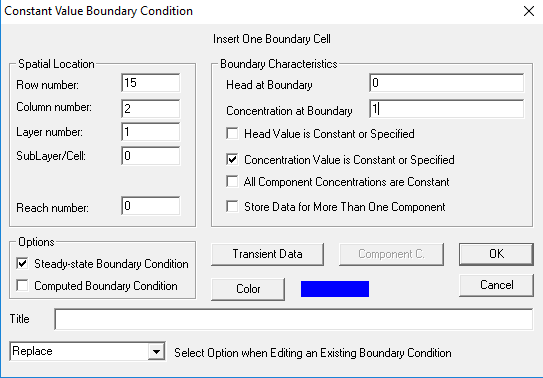
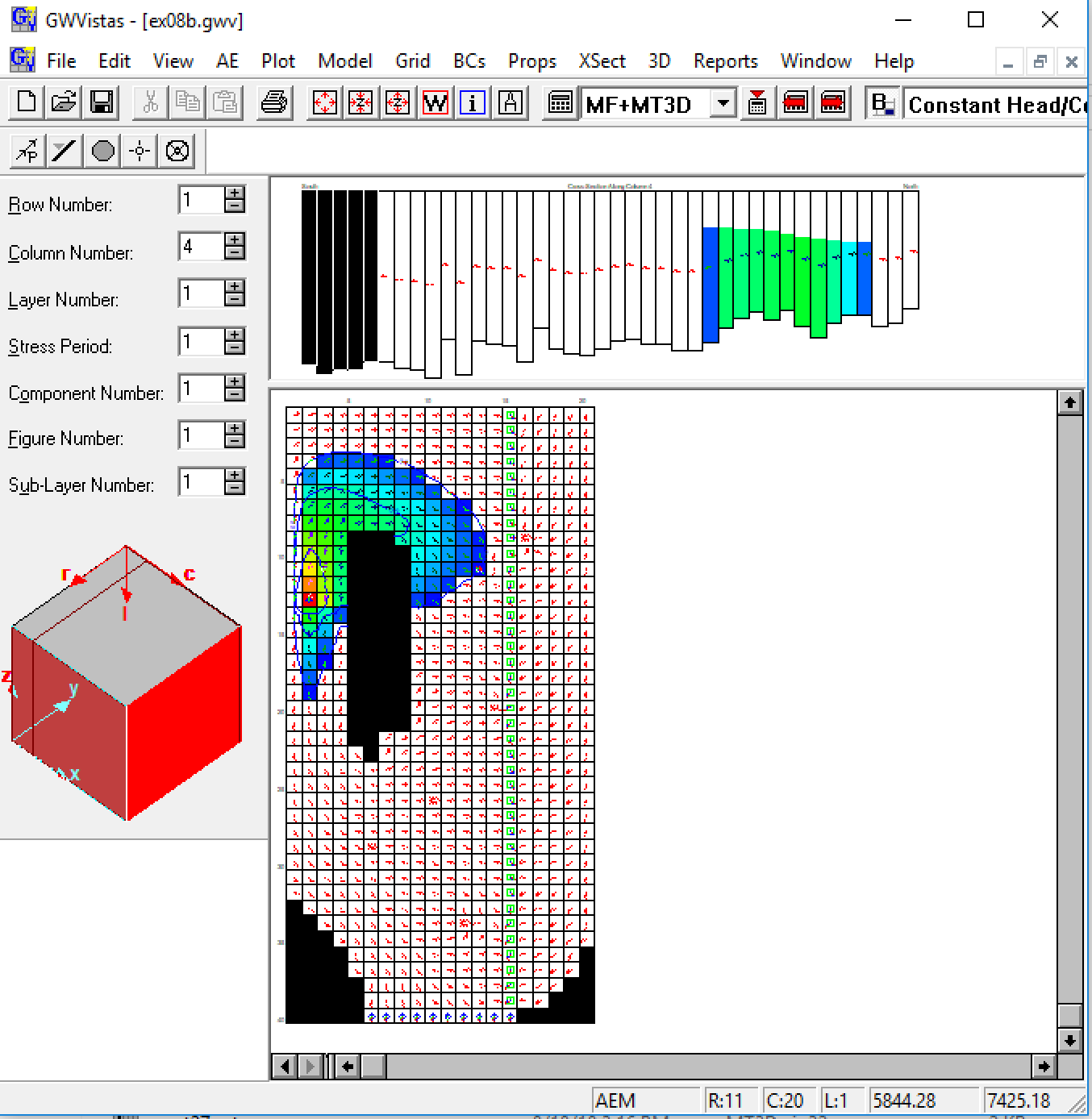
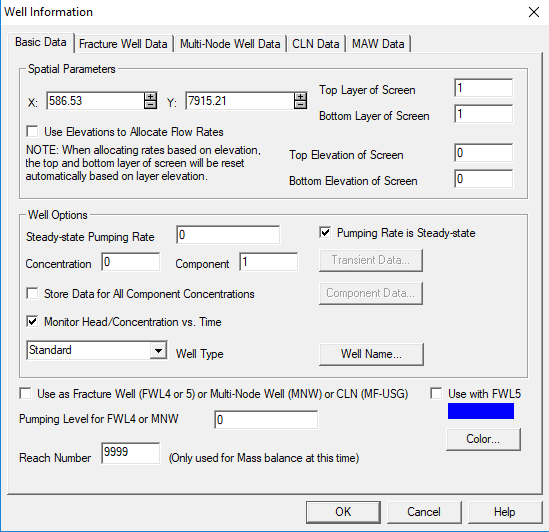
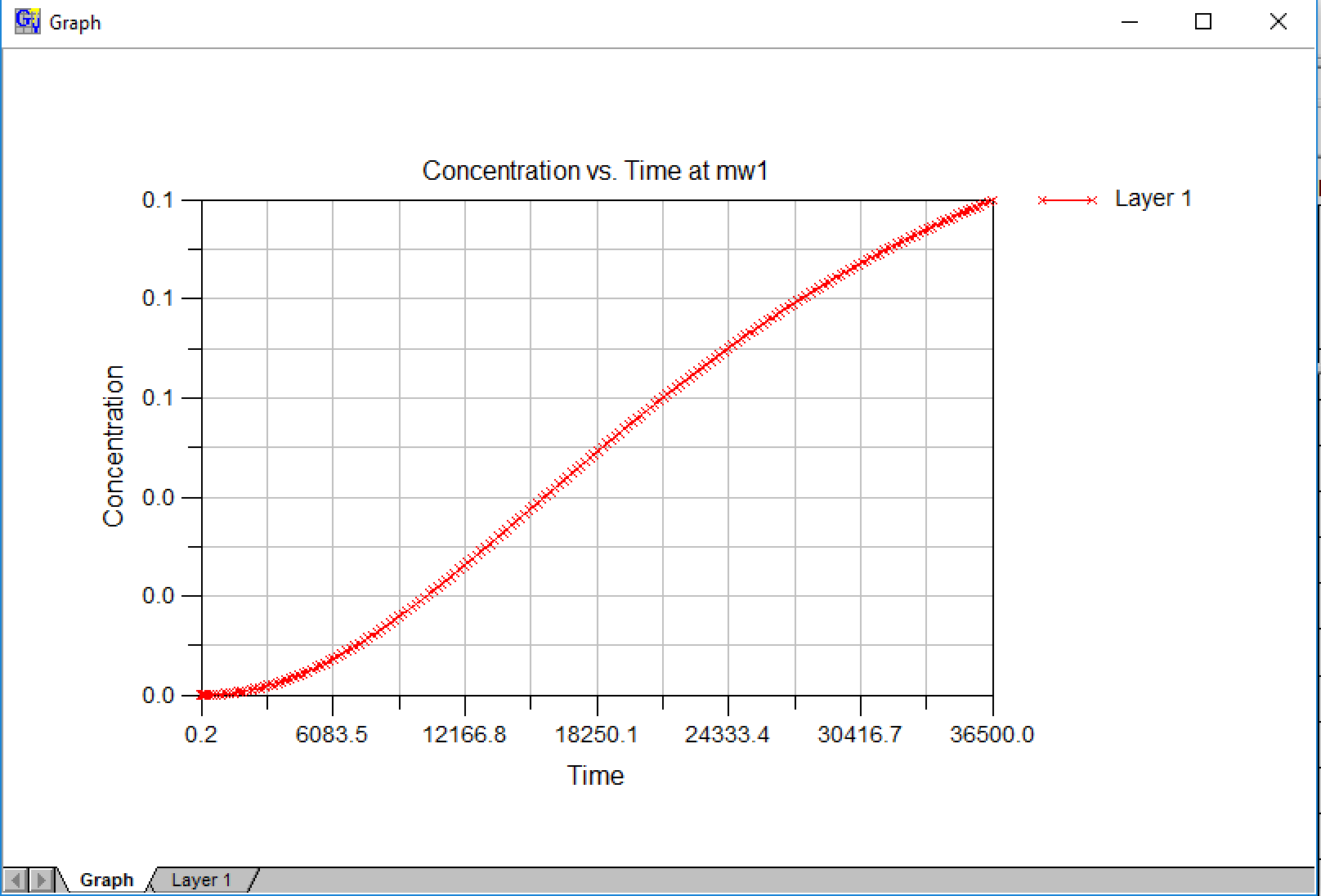
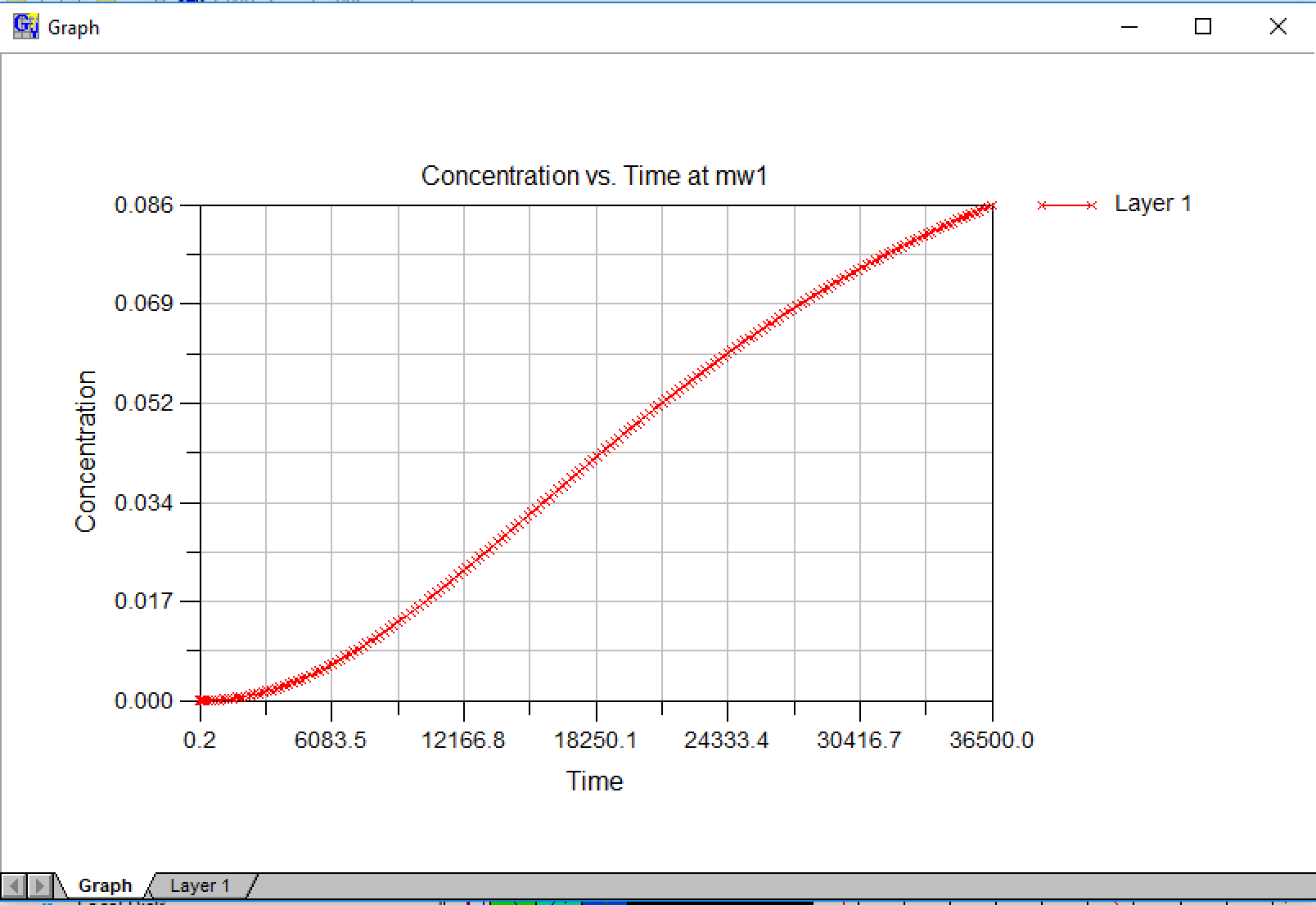
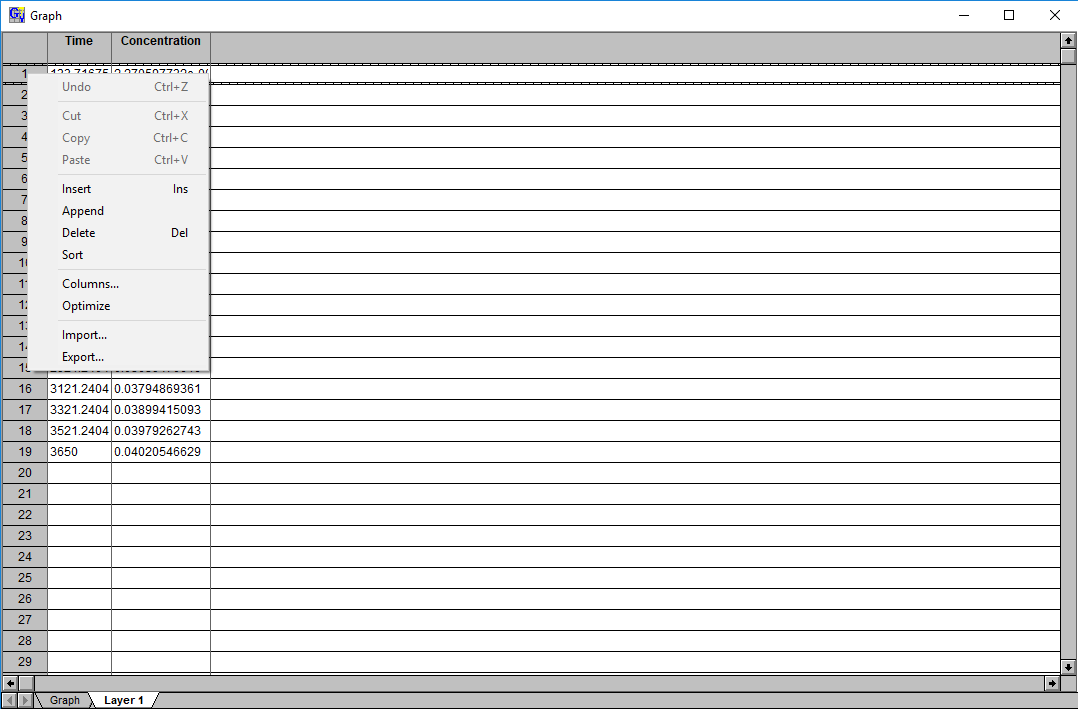
Exercise 8. Simulating advective and dispersive movement of a contaminant plume

**Exercise Description**

The purpose of this exercise is to use MT3D to simulate the transport of a solute. We will represent a plume source using a constant concentration boundary condition. We will then perform a number of simulations to estimate where this plume might go, and what the concentrations might be.

For each part, it might be a good idea to save a new copy of the Groundwater Vistas file (the .gwv file). And make sure to save your Vistas file frequently! Groundwater Vistas can be temperamental.

**Part I. Simulate Advective Transport using MT3D**

1. Copy the Groundwater Vista file that you created in Exercise 05 into the ex08 folder. Open up this file, and make sure to change the paths to models to the ex08 folder.
2. We are going to simulate solute transport for a 100-year period. Change the period length of stress period one to 100 years. This is set in the Model>MODFLOW>Stress Period Setup box.
3. In order to use MT3D, we need MODFLOW to save all of the flows in a special file that can be read by MT3D. For MODFLOW to create this file, you turn on the MT3D Flow Output, which can be found under the Model>Modflow>Packages menu.
4. Run the flow model.
5. Next, we are going to simulate a solute contaminant source as a constant-concentration condition in row 13 and column 2. Put your mouse over that cell and right-click. Here you will enter a concentration value of 1.0; you will uncheck the box that says the head value is constant, and you will check the box that says the concentration value is constant. 
6. The default porosity value in Vistas is 0.01. This is a relatively low value. 0.2 is a more reasonable value. Change the porosity to 0.2
7. At this point you should be ready to run MT3D and simulate solute transport. The procedure is similar to running MODFLOW. First you create the datasets and then you run MT3D.
8. When MT3D is finished, you should be able to post-process the results. Make sure that you import the concentrations; in order to do this, you will need to click the browse button next to the transport time step and pick a transport time step, normally the last one. A color flood is a way to look at the solute concentrations. See if you can make a plot that looks like this: 
9. It is also possible to plot solute concentration breakthrough curves. This is done by adding an analytical element well somewhere along the flow path from the constant concentration boundary. When you add an analytic element well, make sure you check the box that says Monitor Head/Concentration vs. Time. 
10. In order to see a breakthrough curve, you will first rerun MT3D and load the results. Then goto Plot>Hydrograph>Monitoring Well and change the Plot from Head to Concentration and click OK. You should hopefully see a plot that looks like this: 
11. It’s possible that the y-axis does not have enough precision. If this is the case, then double-click on the y-axis and increase the precision of the y-axis so you see something like this: 
12. In order to do comparisons between simulations, you may want to export these concentrations to another file. This can be done by clicking on the Layer 1 tab on the bottom of the chart, and then right-clicking anywhere in the spreadsheet to find an Export… option, which allows you to export to a file (Note that this data is also written to an MT3D observations file, which should be in your paths to models with an obs extension). 
13. Visualize and animate movement of the solute plume using ModelViewer. The default transport time stepping parameters in Model>MT3D/RT3D>General Options>Time Stepping result in transport time steps that are not uniform. This can make a transport animation look a little choppy. Try setting the transport time steps to a uniform value. Make sure that they are not to short, or the simulation will take too long to complete (50 days might be a good value). Note: you can only set the transport time steps if you are using an implicit scheme for advection.

**Part II. Compare the different solute transport schemes**

1. MT3D has very different schemes for simulating advection, and these schemes can give very different answers. Try running this problem using finite difference with both upstream and central-in-space weighting, TVD, and if time permits, try the Method of Characteristic schemes.
2. Make breakthrough curves for these different schemes at some location that becomes contaminated over time. Add these curves to the same plot. Which scheme is the most accurate? Why is that?
3. What do you think are some of the biggest problems with this model?

**Part III. Calculate the Solute Mass at a Pumping Well**

1. Your model should show the solute plume reaching a pumping well, although it arrives at relatively low concentrations. A common question that might be asked is how much solute mass is removed by pumping? If we assume that the constant concentration value of 1.0 represents milligrams per liter, then how much solute mass is removed?
2. How different are your estimates for the different advective solute transport schemes?

**Part III. Run the solute transport model using the true Freyberg parameters**

1. The true Freyberg parameters are included in the ex08/freyberg folder. Can you repeat this exercise using the true Freyberg parameters? How different are your results?

**Part IV. Add more horizontal resolution to this model**

1. If there is enough time, then it might be good to try and add some more horizontal resolution to this model. It would also be a very good idea to add more vertical resolution, but that is not as straightforward to do with Groundwater Vistas.
2. Let’s use the model that you just created using the true Freyberg parameters. Change to the grid editing mode. Once you do that, you can split any row into two and any column into two. Go ahead and try this for the area of interest.
3. Rerun the model. Have things changed?

**Part IV. Oops! What about dispersion? How do we turn on dispersion?**

1. Our previous simulations do not have any dispersion, except for numerical dispersion. Can you turn on dispersion? How does this affect the results?