MT3DMS v5.3

Supplemental User's Guide

Chunmiao Zheng
Department of Geological Sciences
The University of Alabama

The Hydrogeology Group
The University of Alabama

MT3DMS v5.3

a modular three-dimensional multispecies transport model
for simulation of advection, dispersion and chemical reactions
of contaminants in groundwater systems

Supplemental User's Guide

By   Chunmiao Zheng
       Department of Geological Sciences
       The University of Alabama
       Tuscaloosa, Alabama 35487

COPYRIGHT NOTICE, DISCLAIMER
AND CONTACT INFORMATION

This program and the latest updates can be downloaded
from the MT3D home page at:

http://hydro.geo.ua.edu/mt3d

For technical information or to report program errors,
please contact:

Chunmiao Zheng
Department of Geological Sciences
The University of Alabama
202 Bevill Research Building
Tuscaloosa, Alabama 35487, USA
E-mail: czheng@ua.edu

Cover image courtesy of Paul Hsieh of the U.S.
Geological Survey, showing a trichloroethylene
(TCE) plume in an aquifer at the Massachusetts
Military Reservation in Cape Cod, Massachusetts.

Preface

This document describes the cumulative changes and enhancements made since Version 5.0 of the MT3DMS solute fate and transport model. It is intended as a supplement to the original MT3DMS Documentation and User's Guide published by the U.S. Army Corps of Engineers (Zheng and Wang, 1999; available at http://hydro.geo.ua.edu/mt3d). Readers should refer to Zheng and Wang (1999) for complete information on the theoretical background, numerical implementation, input instructions, and application examples for the MT3DMS model.

Table of Contents

(*The blue color indicates new documentation added for Version 5.3.)

1   Introduction

This document describes the cumulative changes and enhancements made since Version 5.0 of the MT3DMS solute fate and transport model and discusses all the new features that have become available since Version 5.0. It is intended as a supplement to the original MT3DMS Documentation and User's Guide (Zheng and Wang, 1999). For the most up-to-date information on MT3DMS, refer to the README file included with the distribution files.

There are a couple of significant organizational changes to Version 5 compared to the previous Version 4. First, the Name-File procedure becomes the only method to start a MT3DMS simulation run since v5.0. A Name-File specifies the names of most input and output files used in a model simulation. In addition, it controls the parts of the model program that are active, i.e., the "packages" that are used in the current simulation. Second, the implicit matrix solver, the General Conjugate-Gradient (GCG) solver, must be used in every simulation since v5.0. In other words, the dispersion, sink/source and reaction terms are now always solved by the implicit finite-difference method, regardless of whether the advection term is solved by the implicit finite-difference method, the mixed Eulerian-Lagrangian methods, or the third-order TVD method.

A major new feature for MT3DMS since v5.0 is the Transport Observation (TOB) Package to save the calculated concentration at any observation location within the model domain and the calculated mass flux at any sink/source location. The calculated values are saved to output files, and optionally, along with the statistics of residuals between the calculated and observed values at the observation locations. The calculated concentrations can be interpolated from the nodal values if the observation point does not coincide with a model node. The calculated mass flux can be defined over any arbitrary group of sink/source cells referred to as a mass flux object.

Another important addition to MT3DMS since v5.0 is the support for the Multi-Node Well (MNW) Package (Halford and Hanson, 2002) used by MODFLOW-2000 (Harbaugh et al., 2000) and MODFLOW-2005 (Harbaugh, 2005) to simulate the flow to a well screened over multiple nodes (layers). The MNW Package automatically partitions the total flow prescribed for a MNW into individual nodes (layers) and determines a single head value in the wellbore. Accordingly, MT3DMS computes a single composite concentration for the same wellbore based on the flux-weighted concentrations of the injected fluid (if any) and the flow rates from different layers.

Since Version 5.0, MT3DMS has also added support for more new MODFLOW sink/source packages, including Drain with Return Flow (DRT) and Evapotranspiration with a Segmented Function (ETS), both documented in Banta (2000). The DRT Package in MODFLOW-2000 simulates the re-injection of a portion of the outflow from a drain cell back into the aquifer. MT3DMS assigns the concentration at the outflow cell as that

of the re-injected source at the return drain-flow location. The ETS package is supported in a manner similar to the original EVT package.

New since Version 5.1 is the capability to simulate zeroth-order reactions in both single- and dual-domain systems. The standard MT3DMS code prior to v5.1 includes only the first-order kinetic reactions in the Chemical Reaction (RCT) Package. However, zeroth-order reactions may be useful for describing certain types of biogeochemical decay or production. In addition, zeroth-order reactions are needed in direct simulation of groundwater ages (e.g., Goode, 1996) and calculation of parameter sensitivities (e.g., Tsai et al., 2003). Thus starting with Version 5.1, zeroth-order reactions are available as a standard option through the RCT Package in either single- or dual-domain formulation.

Version 5.2 introduces a new option to include component-dependent and/or three-dimensional diffusion coefficients in the transport simulation. Prior to v5.2, the molecular diffusion coefficient can only be specified on a layer-by-layer basis (i.e., one uniform diffusion coefficient per model layer). Moreover, all solute components are assumed to have the same diffusion coefficient. Starting with Version 5.2, users are permitted to specify different diffusion coefficients for different solute components on a cell-by-cell basis (i.e., one diffusion coefficient per model cell, if necessary).

Another new feature since Version 5.2 is the capability to simulate a "recirculation well." A recirculation well refers to an injection well whose input concentration is not user-specified, but set internally equal to that of extracted water from a pumping well. This option is convenient for modeling some commonly encountered field situations, such as a "dipole" tracer test.

With MT3DMS Version 5.3, it is possible to specify an arbitrarily defined time-varying mass loading source or boundary condition. This new capability allows the complete mass loading distribution to be specified at a source or boundary without being restricted by the definition of stress periods. The new capability is accomplished through a new package called the Hydrocarbon Spill Source (HSS) Package that has been developed for MT3DMS v5.3 (Zheng et al., 2010).

Another significant change to Version 5.3 is the addition of a steady-state transport simulation option. This steady-state transport option may be useful for long-term transport simulation runs in which the concentration field reaches an equilibrium state with the net change in total mass equal to zero. Also, direct groundwater age simulation usually requires a steady-state concentration solution.

Finally, a new section is added to this documentation to describe the use of MT3DMS for heat transport modeling. Heat is increasingly being recognized as an excellent groundwater tracer because of its usefulness in identifying multiple hydrologic processes and recent availability of improved temperature sensors and measurement technologies. The mathematical equivalency between the heat and solute transport equations makes it possible to use MT3DMS directly and efficiently to simulate heat transport with simple variable conversion.

2. Running MT3DMS Using the Name-File Method

2.1 BACKGROUND

Starting with version 5.0, the only method to start a MT3DMS simulation is through a Name File that is similar to the Name File used by MODFLOW-96, -2000 and -2005. The Name-File method has been available in MT3DMS since version 4 as an optional method of starting a transport simulation (see Zheng et al., 2001). However, it has become the only method to initiate a transport run since MT3DMS v5.0. As a result, the Transport Option (TRNOP) array specified through the Basic Transport (BTN) Package of MT3DMS v5.1 no longer has any effect. However, the input line must still be specified in the BTN input file for backward compatibility reasons.

2.2 INPUT INSTRUCTIONS

The Name File contains the names of most input and output files used in a model simulation and controls the parts of the model program that are active. The Name File is read on unit 99, which is specified in the MT3DMS main program. The Name File is constructed as follows:

FOR EACH SIMULATION

```
1.  Ftype      Nunit      Fname      [options]
```

The Name File contains one of the above records (item 1) for each file. All variables are free format. The length of each record must be 199 characters or less. The records can be in any order except for the record where Ftype (file type) is 'LIST' as described below.

Comment records are indicated by the # character in column 1 and can be located anywhere in the file. Any text characters can follow the # character. Comment records have no effect on the simulation; their purpose is to allow users to provide documentation about a particular simulation. All comment records after the first item-1 record are written in the listing file.

Explanation of Variables in the Name File

Ftype - is the file type, which must be one of the following character values. Ftype may be entered in all uppercase, all lowercase, or any combination.

LIST for the standard MT3DMS output file – the Name File for MT3DMS must always include a record that specifies 'LIST' for Ftype and the LIST record must be the first non-comment record.

BTN for the MT3DMS Basic Transport Package.
FTL for the MODFLOW-produced flow-transport link file.
ADV for the MT3DMS Advection Package.
DSP for the MT3DMS Dispersion Package.
SSM for the MT3DMS Sink/Source Mixing Package.
RCT for the MT3DMS Reaction Package.
GCG for the MT3DMS Generalized Conjugate-Gradient Solver Package.
TOB for the MT3DMS Transport Observation Package.
HSS for the MT3DMS HSS Time-Varying Source Package.

DATA(BINARY) for binary (unformatted) files such as those used for input of concentrations saved in a previous simulation as the initial condition for a continuation run.

DATA for formatted (text) files such as those used to save formatted concentrations at observation points and mass budget summaries or for input of data from files that are separate from the primary package input files.

Various output control options of MT3DMS can be set up to save several optional output files: the unformatted (binary) concentration file, the formatted concentration observation file, the formatted mass budget summary file, and the model configuration file. MT3DMS always assigns default names to these files with the conventions listed below. These default names can be overridden, as explained in the next paragraph.

- **MT3Dnnn.UCN** for the dissolved-phase unformatted concentration files where **nnn** is the species index number such as 001 for species 1, 002 for species 2, and so on;
- **MT3DnnnS.UCN** for the sorbed-phase or immobile-liquid-phase unformatted concentration files where **nnn** is the species index number such as 001 for species 1, 002 for species 2, and so on;
- **MT3Dnnn.OBS** for the formatted concentration observation files;
- **MT3Dnnn.MAS** for the formatted mass budget summary files; and
- **MT3D.CNF** for storing the model configuration (spatial discretization) information needed by post-processing programs. It is always saved along with the UCN files.

Nunit - is the FORTRAN unit to be used when reading from or writing to the file. Any valid unit number on the computer being used can be specified except for the unit numbers that have been internally preserved by the MT3DMS program, as listed in Table 2.1. To use the preserved unit number for a particular file, simply set Nunit associated with that file to 0. If a preserved unit is used for a file for which the unit is not intended,

an error may occur and the program execution will be terminated. To avoid potential errors, avoid using any units between 1 and 20, and any units above 100, when specifying units for those files that do not have a preserved unit (i.e., not listed in Table 2.1).

As pointed out previously, MT3DMS assigns the default file names for 'UCN', 'OBS', 'MAS' and 'CNF' files as MT3Dnnn.UCN, MT3DnnnS.UCN, MT3Dnnn.OBS, MT3Dnnn.MAS, and MT3D.CNF. To keep the results from a previous simulation, these files need to be renamed before starting a new simulation in the same directory. Otherwise, they will be overwritten by the files from the new simulation. It is possible to override these default names. To specify a different name for an unformatted 'UCN' file, for example, add a line as shown below into the MT3DMS Name File:

```
DATA(BINARY)  Nunit   NewRun.UCN
```

where Nunit must be a preserved unit for a particular species. For example, if the NewRun.UCN is intended for saving the unformatted concentration of species 1, then Nunit must be set to 201, i.e., the unit preserved for species 1 (see Table 2.1). Similarly, if the NewRun.UCN is intended for species 2, then Nunit must be set to 202.

To specify a different name for the formatted 'OBS', 'MAS' and 'CNF' files, insert a line as shown below into the MT3DMS Name File

```
DATA   Nunit   NewRun.OBS
```

where again Nunit must be a preserved unit for a particular species. For example, if NewRun.OBS is intended for saving the species 1 concentrations at the observation points, then Nunit must be set to 401, the unit preserved for species 1 (see Table 2.1). Similarly, if the NewRun.OBS is intended for species 2, then Nunit must be set to 402.

Fname - is the name of the input/output file, which is a character value. Pathnames may be specified as part of Fname.

[Options] – optional keywords that may be used for the corresponding input/output file. Currently, only two such keywords, FREE and PRINT, may be specified in conjunction with the flow-transport link (FTL) file. If the keyword FREE is specified, it indicates that the FTL input file for MT3DMS is in list-directed (free) format, i.e., produced by the LMT6 Package with the option OUTPUT_FILE_FORMAT set to formatted. If the keyword PRINT is specified, the content of the flow-transport link file is printed to the standard output file for checking and debugging purposes. Note that if no keyword is specified after the FTL file name, the FTL file is assumed to be unformatted (binary) by default.

Table 2.1. Preserved unit numbers for various file types in MT3DMS.

MT3DMS Input/Output Files	File Type	Preserved Unit
Output Listing File*	LIST	16
Package Options		
Basic Transport*	BTN	1
Flow-Transport Link*	FTL	10
Advection	ADV	2
Dispersion	DSP	3
Sink/Source Mixing	SSM	4
Reaction	RCT	8
Generalized Conjugate Gradient	GCG	9
Transport Observation	TOB	7
HSS Time-Varying Source	HSS	13
Output Files		
Model Configuration File	CNF	17
Unformatted Concentration File (dissolved phase)	UCN	200+species index
Unformatted Concentration File (sorbed/immobile phase)	UCN	300+species index
Concentrations Observation File	OBS	400+species index
Mass Budget Summary File	MAS	600+species index

*Note: these files are always required for every simulation.

An example of the MT3DMS Name File is shown below:

```
#
# MT3DMS Name File for a test problem
#
# Standard output
list       0   test1.m3d
#
# Input files
btn        0   test1.btn
adv        0   test1.adv
dsp        0   test1.dsp
ssm        0   test1.ssm
rct        0   test1.rct
gcg        0   test1.gcg
ftl        0   test1.ftl
```

In the example above, the FTL file 'test1.ftl' is treated as an unformatted (binary) file by default. If 'test1.ftl' is an ascii text file in free format, the last line should be modified as

```
ftl        0  test1.ftl    Free
```

Moreover, if the option print is specified as follows,

```
ftl        0  test1.ftl    print
```

Then the contents of the flow-transport link file read by MT3DMS will be echoed in the standard text output file generated by MT3DMS.

Further note that in the example above, no names are given for the unformatted concentration, formatted concentration observation, mass budget summary, and model configuration files. As a result, these files will be named, by default, MT3D001.UCN, MT3D001S.UCN, MT3D001.OBS, MT3D001.MAS, MT3D.CNF. To associate these files with different names, the Name File shown above can be modified by adding the following lines:

```
data(binary)  201  test1.ucn
data(binary)  301  test1s.ucn
data          401  test1.obs
data          601  test1.mas
data          17   test1.cnf
```

2.3 PROGRAM EXECUTION

To use the Name File method to start an MT3DMS simulation, the Name File must be created first. The name of the Name File can then be passed to MT3DMS through the command line argument as follows:

C:\MT3DMS5\Bin\MT3DMS5x Name-File

where **MT3DMS5x** is the name of the MT3DMS v5 executable program (where **x** denotes either **s** for the executable program using the standard unformatted files; and **b** for the executable program using the true binary files), and **Name-File** is the name of the MT3DMS Name File for a particular simulation. If no command line argument is specified, the user will be prompted to provide the name of the Name File interactively from the keyboard. The name of a Name File is assumed to have the 3-letter extension ".nam" by default if no extension is given when the Name File is specified from the command-line or interactively.

3 Using the New Transport Observation Package

3.1 BACKGROUND

In previous versions of MT3DMS, the user can specify the layer, row and column indices of any model nodes of interest where the calculated concentration values will be saved to one or more optional concentration observation (OBS) output files. If the user needs to obtain the calculated concentration at an observation point that does not coincide with a model nodal point, neighboring nodal concentrations must be saved and an interpolation performed outside the MT3DMS code. This can be quite cumbersome. Moreover, there is no capability in previous versions to save the calculated mass fluxes at any sink/source locations.

A new Transport Observation (abbreviated as "TOB") Package has been added to MT3DMS since v5 to save the calculated concentration at any observation point and the calculated mass flux at any sink/source location. The calculated values can be saved to output files along with the statistics of residuals between the calculated and observed values at the specified locations. The calculated concentrations can be interpolated from the neighboring nodal values if the observation point does not coincide with a model node. The calculated mass flux can be defined over any arbitrary group of sink/source cells referred to as a mass flux object.

3.2 IMPLEMENTING AND ACTIVATING THE TOB PACKAGE

The TOB Package is implemented in MT3DMS v5 by inserting the package name "TOB" into any unused slot of the "NameTRNOP" array in the MT3DMS main program, as illustrated below:

```
 DATA NameTRNOP/'ADV ', 'DSP ', 'SSM ', 'RCT ', 'GCG ',
&                '    ', '    ', '    ', '    ', '    ',
&                'TOB ', '    ', '    ', '    ', '    ',
&                '    ', '    ', '    ', '    ', '    '
```

To activate the TOB Package in a MT3DMS run, insert a line to the Name File of MT3DMS as shown below:

TOB   0   [project.tob]

where **TOB** is the input file type for the TOB Package; integer 0 is the default input unit number for the TOB Package (which is preset to 7 internally by MT3DMS); and `project.tob` is the name of the input file for the TOB Package.

An example of the MT3DMS Name File including the TOB Package:

```
#
# MT3DMS 5 Name File for test case P7
#

# Standard output file
LIST  0 p7.out

# Transport package input files
BTN    0 p7.btn
ADV    0 p7.adv
DSP    0 p7.dsp
SSM    0 p7.ssm
GCG    0 p7.gcg

# Transport observation input file
TOB    0 p7.tob

# Flow-Transport Link input file
FTL    0 p7.ftl  free noprint
```

3.3 INPUT INSTRUCTIONS

Input for the TOB Package is read from a file listed in the Name File with the keyword "TOB" as the file type. The input data are arranged and read sequentially in free format.

For Each Simulation

0.   [#Text]

Item 0 is optional and can include as many lines as desired. Each line needs to begin with the "#" character in the first column.

1.   MaxConcObs, MaxFluxObs, MaxFluxCells

2.   OUTNAM, inConcObs, inFluxObs, inSaveObs

Read items 3, 4, and 5 if [`inConcObs`] > 0

```
3.    nConcObs, CScale, iOutCobs, iConcLOG, iConcINTP

Read items 4 and 5 for [nConcObs] times
4.    COBSNAM, Layer, Row, Column, iComp, TimeObs, Roff,
      Coff, weight, COBS

Read item 5 if [Layer] < 0
5.    mLayer(1), prLayer(1), mLayer(2), prLayer(2), …,
      mLayer(|Layer|), prLayer(|Layer|)
```

Read items 6, 7, 8 and 9 if [`inFluxObs`] > 0

```
6.    nFluxGroup, FScale, iOutFlux

Read items 7, 8 and 9 for [nFluxGroup] times
7.    nFluxTimeObs, nCells, iSSType

Read item 8 [nFluxTimeObs] times
8.    FOBSNAM, iComp, FluxTimeObs, weight_fobs, FluxObs

Read item 9 [nCells] times for current mass flux object
9.    kcell, icell, jcell, factor
```

Text—is a character string (maximum of 79 characters) that starts in column 2. Any characters can be included in Text. The "#" character needs to be in column 1. Text is printed when the file is read and provides an opportunity for the user to include information about the model both in the input file and the associated output file.

MaxConcObs—is the maximum number of concentration observations allowed in the current simulation. This value is used for memory allocation purposes.

MaxFluxObs—is the maximum number of mass-flux observations. It should be set large enough to accommodate all mass-flux observations at different mass-flux objects and observation times. This value is used for memory allocation purposes.

MaxFluxCells—is the maximum number of model cells that makes up a mass-flux object. This value is used for memory allocation purposes.

OUTNAM—a string of one to 78 nonblank characters. OUTNAM specifies the base (root) name for three optional output files. The complete file names are composed of this base name followed by a period and a three-character extension listed below. The specification of lower and upper cases in OUTNAM is preserved in generating the file-name base. OUTNAM can include a path; constraints imposed by a particular computer operating system regarding file names and paths should be considered when specifying OUTNAM.

Output File	Content
`[OUTNAM].OCN`	Output file containing calculated concentrations, and optionally, residuals between the calculated and observed values, at user-defined observation points that are screened either in a single layer or across multiple layers. This text file is generated only if the flag `[inConcObs]` is greater than zero.
`[OUTNAM].MFX`	Output file containing calculated mass fluxes into or out of user-defined mass flux objects, and optionally, residuals between the calculated and observed values. Each mass flux object is defined by a group of model cells containing external sinks/sources such as wells, rivers, drains, recharge, and general-head boundaries. This text file is generated only if the flag `[inFluxObs]` is greater than zero.
`[OUTNAM].PST`	Output file containing calculated concentrations and mass fluxes at user-defined observation points and mass-flux objects. This output file, in binary form, is intended for post-processing purposes or for linkage with other modeling programs. This file is generated only if the flag `[inSaveObs]` is greater than zero.

inConcObs—is an integer flag indicating whether the calculated concentrations at the observation locations should be obtained and saved to the output file `[OUTNAM].OCN`. It also serves as the unit number for the output file `[OUTNAM].OCN`.

inFluxObs—is an integer flag indicating whether the calculated mass fluxes at the mass-flux objects should be obtained and saved to the output file `[OUTNAME].MFX`. It also serves as the unit number for the output file `[OUTNAM].MFX`.

inSaveObs—is an integer flag indicating whether the calculated concentrations and mass fluxes at the user-defined observation points and mass-flux objects should be saved to an unformatted (binary) output file `[OUTNAM].PST`. It also serves as the unit number for the output file `[OUTNAM].PST`.

nConcObs—is the number of concentration observations. Observations made at the same location but different times are considered multiple observations.

CScale—is the multiplier (scaling factor) for the observed concentrations. It is used to convert the unit of observed concentrations to the unit of calculated concentrations used internally in MT3DMS for computing appropriate residuals.

iOutCobs—is an integer flag indicating what type of output should be computed and saved:
iOutCobs = 0,   calculated concentrations at the observation locations are saved to the output file [OUTNAM].OCN;
iOutCobs > 0,   both calculated concentrations and residual errors between the calculated and observed values are saved to the output file [OUTNAM].OCN. The statistics of the residual errors is also computed and saved.

iConcLOG—is an integer flag indicating whether the calculated and observed concentrations should be converted to the common logarithm before computing the residual error and related statistics:
iConcLOG = 0, no conversion is done (residual error = calculated – observed);
iConcLOG > 0, convert the calculated and observed concentration values to the common logarithmic scale before computing the residual error and related statistics (residual error = $\log_{10}$Calculated – $\log_{10}$Observed).

iConcINTP—is an integer flag indicating whether the calculated concentration at an observation location should be interpolated from its neighboring nodal points, if the observation location does not coincide with a nodal point:
iConcINTP = 0, no interpolation is done (the calculated concentration value at the nearest nodal point is used for comparison with the observed value);
iConcINTP > 0, perform bilinear interpolation using four neighboring nodal concentrations in the same model layer.

COBSNAM—is a string of 1 to 12 nonblank characters used to identify the observation. The identifier need not be unique; however, identification of observations in the output files is facilitated if each observation is given a unique COBSNAM.

LAYER—is the layer index of the cell in which the concentration observation is located. If LAYER is less than zero, solute concentrations from multiple layers are combined to calculate a simulated value. The number of layers equals the absolute value of LAYER, or |LAYER|.

ROW—is the row index of the cell in which the concentration observation is located.

COLUMN—is the column index of the cell in which the concentration observation is located.

iComp—is an integer indicating the solute species for which the concentration observation is made. Integer 1 indicates the first species, 2 the second species, and so on.

TimeObs—is the time since the beginning of simulation to the time of the current observation. [TimeObs] should be included in the BTN input file as part of the input array [TIMPRS], i.e., the time to save simulation results. Otherwise, the calculated concentration is obtained from a time specified in [TIMPRS] or the end of a stress period that is closest to [TimeObs]. Note that if [TimeObs] is specified as a *negative* integer, the calculated concentration is saved whenever the number of transport steps is an even multiple of |TimeObs|.

Roff—is the row offset used to locate the observation within a finite-difference cell. The convention is the same as that used by MODFLOW-2000 (see Figure 3.1).

Coff—is the column offset used to locate the observation within a finite-difference cell. The convention is the same as that used by MODFLOW-2000 (see Figure 3.1).

weight—is the user-specified weighting factor for computing the residual error at the current observation, i.e., residual error = (calculated-observed)*weight. If [weight] is assigned a *negative* value, the observed concentration at the target observation point is not used and only the calculated concentration is saved.

COBS—is the concentration observation for the species defined by [iComp]. This input item is required regardless of whether the preceding input item [weight] has been given a positive or negative value.

mLayer(i)—is the ith layer number for a multilayer concentration observation.

prLayer(i)—is the proportion of the simulated solute concentration in layer mLayer(i) that is used to calculate a simulated multilayer concentration. The sum of all prLayer(i) values for a given observation needs to equal 1.0. The convention is the same as that used by MODFLOW-2000 (see Hill et al., 2000, p. 35). Note that if the concentration observation is made at a multi-node well (MNW), the flux-averaged composite concentration for the MNW wellbore can be computed and saved through the SSM Package for MT3DMS v5.

nFluxGroup—is the total number of mass flux objects. A mass flux object is defined as a group of model cells needed to represent one mass flux measurement.

FScale—is the multiplier (scaling factor) for the observed mass flux. It is used to convert the unit of observed mass flux to the unit of calculated mass flux observation used internally in MT3DMS for computing appropriate residuals.

COLUMNS

i-1

i

i+1

ROWS

j - 1   j   j + 1

ROFF = -0.5

COFF = -0.5   COFF = 0.5

B   A   C

E

D   F

ROFF = 0.5

EXPLANATION

☐ Active cell   ▨ Inactive cell   ● Cell center   ○ Point

POINT	ROFF	COFF
A (Cell center)	0.0	0.0
B	-0.25	-0.25
C	-0.2	0.4
D	0.45	-0.45
E	0.25	0.0
F	0.4	0.25

Figure 3.1. Locating points within a finite-difference cell using ROFF and COFF (after Hill et al., 2000).

iOutFlux—is an integer flag indicating what type of output should be computed and saved:

iOutFlux = 0, calculated mass fluxes are saved to the output file [OUTNAM].MFX;

iOutFlux > 0, both calculated mass fluxes and residual errors between the calculated and observed values are saved to the output file [OUTNAM].MFX. The statistics of the residual errors is also computed and saved.

nFluxTimeObs—is the number of times at which mass fluxes are observed for the current mass-flux object.

nCells—is the total number of cells in the current mass-flux object.

iSSType—is an integer code indicating the type of sinks/sources constituting the current mass flux object. The [iSSType] codes used here are the same as those defined for the MT3DMS SSM Package:

iSSType Code	Type of Sink/Source
1	Constant-head
2	Well
3	Drain
4	River
5	Head-dependent boundary
6	(Reserved)
7	Recharge
8	Evapotranspiration
9-14	(Reserved)
15	Mass loading
21	Stream-routing
22	Reservoir
23	Specified flow and head boundary
24	Inter-bed storage
25	Transient leakage
26	Lake
27	Multi-node well
28	Drain with return flow
29	Segmented evapotranspiration
30-49	(Reserved)
50	HSS Mass Loading
51	(Reserved)
52	(Reserved)
53	(Reserved)

FOBSNAM—is a string of 1 to 12 nonblank characters used to identify the mass flux observation. The convention is the same as that for COBSNAM. The first part of FOBSNAM can be used to identify a common group name, while the rest to distinguish different observations for the same group.

FluxTimeObs—is the time since the beginning of simulation to the time of the current mass flux observation. [FluxTimeObs] should be included in the BTN input file as part of the input array [TIMPRS], i.e., the time to save simulation results. Otherwise, the calculated mass flux is obtained from a time specified in [TIMPRS] or the end of a stress period that is closest to [FluxTimeObs]. Note that if [FluxTimeObs] is specified as a *negative* integer, the calculated mass flux is saved whenever the number of transport steps is an even multiple of |FluxTimeObs|.

weight_fobs— is the user-specified weighting factor for computing the residual error at the current mass-flux object, i.e., residual error = (calculated-observed)*weight. If [weight_fobs] is assigned a *negative* value, the observed mass flux at the target mass-flux object is not used and only the calculated mass flux is saved.

FluxObs—is the observed solute mass flux, QC [dimension, MT^{-1}], for the user-specified species [iComp]. The mass flux observation is negative when the mass is leaving the groundwater system, and positive when the mass is entering the groundwater system. This input item is required regardless of whether the preceding input item [weight_fobs] has been given a positive or negative value.

kcell—is the layer index of a sink/source cell included in the current mass flux object.

icell—is the row index of a sink/source cell included in the current mass flux object.

jcell—is the column index of a sink/source cell included in the current mass flux object.

factor—is the weighting factor for the mass flux calculated at the specified cell location (jcell, icell, kcell). [factor] = 1.0 under most circumstances, i.e., the specified cell belongs to a single mass-flux object. However, the mass flux calculated at one specific cell can be assigned to one or more mass-flux objects that cover portions of the cell. In that case, [factor] can be less than 1.0. Regardless, the sum of [factor] values at a single cell for multiple mass-flux objects should add up to 1.0.

3.4 OUTPUT INFORMATION

As pointed out previously, three optional output files may be created depending on how the output options are specified in the input file to the TOB Package. These files are

1) a text file with the 3-letter extension ".OCN" which contains the calculated concentrations, and if requested, the residuals between the calculated and

observed values, at the user-specified observation locations. This text file is generated only if the concentration observation flag [inConcObs] is specified in the TOB input file as greater than zero.

2) a text file with the 3-letter extension ".MFX" which contains the calculated mass fluxes into or out of user-defined mass flux objects, and if requested, the residuals between the calculated and observed values. Each mass flux object is defined by a group of model cells containing external sinks/sources such as wells, rivers, drains, recharge, and general-head boundaries. This text file is generated only if the mass flux observation flag [inFluxObs] is specified in the TOB input file as greater than zero.

3) an unformatted (binary) file with the 3-letter extension ".PST" which contains the calculated concentrations and/or mass fluxes at user-defined observation points.

The records in the PST binary output file are in the form of
[cobsnam, TimeObs, CCal]
for concentration observations where cobsnam is the name of the concentration observation as a string of 12 characters, and TimeObs and CCal are real numbers indicating the observation time and the calculated concentration value.

The records in the PST binary output file are in the form of
[fobsnam, TimeFluxObs, FluxCal]
for mass-flux observations where fobsnam is the name of the mass-flux observation as a string of 12 characters, and TimeFluxObs and FluxCal are real numbers indicating the observation time and the calculated mass flux.

This output file is intended for post-processing purposes or for linkage with other modeling programs. The file is generated only if the output flag [inSaveObs] is specified in the TOB input file as greater than zero.

4 Simulating the Effect of Multi-Node Wells on Solute Transport

4.1 BACKGROUND

Halford and Hanson (2002) developed the drawdown-limited, Multi-Node Well (MNW) Package for MODFLOW. The MNW Package allows MODFLOW users to simulate wells that are screened in multiple aquifers or multiple layers in a single aquifer, and multiple nodes in a horizontal well. A MNW dynamically distributes the total prescribed flow among different layers (or nodes) under either extraction or injection conditions. In addition, intra-wellbore flow between layers are computed under zero-pumping conditions. A single hydraulic head value is determined for the entire wellbore. Although the MNW Package also has the ability to track potential mixes of a water-quality attribute, it is based on the prescribed constant concentrations and does not consider solute transport.

MT3DMS v5 adds the capability to simulate the effect of multi-aquifer or multi-node wells on solute transport. Compatible with the MNW Package, MT3DMS v5 computes a single composite concentration for the entire well borehole based on the flux-weighted concentrations of the injected fluid (if any) and the flow rates from different layers. The average composite concentration at the wellbore at any time is computed from:

$$C_{avg}^t = \frac{\left|Q_w^t\right|C_w^t + \sum_{i=1}^{N}\left|Q_i^t\right|C_i^t}{\left|Q_w^t\right| + \sum_{i=1}^{N}\left|Q_i^t\right|} \tag{4.1}$$

where

C_{avg}^t is the flux-averaged composite concentration inside the wellbore at time t;

Q_w^t is the total prescribed flow **into** the multi-node well (i.e., source) at time t;

C_w^t is the concentration of the injected source Q_w^t at time t;

Q_i^t is the flow rate at node (layer) i **discharging** into the multi-node well (i.e., sink) at time t;

C_i^t is the concentration associated with the sink outflow Q_i^t, i.e., the concentration in the aquifer at node (layer) i at time t; and

N is the total number of nodes (layers) that makes up the multi-node well.

It is noteworthy that the single composite concentration C_{avg}^t inside the wellbore is the actual concentration of the inflow that may go into individual nodes (layers) over which the multi-node well is screened. Further note that even if the total prescribed Q_w^t is zero, i.e., the multi-node well is under zero injection or pumping conditions, there is still potential intra-wellbore flow from layers (nodes) with higher heads to those with lower heads. Correspondingly, mass exchange may occur due to the intra-wellbore flow.

4.2 INPUT INSTRUCTIONS

To simulate the effect of multi-node wells on solute transport, the MNW Package must have been used with MODFLOW-2000 to save the sink/source information in the Flow-Transport link file that is produced after a MODFLOW-2000 run. Although the MNW Package can be used with both MODFLOW-96 and MODFLOW-2000, only version 6.2 or later of the Link-MT3DMS (LMT6) Package added to MODFLOW-2000 since version 1.12 (dated 9/8/2003) has the capability to save the MNW flow term for use by MT3DMS v5.

The input file for the LMT6 Package of MODFLOW-2000 must have the **OUTPUT_FILE_HEADER** option set to **"EXTENDED"**, as shown below:

```
OUTPUT_FILE_NAME     project.ftl
OUTPUT_FILE_UNIT     333
OUTPUT_FILE_HEADER   extended
OUTPUT_FILE_FORMAT   unformatted
```

which instructs MODFLOW-2000 to save the MNW flow term correctly in the flow-transport line file. For more information on the LMT6 Package, refer to Zheng et al. (2001).

If a MNW is a sink (extracting water from the well), it is not necessary to specify a concentration for the MNW. If assigned, it will be ignored anyway. A single composite concentration for all layers within a MNW is determined internally based on equation (1). If a MNW is a source (injecting water into the well), the user may specify any desired concentration for the MNW source. If it is not specified a concentration, the source is assumed, by default, to have a zero concentration. Also, regardless of how many layers (nodes) a MNW may be screened over, it is only necessary to assign the concentration for just one layer (node) within the wellbore of the MNW (all layers or nodes are equivalent). For a MNW under zero pumping or injection conditions, there is nothing to specify.

The input record for specifying the source concentration for a multi-node well in the SSM Package input file is as follows:

For Single-Species Simulation:

Record: K, I, J, CSS, iSSType
Format: 3I10, F10.0, I10

For Multi-Species Simulation:

Record: K, I, J, CSS, iSSType, (CSS(index),index=1,ncomp)
Format: 3I10, F10.0, I10, free

where I and J are the horizontal locations (row and column) and K is the vertical location (layer) of the MNW over which the MNW is screened. CSS is the source concentration if only a single species is simulated. For multi-species simulations, specify one concentration for each of NCOMP species. iSSType is the sink/source type code for MNW. The integer code of iSSType is 27 for MNW.

An example of the SSM input file with the MNW source:

```
HEADER_LINE     [dummy input, not used by MT3DMS v5]
     1000       [MXSS: Max # of Sinks/Sources]
        1       [NSS: # of point sources with specified conc.]
        1        24        24       100.        27   [K,I,J,CSS,iSSType]
```

In the above example, the multi-node well (iSSType=27) is located at two or more model cells, one of which must be at layer K=1, row I=24, and column J=24 with an input concentration of CSS=100. The total prescribed flow to the multi-node well is saved in the Flow-Transport link file produced by MODFLOW-2000.

4.3 OUTPUT INFORMATION

The location and the internally-determined composite concentration for each MNW is saved to the MT3DMS standard text output file generated after each run. The results are saved only at the end of each stress period or at the times specified by the user in the BTN input file. Alternatively, the user can specify a unit number for a MNW-specific output file. This unit can be specified in the second input record of the SSM input file as illustrated below:

Record: MXSS, ISSGOUT
Format: 2I10

where the input value `ISSGOUT` is the unit number for the MNW-specific output file. `ISSGOUT` must be a unique integer that has not be used elsewhere. After `ISSGOUT` is specified, a new line must be inserted to the Name File for the MT3DMS run, as follows:

```
DATA    ISSGOUT     OuputFileName
```

where `DATA` is the keyword not to be modified and `OutputFileName` is the user-specified name for the file to which the location and the internally-determined composite concentration of each MNW will be saved at every transport step.

5 Support for Additional MODFLOW-2000 Sink/Source Packages

In addition to the MNW Package, MT3DMS v5 provides the support for two more MODFLOW sink/source packages, i.e., the Drain with Return Flow (DRT) and Evapotranspiration with Segments (ETS), both documented in Banta (2000). Table 5.1 is an updated list of MODFLOW-2000 flow and sink/source packages and their support status in MT3DMS v5 (the bold typeface indicates those packages newly supported in this version). Interested users should refer to the README file distributed with other MT3DMS distribution files at the MT3D website for the most up-to-date information on the support status of other add-on packages to MODFLOW-2000.

The DRT Package simulates the re-injection of all or a portion of the outflow from a drain back into the aquifer. MT3DMS v5 automatically assigns the concentration at the outflow cell as that of the re-injected source at the return location. The user can specify the proportion of the return drain flow in the DRT input file to MODFLOW, but cannot alter the concentration of the return drain flow, which is controlled by the concentration in the aquifer. The DRT Package is useful in some applications, such as the use of aquifer water discharged into certain drain features directly for agricultural irrigation without any alternation to the water quality.

The ETS Package allows simulation of evapotranspiration with a user-defined relation between evapotranspiration rate and hydraulic head. The ETS provides additional capability and flexibility not available in the EVT Package, which has been available in MODFLOW since its initial release. In MT3DMS v5, the sink flow term produced by the ETS is handled exactly the same way as that produced by the EVT Package. Thus the input instructions for the EVT flow sink (Zheng and Wang, 1999) are directly applicable to the ETS flow sink.

It is noteworthy that in an effort to maintain backward compatibility with previous versions of MT3DMS, the Flow-Transport Link file produced by MODFLOW-2000 has two header options, **"standard"** and **"extended"** (see Zheng et al., 2001). As pointed out previously for the MNW Package, the header option of the Flow-Transport Link file must be set to **"extended"** in the input file to the Link-MT3DMS (LMT6) Package of MODFLOW-2000 for the flow terms of the DRT and ETS Packages to be saved appropriately for use in MT3DMS v5. This is also true for any subsequent MODFLOW-2000 sink/source packages that will be supported in the future.

The structure and contents of the Flow-Transport Link file used by MT3DMS v5 are specified in the Link-MT3DMS (LMT6) Package version 6.2 implemented in MODFLOW-2000 since version 1.12. Developers who wish to add support for a specific MODFLOW sink/source package should follow the same conventions and make appropriate changes to the Flow-Model-Interface (FMI) Package of MT3DMS. Use the pre-defined iSSType code if a sink/source type has been already defined in MT3DMS (Table 5.1). If not, use any unique integer code up to the maximum value of 100.

Table 5.1. List of MODFLOW packages and their support status by MT3DMS v5.

MODFLOW Package Name (documented in Harbaugh and others, 2000, or in references cited in that work)	File Type of MODFLOW-2000 Name File	MT3DMS iSSType Code for Sinks/Sources	Support Status in MT3DMS v5
Basic	BAS6	n/a	Yes
Block-Centered Flow	BCF6	n/a	Yes
Layer Property Flow	LPF1	n/a	Yes
Hydrogeologic Unit Flow	HUF2	n/a	Yes
Horizontal Flow Barrier	HFB	n/a	Yes
Time-Variant Specified Head Boundary	CHD	1	Yes
Well	WEL	2	Yes
Drain	DRN	3	Yes
River	RIV	4	Yes
General Head Dependent Boundary	GHB	5	Yes
Recharge	RCH	7*	Yes
Evapotranspiration	EVT	8*	Yes
Streamflow-Routing	STR	21	Yes
Reservoir	RES	22	Yes
Specified Flow and Head Boundary	FHB	23	Yes
Interbed Storage	IBS	24	No
Transient Leakage	TLK	25	No
Lake	LAK	26	No
Multi-Node Well	**MNW**	**27**	**Yes**
Drain with Return Flow	**DRT**	**28**	**Yes**
Evapotranspiration with Segments	**ETS**	**29**	**Yes**

*Note: these codes are for identification purposes only. Recharge and Evapotranspiration are considered areally distributed sinks/sources rather than point sinks/sources.

6 Simulating Zeroth-Order Reactions in Single- and Dual-Domain Systems

6.1 BACKGROUND

The standard MT3DMS code prior to version 5 includes only the first-order kinetic reactions in the Chemical Reaction (RCT) Package. However, zeroth-order reactions may be useful for describing certain types of biogeochemical decay or production. In addition, zeroth-order reactions are needed in direct simulation of groundwater ages (e.g., Goode, 1996) and calculation of parameter sensitivities (e.g., Tsai et al., 2003). Thus since version 5.1, zeroth-order reactions have been added to the Chemical Reaction Package in both single- and dual-domain formulations.

The governing solute transport equations incorporating the zeroth-order decay or production terms in both dissolved and sorbed phases can be written as

For equilibrium-controlled sorption

$$R\frac{\partial(\theta C)}{\partial t} = \frac{\partial}{\partial x_i}\left(\theta D_{ij}\frac{\partial C}{\partial x_j}\right) - \frac{\partial}{\partial x_i}(q_i C) + q_s C_s - \gamma_1\theta - \gamma_2\rho_b \qquad (6.1)$$

For rate-limited kinetic sorption

$$\frac{\partial(\theta C)}{\partial t} + \frac{\partial(\rho_b\overline{C})}{\partial t} = \frac{\partial}{\partial x_i}\left(\theta D_{ij}\frac{\partial C}{\partial x_j}\right) - \frac{\partial}{\partial x_i}(q_i C) + q_s C_s - \gamma_1\theta - \gamma_2\rho_b \qquad (6.2a)$$

$$\frac{\partial(\rho_b\overline{C})}{\partial t} = \beta\left(C - \frac{\overline{C}}{K_d}\right) - \gamma_2\rho_b \qquad (6.2b)$$

where γ_1 and γ_2 are the zeroth-order rate coefficients for the dissolved phase [ML^{-3}T^{-1}], and the sorbed phase [MM^{-1}T^{-1}], respectively. The conventions for both γ_1 and γ_2 are *positive values for decay* and *negative values for production*. Other terms in equations (6.1) and (6.2) have been defined in Zheng and Wang (1999), i.e., C is the dissolved concentration [ML^{-3}]; θ the porosity, dimensionless; D_{ij} the hydrodynamic dispersion coefficient tensor [L^{2}T^{-1}]; q_i the specific discharge (Darcy flux) [LT^{-1}]; q_s the volumetric flow rate per unit volume of aquifer representing fluid sources (positive) or sinks (negative) [T^{-1}]; C_s the concentration of the fluid source or sink [ML^{-3}]; ρ_b the

bulk density of the subsurface medium [ML^{-1}]; $\overline{C}$ the sorbed concentration [MM^{-1}]; K_d the distribution coefficient of linear sorption isotherm [L^{3}M^{-1}]; R the retardation factor [dimensionless]; and β the first-order rate coefficient between the dissolved and sorbed phases [T^{-1}].

In a dual-domain (dual-porosity) system, the governing transport equations incorporating linear equilibrium-controlled sorption and zeroth-order reactions can be written as

$$R_m \frac{\partial(\theta_m C_m)}{\partial t} + R_{im} \frac{\partial(\theta_{im} C_{im})}{\partial t} = \frac{\partial}{\partial x_i}\left(\theta_m D_{ij} \frac{\partial C_m}{\partial x_j}\right) - \frac{\partial}{\partial x_i}(q_i C_m) + q_s C_s \qquad (6.3a)$$

$$-\gamma_{1,m}\theta_m - \gamma_{1,im}\theta_{im} - \gamma_{2,m} f \rho_b - \gamma_{2,im}(1-f)\rho_b$$

$$R_{im} \frac{\partial(\theta_{im} C_{im})}{\partial t} = \zeta(C_m - C_{im}) - \gamma_{1,im}\theta_{im} - \gamma_{2,im}(1-f)\rho_b \qquad (6.3b)$$

where C_m and C_{im} are the dissolved concentrations in the mobile and immobile domains, respectively [ML^{-3}]; θ_m and θ_{im} are the porosities of the mobile and immobile domains, respectively [dimensionless]; $\gamma_{1,m}$ and $\gamma_{1,im}$ are the zeroth-order rate coefficients for the mobile-liquid and immobile-liquid phases, respectively [ML^{-3}T^{-1}]; $\gamma_{2,m}$ and $\gamma_{2,im}$ are the zeroth-order rate coefficients for the mobile-sorbed and immobile-sorbed phases, respectively [ML^{-3}T^{-1}]; f is the fraction of sorption sites in contact with the mobile fluid phase [dimensionless]; $R_m = 1 + f\rho_b K_d/\theta_m$ and $R_{im} = 1 + (1-f)\rho_b K_d/\theta_{im}$ are the retardation factors for the mobile and immobile domains, respectively [dimensionless]; and ζ is the first-order mass transfer rate coefficient between the mobile and immobile domains [T^{-1}].

6.2 IMPLEMENTATION AND INPUT INSTRUCTIONS

The implementation of zero-order reactions in MT3DMS is done similar to that of first-order reactions. The contributions from the zero-order reactions are directly incorporated into the coefficient matrix [A] and the right-hand-side matrix [RHS] of the linear system of equations resulting from the various solution schemes for the transport model (Zheng and Wang, 1999). In addition, the formulation for the concentration of sorbed phase or immobile-liquid phase is updated to account for the effect of zero-order reactions.

To invoke the zeroth-order reaction option, set the input variable IREACT = **100** in the input file for the RCT Package (see Appendix A). Then, in the input records where the user would normally enter the first-order rate coefficients RC1 for the dissolved phase and RC2 for the sorbed phase, specify the zeroth-order rate coefficients. When sorption is not simulated, set the input variable RC2 to zero. Even though RC2 is not used when sorption is not present, it nevertheless must be specified in the input file.

6.3 BENCHMARK TEST PROBLEMS

6.3.1. Zero-Order Production in a Uniform Flow Field

This benchmark problem involves solute transport in a uniform flow field in the presence of zero-order production. The initial condition for the one-dimensional problem is defined by zero concentration everywhere in the flow field, i.e.,

$$C(x,0) = 0 \qquad x \geq 0 \tag{6.4}$$

and the boundary conditions are specified mass flux (third-type) on the left and specified concentration gradient on the right:

$$qC - D\frac{\partial C}{\partial x}\bigg|_{x=0} = \begin{cases} q_0 C_0 & 0 < t \leq t_0 \\ 0 & t > t_0 \end{cases}$$

$$\frac{\partial C}{\partial x}\bigg|_{x=\infty} = 0, \qquad t > 0 \tag{6.5}$$

A flow and transport model consisting of 101 columns, 1 row and 1 layer is set up to solve the test problem. Other parameters used for the test problem are listed below:

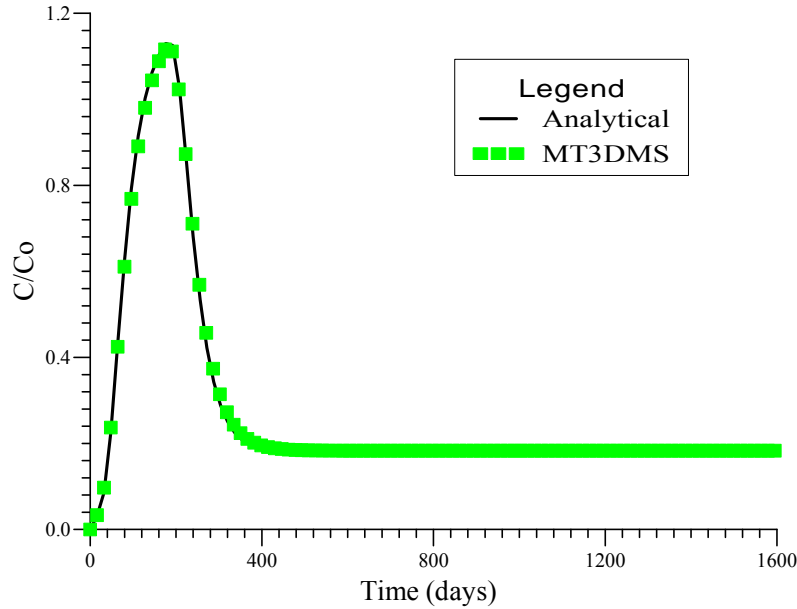| |
|---|
| Cell width along rows (Δx) = 0.16 m |
| Cell width along columns (Δy) = 1 m |
| Layer thickness (Δz) = 1 m |
| Seepage velocity (v) = 0.1 m/day |
| Longitudinal dispersivity = 1.0 m |
| Porosity (θ) = 0.37 |
| Zeroth-order production rate (γ_1) = $- 2 \times 10^{-3}$ mg/L day^{-1} |
| Pulse source duration (t_o) = 160 days |
| Simulation time (t) = 1,000 days |
| Observation location (x) = 8 m |

The numerical solution is obtained using the fully implicit finite-difference option for all transport terms. The transport step size is fixed at 1 day for ease of comparison. The relative concentration closure criterion for the GCG matrix solver is set at 10^{-6} and the MIC pre-conditioner is selected. It is noteworthy that in MT3DMS input conventions, decay coefficients are defined as positive while production rate coefficients as negative.

Figure 6.1 shows the concentration breakthrough curve at the observation point as calculated by MT3DMS (symbols). A corresponding analytical solution is available from

Toride et al. (1995) (solid line). It is evident that the numerical solution by MT3DMS closely agrees with the analytical solution of Toride et al. (1995).

Figure 6.1. Comparison of analytical and numerical solutions for the test problem with zeroth-order reaction.

6.3.2. Zero-Order Production in a Dual-Domain System

This benchmark problem is designed to test the capabilities of MT3DMS to simulate solute transport in a dual-domain system in the presence of zero-order production with or without linear sorption. A general analytical solution for one-dimensional solute transport in a dual-domain system, implemented in the computer code CXTFIT2, is available from Toride et al. (1995). The one-dimensional problem considered in this section involves the following initial condition for both mobile-liquid and immobile-liquid phases:

$$C(0,0) = C_0$$
$$C(x,0) = 0 \qquad x > 0$$

(6.6)

and a first-type (constant-concentration) boundary condition on the left and a second-type (constant concentration gradient) on the right:

$$C(0,t) = \begin{cases} C_0 & 0 < t \le t_0 \\ 0 & t > t_0 \end{cases}$$

$$\frac{\partial C(\infty, t)}{\partial x} = 0 \qquad t > 0$$

(6.7)

28

A numerical flow and transport model consisting of 401 columns, 1 row and 1 layer is used to solve the problem for comparison with the analytical solution for the same initial and boundary conditions as described above. In the flow model, the first and last columns are constant-head boundaries. Arbitrary hydraulic heads are used to establish the required uniform hydraulic gradient. In the transport model, the first column is a constant-concentration boundary with a relative concentration of one. The last column is set sufficiently far away from the source to approximate an infinite one-dimensional flow domain as assumed in the analytical solution. The model parameters used in the simulation are listed below:

Cell width along rows $(\Delta x) = 2.5$ m

Cell width along columns $(\Delta y) = 1$ m

Layer thickness $(\Delta z) = 1$ m

Specific discharge $(q) = 0.06$ m/day

Longitudinal dispersivity $= 10$ m

Porosity of mobile domain $(\theta_m) = 0.2$

Porosity of immobile domain $(\theta_{im}) = 0.05$

Bulk density $(\rho_b) = 4$ g/mL

Distribution coefficient $(K_d) = 0.25$ mL/g

First-order mass transfer rate between
the mobile and immobile domains $(\zeta) = 10^{-3}$ day^{-1}

Fraction of sorption sites
in contact with mobile water $(f) = 0.8$

Source duration $(t_0) = 1,000$ days

Simulation time $(t) = 10,000$ days

Observation location $(x) = 200$ m

Three cases are evaluated with the following zero-order production coefficients and retardation factors:

Case 1: $R_m = R_{im} = 5, \quad \gamma_1 = 0; \; \gamma_2 = -10^{-3}$ g/mL day^{-1};

Case 2: $R_m = R_{im} = 5; \quad \gamma_1 = \gamma_2 = -5 \times 10^{-4}$ g/mL day^{-1};

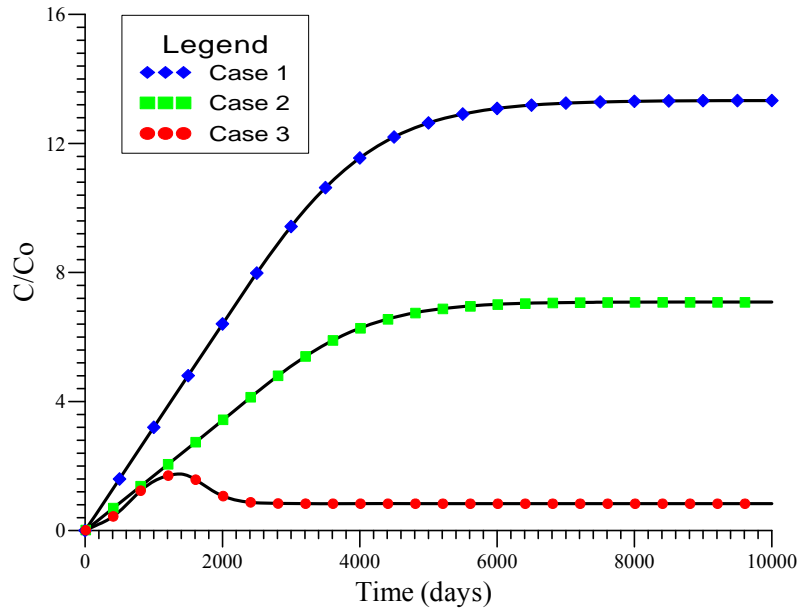Case 3: $R_m = R_{im} = 1 \; (K_d = 0); \; \gamma_1 = -10^{-3}$ g/mL day^{-1}; $\gamma_2 = 0$.

where Case 3 is equivalent to no sorption.

By default it is assumed in MT3DMS that the fraction of sorption sites in contact with mobile water (f) is equal to the ratio of mobile to total porosities so that the retardation factor for the mobile domain is identical to that for the immobile domain. In

addition, it is assumed that the rate constant for the zero-order production is the same for both mobile and immobile domains, i.e., $\gamma_1 = \gamma_{1,m} = \gamma_{1,im}$ and $\gamma_2 = \gamma_{2,m} = \gamma_{2,im}$. However, different rate constants can be specified for the dissolved (liquid) and solid (sorbed) phases.

All cases are solved using the third-order TVD scheme for the advection term and the fully implicit finite-different option for all other terms. The closure criterion for the GCG solver is set at 10^{-6} and the MIC pre-conditioner is selected. For ease of comparison, a fixed transport step size of 1 day is used for all cases.

A close match between the analytical (solid lines) and numerical solutions (symbols) is obtained for all cases (Figure 6.2). Note that only the concentrations of the mobile-liquid phase are plotted. Other three phases, i.e., mobile-sorbed, immobile-liquid, and immobile-sorbed, are tracked internally by MT3DMS that also computes and maintains a global mass budget for all four phases. Negligible mass balance errors are obtained for all the cases considered in this example.

Fig. 6.2. Comparison of the calculated concentrations with the analytical solutions for the one-dimensional dual-domain test problem. The analytical solutions are shown in solid lines with the numerical solutions in different symbols. The reaction parameters for the 3 cases are given in the text.

7 Using the Component-Dependent Diffusion Option

7.1 BACKGROUND

Prior to Version 5.2, the molecular diffusion coefficient can only be specified on a layer-by-layer basis (i.e., one uniform molecular diffusion coefficient for the entire model layer). Moreover, all solute components are assumed to have the same diffusion coefficient. However, in a multi-component simulation, it is possible that different ions and solutes may have noticeably different molecular diffusion coefficients. In addition, the molecular diffusion coefficient may also vary spatially in each layer of the model domain. Thus a new option has been added to Version 5.2 to allow the inclusion of three-dimensional component dependent diffusion coefficients in the transport simulation. The option is implemented through the Dispersion (DSP) Package and the modifications made to the MT3DMS code are limited to within the DSP Package only.

7.2 INPUT INSTRUCTIONS

To activate the *component-dependent diffusion* option, a keyword input record must be inserted to the beginning of the Dispersion (DSP) input file. The symbol $ in the *first* column of an input line signifies a keyword input record containing one or more predefined keywords. Above the keyword input record, comment lines marked by the symbol # in the first column are allowed. Comment lines are processed but have no effect on the simulation. Furthermore, blank lines are also acceptable above the keyword input record. Below the keyword input record, the format of the DSP input file must remain unchanged from the previous versions except for the diffusion coefficient as explained below. If no keyword input record is specified, the input file remains backward compatible with all previous versions of MT3DMS.

The predefined keyword for the component-dependent diffusion option is **MultiDiffusion**. The keyword is case insensitive so "MultiDiffusion" is equivalent to either "Multidiffusion" or "multidiffusion". If this keyword is specified in the keyword input record that has been inserted into the beginning of the DSP input file, the component-dependent diffusion option has been activated and the user needs to specify one diffusion coefficient for *each mobile solute component* and *at each model cell*. This is done by specifying one mobile component at a time, from the first component to the last component (MCOMP). For each mobile component, the real array reader utility (RARRAY) is used to input the 3-D diffusion coefficient array, one model layer at a time. Given below in Table 7.1 is an example of the Dispersion Package input file with the component-dependent diffusion option activated.

Table 7.1. An example DSP input file with the component-dependent diffusion option.

```
#
# DSP input file with keyword line indicated by $ sign in column 1
#
$ MultiDiffusion
        0       10.                             AL-LAYER 1
        0       10.                             AL-      2
        0       10.                             AL-      3
        0       10.                             AL-      4
        0       10.                             AL-      5
        0       10.                             AL-      6
        0       10.                             AL-      7
        0       10.                             AL-      8
        0        .3                             TRPT
        0        .3                             TRPV
        0       1.0                             DMCOEF  L1 Comp-1
        0       1.0                             DMCOEF  L2
        0       1.0                             DMCOEF  L3
        0       1.0                             DMCOEF  L4
        0       1.0                             DMCOEF  L5
        0       1.0                             DMCOEF  L6
        0       1.0                             DMCOEF  L7
        0       1.0                             DMCOEF  L8
        0        .0                             DMCOEF  L1 Comp-2
        0        .0                             DMCOEF  L2
        0        .0                             DMCOEF  L3
        0        .0                             DMCOEF  L4
        0        .0                             DMCOEF  L5
        0        .0                             DMCOEF  L6
        0        .0                             DMCOEF  L7
        0        .0                             DMCOEF  L8
        0        .1                             DMCOEF  L1 Comp-3
        0        .1                             DMCOEF  L2
        0        .1                             DMCOEF  L3
        0        .1                             DMCOEF  L4
        0        .1                             DMCOEF  L5
        0        .1                             DMCOEF  L6
        0        .1                             DMCOEF  L7
        0        .1                             DMCOEF  L8
```

8 Simulating the Effect of a Recirculation Well

8.1 INTRODUCTION

A new option has been added to MT3DMS since Version 5.2 to simulate the effect of a recirculation well on solute transport. A recirculation well is defined as an injection well whose input concentration is equal to the output concentration of an extraction well. This option is implemented in the Sink/Source Mixing (SSM) Package for the type-2 point sources, i.e., injection wells.

To use the *recirculation well* option, set the input concentration (CSS) of the injection well to an negative integer code (IC), i.e.,

$$CSS = -IC$$

with the absolute value of the integer code (IC) given as

$$IC = NCOL \times NROW \times (K-1) + NCOL \times (I-1) + J$$

where NCOL and NROW are the total numbers of columns and rows, respectively, in the model; and K, I, J are the layer, row, and column indexes of the model cell (extraction well) whose concentration is to be used as the input concentration of the recirculation well. For a multi-component transport simulation, the IC value must be specified for the concentrations of different components [CSSMS].

8.2 ILLUSTRATIVE EXAMPLE

As an example, let's assume an extraction well at K = 1, I = 16, and J = 21 (see Figure 8.1). The total numbers of layers, rows and columns in the model are NLAY = 1, NROW = 31 and NCOL = 46. Now let's specify two recirculation wells, one at (K, I, J) = (1, 5, 16) and the other at (K, I, J) = (1, 27, 16). The input concentration to both recirculation wells should be the same as that of the extraction well at (K, I, J) = (1, 16, 21). The IC value for both recirculation wells is
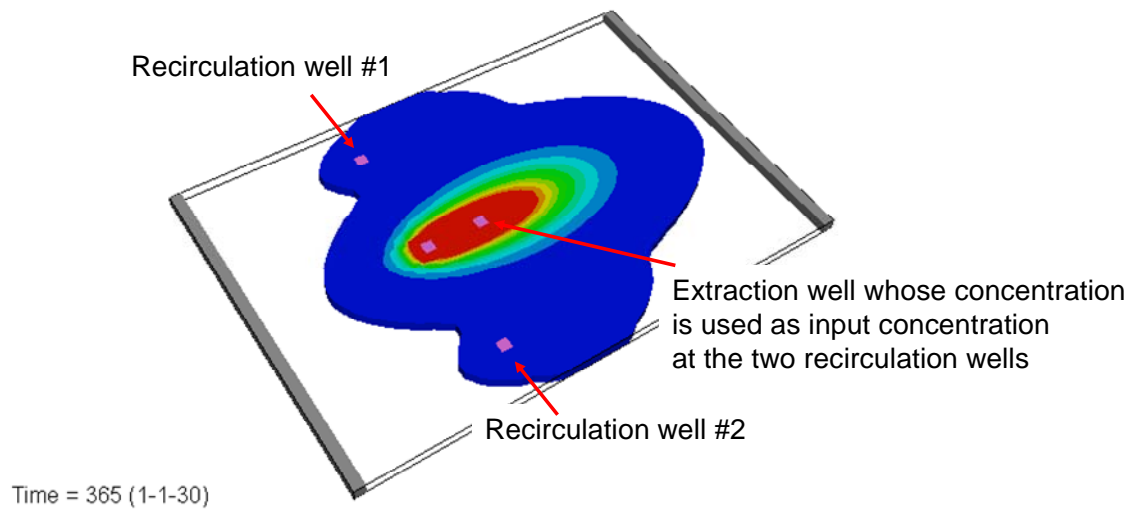
$$IC = 46 \times 31 \times (1-1) + 46 \times (16-1) + 21 = 711$$

Thus to specify the two recirculation wells for the MT3DMS SSM Package, we have

```
K   I   J      CSS          ISSType
1   5   16     -711         2
1   27  16     -711         2
```

where K, I, J are the location indexes of the recirculation wells and ISSType is the MT3DMS sink/source type code (ISSTYPE=2 for the well).

Note that to obtain an accurate solution for the recirculation well option, it is suggested that the maximum outer iteration number of the GCG solver be set to greater than 1 to ensure the concentrations at the extraction and recirculation wells have converged during the iterations.
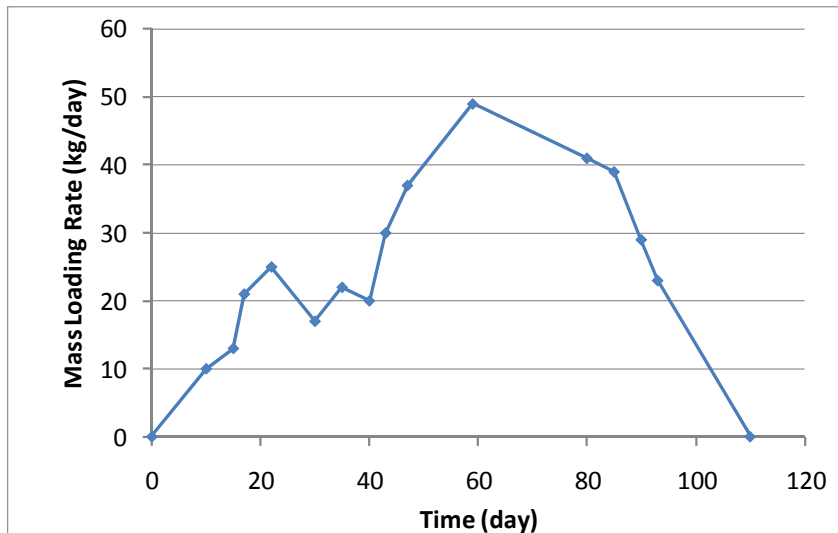

Figure 8.1. Illustration of the recirculation well option.

9 Arbitrary Time-Varying Mass Loading Source or Boundary Condition

9.1 INTRODUCTION

With MT3DMS Version 5.3, it is possible to specify an arbitrarily defined time-varying mass loading source or boundary condition (see, for example, Figure 9.1). In all previous versions, an arbitrary mass loading curve like Figure 9.1 must be discretized into several stress periods, each of which is characterized by a single average mass loading rate. However, a new capability has been added to Version 5.3 to allow the complete mass loading distribution to be specified as a source or boundary condition without being restricted by the definition of stress periods. The new capability is accomplished through a new package called the Hydrocarbon Spill Source (HSS) Package as documented by Zheng et al. (2010).

Figure 9.1. Illustration of an arbitrarily defined time-varying mass loading source or boundary condition.

The HSS package was developed to provide a seamless linkage to MT3DMS for the Hydrocarbon Spill Screening Model (HSSM) (Weaver, 1996). HSSM is a computer code that simulates the vertical migration of a light non-aqueous phase liquid (LNAPL) contaminant through the unsaturated zone and the formation of an oil lens on the water table. It also determines the rate of contaminant mass flux dissolved from the LNAPL

source into groundwater. Through the HSS package, the effect of one or more such LNAPL sources is incorporated into 3-D advective-dispersive-reactive transport simulation using MT3DMS Version 5.3 or later. Moreover, the HSS package can be used to accommodate any user-defined time-varying mass loading sources or boundary conditions in MT3DMS simulations. For more information on the HSS package and complete input instructions, refer to Zheng et al. (2010).

9.2 ILLUSTRATIVE EXAMPLE

As an example, consider an exponentially decaying mass loading source, given by $C(t)=1000e^{-0.01t}$, as illustrated in Figure 9.2. The source is applied to a two-dimensional uniform flow field. Further assume the transport model contains only two stress periods: the first period (1-365 days) with the exponential source and the second period (365-730 days) without any source. Without the HSS package, an average concentration must be calculated for Stress Period 1 and used in the simulation, which will lead to an inaccurate concentration solution (see the dashed line in Figure 9.3 for the concentration breakthrough at the source location). A fix would be to subdivide Stress Period 1 into smaller stress periods, which can improve the solution accuracy but is computationally inefficient (see the blue solid line in Figure 9.3 obtained with Stress Period 1 subdivided into 365 one-day stress periods). The HSS package provides a satisfactory solution to this dilemma by allowing the user to define arbitrary time-varying mass loading sources independent of the stress periods. The concentration solution with the use of the HSS package is also shown in Figure 9.3 as solid dots. It can be seen that the solution based on the HSS package only with one stress period for the first 365 days is as accurate as the reference solution without using the HSS package but under much smaller stress periods.
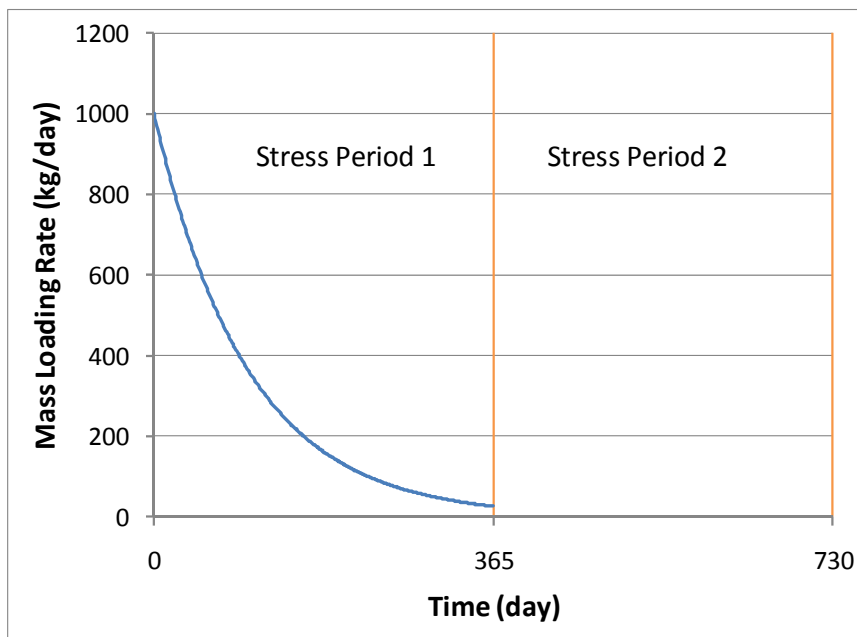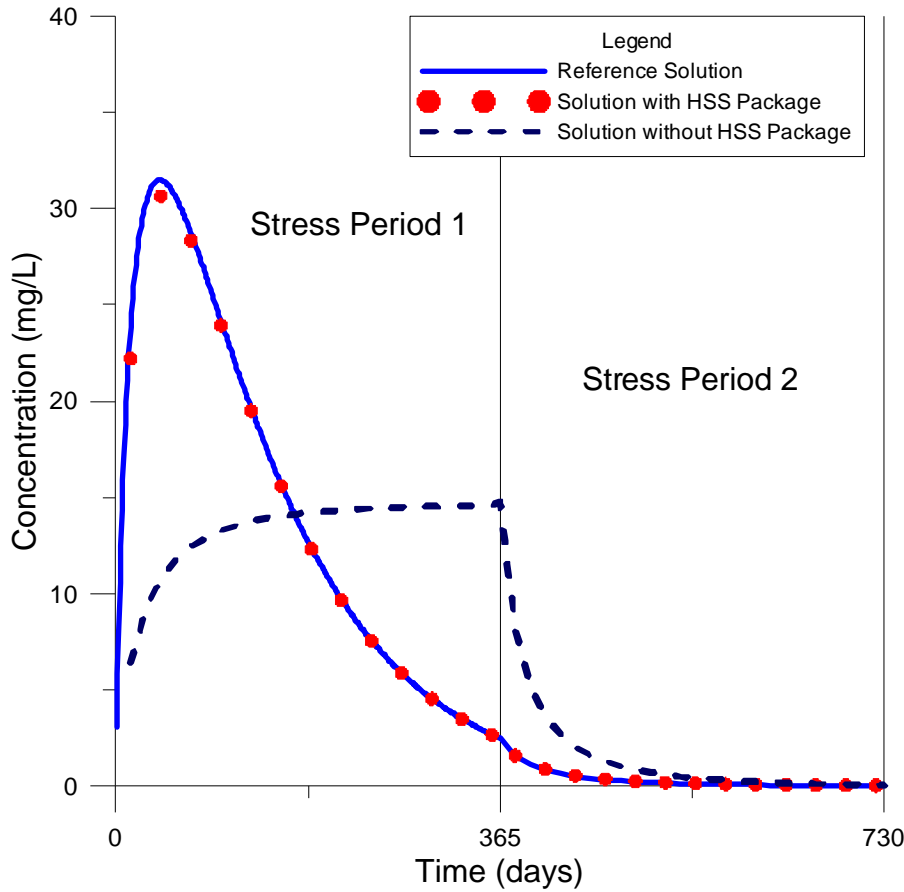

Figure 9.2. An exponentially decaying mass loading source for the illustrative example.

36

Figure 9.3. Comparison of calculated concentration breakthrough curves for the illustrative example.

The solution based on the HSS package is obtained with a "source definition file" as shown in Table 9.1. The first column of the file specifies the time, t, the second column the size, and the third column the mass loading rate at time t. The size can be adjusted to place a single source into multiple model cells. A default value of zero means that the mass loading source is placed at a single model cell, as specified in the input file to the HSS package. See Zheng et al. (2010) for complete input instructions.

Table 9.1 Example of the "source definition file" used by the HSS package

0	0	1000.00
1	0	990.05
2	0	980.20
3	0	970.45
......		
362	0	26.78
363	0	26.52
364	0	26.25
365	0	25.99

10 Running MT3DMS in Steady-State Mode

10.1 INTRODUCTION

A significant change to MT3DMS with Version 5.3 is the addition of a steady-state transport simulation option. This steady-state transport option may be useful for long-term transport simulation runs in which the concentration field reaches an equilibrium state with the net change in total mass equal to zero. For example, hydrocarbon contaminants from a constant source will initially create a growing plume in the contaminated aquifer. At some point, however, the mass released from the source may approach the mass consumed by the biodegradation process. When this happens, the plume might stop expanding and approach a steady state. Also, for the direct groundwater age simulation discussed in a previous section, it is usually the goal of the simulation to obtain a steady-state concentration solution under a zeorth-order production term. Such a steady-state concentration distribution can then be interpreted as the groundwater age distribution.

A steady-state transport solution can be obtained by running the transient transport model for a sufficiently long period of time until the concentration distributions between two successive transport steps cease to change within a predetermined limit. This is simple and easy to do with MT3DMS and it does not require any changes to the computer code. However, it is not computationally efficient to run a transient transport simulation for a long time for the sake of obtaining a steady-state solution. This is especially true if tens of thousands of years are needed to reach the steady state, as in some groundwater age simulations. Thus, it is useful to have a steady-state option for transport simulation, even though under most circumstances transient solutions are the objective of transport modeling studies.

It is noteworthy that the steady-state transport option is implemented in MT3DMS only with the fully implicit finite-difference solution option. This is because the particle tracking based Eulerian-Lagrangian techniques (MOC/MMOC/HMOC) and the third-order TVD scheme require the restriction of transport steps which are no longer used in the steady-state solution. Also, without the need for temporal discretization, the numerical dispersion error proportional to the transport step size is eliminated in a steady-state solution. However, the numerical dispersion error resulting from spatial discretization is still present in the steady-state solution. The magnitude of the numerical dispersion error depends on the Pelect number of the transport problem being simulated. Care should be taken to ensure that the steady-state solution meets the accuracy requirement. If not, it may be necessary to apply finer spatial discretization or run transient simulations with more accurate solution techniques such as TVD.

10.2 INPUT INSTRUCTIONS

To run a steady-state transport simulation with MT3DMS since Version 5.3, follow these two steps:

1. Select the fully implicit finite difference method as the solution option by setting the input variable MIXELM to zero in the input file to the ADV Package.

2. In the input record #A21 of the input file to the BTN Package, insert the keyword **SSTATE** after column 31 of the input line as illustrated below.

Standard transient transport simulation
```
A21         PERLEN     NSTP       TSMULT
Format      F10.0      I10        F10.0
```

Steady-state transport simulation
```
A21         PERLEN     NSTP       TSMULT       [SSflag]
Format      F10.0      I10        F10.0        [free]
```

where PERLEN is the transport simulation time for the current stress period; NSTP and TSMULT are the number of time steps and the time step multiplier used in the flow solution, respectively; and SSFlag is an optional flag for the steady-state transport option.

If SSFlag is set to **SSTATE** (which can be any combination of lower or capital letters as it is case insensitive), the steady-state transport simulation is automatically activated. **SSTATE** can be inserted anywhere after column 31 of the input line. There must be a comma or space after the keyword to separate it from any label following the keyword.
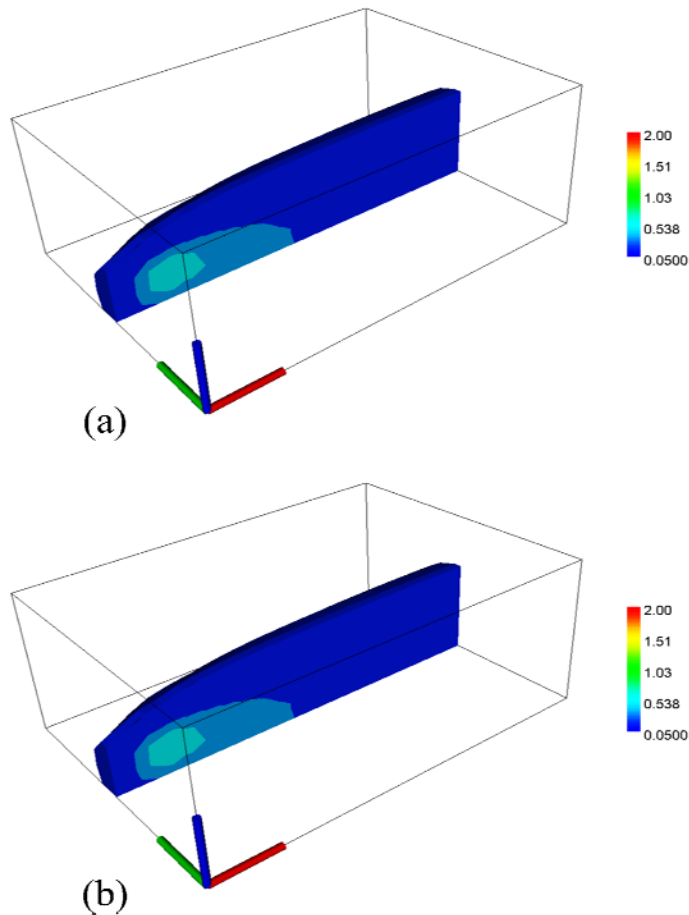
Normally, a steady-state transport solution is associated with a steady-state flow solution, i.e., NSTP=1. If so, PERLEN is the total simulation time assigned to the steady-state transport run. The total simulation time has no physical meaning in a steady-state run other than as a multiplier for mass budgets. For simplicity, PERLEN is best set to unity for a steady-state transport simulation run.

Table 10.1. An example input file for the BTN Package with the state-state transport option activated

```
3D TRANSPORT IN UNIFORM FLOW FIELD WITH CONTINUOUS POINT SOURCE
A test problem for steady-state transport simulation
        8          15         21          1          1          1
 DAY  FT  LB
[transport options]
 0 0 0 0 0 0 0
         0        10.
         0        10.
         0        80.
......
        T          1                                     !CHKMAS,NPRMAS
        1.         1          1.       SState    !PERLEN,NSTP,TSMULT,[SSflag]
        0.      10000         1.        3.5      !DT0,MXSTRN,TTSMULT,TTSMAX
```

10.3 ILLUSTRATIVE EXAMPLE

Using the benchmark problem #7 (p. 145-146) in the original MT3DMS manual (Zheng and Wang, 1999) as an illustrative example, we run the transient transport model for a sufficiently long time until the steady state is reached. The concentration distribution at the end of simulation is shown Figure 10.1(a). In a separate run, the steady-state transport option is activated and the transport solution is shown in Figure 10.1(b). It can be seen that the two solutions are essentially identical. However, the steady-state option requires only a fraction of the runtime needed by the simulation in transient mode.

Figure 10-1. Comparison of simulated concentration distributions by running the transient model for a sufficiently long time (a) and by running MT3DMS in the steady-state mode (b).

11 Using MT3DMS for Heat Transport Simulation[1]

11.1 INTRODUCTION

Heat is increasingly being recognized as an excellent groundwater tracer because of its usefulness in identifying multiple hydrologic processes and recent availability of improved temperature sensors and measurement technologies (Anderson, 2005). The mathematical equivalency between the heat and solute transport equations has been established in numerous studies (e.g., Healy and Ronan, 1996; Kim et al., 2005; Prommer and Stuyfzand, 2005; Thorne et al., 2006). As a result, MT3DMS can be used directly and efficiently to simulate heat transport with simple variable conversion. Figure 11.1 shows a heat plume from an injection of cool water into warmer groundwater, as part of a pre-modeling exercise to guide the subsequent actual field heat tracer test.

The use of MT3DMS for heat transport modeling is based on the assumption that the changes in fluid density and viscosity induced by temperature variation are negligible so that flow and transport simulations can be decoupled for greater computational efficiency. However, when the temperature causes significant changes in fluid density and viscosity, the SEAWAT code (Langevin et al., 2007) is more suitable because it can account for the effects of variable fluid density and viscosity during heat transport. Ma and Zheng (2010) have conducted a series of numerical experiments to quantify the conditions under which the effects of variable density and viscosity can be neglected without any significant loss of computational accuracy.

11.2 ANALOGY BETWEEN SOLUTE AND HEAT TRANSORT

The general form of solute transport equation solved by MT3DMS is as follows (see Zheng and Wang, 1999):

$$(1 + \frac{\rho_b}{\theta} K_d) \frac{\partial(\theta C)}{\partial t} = \nabla \cdot (\theta \mathbf{D} \cdot \nabla C) - \nabla \cdot (\mathbf{q} C) + q_s C_s + \sum R_n \tag{11.1}$$

where C is the dissolved concentration; t is time; θ is the porosity of the subsurface medium; ρ_b the is bulk density of the rock matrix, $\rho_b = \rho_s(1 - \theta)$, where ρ_s is the density of solid; K_d is the distribution coefficient; $\mathbf{q}$ is the specific discharge vector); $\mathbf{D}$ is the hydrodynamic dispersion coefficient tensor, $\mathbf{D} = \mathbf{D_{mech}} + D_m$, where $\mathbf{D_{mech}}$ is the mechanical dispersion coefficient tensor and D_m is the molecular diffusion coefficient; q_s is the volumetric flow rate per unit volume of the aquifer representing fluid sources or

---

[1] This section is adapted from Zheng (2009) and Ma and Zheng (2010).

sinks; C_s is the concentration of the source or sink flux; and $\sum R_n$ is the chemical reaction term.

Heat transport in the aquifer is generally governed by heat convection along with the fluid phase, heat conduction and dispersion through the fluid and aquifer sediment, and heat exchange between the aqueous phase and the aquifer sediment. Assuming thermal equilibrium between the aquifer sediment and fluid, and neglecting the vapor phase in the aquifer, the general equation to describe the heat transport in the aquifer system can be expressed as (e.g., Healy and Ronan, 1996; Anderson, 2005; Thorn et al., 2006; Langevin et al., 2007):

$$\left(1 + \frac{1-\theta}{\theta}\frac{\rho_s}{\rho_w}\frac{c_s}{c_w}\right)\frac{\partial(\theta T)}{\partial t} = \nabla \cdot \left(\theta\left(\frac{\kappa_o}{\theta\rho_w c_w} + \mathbf{D_{mech}}\right) \cdot \nabla T\right) - \nabla \cdot (\mathbf{q}T) + q_s T_s \qquad (11.2)$$

where T is the temperature; c_s is the specific heat capacity of the solid; c_w is the specific heat capacity of the fluid; ρ_w is the density of the fluid; κ_o is the bulk thermal conductivity of the rock matrix, $\kappa_o = \theta\kappa_w + (1 - \theta)\kappa_s$, where κ_w and κ_s are the fluid and solid phase thermal conductivities, respectively; and T_S is the source temperature.

The analogy between solute transport and heat transport equations is apparent from a comparison of equation (11.1) with equation (11.2). The first term on the right side in equation (11.2), representing transport of heat by conduction and thermal dispersion, is analogous to molecular diffusion and mechanical dispersion in solute transport equation (11.1). The second term in equation (11.2), which represents transport of heat by groundwater flow, is analogous to the advection term in solute transport equation (11.1). The third term in equation (11.2) represents thermal sinks/sources, and is analogous to fluid solute sinks/sources in equation (11.1).

Based on the analogy between solute transport and heat transport equations, variables for the MT3DMS solute transport simulator can be re-interpreted for heat transport. The solute concentration becomes the temperature. The distribution coefficient (K_d) for solute transport is replaced by the thermal distribution factor which can be calculated for temperature using the following equation:

$$K_d = \frac{c_s}{c_w\rho_w} \qquad (11.3)$$

The molecular diffusion in solute transport can be represented by the thermal conduction in heat transport. Thus, the molecular diffusion coefficient reinterpreted for heat transport is:

$$D_m = \frac{\kappa_o}{\theta\rho_w c_w} \qquad (11.4)$$

This thermal conduction term is referred to as the bulk thermal diffusivity. Additional information on using solute transport simulation for heat transport modeling can be found in Kim et al. (2005) and Thorn et al. (2006).
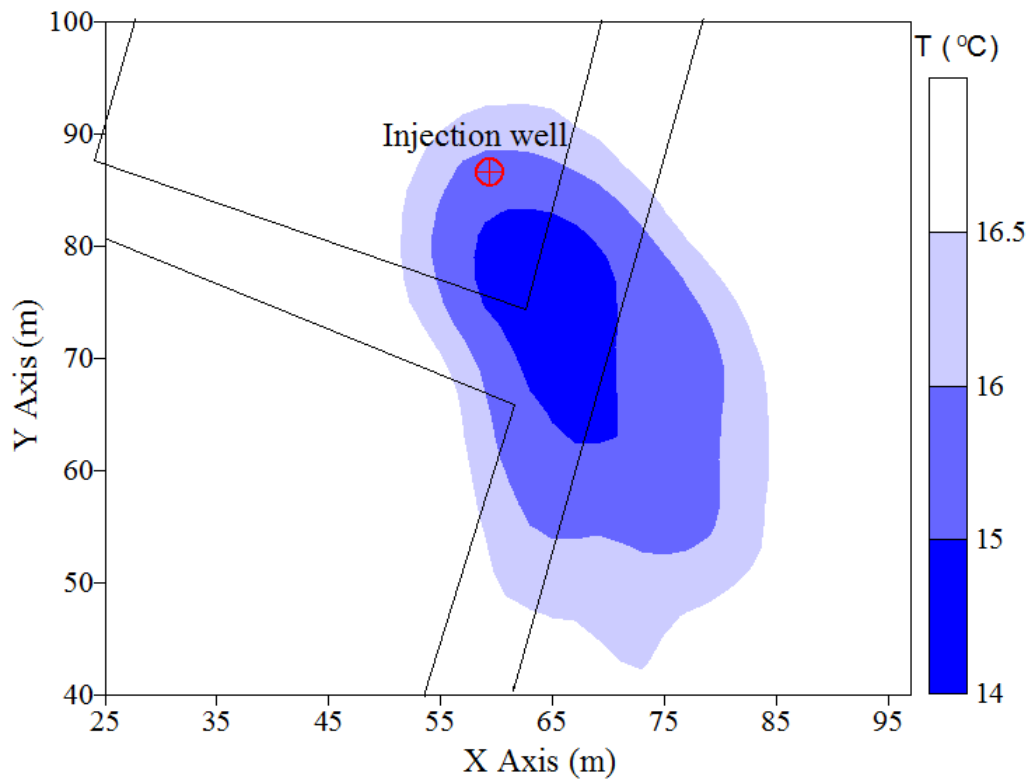
Figure 11.1 Simulation of a heat plume following injection of cool water into warmer groundwater using MT3DMS at the Hanford 300A site in Washington State (after Ma and Zheng, 2010).

References

Anderson, M.P. 2005. Heat as a ground water tracer. *Ground Water,* 43(6): 951–968.

Banta, E. R. 2000. MODFLOW-2000, The U.S. Geological Survey Modular Ground-Water Model—Documentation of Packages for Simulating Evapotranspiration with a Segmented Function (ETS1) and Drains with Return Flow (DRT1): U.S. Geological Survey Open-File Report 00–466, Reston, Virginia.

Goode, D. J. 1996. Direct simulation of groundwater age, *Water Resour. Res.*, 32(2): 289-296.

Halford, K. J. and R.T. Hanson. 2002. User Guide for the Drawdown-Limited, Multi-Node Well (MNW) Package for the U.S. Geological Survey's Modular Three-Dimensional Finite-Difference Ground-Water Flow Model, Versions MODFLOW-96 and MODFLOW-2000: US Geological Survey Open-File Report 02-293, Reston, Virginia.

Harbaugh, A.W., E.R. Banta, M.C. Hill, and M.G. McDonald. 2000, MODFLOW-2000, the U.S. Geological Survey Modular Ground-Water Model—User guide to modularization concepts and the ground-water flow processes: U.S. Geological Survey Open-File Report 00-92, 121 p.

Harbaugh, A.W. 2005. MODFLOW-2005, the U.S. Geological Survey modular ground-water model – the Ground-Water Flow Process: U.S. Geological Survey Techniques and Methods 6-A16.

Healy, R.W., and A.D. Ronan. 1996. Documentation of computer program VS2DH for simulation of energy transport in variably saturated porous media — modification of the U.S. Geological Survey's computer program VS2DT. Water-Resources Investigations Report 96-4230. U.S. Geological Survey, Denver, Colorado.

Hill, M.C., E.R. Banta, A.W. Harbaugh, and E.R. Anderman. 2000. MODFLOW-2000, the U.S. Geological Survey Modular Ground-Water Model—User guide to the observation, sensitivity, and parameter-estimation processes and three post-processing programs: U.S. Geological Survey Open-File Report 00-184, 210 p.

Kim J., Y. Park, and T.C. Harmon. 2005. Real-time model parameter estimation for analyzing transport in porous media. *Ground Water Monitoring & Remediation*, 25(2): 78–86.

Langevin, C.D., D.T. Thorne, A.M. Dausman, M.C. Sukop, and W. Guo. 2007. SEAWAT Version 4: A computer program for simulation of multi-species solute and heat transport: U.S. Geological Survey Techniques and Methods Book 6, Chapter A22, 39 p.

Ma, R. and C. Zheng. 2010. Effects of density and viscosity in modeling heat as a groundwater tracer, *Ground Water*, doi: 10.1111/j.1745-6584.2009.00660.x.

Prommer, H., and P.J. Stuyfzand. 2005. Identification of temperature-dependent water quality changes during a deep well injection experiment in a pyritic aquifer. *Environ. Sci. Technol.,* 39(7): 2200-2209.

Thorne, D., C.D. Langevin, and M.C. Sukop. 2006. Addition of simultaneous heat and solute transport and variable fluid viscosity to SEAWAT. *Computers & Geosciences,* 32: 1758–1768.

Toride, E., F.J. Leij, and M. Th. Van Genuchten. 1995. The CXTFIT Code for Estimating Transport Parameters from Laboratory or Field Tracer Experiments, Version 2.0, Research Report 137, U.S. Salinity Laboratory, Riverside, California.

Tsai, F. T.-C., N.-Z. Sun, and W. W.-G. Yeh. 2003. Global-local optimization for parameter structure identification in three-dimensional groundwater modeling, *Water Resour. Res.,* 39(2), 1043, doi:10.1029/2001WR001135.

Weaver, J. 1996. The Hydrocarbon Spill Screening Model (HSSM) Volume 1 User's Guide (Version 1.1 Rev. October 1996). Athens, Georgia: U.S. Environmental Protection Agency, Office of Research and Development.

Zheng, C. 2009. Recent developments and future directions for MT3DMS and related transport codes, *Ground Water*, 47(5): 620-625, doi: 10.1111/j.1745-6584.2009.00602.x.

Zheng, C., and P.P. Wang. 1999. MT3DMS: A modular three-dimensional multi-species transport model for simulation of advection, dispersion and chemical reactions of contaminants in groundwater systems; Documentation and user's guide: Contract report SERDP-99-1: U.S. Army Engineer Research and Development Center, Vicksburg, MS, 169 p.

Zheng, C., M.C. Hill, and P.A. Hsieh. 2001. MODFLOW-2000, The U.S. Geological Survey Modular Ground-Water Model—User Guide to the LMT6 Package, the Linkage with MT3DMS for Multi-Species Mass Transport Modeling: US Geological Survey Open-File Report 01-82, Reston, Virginia.

Zheng, C., J. Weaver, and M. Tonkin. 2010. MT3DMS, A Modular Three-dimensional Multispecies Transport Model—User Guide to the Hydrocarbon Spill Source (HSS) Package. Athens, Georgia: U.S. Environmental Protection Agency.

Appendix A
Changes to the Input Instructions since MT3DMS v5.0

I. CHANGES FOR THE BTN PACKAGE

Prior to version 5.0

    A5   Record: TRNOP(10)

           Format: 10L2

- TRNOP contains logical flags (true or false) for major transport and solution options.

Changes since version 5.0

    A5   Record: Dummy input line

           Format: n/a

- TRNOP is no longer used since MT3DMS v5.0. However, a dummy input line must still be specified in the input file. A blank line is acceptable.

Prior to version 5.3

    A21   Record: PERLEN, NSTP, TSMULT

            Format: F10.0,   I10,   F10.0

- PERLEN is the length of simulation time for the current stress period; NSTP and TSMULT are the number of time steps and the time step multiplier used in the flow solution, respectively.

Changes since version 5.3

    A21   Record: PERLEN, NSTP, TSMULT, **[SSFlag]**

            Format: F10.0,   I10,   F10.0,   **[free]**

- SSFlag is an optional flag to indicate whether the steady-state transport option should be activated. The option is activated if SSFlag is set to the keyword SSTATE, which can be any combination of lower or capital letters.

II. CHANGES FOR THE DSP PACKAGE

Prior to version 5.2

    C1   Array: AL(NCOL,NROW)   (One array for each layer).
            Reader: RARRAY
- AL is the longitudinal dispersivity, α_L, for every cell of the model grid (unit: L).

……
    C4   Array: DMCOEF(NLAY)
            Reader: RARRAY
- DMCOEF is the effective molecular diffusion coefficient (unit: L^2T^{-1}). Set DMCOEF = 0 if the effect of molecular diffusion is considered unimportant. Each value in the array corresponds to one model layer.

Changes since version 5.2

    C0   Record: One or more optional keywords.
            Format: [Free]
- The only keyword currently supported is **MultiDiffusion**, case insensitive. **The keyword must be specified in an input line with the $ sign in the first column**. The keyword input record is optional.

    C1   Array: AL(NCOL,NROW)   (One array for each layer).
            Reader: RARRAY
- No change.

……
If no keyword is defined:
    C4   Array: DMCOEF(NLAY)
            Reader: RARRAY
- DMCOEF is the effective molecular diffusion coefficient (unit: L^2T^{-1}). Set DMCOEF = 0 if the effect of molecular diffusion is considered unimportant. Each value in the array corresponds to one model layer. **Enter one array for all solute components.**

If keyword [MultiDiffusion] is defined:
    C4   Array: DMCOEF(NCOL,NROW)   (One array for each layer)
            Reader: RARRAY
- DMCOEF is the effective molecular diffusion coefficient (unit: L^2T^{-1}). Set DMCOEF = 0 if the effect of molecular diffusion is considered unimportant. Each value in the array corresponds to one model cell. **Repeat the input for each mobile component.**

III. CHANGES FOR THE SSM PACKAGE

Prior to version 5.0

> D1   Record: FWEL, FDRN, FRCH, FEVT, FRIV, FGHB, (FNEW(n), n=1,4)
>   Format: 10L2
>   • These are logical flags (true or false) to indicate whether various MODFLOW sink/source packages are used in the flow simulation.

Changes since version 5.0

> D1   Record: Dummy input line
>   Format: n/a
>   • These logical flags are no longer needed as the status of various flow sink/source packages is obtained by MT3DMS through the Flow-Transport Link File produced by MODFLOW. However, a dummy input line must still be specified in the input file. A blank line is acceptable.

Prior to version 5.0

> D2   Record: MXSS
>   Format: I10
>   • MXSS is the maximum allowed number of all point sinks and sources included in the flow model. This input value is used for the memory allocation purpose only.

Changes since version 5.0

> D2   Record: MXSS, **ISSGOUT**
>   Format: 2I10
>   • ISSGOUT is the unit number for an optional output file to save the calculated flux-averaged composite concentrations at multi-node wells. The name of the output file must be specified through the Name File as in **"DATA   ISSGOUT   FileName".**

Prior to version 5.0

> D8   Record: KSS, ISS, JSS, CSS, ISSTYPE, [CSSMS(n), n=1, NCOMP]
>   Format: 3I10, F10.0, I10, [free]
>   • This record is used to specify the point sources of specified concentrations. Note that those fluid sources included in the MODFLOW simulation but not specified herein are assumed to have zero concentration by default.

Changes since version 5.0

D8   Record: KSS, ISS, JSS, CSS, ISSTYPE, [CSSMS(n), n=1, NCOMP]
       Format: 3I10, F10.0, I10, [free]

- No changes in the meaning and number of the input variables as well as their input format; however, the range of allowed ISSTYPE codes has increased since the initial release of MT3DMS (see Table 5.1).

  Since version 5.2, users can specify the input concentration of an injection well (ISSTYPE=2), i.e., CSS or CSSMS, as a negative integer code (IC). The absolute value of the integer code is the single cell location indicator of the extraction well whose output concentration is used as the input concentration for the injection well. For an extraction well located at layer K, row I, and column J, IC is computed as,

  $$IC = NCOL \times NROW \times (K-1) + NCOL \times (I-1) + J .$$

  where NCOL and NROW are the total numbers of columns and rows.

IV. CHANGES FOR THE RCT PACKAGE

Prior to version 5.1

E1   Record: ISOTHM, IREACT, IRCTOP, IGETSC
          Format: 4I10
- IREACT is an integer indicating the type of kinetic reaction simulated:
IREACT =0, no kinetic rate reaction is simulated;
          =1, first-order irreversible reaction.

Changes since version 5.1

E1   Record: ISOTHM, **IREACT,** IRCTOP, IGETSC
          Format: 4I10
- IREACT is an integer indicating the type of kinetic reaction simulated:
IREACT =0, no kinetic rate reaction is simulated;
          =1, first-order irreversible reaction;
          =100, zeroth-order reaction (decay or production).

Prior to version 5.1

E5   Array: RC1(NCOL, NROW) (one array for each layer)
          Reader: RARRAY
- RC1 is the first-order reaction rate coefficient for the dissolved (liquid) phase (unit: T^{-1}). If a dual-domain system is simulated, the rate coefficients for the liquid phase in the mobile and immobile domains are assumed to be equal.

Changes since version 5.1

E5   Array: RC1(NCOL, NROW) (one array for each layer)
          Reader: RARRAY
- If IREACT=1 (first-order kinetic reactions),
RC1 is the first-order rate coefficient for the dissolved (liquid) phase (T^{-1}). If a dual-domain system is simulated, the rate coefficients for the liquid phase in the mobile and immobile domains are assumed equal.
If IREACT=100 (zeroth-order decay or production),
RC1 is the zeroth-order reaction rate coefficient for the dissolved (liquid) phase $(ML^{-3}T^{-1})$ (*positive for decay and negative for production).* If a dual-domain system is simulated, the rate coefficients for the liquid phase in the mobile and immobile domains are assumed equal.

<u>Prior to version 5.1</u>

E6   Array: RC2(NCOL, NROW)   (one array for each layer)
        Reader: RARRAY

- RC2 is the first-order reaction rate coefficient for the sorbed (solid) phase (T^{-1}). If a dual-domain system is simulated, the rate coefficients for the sorbed phase in the mobile and immobile domains are assumed equal. Note that RC2 must be still specified although it is not used, if no sorption is included in the simulation.

<u>Changes since version 5.1</u>

E6   Array: RC2(NCOL, NROW)   (one array for each layer)
        Reader: RARRAY

- If IREACT=1 (first-order kinetic reactions),
RC2 is the first-order rate coefficient for the sorbed (solid) phase (T^{-1}). If a dual-domain system is simulated, the rate coefficients for the sorbed phase in the mobile and immobile domains are assumed equal.
If IREACT=100 (zeroth-order decay or production),
RC2 is the zeroth-order reaction rate coefficient for the sorbed (solid) phase ($MM^{-1}T^{-1}$) (*positive for decay and negative for production)*. If a dual-domain system is simulated, the rate coefficients for the liquid phase in the mobile and immobile domains are assumed equal.