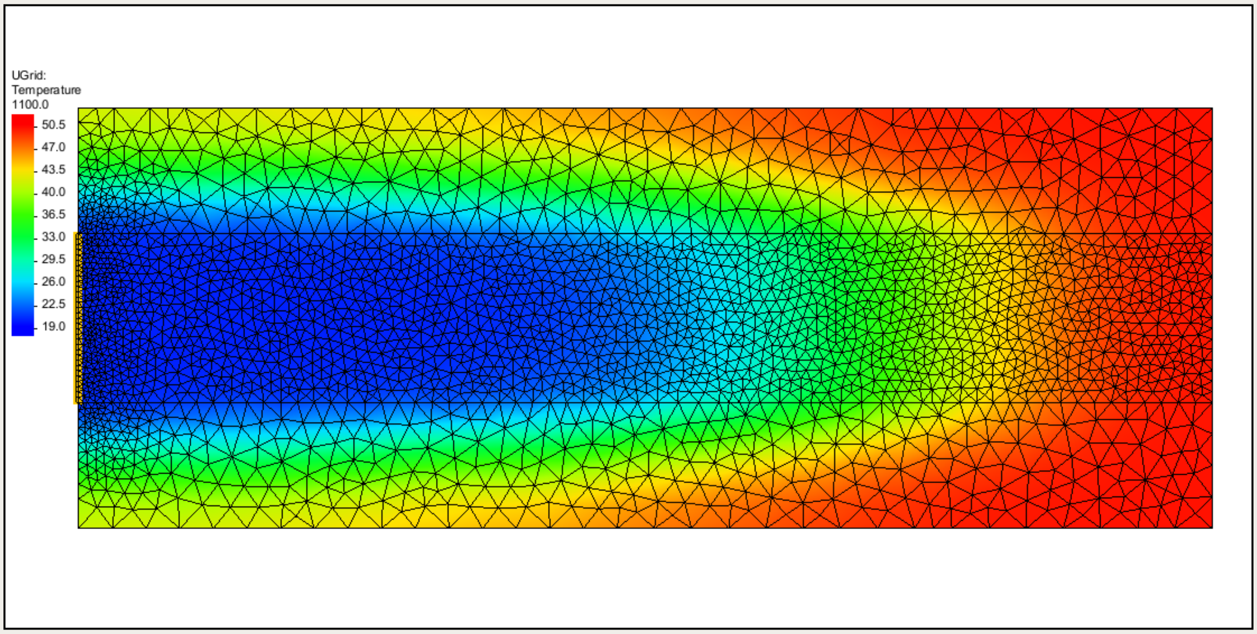
GMS 10.9 Tutorial

MODFLOW 6 – Groundwater Energy

Build a MODFLOW 6 model using the Groundwater Energy (GWE) model



Objectives

This tutorial demonstrates adding evapotranspiration data to a MODFLOW 6 simulation.

Prerequisite Tutorials

- Getting Started
- MODLFOW 6 Grid Approach

Required Components

- GMS Core
- MODFLOW-USG Model & Interface

Time

- 25–45 minutes

1 Introduction

This tutorial illustrates how to add a Groundwater Energy (GWE) model to an existing MODFLOW 6 simulation. The tutorial is based on the ATES example that is included with MODFLOW 6.

"Aquifer thermal energy storage (ATES) systems use groundwater wells to store and extract energy from aquifers for reducing energy costs associated with heating, ventilating, and air conditioning (HVAC) systems. For example, during cold winter months, an ATES system pumps air-chilled water into an aquifer for later recovery (extraction) during hot summer months, thereby reducing energy costs associated with cooling. Furthermore, when the water is pulled out for cooling purposes during hot summer months and is warmed as a result, it can subsequently be stored in a different part of the aquifer and used for heating purposes during cold winter months.1"

2 Getting Started

Do as follows 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.

2.1 Opening the Existing Model

Start with a previously created project.

---

[1] MODFLOW 6 Development Team, with contributions from Chieh Ying Chen and Mike Toews, 2017, MODFLOW 6 – Example problems

1. Click **Open** to bring up the *Open* dialog.
2. Select “Project Files (*.gpr)” from the *Files of type* drop-down.
3. Browse to the *mf6_gwe\start* folder and select “ates-gwf.gpr”.
4. Click **Open** to import the project and exit the *Open* dialog.

The project should be visible in the Graphics Window (Figure 1). The project contains a MODFLOW 6 simulation along with a 3D UGrid. wells, drains, recharge, and constant head boundary conditions have already been defined. A solution also exists.

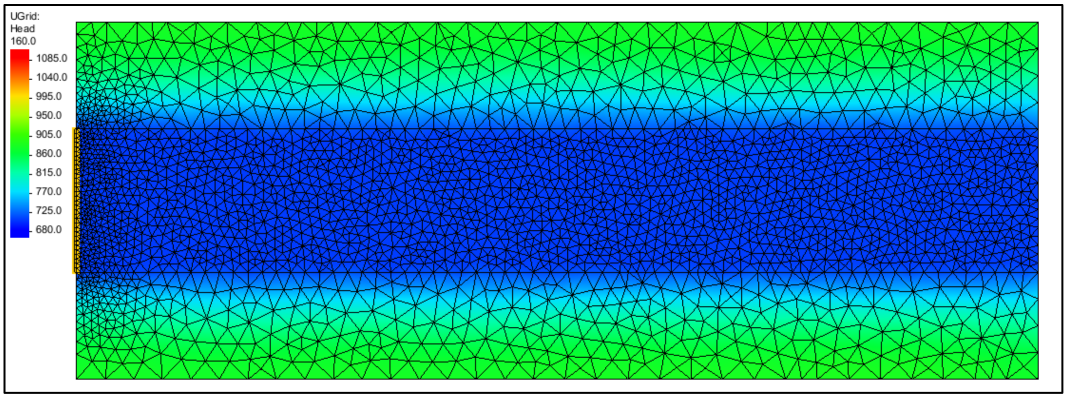


Figure 1 Initial project

The model uses a DISV grid that has one layer, but it is actually modeling a three-layer aquifer system by having the model Y dimension represent the real world Z dimension (Figure 2). “Water is extracted and re-injected into the left side of the middle layer” using the WEL package. The middle layer has a much higher hydraulic conductivity than the upper and lower layers.

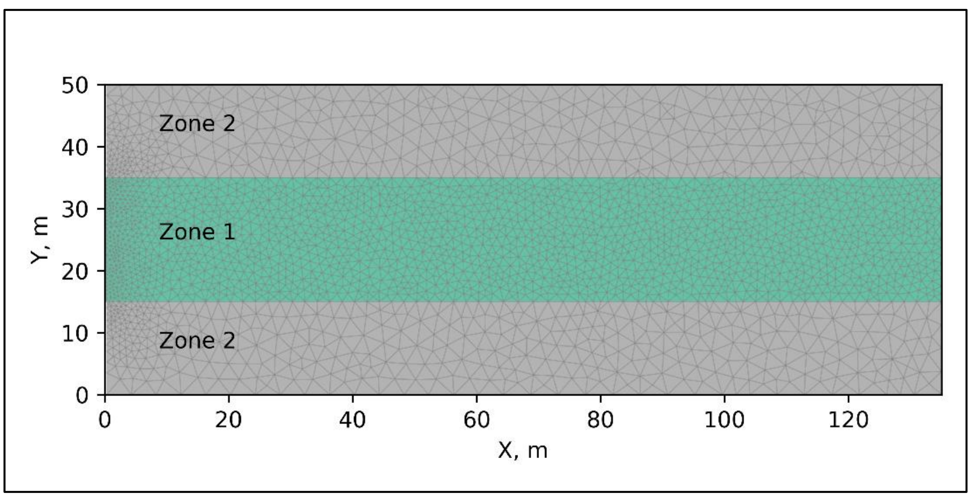


Figure 2 Grid zones

The graph in Figure 3 shows “the extraction and injection pumping intervals”.

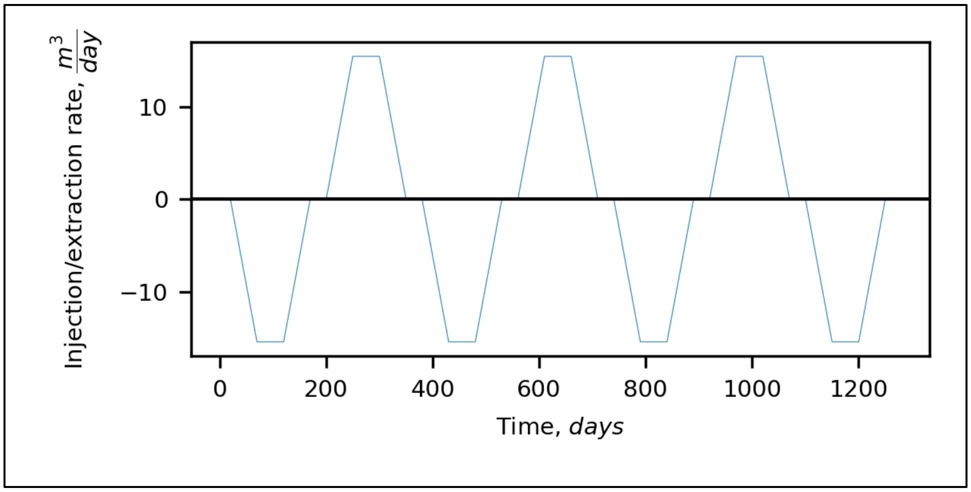


Figure 3 Extraction and injection pumping intervals

3 Saving the Project

Before making any changes, save the project under a new name.

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 "ates.gpr" for the *File name*.
4. Click **Save** to save the project under the new name and close the *Save As* dialog.

It is recommended to periodically **Save** while working through the tutorial and while working on any project.

4 Examine the Flow Solution

Before proceeding, examine the current solution.To do this:

1. In the Project Explorer, below " UGrid", select the " Head" dataset.
2. In the Time Steps Window, use the up/down arrow keys to step through solution time steps.

The contours show how the head changes in accordance with the injection/extraction periods. The greatest change is in the middle layer, where K is high.

5 Adding a GWE Model

The next step is to add a GWE model to the existing simulation. This means the flow and energy models will be "coupled".

1. Right-click the " ates" simulation and select *New Package* | **GWE** to bring up the *New Groundwater Energy Transport (GWE) Model* dialog
2. In the dialog, change the *Model name* to "gwe-ates".
3. Under the GWE – Groundwater Energy Transport Model section, turn on:

- ADV
- CND
- EST
- SSM

4. Click **OK** to close the New Groundwater Energy Transport (GWE) Model dialog.

The new " gwe-ates" model will appear in the Project Explorer.

6 Add a GWF-GWE Exchange

Because the flow and energy models are coupled in the same simulation, a GWF-GWE exchange is needed. Add this by completing the following:

1. Right-click the " ates" simulation and select *New Package* | **GWF-GWE**.

The exchange, " GWF-GWE", will appear in the Project Explorer.

7 Add an IMS Package

The GWE model requires its own IMS package. To create this, complete the following:

1. Right-click the " ates" simulation and select *New Package* | **IMS**
2. Right-click on the new " IMS" package and select **Rename**.
3. Enter "IMS-gwe-ates" as the new name.
4. Double-click on the " IMS-gwe-ates" package to open the *Iterative Model Simulation (IMS)* dialog.

The IMS package for the GWE model requires a lot of changes from the default values. In the interest of time, an existing package will be imported.

5. Click the **Import…** button to open an *Import* dialog.
6. Browse to the *mf6_gwe_ates* folder and select the "gwe-ates.ims" file.
7. Click **Open** to close the *Import* dialog.
8. Click **OK** at the message that the file has been imported successfully.
9. Click **OK** to close the Iterative Model Simulation (IMS) dialog.

8 Set Simulation EXCHANGES and SOLUTIONGROUPST

The simulation must be updated to handle the new exchange and IMS package.

1. Double-click the " ates" simulation to open the *Simulation Options* dialog.
2. Under *Sections*, turn on the *EXCHANGES* and *SOLUTIONGROUP* sections.
3. In the *EXCHANGES* section, under *EXGMNAMEA* click the **(none selected)** field to open the *Select Model* dialog.
4. Select the "gwf-ates" model and click **OK** to close the *Select Model* dialog.

5. Under *EXGMNAMEB* click the **(none selected)** field to open the *Select Model* dialog.
6. Select the “gwe-ates” model and click **OK** to close the *Select Model* dialog.
7. In the *SOLUTIONGROUPS* section, under *SLNMNAMES* click the **(none selected)** field in the second row to bring up the *Select Model(s)* dialog.
8. Select “gwe-ates” and click **OK** to close the *Select Model(s)* dialog.
9. Click **OK** to close the *Simulations Options* dialog.

9 GWE Parameters

The default settings for the IMS package are now inadequate. Change these settings by doing the following:

9.1 Set IC Parameters

The initial temperature needs to be set in the IC package for the “gwe-ates” model. To do this:

1. Double-click on the “IC” package to open the *Initial Conditions (IC) Package* dialog.
2. Under the *GRIDDATA* section, set *Constant* to “50” for the initial temperature.
3. Click **OK** to close the Initial Conditions (IC) Package dialog.

9.2 Set ADV Parameters

The scheme used to solve advection needs to be updated by doing the following:

1. Double-click on the “ADV” package to open the *Advection (ADV) Package* dialog.
2. Under the *OPTIONS* section, set *SCHEME* to “TVD”.
3. Click **OK** to close the *Advection (ADV) Package* dialog.

9.3 Set up Conceptual Model

Some parameters in the EST and CND packages require different values for the middle zone than the upper and lower zones. The simplest way to do this is with a conceptual model. All that is needed is a coverage with different polygons for the different zones. This is simple to create, but in the interest of time, one has already been created.

1. In the Project Explorer, expand “Map Data” and turn on the “zones” coverage.

9.4 Add Dataset Properties

The conceptual model will be used to create datasets on the UGrid, that will then be used in the GWE model.

1. Double-click the " zones" coverage to open the *Coverage Setup* dialog
2. At the bottom of the *Areal Properties* section, turn on the *Datasets* option.
3. Click the **Datasets…** button to open the *Datasets* dialog.
4. Click the **Insert Row** button three times to add three rows.
5. For the three rows, enter the following names:
   - "porosity"
   - "heat-capacity"
   - "kts"
6. Click **OK** to exit the *Datasets* dialog.
7. Click **OK** to exit the *Coverage Setup* dialog.

9.5 Set Dataset Values for the Zones

The polygons can now accept dataset values. Add these values

1. Using the **Select Polygon** tool and holding the *Shift* key, select both the upper and lower zone polygons.
2. Right-click and select **Attribute Table…** to open the *Attribute Table* dialog.
3. In the *All* row, set the following:
   - *porosity* to "0.1",
   - *heat-capacity* to "850.0",
   - *kts* to "120960.0".
4. Click **OK** to close the *Attribute Table* dialog.
5. Double-click the middle zone polygon to open the *Attribute Table* dialog.
6. In the *All* row, set the following:
   - *porosity* to "0.01",
   - *heat-capacity* to "1100.0",
   - *kts* to "259200.0".
7. Click **OK** to close the *Attribute Table* dialog.
8. Right-click the " zones" coverage and select *Map To* | **UGrid Datasets**.

Note the new datasets below the " UGrid" named " porosity", " heat-capacity", and " kts".

9. Turn off the " zones" coverage.

10 Set EST Parameters

Next to set the energy storage and transfer parameters for the simulation.

1. Double-click on the " EST" package to open the *Energy Storage and Transfer (EST) Package* dialog.

2. In the *GRIDDATA* tab, select the *POROSITY* tab.
3. Click the **Dataset to Array…** button to open the *Select Dataset* dialog.
4. Select the “porosity” dataset and click **OK** to close the *Select Dataset* dialog.
5. Click **OK** at the message stating the dataset has been applied.

The values have some rounding error but it does not matter.

Similarly for HEAT_CAPACITY_SOLID, use the heat-capacity dataset to set the values.

6. In the *GRIDDATA* tab, select the *HEAT_CAPACITY_SOLID* tab.
7. Click the **Dataset to Array…** button to open the *Select Dataset* dialog.
8. Select the “heat-capcity” dataset and click **OK** to close the *Select Dataset* dialog.
9. Click **OK** at the message stating the dataset has been applied.
10. In the *GRIDDATA* tab, select the *DENISITY_SOLID* tab.
11. Set the *Constant* field to “2500.0”.
12. In the *Section* list, turn on *OPTIONS*.
13. Under the *OPTIONS* section, turn on the *DENSITY_WATER* option.
14. Turn on the *HEAT_CAPACITY_WATER* option.
15. Turn on the *LATENT_HEAT_VAPORIZATION* option and change the value to “2500”.
16. Click **OK** to close the Energy Storage and Transfer (EST) Package dialog.

11 Set CND parameters

Now to define the conduction and dispersion properties by completing the following:

1. Double-click on the “ CND” package to open the *Conduction and Dispersion (CND) Package* dialog.
2. In the *GRIDDATA* tab, select the *ALH* tab.
3. Turn on *Define* and set the *Constant* field to “0.1”.
4. Select the *ATH1* tab.
5. Turn on *Define* and set the *Constant* field to “0.01”.
6. Select the *KTW* tab.
7. Turn on *Define* and set the *Constant* field to “0.58”.
8. Select the *KTS* tab.
9. Turn on *Define* and click the **Dataset to Array…** button to open the *Select Dataset* dialog.
10. Select the “kts” dataset and click **OK** to close the *Select Dataset* dialog.
11. Click **OK** at the message stating the dataset has been applied.
12. In the *Section* list, turn on *OPTIONS*.
13. Under the *OPTIONS* section, turn on the *XT3D_OFF* option.
14. Click **OK** to close the Conduction and Dispersion (CND) Package dialog.

12 Set SSM parameters

To set the SSM package parameters, first create an auxiliary variable on the WEL package in the flow model.

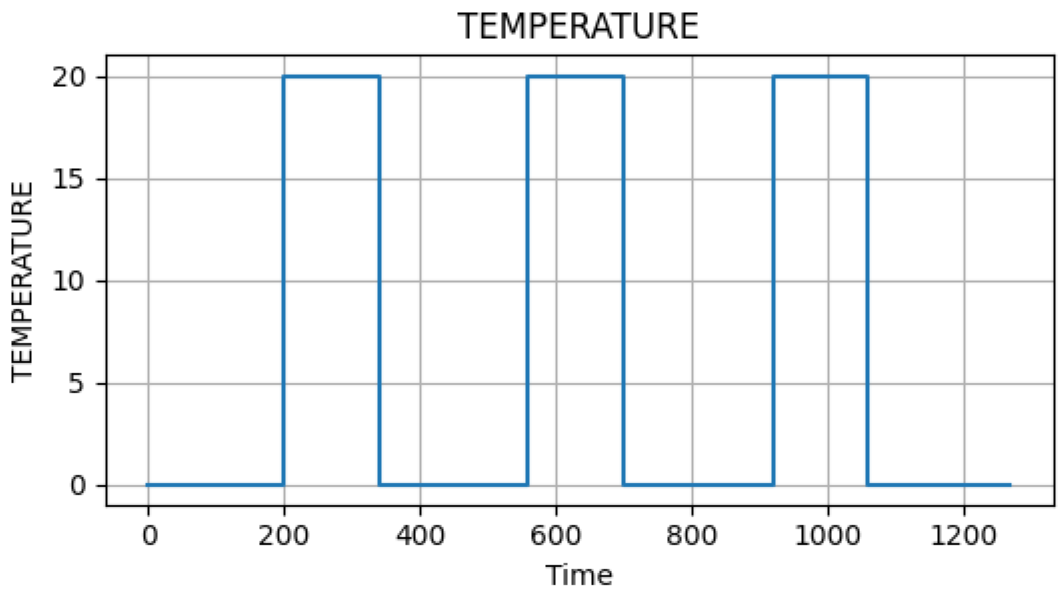
1. Under the " gwf-ates" simulation, right-click on the "wel" package and select **Unlock**.
2. Double-click on the " wel" package to open the *Well (WEL) Package* dialog.

In the interest of time, the TEMPERATURE auxiliary variable has already been defined.

3. Select any cell in the *TEMPERATURE* column.
4. Click the **Plot All Periods** button to open the *XY Series Editor*.

Notice the temperature is 0.0 during extraction periods and 20 during injection periods.

5. Click **Cancel** to exit the *XY Series Editor*.
6. Click **Cancel** to exit the *Well (WEL) Package* dialog.



Open the SSM package

7. Under the " gwe-ates" simulation, double-click on the " SSM" package to open the *Source and Sink Mixing (SSM) Package* dialog.

A row for the WEL package could be added manually, but it's easier to let GMS do it.

8. Under the *Sources* section, click the **Set Up From Flow Model** button to open the *Select GWF6* dialog.
9. Select the "gwf-ates" flow model.
10. Click **OK** to close the *Select GWF6* dialog.

GMS found the WEL package with the TEMPERATURE auxiliary variable and added a row to the table.

11. Click **OK** to close the Source and Sink Mixing (SSM) Package dialog.

13 Saving the Simulation

Before running the model simulation, the data needs to be saved out. Start with saving the project file.

1. Click the **Save** macro to save the project.

The project file has now been saved. However, the simulation files needed to run MODFLOW 6 have not yet been exported. To export these files, do the following:

2. In the Project Explorer, right-click on " ates" and select **Save Simulation**.

The files for the simulation have now been exported.

14 Running MODFLOW 6

It is now possible to run MODFLOW:

1. Right-click on " ates" and select **Run Simulation** to bring up a warning message.

Because a solution was already loaded into the project, this solution will have to be unloaded in order for MODFLOW 6 to run.

2. Click **OK** to close the warning dialog and start the *Simulation Run Queue* model wrapper dialog.

The *Simulation Run Queue* shows all simulation model runs currently in progress. Since this project only has one simulation, only one is shown.

3. When MODFLOW 6 finishes, click **Load Solution**.
4. Click **Close** to exit the *Simulation Run Queue* dialog.

15 Examine the Solution

Review the flow budget from the solution by doing the following:

1. Under the " Solution" folder, expand the " gwe-ates" folder and select the " Temperature" dataset
2. Change time steps using the up/down arrow keys and see how contours show the temperature changing over time.

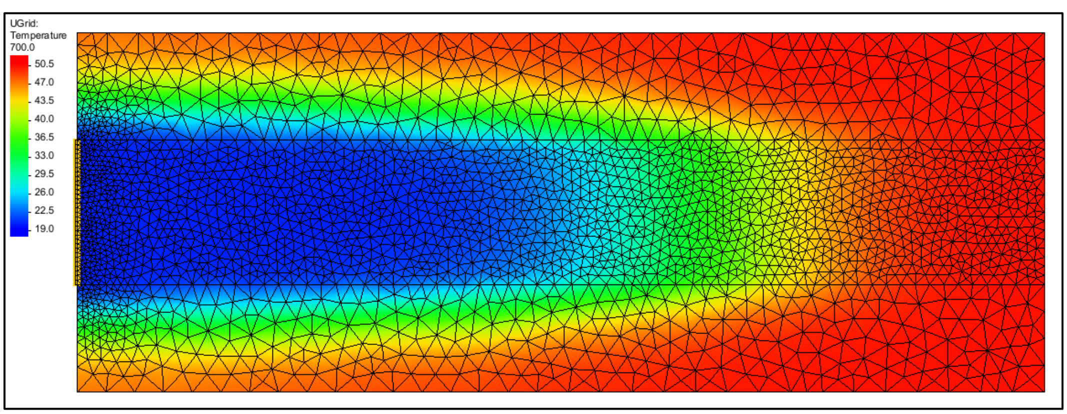


Figure 4 Temperature solution

16 Conclusion

This concludes the "MODFLOW 6 – Groundwater Energy (GWE) Package" tutorial. The following topics were discussed and demonstrated:

- A Groundwater Energy (GWE) model can be added to an existing flow simulation.
- A GWF-GWE exchange and an IMS package must be added to the simulation.
- UGrid datasets can be created with a conceptual model and the *Map To* | **UGrid Datasets** command. These datasets can be used to set array values in MODFLOW 6.
- The SSM package lists packages with auxiliary variables named TEMPERATURE.

