

2D Transport Tutorial Part 1: Setting Up flow model using MODFLOW-2005 and MODPATH

1. Model Configuration

Directory Preparation

Before setting up the simulation, we will create a structured set of folders to organize our different model versions.

Each combination of grid type (coarse/fine) and solver method (FD, TVD, MOC, MMOC) will have its own subdirectory. This organization will help us later when comparing simulation results with the analytical solution.

```
Coarse/  
  coarse_FD/  
  coarse_MOC/
```

```
Fine/  
  fine_FD/  
  fine_MOC/
```

Creating a New MODFLOW Model in ModelMuse

- Launch ModelMuse: Open the ModelMuse on your computer.
- Select "Create New MODFLOW Model"
- Click "Next"
- Geo Reference and Model Description
 - Keep all default settings.
 - Ensure **Length Unit** is **meters** and **Time Unit** is **seconds**.
 - In the Model Description field, enter: **2D Transport Model**.
- Click "Next" to proceed

This initiates the setup process for building a new MODFLOW simulation model.

Grid Definition Dialog Box

- **MODFLOW VERSION:** MODFLOW 2005
- **Number of Columns:** 25
- **Column Width:** 100 units
- **Number of Rows:** 11
- **Row Width:** 100 units
- **Number of Layers:** 1

ModelTop and Upper aquifer can be 0 and -10 respectively.

Leave other parameters to Default Settings. These default settings are appropriate for most model domains unless a specific coordinate transformation is required.

Click: "Finish".

Basic Settings

- Go to "Model" in the menu bar.
- Set **MODFLOW Time** (in seconds):
 - "Starting Time": 0
 - "Ending Time": 86,400,000
 - "Max First Time Step Length": 86,400,000
 - Click "OK"

2. Package Selection and Hydraulic Parameters

Activate MODFLOW Packages

- Go to "Model > MODFLOW Packages and Programs".
- Under "Boundary Conditions", select "Specified Head > CHD" and Click "OK".

Define Hydraulic Conductivity

- Navigate to "Data > Edit Data Sets > Required".
- Under "Hydrology", set: "Kx = 0.001 m/s"
- Click "Apply"
- Click "Close"

3. Boundary Conditions

Left Boundary - Specified Head

1. Select "Create straight-line object" (stairs icon below "Customize").
2. Click on the **upper-left cell** in the top view, drag to the **lower-left cell**, and double-click.
3. Name the object: "left_CHD"
4. In the dialog box switch to MODFLOW Features tab
5. In the MODFLOW features tab check the box next to "CHD" (i.e. "MODFLOW Features > CHD"), then set:
 - "Starting Time": 0
 - "Ending Time": 86,400,000
 - "Starting Head": 22
 - "Ending Head": 22
6. Click "OK".

Right Boundary - Specified Head

1. Repeat previous steps to create a new straight-line object.
2. Click on the **upper-right cell**, drag to the **lower-right cell**, and double-click.
3. Name the object: "right_CHD"
4. In the dialog box switch to MODFLOW Features tab
5. In the MODFLOW features tab check the box next to "CHD" (i.e. "MODFLOW Features > CHD"), then set:
 - "Starting Time": 0
 - "Ending Time": 86,400,000
 - "Starting Head": 10
 - "Ending Head": 10
6. Click "OK".

4. A Note About "Warnings" in ModelMuse

You may see a warning message in the next step when you run the model. This can be safely ignored—simply click "Close" in the warning box.

The most common warning is related to "missing georeference information". When setting up the grid, there is an option to define a georeference, which is especially useful when working with real-world spatial data. If this step is skipped, ModelMuse may issue a warning, but it does not affect the model's functionality for general or conceptual use.

In general, warnings in ModelMuse are notifications rather than critical errors. While it is good practice to read them and understand their context, they can often be safely ignored unless they are persistent or impact model results.

5. Running the Simulation

Run MODFLOW

1. Click the **green triangle** below "Grid" to run MODFLOW-2005.
2. Navigate to the appropriate subdirectory you created earlier (e.g., **Coarse/coarse_FD/**).
3. Save the model as **coarse.nam**.
4. Confirm and run the simulation.
5. After execution:
 - Check the **ModelMonitor** (green smileys indicate success).
 - Close the monitor window.
 - Review the **listing file**, especially the **Budget** section.
 - Compare the results with the Excel sheet.

6. Setting up MODPATH for Particle Tracking

MODPATH Configuration

1. Go to "Model > MODFLOW Packages and Programs > Post Processors > MODPATH".
2. Check the box next to "MODPATH".
3. Set:
 - MODPATH Version: 6
 - Output Mode: Pathlines
 - Reference Time: 0
 - Tracking Direction: Forward

4. Under "Version 6 & 7 Options", set:
 - "StopOption: Stop at termination points (Steady State (2))"
5. Click "OK".

7. Running MODPATH

Initial Particle Placement

1. Select "Select objects" (red cursor tool).
2. Double-click on the left_CHD object.
3. Go to "MODFLOW Features > MODPATH".
4. Choose Initial Particle Placement: Grid.
5. Click "OK".

Prepare Output

1. Go to "Model > MODFLOW Output Control > Head".
2. Set "External File Type: Binary".
3. Click "OK".
4. Save the model "Ctrl + S".

Run Simulation

- Click the green triangle below "Grid" to run MODFLOW.
- A warning may appear about MODPATH version 7. Since we already selected version 6 earlier, simply click "Yes" to continue.
- Check ModelMonitor and listing file.
- Close the command window.

8. Visualization

Visualize Pathlines

1. Select "Data Visualization" (colored icon in second row).
2. Choose "MODPATH Pathlines".
3. Set:
 - MODPATH Pathline File: *.path
4. Click "Apply", then "Close".
5. Review and interpret the particle pathlines.