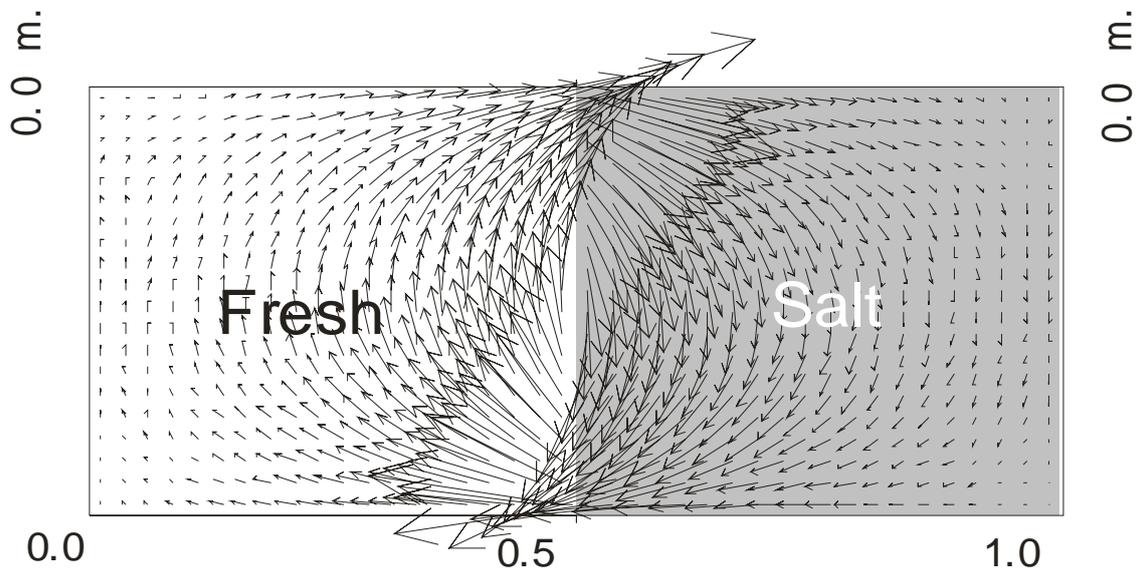


Vertical interface between fresh and saline groundwater

Variable-density groundwater flow modelling with SEAWAT



Gualbert Oude Essink
Deltares/Utrecht University
Unit Subsurface and Groundwater Systems
gualbert.oudeessink@deltares.nl



Introduction

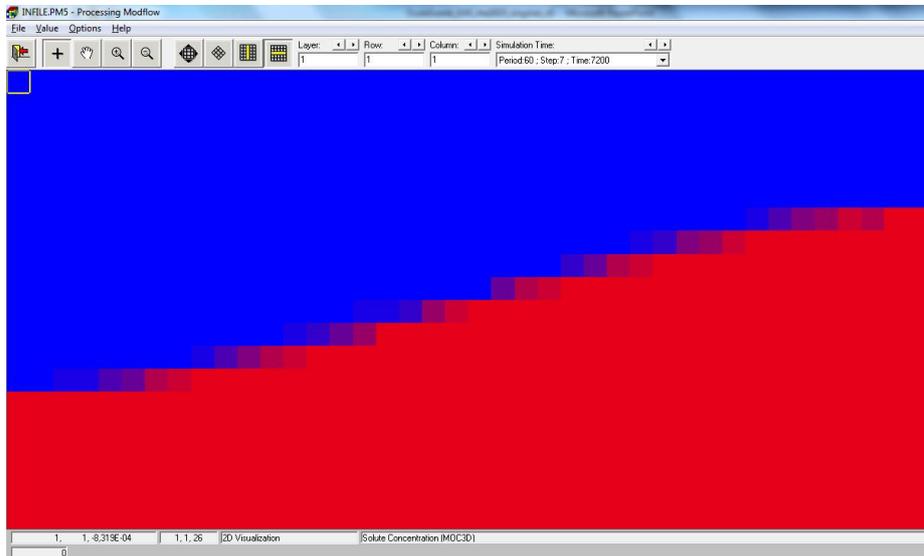
A hypothetical problem: a vertical fresh-saline interface in a homogeneous aquifer with the following geometry: horizontal $L=1.0$ m by vertical $D=0.5$ m. Interface approximation is simulated, which means: $D_{mol}=0$ m²/s, $\alpha_L=\alpha_{TH}=\alpha_{TV}=0$ m en $R_d=1$ (so no retardation). The other soil parameters are: hydraulic conductivity $k=10^{-3}$ m/s and porosity $n_e=0.1$.

Parameters

Layers	20	K_{hor}	$1 \cdot 10^{-3}$ m/s
Rows	1	T (= K_{hor} * thickness cell)	$2.5 \cdot 10^{-5}$ m ² /s
columns	40	Anisotropy K_{hor}/K_{ver}	1
Δx	0.025 m	n_e	0.1
Δy	1 m	α_L	0 m
Δz	0.025 m	α_T	0 m
Stress periods	15		
Initial concentration	0 and 35000 mg/l		
density buoyancy	0.025		

The total simulation consists of 720 time steps Δt of 30 s, divided over 15 'stress periods': the simulation time is 360 min=0.25 day.

In past IHE courses, this conceptual model has always been simulated with MOCDENS3D. This code (which is MOC3D (Konikow et al., 1996) adapted for density differences) is similar to SEAWAT, and pretty powerful in solving salt transport issues. The result with this code is after 7200sec is:



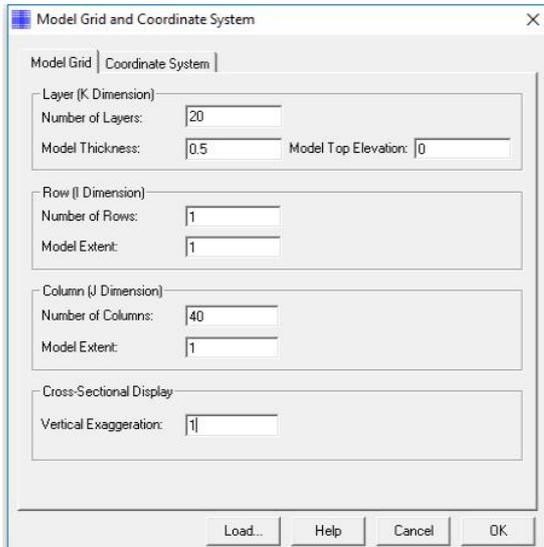
The number of initial particles is 16, which appears to be enough for this case.

Some tips on modelling

When numerical modelling, learn to be as neatly as possible because this behaviour helps you avoiding vague modelling results with you cannot explain nor understand anymore. In modelling practices, you normally forget within days where a specific model with specific model input parameters and variables run was about. Try to keep up a smart and clear logbook system (e.g. in EXCEL) explaining about the changed model input data files, and describe what you see as significant or major modelling results that are different than in the previous model run(s). Create a new subdirectory for every new model run, e.g. by copying an old previous one and rename it. Important is to set up about a proper naming system: number every new model with enough digits, e.g. vertint001, vertint002, etc. Avoid spaces in names in files and subdirectories. Under normal circumstances running big 3D models, you also have to limit model output which easily can end up in hundreds of Gb's of memory when you testing around. Better zip input data of old model runs. Big modelling output files could be zipped and saved too, but those output files are not essential (a new model run could be simulated again). By experience, it is strongly recommended to use Total Commander (freeware: [link](#)), which helps you in better manage model runs (some hotkeys: [link](#)), use short cuts like file and directory content comparison. Note you also need good freeware text editors like the Med Editor, Textpad and Notepad ++.

Exercise 000: making of the model

1. Go to: File, New Model
Name the new model: e.g. 'vertint000'
2. Go to: Grid, Mesh Size



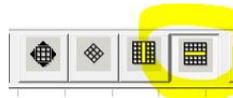
3. Go to: Grid, Layer Property
Make all 20 model layers '0: Confined'

Layer	Type	Horizontal Anisotropy	Vertical Anisotropy	Transmissivity	Leakance	Storage Coefficient	Interbe Storage
1	0: Confined	1	VK	Calculated	Calculated	Calculated	<input type="checkbox"/>

Make sure that Transmissivity and Leakance are 'Calculated' for all model layers

Layer	Type	Horizontal Anisotropy	Vertical Anisotropy	Transmissivity	Leakance	Storage Coefficient	Interbe Storage
1	0: Confined	1	VK	Calculated	Calculated	Calculated	<input type="checkbox"/>
2	0: Confined	1	VK	User Specified	Calculated	Calculated	<input type="checkbox"/>

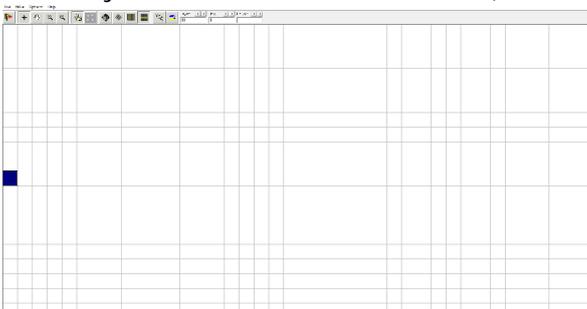
4. Place a fixed head in the model



use this view (as we are 2D vertical):

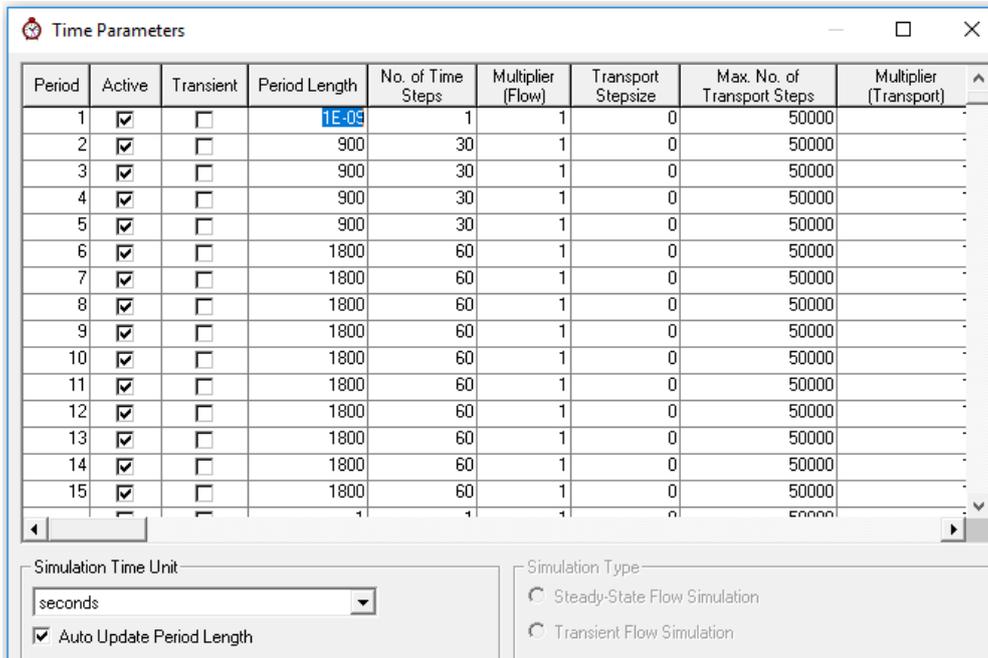
Go to: Grid, Cell Status, IBOUND (MODFLOW)

Make layer=11, Row=1, Column=1: -1 (now this cell has a fixed head)

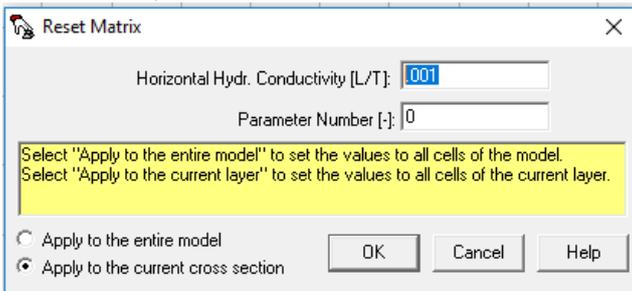


5. Go to: Parameter, Time
Make 15 stress periods
SP1: period length 1E-09 sec, No. of Time Steps 1

SP2-5 period length 900 sec, No. of Time Steps 30
 SP6-15 period length 1800 sec, No. of Time Steps 60

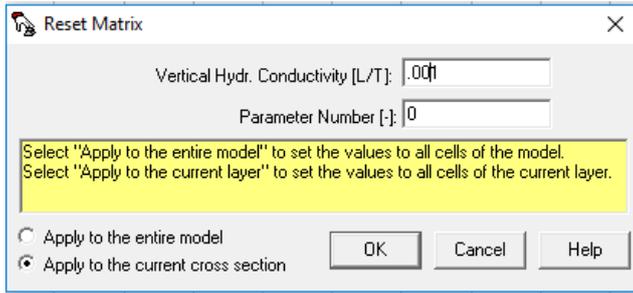


6. Go to: Parameter, Initial & Prescribed Hydraulic Heads [L]
 add "0"m to this cell where the head is fixed (IBOUND=-1)
 Leave Editor, save (so now the value is saved).
7. Go to: Parameter, Horizontal Hydraulic Conductivity
 Go to: Value, Reset Matrix
 Set to 0.001 (in 2D, 'Apply to the entire model' or 'Apply to the current cross section is the same')



Leave Editor, save

8. Go to: Parameter, Vertical Hydraulic Conductivity
 Go to: Value, Reset Matrix
 Set to 0.001

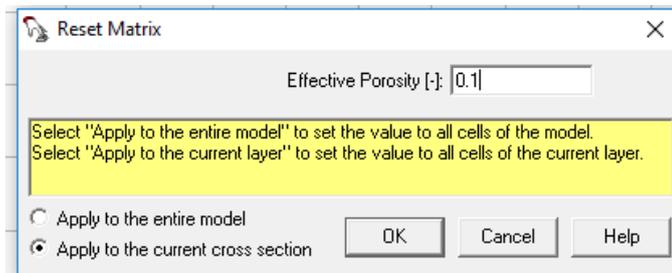


Leave Editor, save

9. Go to: Parameter, Effective Porosity

Go to: Value, Reset Matrix

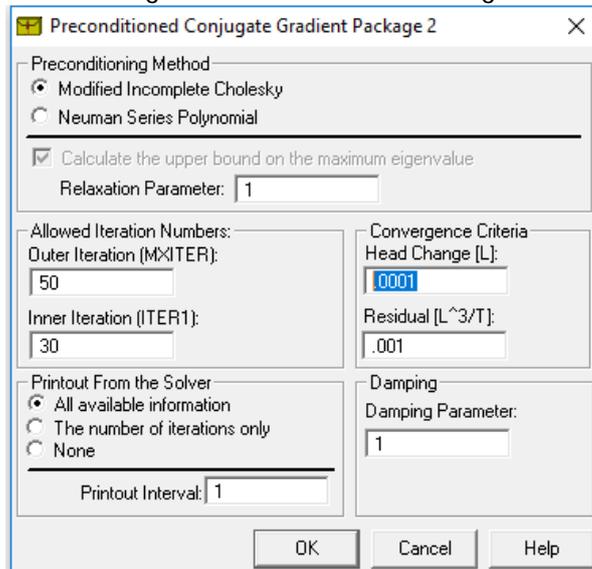
Set to 0.1



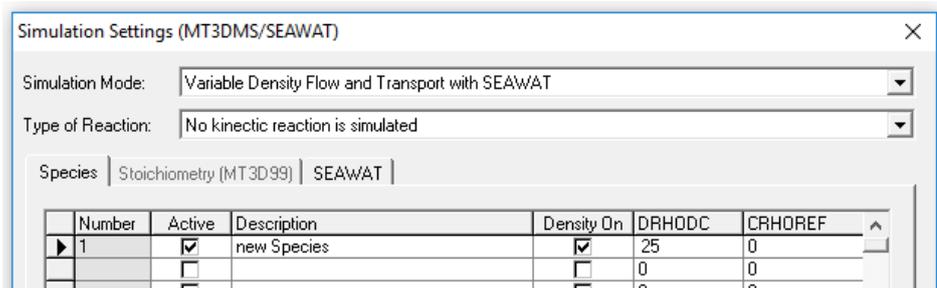
Leave Editor, save

10. Go to Model, MODFLOW (Flow Simulation), Solvers, PCG2

Set Convergence Criteria, Head Change to 0.001



11. Go to Model, MT3DMS/SEAWAT, Simulation setting



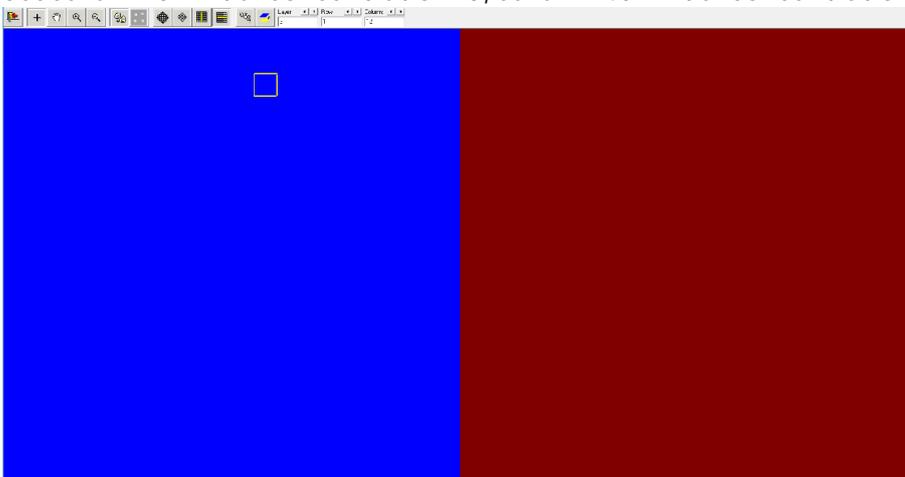
Set Simulation Mode: Variable Density Flow and Transport with SEAWAT

Set Species, Number 1, Active, 'new Species', Density 'On'

Set DRHODC=25 (this is the DRHODC term, Equation of State: $\partial\rho/\partial C$)

12. Go to Model, MT3DMS/SEAWAT, Initial Concentration

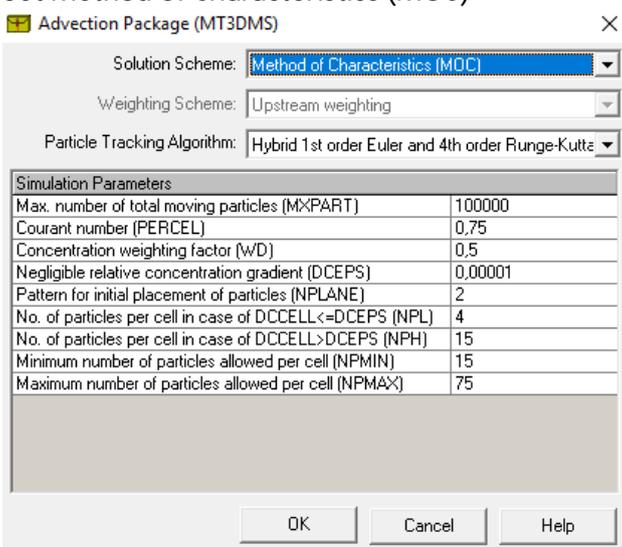
Set cells 1-20: Initial concentration=0, cells 21-40: Initial concentration=1



Leave Editor, save

13. Go to Model, MT3DMS/SEAWAT, Advection

Set Method of Characteristics (MOC)

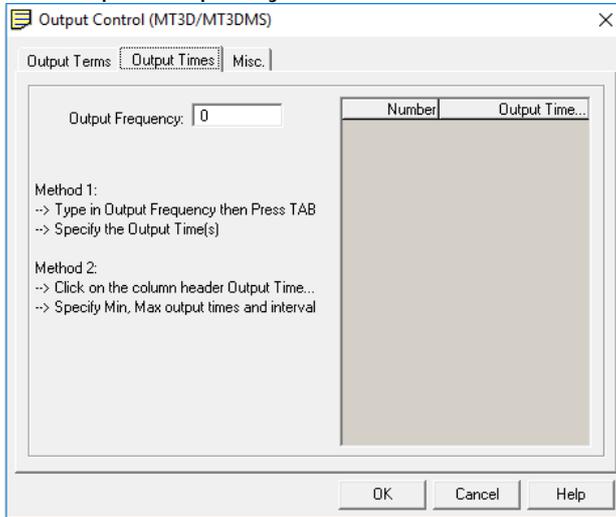


14. Go to Model, MT3DMS/SEAWAT, Dispersion, Edit, Ok

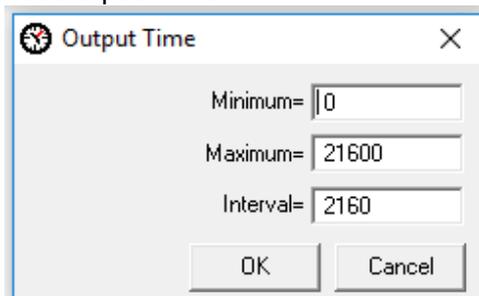
Go to Value, Reset Matrix Longitudinal Dispersivity=0 (m), Ok

Leave Editor, save

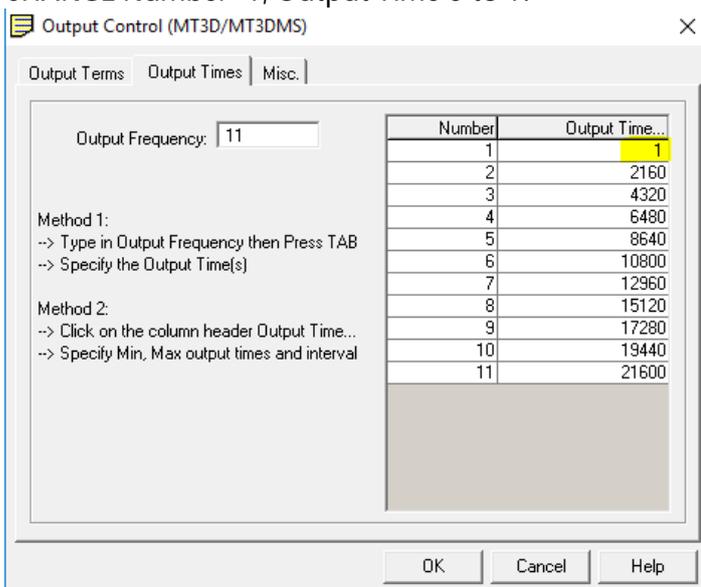
15. Go to: Model, MT3DMS/SEAWAT, Output Control, Output Times
Set Output frequency=11



Set Output Time...

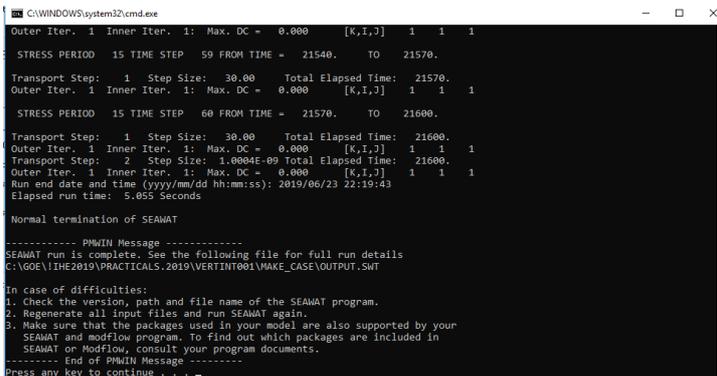
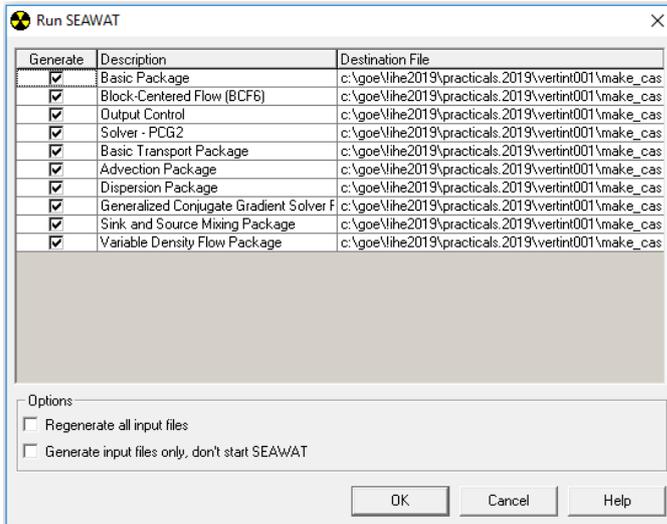


CHANGE Number=1, Output Time 0 to 1!



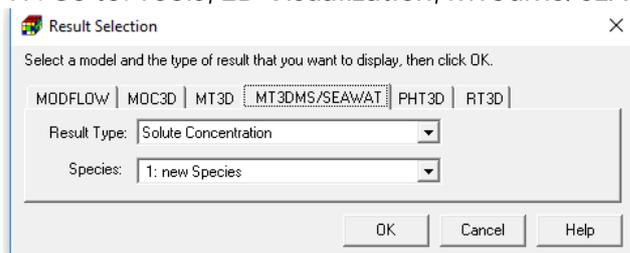
16. Run the model

Go to: Model, MT3DMS/SEAWAT, Run



Output:

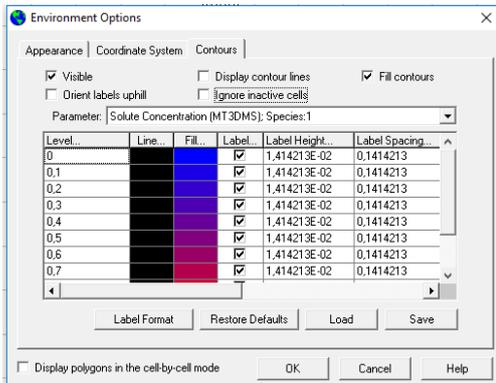
17. Go to: Tools, 2D-Visualization, MT3dMS/SEAWAT, Solute Concentration



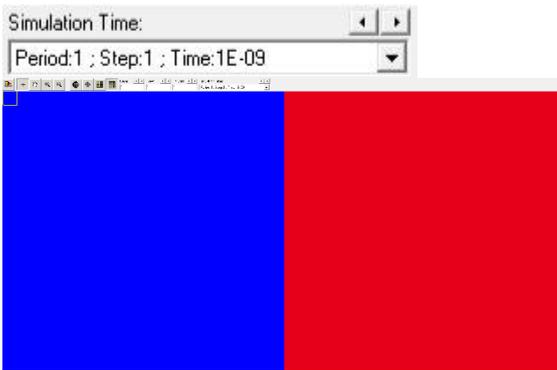
18. Go to: Options, Environment

Set Visible, Fill Contours

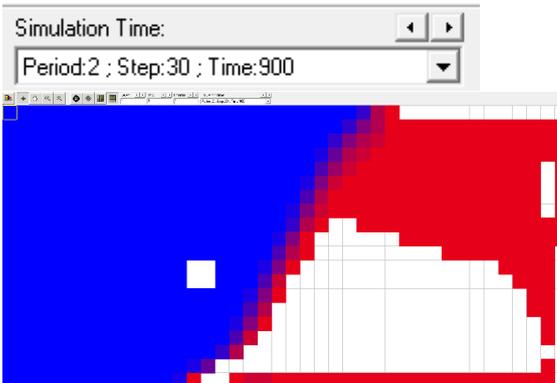
Make for instance red and blue



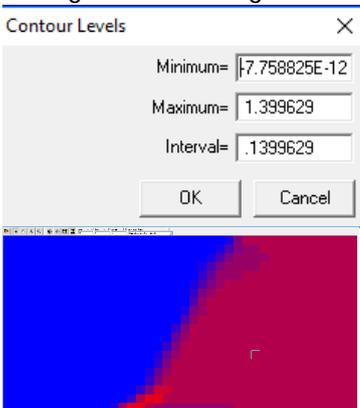
19. Time=1E-09 sec



Time=900sec



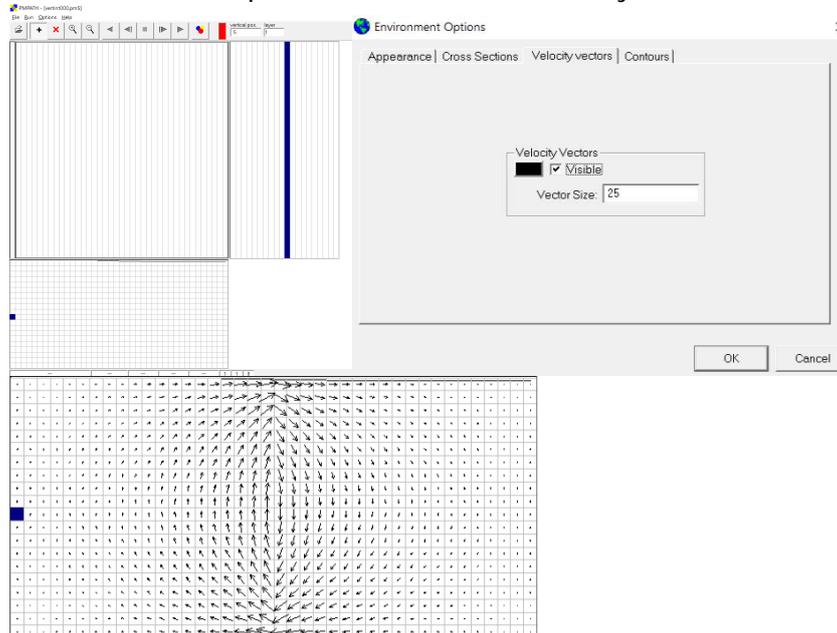
20. Over and undershooting:
Change the color legend:



21. Printing the velocity field

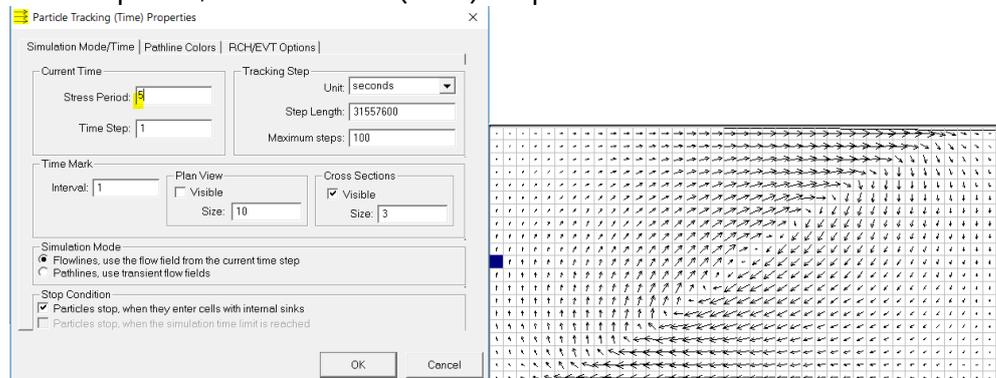
Go to: Models, PMPATH (Advective Transport)

Go to: PMPATH: Options, Environment, Velocity vectors, click on Visible



for a different Stress Period:

Go to: Options, Particle Track (Time) Properties



HERE FOLLOWS THE ORIGINAL PRACTICAL EXERCISES (using vertint001.zip)

Exercise 001: geometry of the problem

- This model vertint001 has already been run.
- Check in PMWIN the input files: heads, time-characteristics, IBOUND, MODFLOW; SEAWAT parameters. You still understand the input files?
- Check the concentration and the flow face results and explain what is going on.
- Why do cells exist with a concentration not equal to fresh or saline groundwater? Why does a large number of these cells disappear as a function of time?
- Is the steady-state time reached after 0.25 day (21600sec), based on the concentration distribution?

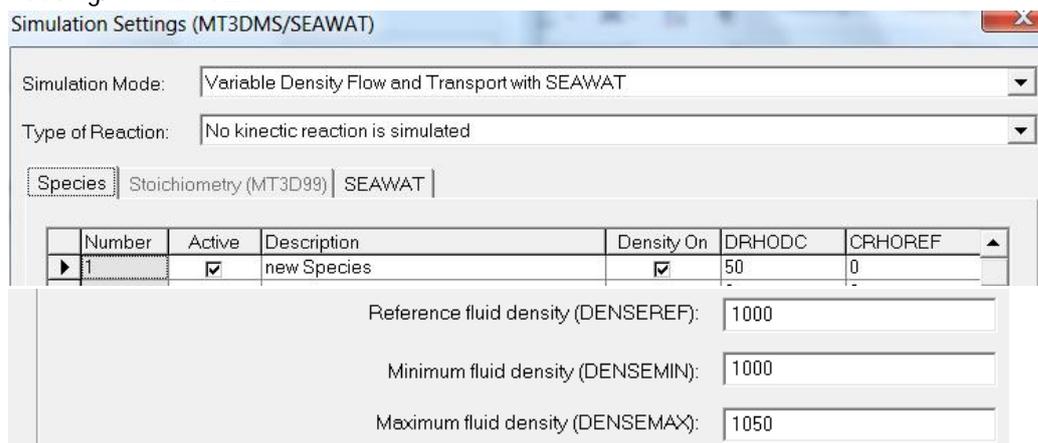
Exercise 002: on the best solute solver to reproduce the SEAWAT result

As can be seen checking the results with the default Advection solver ('ULTIMATE'), it is not easy to get a sharp interface (remember that no hydrodynamic dispersion is simulated!). Try to find a proper solver among the Advection solvers ('Models'->'MT3DMS / SEAWAT'->'Advection'): Finite Difference Method, MOC, HMOC, MMOC, ULTIMATE. You must change parameters within the solvers themselves, such as number as particles in the MOC versions; if you want to know more about the different parameters, check e.g. the PMWIN manual of the MT3DMS manual¹. Run SEAWAT.

NOTE: to be sure the old model runs remain and that you can compare the results; make new model runs in new subdirectories: e.g. vertint002, vertint003, etc.!

Exercise 003: effect of a larger density of saline groundwater

Use the best solute solver from Exercise 2. Change within 'MT3DMS / SEAWAT'-> 'Simulation settings' the value DRHODC in such a way that the saline/brine groundwater has a density of 1050 kg/m³. Run SEAWAT.



- Check the concentration results.

NOTE: to be sure the old model runs remain and that you can compare the results; make new model runs in new subdirectories!

Exercise 004: effect of dispersion

Use the best solute solver from Exercise 2. Add dispersion to the system: $\alpha_L=1.0$ m; $\alpha_{TH}=\alpha_{TV}=0.1$ m. 'MT3DMS / SEAWAT'-> 'Dispersion'. Run SEAWAT.

- Analyse the effect due to the differences compared with the reference case.

NOTE: to be sure the old model runs remain and that you can compare the results; make new model runs in new subdirectories!

¹ Zheng, C., & Wang, P. (1999). *MT3DMS: A modular three-dimensional multispecies transport model for simulation of advection, dispersion, and chemical reactions of contaminants in groundwater systems. Technical report, Waterways Experiment Station, US Army Corps of Engineers.*