

# 2D Transport Tutorial Part 2: Setting Up the Transport Model

## 1. Setting Up the Transport Model

### 1.1 Activating Solute Transport Packages

- To begin configuring the transport model, navigate to:  
"Model > MODFLOW Packages and Programs > Groundwater transport (Expand) > MT3DMS or MT3D-USGS"
- From the available list, select the following packages: BTN, ADV, DSP, SSM, and GCG.
- Additional changes are required within some of these packages. Follow the step-by-step instructions below:
  - **BTN Package:** Under MT3D Options, change the MT3D version from MT3D-USGS to MT3DMS.
  - **ADV Package**  
Advection 1 > Advection Solution Scheme > Standard Finite Difference  
*Note: We will explore other solution schemes in subsequent runs.*
- Click "OK" to close the packages window.

### 1.2 Configuring Transport Model Time Discretization

When prompted to configure MT3D time settings, proceed as follows:

- Go to Model > MODFLOW Time
- Choose MT3DMS or MT3D-USGS
- Set the following parameters:
  - Starting Time: 0
  - Ending Time: 86,400,000
  - Initial Time Step: 86400
  - Max Transport Steps per Flow Step: 1000
- Click "OK"

## 1.3 Defining Contaminant Source and Observation Points

### Source

1. Select **Create point object** (dot icon below “scissors“ icon).
2. Placing the source: Click on the 6th row, 4th column.
3. Name the object **source**.
4. Go to **MODFLOW Features > SSM**, then set:
  - **Check Specified Concentration**
  - To input the values, click directly on the text fields. They may appear inactive at first, but will become editable when double-clicked. If this doesn't work, check the bottom-left of the dialog box—there is a setting labeled “**Number of times.**”. If it is set to 0, change it to 1. This will enable the input fields for entering various parameter values.
  - **Starting Time:** 0
  - **Ending Time:** 86,400,000
  - **Chem concentration:** 1
5. Click "OK".

**Observation Points** All observation points are in the 6th row and placed at columns: 5, 7, 9, and 14 corresponding to 100m, 300m, 500m, and 1000m respectively from the source (cell size = 100m).

Repeat the following for each:

1. Select **Create point object**
2. Click on the appropriate cell (e.g., 6th row, 5th column for 100m)
3. Name the object (e.g., **Obs100**)
4. Go to **Data Sets > Required**, expand **MT3DMS** or **MT3D-USGS**
5. Check **MT3DMS\_Observation\_Location** and set it to **TRUE**
6. Press "OK"

## 1.4 Setting Longitudinal Dispersivity

- Go to **Data > Edit Data Sets>Required**
- Expand **MT3DMS**, **MT3D-USGS**, or **GWT**
- Locate **Longitudinal\_Dispersivity**

- Set the value to 10
- Click "Apply"
- Click "Close"

## 1.5 Executing the Simulation

**Note:** Before running the transport model, the flow model must be re-executed to generate the Flow Transport Link (.ftl) file. This step is essential because new boundary and source conditions have been added.

### Running MODFLOW

- Click the green triangle below **Grid**
- Navigate to **Coarse/coarse\_FD/**
- Save as **coarse.nam**
- Run the simulation

### Running MT3DMS

- Click the dropdown next to the green triangle
- Click **Export MT3D Input Files**
- Name the file **coarse.mtnam**
- Check **ModelMonitor** and the listing file
- Close the command window

### Checklist:

- Check percent discrepancy in **.lst** file
- Note simulation run time
- Review **Mass Transport Observation(.MT0)** file for observation data:
  - Format: time step, cumulative time, layer (K), row (I), column (J), concentration

## 1.6 Comparing Against Analytical Solution

- Open the Excel sheet provided
- Go to the **Coarse** worksheet
- Copy-paste contents of **.mto** file
- View superimposed plot of numerical and analytical solutions for all observation points

## 1.7 Visualizing the Results

1. Click **Import and display result** (colored icon next to the green simulation run button)
2. Navigate to **Coarse\_FD/** and double-click the **.ucn** file
3. Select the final transport time step.
4. In the **Select Model Results to Import** window:
  - Initially, the **Classification** dropdown shows **Model Result**, and the adjacent **Prefix** field is disabled
  - Change **Classification** to **User Define**
  - The **Prefix** field becomes active; enter: **FD\_Coarse**
  - Select **"Contour Grid"**
  - Click **"OK"**

**Note:** .UCN files store domain-wide concentration data in binary format, whereas .MTO files contain data only at defined observation points.