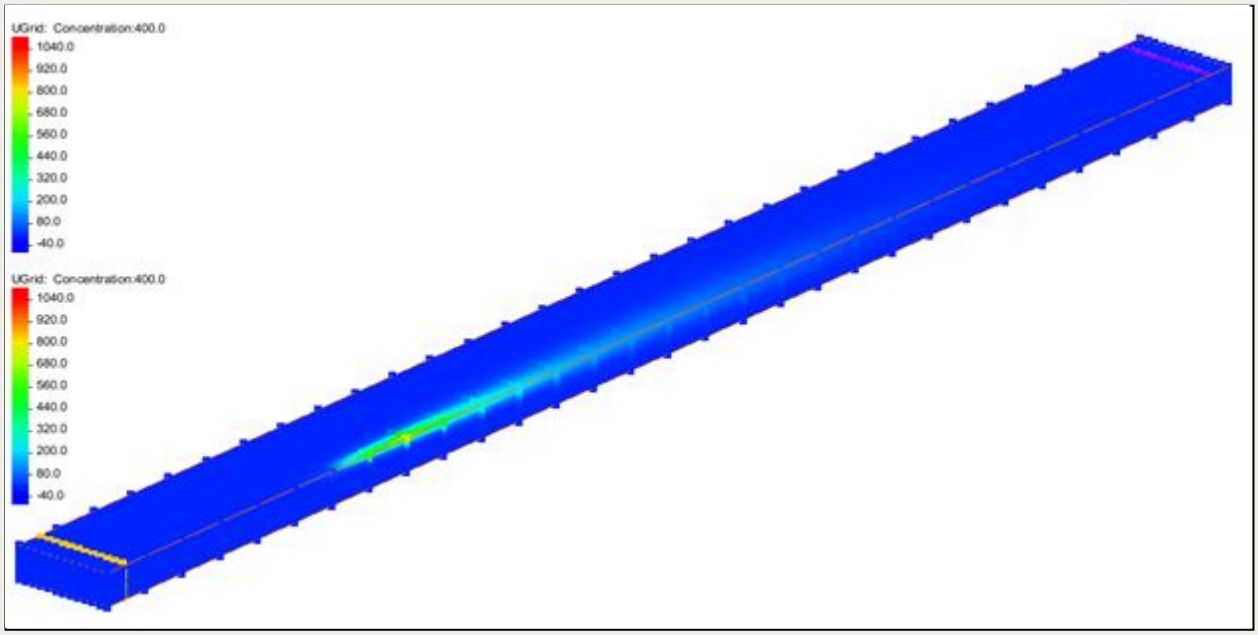
GMS 10.7 Tutorial

MODFLOW 6 – Transport Uncoupled

Run a transport simulation using the solution from an uncoupled flow simulation linked via the FMI package



Objectives

Learn how to import a flow and transport simulation and link them together using the FMI package.

Prerequisite Tutorials

- MODFLOW 6 – Grid Approach

Required Components

- GMS Core
- MODFLOW-USG Model & Interface

Time

- 10–20 minutes

AQUAVEO™

1 Introduction

GMS can allow for models to be uncoupled, with the necessary data re-linked back into models at a later point. This workflow is described in this tutorial. The FMI package is used to link separate flow data from one MODFLOW 6 simulation back to the transport model so it can be utilized in the transport simulation.

For some groundwater projects it is useful to uncouple models from each other and link the data between models later on. This tutorial will demonstrate uncoupling models using MODFLOW 6 simulations. The tutorial makes use of a simple UGrid for demonstration purposes.

This tutorial workflow will involve importing existing flow and transport simulations and running the simulations to get results. The Groundwater Transport (GWT) Model simulates three-dimensional transport of a single solute species in flowing groundwater.

The flow and transport model can be uncoupled. The FMI package is used to link the flow data to the transport model. The mass loading is specified in the mass source loading (SRC) package. Positive value means addition and negative value means removal from the model.

1.1 Getting Started

Do the following to get started:

1. If necessary, launch GMS.
2. If GMS is already running, select *File* | **New** to ensure that the program settings are restored to their default state.
3. If asked to save changes, click **Don't Save** to close the dialog and restore GMS to a default state.

2 Importing the Models

First, import the models:

1. Outside of GMS, browse to the *mf6_transport_uncoupled\ex-gwt-moc3d-p02\mf6gwf* directory and select "mfsim.nam".

2. Drag the “mfsim.nam” file into GMS. This will import the flow model into GMS. A dialog should briefly appear and disappear as it is importing.

Once it is imported, the “ flow” MODFLOW simulation should be visible in the Project Explorer. Now, import the transport model. Note the subtle change in directory name:

3. Outside of GMS, browse to the *mf6_transport_uncoupled\ex-gwt-moc3d-p02\mf6gwt* directory and select “mfsim.nam”.
4. Drag the “mfsim.nam” file into GMS. This will import the transport model into GMS. A dialog should briefly appear and disappear as it is importing.

Once it is imported, the “ trans” MODFLOW simulation should be visible in the Project Explorer. The starting project should look like Figure 1.

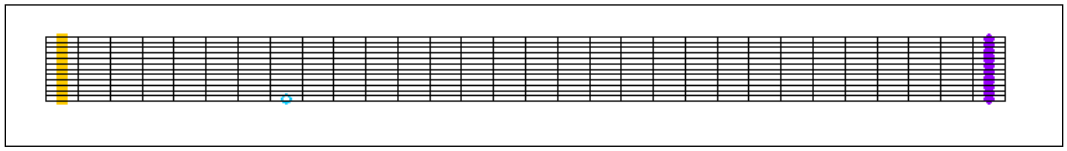


Figure 1 Initial project

3 Saving the Project

At this point, make sure that the project is saved.

1. Select *File* | **Save As…** to bring up the *Save As* dialog.
2. Select “Project Files (*.gpr)” from the *Save as type* drop-down.
3. Enter “mf6-uncoupled.gpr” as the *File name*.
4. Click **Save** to close the *Save As* dialog.

Save the project regularly so that progress isn’t lost if something goes wrong.

4 Delete the Second UGrid

A UGrid was created for each imported simulation, so there are now two, but they are identical. It is possible to continue with two UGrids, but for the sake of simplicity, one UGrid will be reused for both simulations.

1. Right-click on “ UGrid (2)” in the Project Explorer and select **Delete** to remove the unnecessary UGrid.
2. In the Project Explorer, expand “ trans” and “ trans”.
3. Drag “ UGrid” in the Project Explorer to be under “ trans” (the model, not the simulation) to link it to the trans model.
4. Save the project.

5 Run the Flow Model

Now that the redundant UGrid has been removed, the flow model should be run to generate the needed data.

The flow model has the injection well on the left side and the constant head boundary conditions on the right side. This means the flow is going from left to right.

5.1 Saving the Simulation

Before the flow simulation is run, it should be saved:

1. Right-click on the "flow" MODFLOW simulation and select **Save Simulation**. A dialog should briefly appear and disappear as it is saving.

5.2 Checking the Simulation

The flow simulation should also be checked before it is run in case there are any unresolved errors. To check for errors, do the following:

1. Right-click on the "flow" MODFLOW simulation and select the **Check Simulation…** command. This will bring up the *Check MODFLOW 6 Simulation* dialog.
2. There should be no errors. If this is the case, click **OK** to close the *Check MODFLOW 6 Simulation* dialog. If there are errors, ensure that they are resolved before moving forward with the tutorial.

5.3 Running the Simulation

With the flow simulation having been saved and checked for errors, it is now time to run through it.

1. Right-click on the "flow" MODFLOW simulation and select **Run Simulation**. This will bring up the *Simulation Run Queue* dialog.
2. When the simulation is completed, click on the **Load Solution** button to load the solution into GMS.
3. Click on the **Close** button to close the *Simulation Run Queue* dialog.
4. Select *Display* | **Display Options…** to bring up the *Display Options* dialog.
5. Select *UGrid: UGrid – [Active]* from the list on the left.
6. Turn on *Face contours* to see the solutions.
7. Turn off *Cell edges* and turn on *UGrid boundary*.
8. Click **OK** to close the *Display Options* dialog.

The Graphics Window should appear similar to Figure 2.

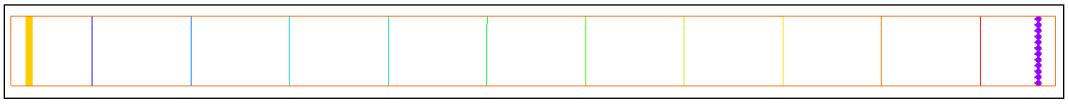


Figure 2 Project after the flow model has been run

6 Run the Transport Model

With the data generated for the flow model, the next step is to prepare to run the transport model. The data from the flow model run will be linked to the transport model to inform it.

6.1 Examine the Mass Source Loading Data

First to examine the mass source loading data:

1. If necessary, in the Project Explorer, expand out the "trans" MODFLOW simulation.
2. Double-click on "src" to open the *Mass Source Loading (SRC) Package* dialog.

Notice the spreadsheet in the PERIODS section. In the SMASSRATE column, there should be a value of "2.5" followed by zeros. This is the mass source loading rate, which is in a unit of mass/time. A positive value indicates addition of solute mass, and a negative value indicates removal of solute mass. In this example, an addition of mass is introduced to the model at row 12, column 8, layer 1, with a rate of 2.5.

3. When done observing, click **OK** to close the *Mass Source Loading (SRC) Package* dialog.

6.2 Specify the Flow Model Data

Next, the FMI package will be used to inform the transport simulation of the flow solution.

1. Double-click on "fmi" to open the *Flow Model Interface (FMI) Package* dialog.
2. Turn on *Let GMS determine the paths from the flow simulation when saving*.
3. Click the **Select Flow Simulation…** button to open the *Select Flow Simulation* dialog.
4. Click the checkbox next to the "flow" MODFLOW simulation.
5. Click **OK** to close the *Select Flow Simulation* dialog.
6. Click **OK** to close the *Flow Model Interface (FMI) Package* dialog.

6.3 Saving and Running the Simulation

With the flow model data specified for the transport model, the simulation can now be saved and run:

1. Right-click on the "trans" MODFLOW simulation and select the **Save Project, Simulation and Run** command to open the *Simulation Run Queue* dialog.
2. When the simulation is completed within the dialog, click on the **Load Solution** button to load the simulation solution data into GMS.
3. Once the dialog is empty of data after loading in the solution data, click on the **Close** button to close the *Simulation Run Queue* dialog.

6.4 Viewing the Results

With the solution data now generated, examine the results:

1. In the Project Explorer, under "Solution", expand "trans" and select the "Concentration" dataset.

The results should now be displayed in the Graphics Window.

2. Scroll through different time steps to observe changes in the display.

As time goes on, the concentration should be seen spreading out from the mass loading source. The Graphics Window on the last step should appear similar to Figure 3.

Figure 3 Concentration dataset on final time step

7 Conclusion

This concludes the "MODFLOW 6 – Transport Uncoupled" tutorial. The following key topics were discussed and demonstrated:

- Use the GWT model to simulate three-dimensional transport of a single solute species in flowing groundwater.
- Uncoupling a flow and transport model.
- Using the FMI package to link the flow data to the transport model.
- GMS can automatically determine inputs for the FMI package when the flow model is selected.
- Specifying mass loading in the SRC package. Positive value means addition and negative value means removal from the model.