

# BW

This is a file is written using a pre-release version of the meta-flopy-scripting package

Using flopy version 3.2.10

```
In [ ]: import flopy
import numpy as np
from numpy import rec
```

## flopy.modflow

```
In [ ]: # Name of model.  this string will be used to name the modflow input that are created with
# write_model. (the default is 'modflowtest')
modelname = 'BW'
# Extension for the namefile (the default is 'nam')
namefile_ext = 'nam'
# Version of modflow to use (the default is 'mf2005').
version = 'mf2005'
# The name of the executable to use (the default is 'mf2005').
exe_name = 'mf2005.exe'
structured = True
# Unit number for the list file (the default is 2).
listunit = 2
# Model workspace.  directory name to create model data sets. (default is the present working
# directory).
model_ws = '.'
# Location for external files (default is none).
external_path = None
# Print additional information to the screen (default is false).
verbose = False

modflow = flopy.modflow.mf.Modflow(modelname=modelname, namefile_ext=namefile_ext, version=version,
                                   exe_name=exe_name, structured=structured, listunit=listunit,
                                   model_ws=model_ws, external_path=external_path, verbose=verbose)
```

## DIS

[illegible]

```
In [ ]:
```

```
# The model object (of type :class:`flopymodflow.mf.modflow`) to which this package will be added.  
model = modflow  
  
# The ibound array (the default is 1).  
ibound = np.broadcast_to([[[[ 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1,  
    1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1,  
    1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1,  
    1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1,  
    1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, -1]]]], (1, 60, 100))  
  
# An array of starting heads (the default is 1.0).  
strt = 10.0  
  
# Indicates whether or not packages will be written as free format.  
ifrefm = True  
  
# Indication of whether model is cross sectional or not (the default is false).  
ixsec = False  
  
# Flag indicating that flows between constant head cells should be calculated (the default is  
#   false).  
ichflg = False  
  
# Percent discrepancy that is compared to the budget percent discrepancy continue when the solver  
#   convergence criteria are not met. execution will unless the budget percent discrepancy is  
#   greater than stoper (default is none). modflow-2005 only  
stoper = None  
  
# Head value assigned to inactive cells (default is -999.99).  
hnflo = 1e+30  
  
bas6 = flopymodflow.mfbas.ModflowBas(model=model, ibound=ibound, strt=strt, ifrefm=ifrefm,  
                                         ixsec=ixsec, ichflg=ichflg, stoper=stoper, hnflo=hnflo)
```

[illegible]

[illegible]

[illegible]

[illegible]

[illegible]

[illegible]



[illegible]

```
# Is specific storage unless the storagecoefficient option is used. when storagecoefficient is
# used, ss is confined storage coefficient. (default is 1.e-5).
ss = 0.0
# Is specific yield. (default is 0.15).
sy = 0.0
# Is the vertical hydraulic conductivity of a quasi-three-dimensional confining bed below a layer.
```

```

# (default is 0.0). note that if an array is passed for vkcb it must be of size (nlay, nrow,
# ncol) even though the information for the bottom layer is not needed.
vkcb = 0.0
# Is a combination of the wetting threshold and a flag to indicate which neighboring cells can
# cause a cell to become wet. (default is -0.01).
wetdry = 0.0
# Indicates that variable ss and ss parameters are read as storage coefficient rather than specific
# storage. (default is false).
storagecoefficient = False
# indicates that vertical conductance for an unconfined cell is computed from the cell thickness
# rather than the saturated thickness. the constantcv option automatically invokes the
# nocvcorrection option. (default is false).
constantcv = False
# Indicates that layers having a negative laytyp are confined, and their cell thickness for
# conductance calculations will be computed as strt-bot rather than top-bot. (default is false).
thickstrt = False
# Indicates that vertical conductance is not corrected when the vertical flow correction is
# applied. (default is false).
nocvcorrection = False
# turns off the vertical flow correction under dewatered conditions. this option turns off the
# vertical flow calculation described on p. 5-8 of usgs techniques and methods report 6-a16 and
# the vertical conductance correction described on p. 5-18 of that report. (default is false).
novfc = False

lpf = flopy.modflow.mflpf.ModflowLpf(model=model, laytyp=laytyp, layavg=layavg, chani=chani,
    layvka=layvka, laywet=laywet, ipakcb=ipakcb, hdry=hdry,
    iwdflg=iwdflg, wetfct=wetfct, iwetit=iwetit, ihdwet=ihdwet,
    hk=hk, hani=hani, vka=vka, ss=ss, sy=sy, vkcb=vkcb,
    wetdry=wetdry, storagecoefficient=storagecoefficient,
    constantcv=constantcv, thickstrt=thickstrt,
    nocvcorrection=nocvcorrection, novfc=novfc)

```

## WEL

```

In [ ]: # The model object (of type :class:`flopy.modflow.mf.modflow`) to which this package will be added.
model = modflow
# A flag that is used to determine if cell-by-cell budget data should be saved. if ipakcb is
# non-zero cell-by-cell budget data will be saved. (default is 0).
ipakcb = 53
# Dictionary of boundaries each well is defined through definition of layer (int), row (int),
# column (int), flux (float). the simplest form is a dictionary with a lists of boundaries for
# each stress period, where each list of boundaries itself is a list of boundaries. indices of
# the dictionary are the numbers of the stress period. this gives the form of:
stress_period_data = {0: rec.array([(0, 0, 0, 0.0875), (0, 1, 0, 0.0875), (0, 2, 0, 0.0875),\
    (0, 3, 0, 0.0875), (0, 4, 0, 0.0875), (0, 5, 0, 0.0875), (0, 6, 0,\
    0.0875), (0, 7, 0, 0.0875), (0, 8, 0, 0.0875), (0, 9, 0, 0.0875), (0,\
    10, 0, 0.0875), (0, 11, 0, 0.0875), (0, 12, 0, 0.0875), (0, 13, 0,\
    0.0875), (0, 14, 0, 0.0875), (0, 15, 0, 0.35 ), (0, 16, 0, 0.35 ), (0,\
    17, 0, 0.35 ), (0, 18, 0, 0.35 ), (0, 19, 0, 0.35 ), (0, 20, 0, 0.35\
    ), (0, 21, 0, 0.35 ), (0, 22, 0, 0.35 ), (0, 23, 0, 0.35 ), (0, 24, 0,\
    0.35 ), (0, 25, 0, 0.35 ), (0, 26, 0, 0.35 ), (0, 27, 0, 0.35 ), (0,\
    28, 0, 0.35 ), (0, 29, 0, 0.35 ), (0, 30, 0, 0.35 ), (0, 31, 0, 0.35\
    ), (0, 32, 0, 0.35 ), (0, 33, 0, 0.35 ), (0, 34, 0, 0.35 ), (0, 35, 0,\
    0.35 ), (0, 36, 0, 0.35 ), (0, 37, 0, 0.35 ), (0, 38, 0, 0.35 ), (0,\
    39, 0, 0.35 ), (0, 40, 0, 0.35 ), (0, 41, 0, 0.35 ), (0, 42, 0, 0.35\
    ), (0, 43, 0, 0.35 ), (0, 44, 0, 0.35 ), (0, 45, 0, 0.35 ), (0, 46, 0,\
    0.35 ), (0, 47, 0, 0.35 ), (0, 48, 0, 0.35 ), (0, 49, 0, 0.35 ), (0,\
    50, 0, 0.35 ), (0, 51, 0, 0.35 ), (0, 52, 0, 0.35 ), (0, 53, 0, 0.35\
    ), (0, 54, 0, 0.35 ), (0, 55, 0, 0.35 ), (0, 56, 0, 0.35 ), (0, 57, 0,\
    0.35 ), (0, 58, 0, 0.35 ), (0, 59, 0, 35.    )]), dtype=[('k',\
    '<i8'), ('i', '<i8'), ('j', '<i8'), ('flux', '<f4')])}
# If none the default well datatype will be applied (default is none).
dtype = np.dtype([('k', '<i8'), ('i', '<i8'), ('j', '<i8'), ('flux', '<f4')])
# Package options (default is none).
options = []
binary = False

wel = flopy.modflow.mfwel.ModflowWel(model=model, ipakcb=ipakcb,
    stress_period_data=stress_period_data, dtype=dtype,
    options=options, binary=binary)

```

## PCG

```

In [ ]: # The model object (of type :class:`flopymodflow.mf.modflow`) to which this package will be added.
model = modflow
# Maximum number of outer iterations. (default is 50)
mxiter = 30
# Maximum number of inner iterations. (default is 30)
iter1 = 30
# Flag used to select the matrix conditioning method. (default is 1). specify npcond = 1 for
#   modified incomplete cholesky. specify npcond = 2 for polynomial.
npcond = 1
# Is the head change criterion for convergence. (default is 1e-5).
hclose = 0.0001
# Is the residual criterion for convergence. (default is 1e-5)
rclose = 0.0001
# Is the relaxation parameter used with npcond = 1. (default is 1.0)
relax = 1.0
# Is only used when npcond = 2 to indicate whether the estimate of the upper bound on the maximum
#   eigenvalue is 2.0, or whether the estimate will be calculated. nbpol = 2 is used to specify the
#   value is 2.0; for any other value of nbpol, the estimate is calculated. convergence is
#   generally insensitive to this parameter. (default is 0).
nbpol = 0
# Solver print out interval. (default is 0).
iprpcg = 0
# If mutpcg = 0, tables of maximum head change and residual will be printed each iteration. if
#   mutpcg = 1, only the total number of iterations will be printed. if mutpcg = 2, no information
#   will be printed. if mutpcg = 3, information will only be printed if convergence fails. (default
#   is 3).
mutpcg = 1
# Is the steady-state damping factor. (default is 1.)
damp = 1.0
# Is the transient damping factor. (default is 1.)
dampt = 0.0
# Is a flag that determines what happens to an active cell that is surrounded by dry cells.
#   (default is 0). if ihcofadd=0, cell converts to dry regardless of hcof value. this is the
#   default, which is the way pcg2 worked prior to the addition of this option. if ihcofadd<>0,
#   cell converts to dry only if hcof has no head-dependent stresses or storage terms.
ihcofadd = 0

pcg = flopymodflow.mfpcg.ModflowPcg(model=model, mxiter=mxiter, iter1=iter1, npcond=npcond,
                                     hclose=hclose, rclose=rclose, relax=relax, nbpol=nbpol,
                                     iprpcg=iprpcg, mutpcg=mutpcg, damp=damp, dampt=dampt,
                                     ihcofadd=ihcofadd)

```

OC

```
In [ ]: # The model object (of type :class:`flopy.modflow.mf.modflow`) to which this package will be added.
model = modflow
# Is a code for the format in which heads will be printed. (default is 0).
ihedfm = 0
# Is a code for the format in which drawdown will be printed. (default is 0).
iddnfm = 0
# Is a character value that specifies the format for saving heads. the format must contain 20
# characters or less and must be a valid fortran format that is enclosed in parentheses. the
# format must be enclosed in apostrophes if it contains one or more blanks or commas. the
# optional word label after the format is used to indicate that each layer of output should be
# preceded with a line that defines the output (simulation time, the layer being output, and so
# forth). if there is no record specifying chedfm, then heads are written to a binary
# (unformatted) file. binary files are usually more compact than text files, but they are not
# generally transportable among different computer operating systems or different fortran
# compilers. (default is none)
chedfm = None
# Is a character value that specifies the format for saving drawdown. the format must contain 20
# characters or less and must be a valid fortran format that is enclosed in parentheses. the
# format must be enclosed in apostrophes if it contains one or more blanks or commas. the
# optional word label after the format is used to indicate that each layer of output should be
# preceded with a line that defines the output (simulation time, the layer being output, and so
# forth). if there is no record specifying cddnfm, then drawdowns are written to a binary
# (unformatted) file. binary files are usually more compact than text files, but they are not
# generally transportable among different computer operating systems or different fortran
# compilers. (default is none)
cddnfm = None
# Is a character value that specifies the format for saving ibound. the format must contain 20
# characters or less and must be a valid fortran format that is enclosed in parentheses. the
# format must be enclosed in apostrophes if it contains one or more blanks or commas. the
# optional word label after the format is used to indicate that each layer of output should be
# preceded with a line that defines the output (simulation time, the layer being output, and so
# forth). if there is no record specifying cboufm, then ibounds are written to a binary
# (unformatted) file. binary files are usually more compact than text files, but they are not
# generally transportable among different computer operating systems or different fortran
# compilers. (default is none)
cboufm = None
# Save results in compact budget form. (default is true).
compact = True
# Dictionary key is a tuple with the zero-based period and step (iperoc, itsoc) for each
# print/save option list. if stress_period_data is none, then heads are saved for the last time
# step of each stress period. (default is none) the list can have any valid modflow oc
# print/save option:      print head      print drawdown      print budget      save head      save
# drawdown      save budget      save ibound      the lists can also include (1) ddreference in
# the list to reset      drawdown reference to the period and step and (2) a list of layers
# for print head, save head, print drawdown, save drawdown, and      save ibound.
stress_period_data = {(0, 0): ['save head', 'save budget']}
label = 'LABEL'

oc = flopy.modflow.mfoc.ModflowOc(model=model, ihedfm=ihedfm, iddnfm=iddnfm, chedfm=chedfm,
                                   cddnfm=cddnfm, cboufm=cboufm, compact=compact,
                                   stress_period_data=stress_period_data, label=label)
```

**flopy.mt3d**

```

In [ ]: # Name of model.  this string will be used to name the modflow input that are created with
# write_model. (the default is 'mt3dtest')
modelname = 'BW'
# Extension for the namefile (the default is 'nam')
namefile_ext = 'nam'
# This is a flopy modflow model object upon which this mt3dms model is based. (the default is none)
modflowmodel = modflow
ftlfilename = 'mt3d_link.ftl'
# Version of mt3dms to use (the default is 'mt3dms').
version = 'mt3dms'
# The name of the executable to use (the default is 'mt3dms.exe').
exe_name = 'mt3dms.exe'
structured = True
# Unit number for the list file (the default is 2).
listunit = None
# Model workspace.  directory name to create model data sets. (default is the present working
# directory).
model_ws = '.'
# Location for external files (default is none).
external_path = None
# Print additional information to the screen (default is false).
verbose = False
# (default is true).
load = True
# (default is 0)
silent = 0

mt3d = flopy.mt3d.mt.Mt3dms(modelname=modelname, namefile_ext=namefile_ext,
                             modflowmodel=modflowmodel, ftlfilename=ftlfilename, version=version,
                             exe_name=exe_name, structured=structured, listunit=listunit,
                             model_ws=model_ws, external_path=external_path, verbose=verbose,
                             load=load, silent=silent)

```

**BTN**

```

In [ ]: # The model object (of type :class:`flopymt3dms.mt.mt3dms`) to which this package will be added.
model = mt3d
# The total number of chemical species in the simulation. (default is none, will be changed to 1 if
# sconc is single value)
ncomp = 2
# The total number of 'mobile' species (default is 1). mcomp must be equal or less than ncomp.
mcomp = 2
# The name of unit for time (default is 'd', for 'days'). used for identification purposes only.
tunit = 'D'
# The name of unit for length (default is 'm', for 'meters'). used for identification purposes
# only.
lunit = 'M'
# The name of unit for mass (default is 'kg', for 'kilograms'). used for identification purposes
# only.
munit = 'KG'
# The effective porosity of the porous medium in a single porosity system, or the mobile porosity
# in a dual-porosity medium (the immobile porosity is defined through the chemical reaction
# package. (default is 0.25).
prsimty = 0.34999999940395355
# The icbund array specifies the boundary condition type for solute species (shared by all
# species). if icbund = 0, the cell is an inactive concentration cell; if icbund < 0, the cell is
# a constant-concentration cell; if icbund > 0, the cell is an active concentration cell where
# the concentration value will be calculated. (default is 1).
icbund = 1
# Sconc is the starting concentration for the first species. to specify starting concentrations
# for other species in a multi-species simulation, include additional keywords, such as sconc2,
# sconc3, and so forth.
sconc = 0.0
# The value for indicating an inactive concentration cell. (default is 1e30).
cinact = -1000.0
# The minimum saturated thickness in a cell, expressed as the decimal fraction of its thickness,
# below which the cell is considered inactive. (default is 0.01).
thkmin = 0.01
# A flag/format code indicating how the calculated concentration should be printed to the standard
# output text file. format codes for printing are listed in table 3 of the mt3dms manual. if
# ifmtcn > 0 printing is in wrap form; ifmtcn < 0 printing is in strip form; if ifmtcn = 0
# concentrations are not printed. (default is 0).
ifmtcn = 0
# A flag/format code indicating how the number of particles should be printed to the standard
# output text file. the convention is the same as for ifmtcn. (default is 0).
ifmtnp = 0
# A flag/format code indicating how the calculated retardation factor should be printed to the
# standard output text file. the convention is the same as for ifmtcn. (default is 0).
ifmtrf = 0
# A flag/format code indicating how the distance-weighted dispersion coefficient should be printed
# to the standard output text file. the convention is the same as for ifmtcn. (default is 0).
ifmtdp = 0
# A logical flag indicating whether the concentration solution should be saved in an unformatted
# file. (default is true).
savucn = True
# A flag indicating (i) the frequency of the output and (ii) whether the output frequency is
# specified in terms of total elapsed simulation time or the transport step number. if nprs > 0
# results will be saved at the times as specified in timprs; if nprs = 0, results will not be
# saved except at the end of simulation; if nprs < 0, simulation results will be saved whenever
# the number of transport steps is an even multiple of nprs. (default is 0).
nprs = -999999999
# The total elapsed time at which the simulation results are saved. the number of entries in timprs
# must equal nprs. (default is none).
timprs = None
obs = None
nprobs = 0
chkmas = True
nprmas = 1
ssflag = 288 * [' ']
dt0 = 0.0
mxstrn = 50000
ttsmult = 1.0
ttsmax = 0.0
species_names = []
# Sconc is the starting concentration for the first species. to specify starting concentrations
# for other species in a multi-species simulation, include additional keywords, such as sconc2,
# sconc3, and so forth.
sconc2 = 12.0

btn = flopymt3d.mtbtn.Mt3dBtn(model=model, ncomp=ncomp, mcomp=mcomp, tunit=tunit, lunit=lunit,
                               munit=munit, prsimty=prsimty, icbund=icbund, sconc=sconc,
                               cinact=cinact, thkmin=thkmin, ifmtcn=ifmtcn, ifmtnp=ifmtnp,
                               ifmtrf=ifmtrf, ifmtdp=ifmtdp, savucn=savucn, nprs=nprs,
                               timprs=timprs, obs=obs, nprobs=nprobs, chkmas=chkmas, nprmas=nprmas,
                               ssflag=ssflag, dt0=dt0, mxstrn=mxstrn, ttsmult=ttsmult,
                               ttsmax=ttsmax, species_names=species_names, sconc2=sconc2)

```

```

In [ ] : # The model object (of type :class:`flopy.mt3d.mt.mt3dms`) to which this package will be added.
model = mt3d
# Mixelm is an integer flag for the advection solution option. mixelm = 0, the standard
# finite-difference method with upstream or central-in-space weighting, depending on the value of
# nadvfd; = 1, the forward-tracking method of characteristics (moc); = 2, the backward-tracking
# modified method of characteristics (mmoc); = 3, the hybrid method of characteristics (hmoc)
# with moc or mmoc automatically and dynamically selected; = -1, the third-order tvd scheme
# (ultimate).
mixelm = -1
# Percel is the courant number (i.e., the number of cells, or a fraction of a cell) advection will
# be allowed in any direction in one transport step. for implicit finite-difference or
# particle-tracking-based schemes, there is no limit on percel, but for accuracy reasons, it is
# generally not set much greater than one. note, however, that the percel limit is checked over
# the entire model grid. thus, even if percel > 1, advection may not be more than one cell's
# length at most model locations. for the explicit finite-difference or the third-order tvd
# scheme, percel is also a stability constraint which must not exceed one and will be
# automatically reset to one if a value greater than one is specified.
percel = 1.0
# Mxpart is the maximum total number of moving particles allowed and is used only when mixelm = 1
# or 3.
mxpart = 0
# Nadvfd is an integer flag indicating which weighting scheme should be used; it is needed only
# when the advection term is solved using the implicit finite-difference method. nadvfd = 0 or
# 1, upstream weighting (default); = 2, central-in-space weighting.
nadvfd = 0
# Itrack is a flag indicating which particle-tracking algorithm is selected for the
# eulerian-lagrangian methods. itrack = 1, the first-order euler algorithm is used. = 2, the
# fourth-order runge-kutta algorithm is used; this option is computationally demanding and may be
# needed only when percel is set greater than one. = 3, the hybrid first- and fourth-order
# algorithm is used; the runge-kutta algorithm is used in sink/source cells and the cells next to
# sinks/sources while the euler algorithm is used elsewhere.
itrack = None
# Is a concentration weighting factor between 0.5 and 1. it is used for operator splitting in the
# particle-tracking-based methods. the value of 0.5 is generally adequate. the value of wd may
# be adjusted to achieve better mass balance. generally, it can be increased toward 1.0 as
# advection becomes more dominant.
wd = None
# Is a small relative cell concentration gradient below which advective transport is considered
dceps = None
# Nplane is a flag indicating whether the random or fixed pattern is selected for initial placement
# of moving particles. if nplane = 0, the random pattern is selected for initial placement.
# particles are distributed randomly in both the horizontal and vertical directions by calling a
# random number generator (figure 18b). this option is usually preferred and leads to smaller
# mass balance discrepancy in nonuniform or diverging/converging flow fields. if nplane > 0, the
# fixed pattern is selected for initial placement. the value of nplane serves as the number of
# vertical 'planes' on which initial particles are placed within each cell block (figure 18a).
# the fixed pattern may work better than the random pattern only in relatively uniform flow
# fields. for two-dimensional simulations in plan view, set nplane = 1. for cross sectional or
# three-dimensional simulations, nplane = 2 is normally adequate. increase nplane if more
# resolution in the vertical direction is desired.
nplane = None
# Npl is the number of initial particles per cell to be placed at cells where the relative cell
# concentration gradient is less than or equal to dceps. generally, npl can be set to zero since
# advection is considered insignificant when the relative cell concentration gradient is less
# than or equal to dceps. setting npl equal to nph causes a uniform number of particles to be
# placed in every cell over the entire grid (i.e., the uniform approach).
npl = None
# Nph is the number of initial particles per cell to be placed at cells where the relative cell
# concentration gradient is greater than dceps. the selection of nph depends on the nature of the
# flow field and also the computer memory limitation. generally, a smaller number should be used
# in relatively uniform flow fields and a larger number should be used in relatively nonuniform
# flow fields. however, values exceeding 16 in two-dimensional simulation or 32 in three-
# dimensional simulation are rarely necessary. if the random pattern is chosen, nph particles are
# randomly distributed within the cell block. if the fixed pattern is chosen, nph is divided by
# nplane to yield the number of particles to be placed per vertical plane, which is rounded to
# one of the values shown in figure 30.
nph = None
# Is the minimum number of particles allowed per cell. if the number of particles in a cell at the
# end of a transport step is fewer than npmin, new particles are inserted into that cell to
# maintain a sufficient number of particles. npmin can be set to zero in relatively uniform flow
# fields and to a number greater than zero in diverging/converging flow fields. generally, a
# value between zero and four is adequate.
npmin = None
# Npmax is the maximum number of particles allowed per cell. if the number of particles in a cell
# exceeds npmax, all particles are removed from that cell and replaced by a new set of particles
# equal to nph to maintain mass balance. generally, npmax can be set to approximately two times
# of nph.
npmax = None
# S a flag indicating whether the random or fixed pattern is selected for initial placement of
# particles to approximate sink cells in the mmoc scheme. the convention is the same as that for
# nplane. it is generally adequate to set nlsink equivalent to nplane.
nlsink = None
# Is the number of particles used to approximate sink cells in the mmoc scheme. the convention is
# the same as that for nph. it is generally adequate to set npsink equivalent to nph.
npsink = None
# Dchmoc is the critical relative concentration gradient for controlling the selective use of
# either moc or mmoc in the hmoc solution scheme. the moc solution is selected at cells where the
# relative concentration gradient is greater than dchmoc. the mmoc solution is selected at cells
# where the relative concentration gradient is less than or equal to dchmoc.

```



dchmoc = **None**

```
adv = flopy.mt3d.mtadv.Mt3dAdv(model=model, mixelm=mixelm, percel=percel, mxpart=mxpart,  
                                nadvfd=nadvfd, itrack=itrack, wd=wd, dceps=dceps, nplane=nplane,  
                                npl=npl, npn=nph, npmin=npmin, npmax=npmax, nlsink=nlsink,  
                                npsink=npsink, dchmoc=dchmoc)
```

**DSP**

[illegible]

[illegible]

[illegible]

[illegible]

[illegible]

[illegible]

[illegible]



[illegible]





```

In [ ]: # The model object (of type :class:`flopy.mt3d.mt.mt3dms`) to which this package will be added.
model = mt3d
# Crch is the concentration of recharge for species 1. if the recharge flux is positive, it acts as
# a source whose concentration can be specified as desired. if the recharge flux is negative, it
# acts as a sink (discharge) whose concentration is always set equal to the concentration of
# groundwater at the cell where discharge occurs. note that the location and flow rate of
# recharge/discharge are obtained from the flow model directly through the unformatted
# flow-transport link file. crch can be specified as an array, if the array is constant for the
# entire simulation. if crch changes by stress period, then the user must provide a dictionary,
# where the key is the stress period number (zero based) and the value is the recharge array.
# the recharge concentration can be specified for additional species by passing additional
# arguments to the mt3dssm constructor. for example, to specify the recharge concentration for
# species two one could use crch2={0: 0., 1: 10*np.ones((nrow, ncol), dtype=np.float)} as and
# additional keyword argument that is passed to mt3dssm when making the ssm object.
crch = None
# Is the concentration of evapotranspiration flux for species 1. evapotranspiration is the only
# type of sink whose concentration may be specified externally. note that the concentration of a
# sink cannot be greater than that of the aquifer at the sink cell. thus, if the sink
# concentration is specified greater than that of the aquifer, it is automatically set equal to
# the concentration of the aquifer. also note that the location and flow rate of
# evapotranspiration are obtained from the flow model directly through the unformatted
# flow-transport link file. for multi-species simulations, see crch for a description of how to
# specify additional concentrations arrays for each species.
cevt = None
mxss = 121
# Keys in the dictionary are stress zero-based stress period numbers; values in the dictionary are
# recarrays of ssm boundaries. the dtype for the recarray can be obtained using ssm.dtype (after
# the ssm package has been created). the default dtype for the recarray is np.dtype([('k',
# np.int), ('i', np.int), ('j', np.int), ('css', np.float32), ('itype', np.int), ((cssms(n),
# np.float), n=1, ncomp)]) if there are more than one component species, then additional entries
# will be added to the dtype as indicated by cssm(n). note that if the number of dictionary
# entries is less than the number of stress periods, then the last recarray of boundaries will
# apply until the end of the simulation. full details of all options to specify
# stress_period_data can be found in the flopy3_multi-component_ssm ipython notebook in the
# notebook subdirectory of the examples directory. css is the specified source concentration or
# mass-loading rate, depending on the value of itype, in a single-species simulation, (for a
# multispecies simulation, css is not used, but a dummy value still needs to be entered here.)
# note that for most types of sources, css is interpreted as the source concentration with the
# unit of mass per unit volume (ml-3), which, when multiplied by its corresponding flow rate
# (l3t-1) from the flow model, yields the mass-loading rate (mt-1) of the source. for a special
# type of sources (itype = 15), css is taken directly as the mass-loading rate (mt-1) of the
# source so that no flow rate is required from the flow model. furthermore, if the source is
# specified as a constant-concentration cell (itype = -1), the specified value of css is assigned
# directly as the concentration of the designated cell. if the designated cell is also associated
# with a sink/source term in the flow model, the flow rate is not used. itype is an integer
# indicating the type of the point source. an itype dictionary can be retrieved from the ssm
# object as itype = mt3d.mt3dssm.itype_dict() (cssms(n), n=1, ncomp) defines the concentrations
# of a point source for multispecies simulation with ncomp>1. in a multispecies simulation, it is
# necessary to define the concentrations of all species associated with a point source. as an
# example, if a chemical of a certain species is injected into a multispecies system, the
# concentration of that species is assigned a value greater than zero while the concentrations of
# all other species are assigned zero. cssms(n) can be entered in free format, separated by a
# comma or space between values. several important notes on assigning concentration for the
# constant-concentration condition (itype = -1) are listed below: the constant-concentration
# condition defined in this input file takes precedence to that defined in the basic transport
# package input file. in a multiple stress period simulation, a constant-concentration cell, once
# defined, will remain a constant- concentration cell in the duration of the simulation, but its
# concentration value can be specified to vary in different stress periods. in a multispecies
# simulation, if it is only necessary to define different constant-concentration conditions for
# selected species at the same cell location, specify the desired concentrations for those
# species, and assign a negative value for all other species. the negative value is a flag used
# by mt3dms to skip assigning the constant-concentration condition for the designated species.
stress_period_data = {0: rec.array([(0, 0, 0, 0., 2, 0., 12. ), (0, 1, 0, 0., 2, 0., \
12. ), (0, 2, 0, 0., 2, 0., 12. ), (0, 3, 0, 0., 2, 0., 12.\
), (0, 4, 0, 0., 2, 0., 12. ), (0, 5, 0, 0., 2, 0., 12. ), \
(0, 6, 0, 0., 2, 0., 12. ), (0, 7, 0, 0., 2, 0., 12. ), (0, \
8, 0, 0., 2, 0., 12. ), (0, 9, 0, 0., 2, 0., 12. ), (0, 10, \
0, 0., 2, 0., 12. ), (0, 11, 0, 0., 2, 0., 12. ), (0, 12, 0, \
0., 2, 0., 12. ), (0, 13, 0, 0., 2, 0., 12. ), (0, 14, 0, 0., \
2, 0., 12. ), (0, 15, 0, 0., 2, 0., 12. ), (0, 16, 0, 0., 2, \
0., 12. ), (0, 17, 0, 0., 2, 0., 12. ), (0, 18, 0, 0., 2, 0., \
12. ), (0, 19, 0, 0., 2, 0., 12. ), (0, 20, 0, 0., 2, 0., 12.\
), (0, 21, 0, 0., 2, 0., 12. ), (0, 22, 0, 0., 2, 0., 12. ), \
(0, 23, 0, 0., 2, 0., 12. ), (0, 24, 0, 0., 2, 0., 12. ), (0, \
25, 0, 0., 2, 0., 12. ), (0, 26, 0, 0., 2, 0., 12. ), (0, 27, \
0, 0., 2, 0., 12. ), (0, 28, 0, 0., 2, 0., 12. ), (0, 29, 0, \
0., 2, 0., 12. ), (0, 30, 0, 0., 2, 0., 12. ), (0, 31, 0, 0., \
2, 0., 12. ), (0, 32, 0, 0., 2, 0., 12. ), (0, 33, 0, 0., 2, \
0., 12. ), (0, 34, 0, 0., 2, 0., 12. ), (0, 35, 0, 0., 2, 0., \
12. ), (0, 36, 0, 0., 2, 0., 12. ), (0, 37, 0, 0., 2, 0., 12.\
), (0, 38, 0, 0., 2, 0., 12. ), (0, 39, 0, 0., 2, 0., 12. ), \
(0, 40, 0, 0., 2, 0., 12. ), (0, 41, 0, 0., 2, 0., 12. ), (0, \
42, 0, 0., 2, 0., 12. ), (0, 43, 0, 0., 2, 0., 12. ), (0, 44, \
0, 0., 2, 0., 12. ), (0, 45, 0, 0., 2, 0., 12. ), (0, 46, 0, \
0., 2, 0., 12. ), (0, 47, 0, 0., 2, 0., 12. ), (0, 48, 0, 0., \
2, 0., 12. ), (0, 49, 0, 0., 2, 0., 12. ), (0, 50, 0, 0., 2, \
0., 12. ), (0, 51, 0, 0., 2, 0., 12. ), (0, 52, 0, 0., 2, 0., \
12. ), (0, 53, 0, 0., 2, 0., 12. ), (0, 54, 0, 0., 2, 0., 12.\
), (0, 55, 0, 0., 2, 0., 12. ), (0, 56, 0, 0., 2, 0., 12. ), \

```

```

(0, 57, 0, 0., 2, 0., 12. ), (0, 58, 0, 0., 2, 0., 12. ), (0, \
59, 0, 0., 2, 0., 12. ), (0, 7, 20, 0., 15, 0., 0.316098)], \
dtype=[('k', '<i8'), ('i', '<i8'), ('j', '<i8'), ('css', '<f4'), ('itype', \
'<i8'), ('cssm(01)', '<f4'), ('cssm(02)', '<f4')]], 144: rec.array([(0, 0, \
0, 0., 2, 0., 12.), (0, 1, 0, 0., 2, 0., 12.), (0, 2, 0, 0., 2, 0., \
12.), (0, 3, 0, 0., 2, 0., 12.), (0, 4, 0, 0., 2, 0., 12.), (0, 5, 0, \
0., 2, 0., 12.), (0, 6, 0, 0., 2, 0., 12.), (0, 7, 0, 0., 2, 0., 12.), \
(0, 8, 0, 0., 2, 0., 12.), (0, 9, 0, 0., 2, 0., 12.), (0, 10, 0, 0., \
2, 0., 12.), (0, 11, 0, 0., 2, 0., 12.), (0, 12, 0, 0., 2, 0., 12.), (0, \
13, 0, 0., 2, 0., 12.), (0, 14, 0, 0., 2, 0., 12.), (0, 15, 0, 0., 2, \
0., 12.), (0, 16, 0, 0., 2, 0., 12.), (0, 17, 0, 0., 2, 0., 12.), (0, 18, \
0, 0., 2, 0., 12.), (0, 19, 0, 0., 2, 0., 12.), (0, 20, 0, 0., 2, 0., \
12.), (0, 21, 0, 0., 2, 0., 12.), (0, 22, 0, 0., 2, 0., 12.), (0, 23, 0, \
0., 2, 0., 12.), (0, 24, 0, 0., 2, 0., 12.), (0, 25, 0, 0., 2, 0., 12.), \
(0, 26, 0, 0., 2, 0., 12.), (0, 27, 0, 0., 2, 0., 12.), (0, 28, 0, 0., \
2, 0., 12.), (0, 29, 0, 0., 2, 0., 12.), (0, 30, 0, 0., 2, 0., 12.), (0, \
31, 0, 0., 2, 0., 12.), (0, 32, 0, 0., 2, 0., 12.), (0, 33, 0, 0., 2, \
0., 12.), (0, 34, 0, 0., 2, 0., 12.), (0, 35, 0, 0., 2, 0., 12.), (0, 36, \
0, 0., 2, 0., 12.), (0, 37, 0, 0., 2, 0., 12.), (0, 38, 0, 0., 2, 0., \
12.), (0, 39, 0, 0., 2, 0., 12.), (0, 40, 0, 0., 2, 0., 12.), (0, 41, 0, \
0., 2, 0., 12.), (0, 42, 0, 0., 2, 0., 12.), (0, 43, 0, 0., 2, 0., 12.), \
(0, 44, 0, 0., 2, 0., 12.), (0, 45, 0, 0., 2, 0., 12.), (0, 46, 0, 0., \
2, 0., 12.), (0, 47, 0, 0., 2, 0., 12.), (0, 48, 0, 0., 2, 0., 12.), (0, \
49, 0, 0., 2, 0., 12.), (0, 50, 0, 0., 2, 0., 12.), (0, 51, 0, 0., 2, \
0., 12.), (0, 52, 0, 0., 2, 0., 12.), (0, 53, 0, 0., 2, 0., 12.), (0, 54, \
0, 0., 2, 0., 12.), (0, 55, 0, 0., 2, 0., 12.), (0, 56, 0, 0., 2, 0., \
12.), (0, 57, 0, 0., 2, 0., 12.), (0, 58, 0, 0., 2, 0., 12.), (0, 59, 0, \
0., 2, 0., 12.), (0, 7, 20, 0., 15, 0., 0.)]), dtype=[('k', \
'<i8'), ('i', '<i8'), ('j', '<i8'), ('css', '<f4'), ('itype', '<i8'), \
('cssm(01)', '<f4'), ('cssm(02)', '<f4')]])}

# Dtype to use for the recarray of boundaries. if left as none (the default) then the dtype will
# be automatically constructed.
dtype = np.dtype([('k', '<i8'), ('i', '<i8'), ('j', '<i8'), ('css', '<f4'), ('itype', '<i8'),
('cssm(01)', '<f4'), ('cssm(02)', '<f4')]))

ssm = flopy.mt3d.mtssm.Mt3dSsm(model=model, crch=crch, cevt=cevt, mxss=mxss,
stress_period_data=stress_period_data, dtype=dtype)

```

## GCG

```

In [ ]: # The model object (of type :class:`flopy.mt3d.mt.mt3dms`) to which this package will be added.
model = mt3d
# Is the maximum number of outer iterations; it should be set to an integer greater than one only
# when a nonlinear sorption isotherm is included in simulation. (default is 1)
mxiter = 1
# Is the maximum number of inner iterations; a value of 30-50 should be adequate for most problems.
# (default is 50)
iter1 = 100
# Is the type of preconditioners to be used with the lanczos/orthomin acceleration scheme: = 1,
# jacobi = 2, ssor = 3, modified incomplete cholesky (mic) (mic usually converges faster, but it
# needs significantly more memory) (default is 3)
isolve = 3
# Is an integer flag for treatment of dispersion tensor cross terms: = 0, lump all dispersion cross
# terms to the right-hand-side (approximate but highly efficient). = 1, include full dispersion
# tensor (memory intensive). (default is 0)
ncrs = 1
# Is the relaxation factor for the ssor option; a value of 1.0 is generally adequate. (default is
# 1)
accl = 1.0
# Is the convergence criterion in terms of relative concentration; a real value between 10-4 and
# 10-6 is generally adequate. (default is 1.e-5)
cclose = 1e-05
# Iprgcg is the interval for printing the maximum concentration changes of each iteration. set
# iprgcg to zero as default for printing at the end of each stress period. (default is 0)
iprgcg = 0

gcg = flopy.mt3d.mtgcg.Mt3dGcg(model=model, mxiter=mxiter, iter1=iter1, isolve=isolve, ncrs=ncrs,
accl=accl, cclose=cclose, iprgcg=iprgcg)

```

## RCT

[illegible]

[illegible]

[illegible]



[illegible]

```
# Prsity2 is the porosity of the immobile domain (the ratio of pore spaces filled with immobile
```

```

# fluids over the bulk volume of the aquifer medium) when the simulation is intended to represent
# a dual-domain system. prsity2 is used if isothm = 5 or 6. if prsity2 is not user- specified and
# isothm = 5 or 6 then prsity2 is set to 0.1. (default is none)
prsity2 = 0.10000000149011612
# Srconc is the user-specified initial concentration for the sorbed phase of the first species if
# isothm = 4 (unit, mm-1). note that for equilibrium-controlled sorption, the initial
# concentration for the sorbed phase cannot be specified. srconc is the user-specified initial
# concentration of the first species for the immobile liquid phase if isothm = 5 or 6 (unit,
# ml-3). if srconc is not user-specified and isothm = 4, 5, or 6 then srconc is set to 0.
# (default is none).
srconc = 0.0
# Sp1 is the first sorption parameter for the first species. the use of sp1 depends on the type of
# sorption selected (the value of isothm). for linear sorption (isothm = 1) and nonequilibrium
# sorption (isothm = 4), sp1 is the distribution coefficient (kd) (unit, l3m-1). for freundlich
# sorption (isothm = 2), sp1 is the freundlich equilibrium constant (kf) (the unit depends on the
# freundlich exponent a). for langmuir sorption (isothm = 3), sp1 is the langmuir equilibrium
# constant (kl) (unit, l3m-1 ). for dual-domain mass transfer without sorption (isothm = 5), sp1
# is not used, but still must be entered. for dual-domain mass transfer with sorption (isothm =
# 6), sp1 is also the distribution coefficient (kd) (unit, l3m-1). if sp1 is not specified and
# isothm > 0 then sp1 is set to 0. (default is none).
sp1 = 0.0
# Sp2 is the second sorption or dual-domain model parameter for the first species. the use of sp2
# depends on the type of sorption or dual-domain model selected. for linear sorption (isothm =
# 1), sp2 is read but not used. for freundlich sorption (isothm = 2), sp2 is the freundlich
# exponent a. for langmuir sorption (isothm = 3), sp2 is the total concentration of the sorption
# sites available ( s ) (unit, mm-1). for nonequilibrium sorption (isothm = 4), sp2 is the
# first-order mass transfer rate between the dissolved and sorbed phases (unit, t-1). for
# dual-domain mass transfer (isothm = 5 or 6), sp2 is the first-order mass transfer rate between
# the two domains (unit, t-1). if sp2 is not specified and isothm > 0 then sp2 is set to 0.
# (default is none).
sp2 = 0.0
# Rc1 is the first-order reaction rate for the dissolved (liquid) phase for the first species
# (unit, t-1). rc1 is not used ireact = 0. if a dual-domain system is simulated, the reaction
# rates for the liquid phase in the mobile and immobile domains are assumed to be equal. if rc1
# is not specified and ireact > 0 then rc1 is set to 0. (default is none).
rc1 = 0.0
# Rc2 is the first-order reaction rate for the sorbed phase for the first species (unit, t-1). rc2
# is not used ireact = 0. if a dual-domain system is simulated, the reaction rates for the sorbed
# phase in the mobile and immobile domains are assumed to be equal. generally, if the reaction is
# radioactive decay, rc2 should be set equal to rc1, while for biodegradation, rc2 may be
# different from rc1. note that rc2 is read but not used, if no sorption is included in the
# simulation. if rc2 is not specified and ireact > 0 then rc2 is set to 0. (default is none).
rc2 = 0.0
# Sp1 is the first sorption parameter for the first species. the use of sp1 depends on the type of
# sorption selected (the value of isothm). for linear sorption (isothm = 1) and nonequilibrium
# sorption (isothm = 4), sp1 is the distribution coefficient (kd) (unit, l3m-1). for freundlich
# sorption (isothm = 2), sp1 is the freundlich equilibrium constant (kf) (the unit depends on the
# freundlich exponent a). for langmuir sorption (isothm = 3), sp1 is the langmuir equilibrium
# constant (kl) (unit, l3m-1 ). for dual-domain mass transfer without sorption (isothm = 5), sp1
# is not used, but still must be entered. for dual-domain mass transfer with sorption (isothm =
# 6), sp1 is also the distribution coefficient (kd) (unit, l3m-1). if sp1 is not specified and
# isothm > 0 then sp1 is set to 0. (default is none).
sp12 = 0.00020732000120915473
# Sp2 is the second sorption or dual-domain model parameter for the first species. the use of sp2
# depends on the type of sorption or dual-domain model selected. for linear sorption (isothm =
# 1), sp2 is read but not used. for freundlich sorption (isothm = 2), sp2 is the freundlich
# exponent a. for langmuir sorption (isothm = 3), sp2 is the total concentration of the sorption
# sites available ( s ) (unit, mm-1). for nonequilibrium sorption (isothm = 4), sp2 is the
# first-order mass transfer rate between the dissolved and sorbed phases (unit, t-1). for
# dual-domain mass transfer (isothm = 5 or 6), sp2 is the first-order mass transfer rate between
# the two domains (unit, t-1). if sp2 is not specified and isothm > 0 then sp2 is set to 0.
# (default is none).
sp22 = 0.0
# Rc1 is the first-order reaction rate for the dissolved (liquid) phase for the first species
# (unit, t-1). rc1 is not used ireact = 0. if a dual-domain system is simulated, the reaction
# rates for the liquid phase in the mobile and immobile domains are assumed to be equal. if rc1
# is not specified and ireact > 0 then rc1 is set to 0. (default is none).
rc12 = 0.0
# Rc2 is the first-order reaction rate for the sorbed phase for the first species (unit, t-1). rc2
# is not used ireact = 0. if a dual-domain system is simulated, the reaction rates for the sorbed
# phase in the mobile and immobile domains are assumed to be equal. generally, if the reaction is
# radioactive decay, rc2 should be set equal to rc1, while for biodegradation, rc2 may be
# different from rc1. note that rc2 is read but not used, if no sorption is included in the
# simulation. if rc2 is not specified and ireact > 0 then rc2 is set to 0. (default is none).
rc22 = 0.0
# Srconc is the user-specified initial concentration for the sorbed phase of the first species if
# isothm = 4 (unit, mm-1). note that for equilibrium-controlled sorption, the initial
# concentration for the sorbed phase cannot be specified. srconc is the user-specified initial
# concentration of the first species for the immobile liquid phase if isothm = 5 or 6 (unit,
# ml-3). if srconc is not user-specified and isothm = 4, 5, or 6 then srconc is set to 0.
# (default is none).
srconc2 = 0.0

rct = floppy.mt3d.mtrct.Mt3dRct(model=model, isothm=isothm, ireact=ireact, igetsc=igetsc, rhob=rhob,
    prsity2=prsity2, srconc=srconc, sp1=sp1, sp2=sp2, rc1=rc1, rc2=rc2,
    sp12=sp12, sp22=sp22, rc12=rc12, rc22=rc22, srconc2=srconc2)

```

## flopy.seawat

```
In [ ]: # Name of model.  this string will be used to name the seawat input that are created with
# write_model. (the default is 'swttest')
modelname = 'BW'
# Extension for the namefile (the default is 'nam')
namefile_ext = 'nam'
modflowmodel = modflow
mt3dmodel = mt3d
# Version of seawat to use (the default is 'seawat').
version = 'seawat'
# The name of the executable to use (the default is 'swt_v4.exe').
exe_name = 'swt_v4'
structured = True
# Unit number for the list file (the default is 2).
listunit = 2
# Model workspace.  directory name to create model data sets. (default is the present working
# directory).
model_ws = '.'
# Location for external files (default is none).
external_path = None
# Print additional information to the screen (default is false).
verbose = False
# (default is true).
load = True
# (default is 0)
silent = 0

seawat = flopy.seawat.swt.Seawat(modelname=modelname, namefile_ext=namefile_ext,
                                modflowmodel=modflowmodel, mt3dmodel=mt3dmodel, version=version,
                                exe_name=exe_name, structured=structured, listunit=listunit,
                                model_ws=model_ws, external_path=external_path, verbose=verbose,
                                load=load, silent=silent)
```

## VSC

```
In [ ]: # The model object (of type :class:`flop.py.seawat.swt.seawat`) to which this package will be added.
model = seawat
mt3dmuflg = -1
# Is the minimum fluid viscosity. if the resulting viscosity value calculated with the equation is
# less than viscmín, the viscosity value is set to viscmín. if viscmín = 0, the computed fluid
# viscosity is not limited by viscmín (this is the option to use for most simulations). if
# viscmín > 0, a computed fluid viscosity less than viscmín is automatically reset to viscmín.
viscmín = 0.0
# Is the maximum fluid viscosity. if the resulting viscosity value calculated with the equation is
# greater than viscmáx, the viscosity value is set to viscmáx. if viscmáx = 0, the computed fluid
# viscosity is not limited by viscmáx (this is the option to use for most simulations). if
# viscmáx > 0, a computed fluid viscosity larger than viscmáx is automatically reset to viscmáx.
viscmáx = 0.0
# Is the fluid viscosity at the reference concentration and reference temperature. for most
# simulations, viscref is specified as the viscosity of freshwater.
viscref = 0.001266
nsmueos = 1
# Is a flag that specifies the option for including the effect of temperature on fluid viscosity.
# if mutempopt = 0, the effect of temperature on fluid viscosity is not included or is a simple
# linear relation that is specified in item 3c. if mutempopt = 1, fluid viscosity is calculated
# using equation 18. the size of the amucoeff array in item 3e is 4 (muncoeff = 4). if mutempopt
# = 2, fluid viscosity is calculated using equation 19. the size of the amucoeff array in item 3e
# is 5 (muncoeff = 5). if mutempopt = 3, fluid viscosity is calculated using equation 20. the
# size of the amucoeff array in item 3e is 2 (muncoeff = 2). if nsmueos and mutempopt are both
# set to zero, all fluid viscosities are set to viscref.
mutempopt = 2
# Is the mt3dms species number corresponding to the adjacent dmudc and cmuref.
mtmuspec = 1
# Is the slope of the linear equation that relates fluid viscosity to solute concentration.
dmudc = 1.923e-06
# Is the reference concentration.
cmuref = 0.0
mtmutempspec = 2
# Is the coefficient array of size muncoeff. amucoeff is a in equations 18, 19, and 20.
amucoeff = [0.001, 1.0, 0.015512, -20.0, -1.572]
# Is a flag. invisc is read only if mt3dmuflg is equal to zero. if invisc < 0, values for the visc
# array will be reused from the previous stress period. if it is the first stress period, values
# for the visc array will be set to viscref. if invisc = 0, values for the visc array will be set
# to viscref. if invisc >= 1, values for the visc array will be read from item 5. if invisc = 2,
# values read for the visc array are assumed to represent solute concentration, and will be
# converted to viscosity values.
invisc = None
# Is the fluid viscosity array read for each layer using the modflow-2000 u2drel array reader. the
# visc array is read only if mt3dmuflg is equal to zero. the visc array may also be entered in
# terms of solute concentration (or any other units) if invisc is set to 2, and the simple linear
# expression in item 3 can be used to represent the relation to viscosity.
visc = None

vsc = flop.py.seawat.swtvsc.SeawatVsc(model=model, mt3dmuflg=mt3dmuflg, viscmín=viscmín,
                                       viscmáx=viscmáx, viscref=viscref, nsmueos=nsmueos,
                                       mutempopt=mutempopt, mtmuspec=mtmuspec, dmudc=dmudc,
                                       cmuref=cmuref, mtmutempspec=mtmutempspec, amucoeff=amucoeff,
                                       invisc=invisc, visc=visc)
```

## Run this thing!

```
In [ ]: seawat.write_input()
# seawat.run_model()
```