

First AnAqSim Tutorial

Building a single-layer heterogeneous, anisotropic model with various boundary conditions

Contents

Introduction	2
Starting AnAqSim	2
Interface Overview	2
Setting up a Basemap	3
Display the Basemap.....	3
General Input.....	4
Units in AnAqSim	4
Domain Input	4
Adding the Lake Boundary.....	5
Digitizing a line boundary	6
Adding the Impermeable Aquifer Boundary.....	6
Plotting the Elements and Examining Details	8
Graphically Editing a Line Boundary	8
Adding River Line Boundaries	8
Adding the Interdomain Boundary	9
Adding the Well	11
Adding Uniform Recharge.....	11
Solution Settings	12
Solving the Model.....	13
Plot Input Settings.....	13
What to Plot.....	13
Contour Settings	13
Vector Settings.....	14
Making a Plot	14
Data at Cursor Location and Zooming and Panning	15
Analysis of the Solution	15
Well Heads and Discharges.....	15
Checking Line Boundary Conditions	16

Changing Line Boundary Accuracy with Parameters per Line	18
Making a Plot in a Smaller Window	20
Adding Pathlines to the Plot	20
Pathline Settings	21
Well Pathlines	21
Vertical Profile with Pathlines.....	22
Line Pathlines.....	26
Capture Constraints on Pathlines	28
Wrap up and Next Steps	29

Introduction

This first tutorial will introduce you to AnAqSim’s user interface and walk you through building a single-layer steady flow model with a variety of boundary conditions. In places, the tutorials will have links to short videos on the Fitts Geosolutions web site that demonstrate things in that mode. All tutorial models may be built with either AnAqSim or AnAqSimEDU.

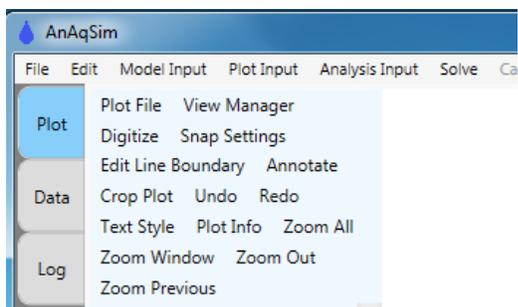
Before starting the tutorials, you should read the User Guide (opens when you press *Help* on the main menu) up through the section titled “User Interface”. These tutorials presume you have already read these sections of the User Guide. It is also assumed that you have installed AnAqSim and set up the license or installed AnAqSimEDU (no license needed). Please see the [Downloads, Licensing web page](#) for instructions on downloading, installing and licensing.

Starting AnAqSim

AnAqSim can be started by clicking on the Windows Start button at the lower left of your screen. In most cases, AnAqSim will be one of the programs pinned to the start menu and you can just select it there. Alternatively, you click on the Windows Start button, then click on *All Programs*, then find AnAqSim in the list and select it there.

Interface Overview

The user interface consists of three tabs: *Plot*, *Data*, and *Log*. You can move between them by clicking on the tab at left. The currently displayed tab is shown in blue:



The *Plot* tab displays various plots in the X-Y plane, the *Data* tab is where you enter and edit input data, and the *Log* tab displays text outputs from AnAqSim.

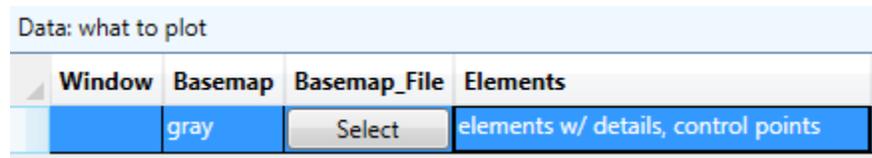
There is a main menu across the top (*File, Edit, Model Input...*) and in the *Plot* tab there is another menu at the left below the main menu (*Plot File, View Manager, Digitize...*). In addition, both the *Plot* and *Data* tabs have right-click menus that pop up and offer many options.

Setting up a Basemap

It's very handy to have a basemap and then digitize model elements over the basemap. AnAqSim uses DXF basemaps, a format that can typically be exported from CAD and GIS programs.

Select *Plot Input/What to Plot* from the main menu.

Edit the *Basemap* and *Elements* cells of data grid so it looks like the following image. See this [short video to learn the basics of editing data in the Data tab](#).



With these settings, the basemap will be displayed gray (or you could chose to display it in its original colors) and when elements are displayed, they will show control points and have pop-up text explaining details.

Click the *Select* button in the *Basemap_File* column, then find and select the basemap file, which is in one of these directories:

- Program Files/Fitts Geosolutions/AnAqSim/Documentation/basemap.dxf (64-bit AnAqSim)
- Program Files (x86)/Fitts Geosolutions/AnAqSim/Documentation/basemap.dxf (32-bit AnAqSim or AnAqSimEDU)

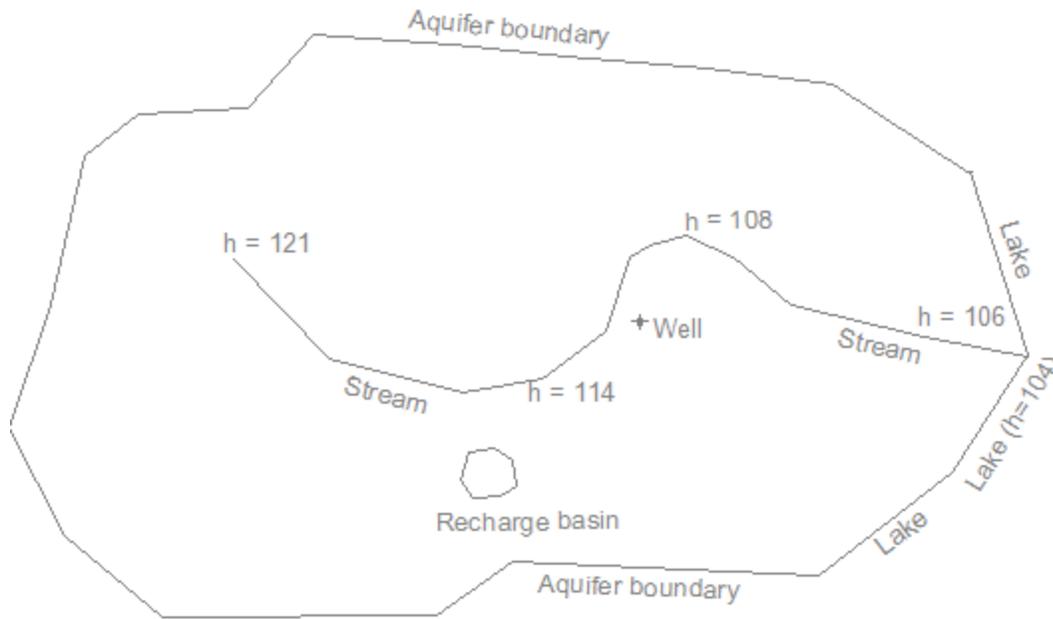
By leaving the *Window* cell blank, plots will include the entire area covered by the model elements. If you want to make a plot of a smaller window, coordinates can be entered in this cell.

Display the Basemap

Select *Make Plot/Model Elements Only* and then press the *OK* button in the dialog that shows.

Select *Zoom All* in the plot view menu, and you will see the basemap with these features:

- a lake shore along the east side
- an impermeable aquifer boundary to the south, west, and north
- a polygon recharge basin area
- a well
- a stream that goes from stage 121 in the west to 106 in the east



At this point, save your input by selecting *File/Save As*, and then choose a file name and path that suits you.

General Input

Begin creating model input by selecting *Model Input/General* under the main menu. Check the *Steady* cell since this will be a steady-state, not transient, model. Enter the units you choose for the model and a comment to characterize the model; these are just optional text entries to help document your model. Here is a completed example:

Data: general

Steady	Length_unit	Time_unit	Comments
<input checked="" type="checkbox"/>	m	days	First Tutorial

Units in AnAqSim

In AnAqSim, all input is in a consistent set of units. For example with meter and day units, discharges would be in m^3/day and hydraulic conductivities would be in m/day . AnAqSim doesn't keep track of what units you are using, so it is up to you to make sure all input is in terms of one consistent system of units based on the same length and time units.

Domain Input

Before entering other model inputs, we must first define the properties of the aquifers or regions within the model. AnAqSim uses the term *domain* to refer to regions within which the aquifer properties are constant. These can vary by layer in multilayer models and by location within a layer in heterogeneous models.

Select *Model Input/Domains/Confined* and/or *Unconfined* and edit the table so it has two completed rows and looks like the following.

Data: confined and/or unconfined domains

Label_unique	Level	Domain_Ty	Top_ele	Bottom_ele	Average_head	Porosity	Storativity	Specific_yi	K1_hori	K2_hor	Angle_K1_to_x	K3_vertical_to	K3_vertical_bot
all	1	unconfined	110	82	110	0.11	0.001	0.1	3	0.6	30	0.1	0.1
recharge basin	1	unconfined	110	82	110	0.11	0.001	0.1	3	0.6	30	0.1	0.1

The domain labels need to be unique because they are referenced by other kinds of model input such as wells and line boundaries. The “all” domain will span all of the model, except for the small polygon area that will be the “recharge basin” domain. These two domains are identical in their properties, but defining them separately allows us to define different recharge rates in the two domains.

The domains are both unconfined with a base at elevation 82. With unconfined domains the top elevation is not used.

Average_head is your best estimate of the average head in the domain, and this determines a constant that is added to the solution. Your choice of *Average_head* will not matter so long as it is close to the middle of the range of simulated heads in the domain.

Porosity is used in computations of travel time along pathlines. The storage parameters *Storativity* and *Specific_yield* have no bearing on a steady model.

The horizontal hydraulic conductivity is anisotropic, with $K_1=3$ and $K_2=0.6$. The K_1 direction is oriented 30 degrees counterclockwise from the positive x axis (30 degrees N from due E). The K_2 direction is normal to the K_1 direction.

The values of K_3 define the resistance to vertical flow in multi-level models or in single-level models with head-dependent leakage. Since this model has neither, the values of K_3 are not used.

Now you have defined the properties of the two domains. The location and extent of these domains are defined by the external line boundaries, which we will enter in the next sections.

Save your model again using *File/Save* or by typing the keyboard shortcut *Ctrl-S*.

Adding the Lake Boundary

Next we will add a constant head boundary at the lake shore using the digitizing features in AnAqSim. Before beginning that, please watch this [video about the plot tab](#), which describes the plot menus, zooming, panning, and data at the cursor.

Select *Model Input/Line Boundaries/Head-Specified* to bring up that data table. Edit the table so it looks like this:

Data: head-specified lines

Label	Domain	Parameters_per_line	Domain_Boundary	h_start	h_end	Coordinates	Off_periods
lake	all	3	<input checked="" type="checkbox"/>	104	104	Edit	
			<input type="checkbox"/>			Edit	

This line boundary is an external boundary to the “all” domain, hence the *Domain_Boundary* cell is checked. If this boundary was internal to the domain, this would not be checked. The specified head is set to 104 along this entire boundary.

The *Parameters_per_line* entry defines how many degrees of freedom each segment of the boundary will have and at the number of points on each segment where the specified head condition will be met perfectly. Larger numbers here will make for more accurate boundary conditions at the cost of more equations and

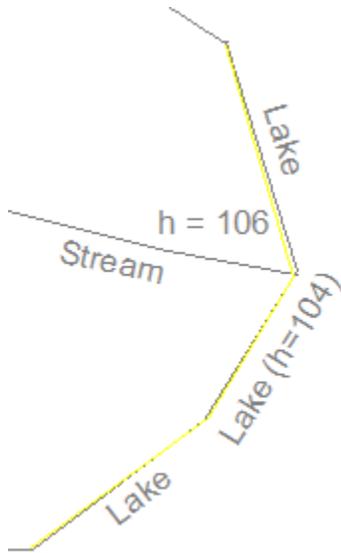
longer solve computation time. In transient models, you can turn the boundary condition off during the *Off_periods* you enter, but that has no bearing in this steady model.

Digitizing a line boundary

Next, you will digitize the coordinates of the line boundary. Before starting, please watch this [video about digitizing and snap settings](#).

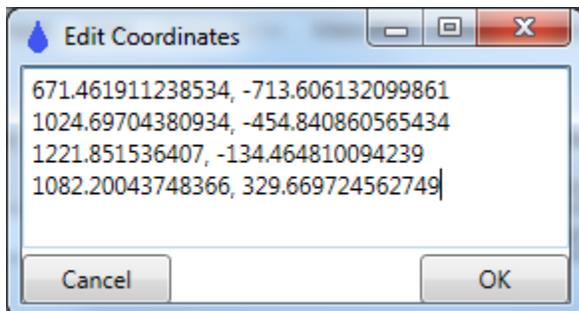
Click on the *Plot* tab to display the plot and basemap.

Digitize the lakeshore boundary either from N to S or S to N (for head-specified boundaries, direction doesn't matter). After digitizing, your digitized line (yellow) should look something like this:



Click on the *Data* tab to display the data table for head-specified line boundaries.

Click on the *Edit* button in the *Coordinates* column to show the coordinates window and then paste (*Ctrl-V*) the digitized coordinates in, resulting in something like this:



Press *OK* to save and close this window.

Save your model again using *File/Save* or by typing the keyboard shortcut *Ctrl-S*.

Adding the Impermeable Aquifer Boundary

Now we will add the impermeable aquifer boundary that forms the S, W, and N boundaries of the aquifer.

Select *Model Input/Normal Flux-Specified* to open the data table for this kind of boundary. Edit the table so it looks like this:

Data: normal flux-specified lines

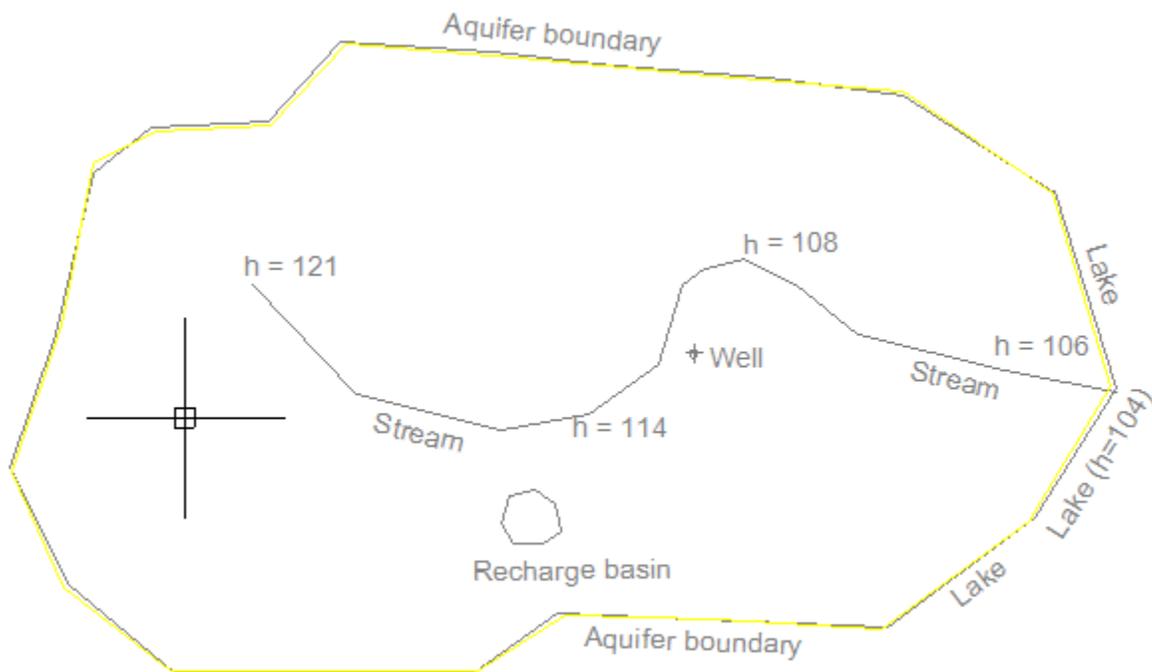
Label	Domain	Parameters_per_line	Domain_Boundary	Normal_flux_start	Normal_flux_end	Coordinates
outer	all	3	<input checked="" type="checkbox"/>	0	0	Edit
			<input type="checkbox"/>			Edit

This boundary is also external to the domain. It will be impermeable because the normal flux is set to zero along it. It is also possible to specify a non-zero flux in/out of the domain boundary.

The 3 parameters per line means there are three degrees of freedom per line segment, and the normal flux will be perfectly set to zero across three different sub-segments of each line segment.

Click on the *Plot* tab. Make sure the *Snap Settings* are checked for *Snap to Elements* and *Snap to Digitized Data*. This will assure that your digitized boundaries match perfectly at common vertices.

Digitize the "Aquifer Boundary" shown in the basemap, from the north end of the lake shore boundary (NW of the northernmost "Lake" on the basemap), going counterclockwise around the N, W, then S parts of this boundary until it ends at the south end of the lake shore boundary (SW of the southernmost "Lake" on the basemap)". Digitizing this type of boundary must be done counterclockwise, since the boundary conditions are imposed on the left side of the boundary as you proceed from the first toward the last vertex. See the User Guide (*Help*) for more information on this type of boundary. After digitizing, the plot will look like this:



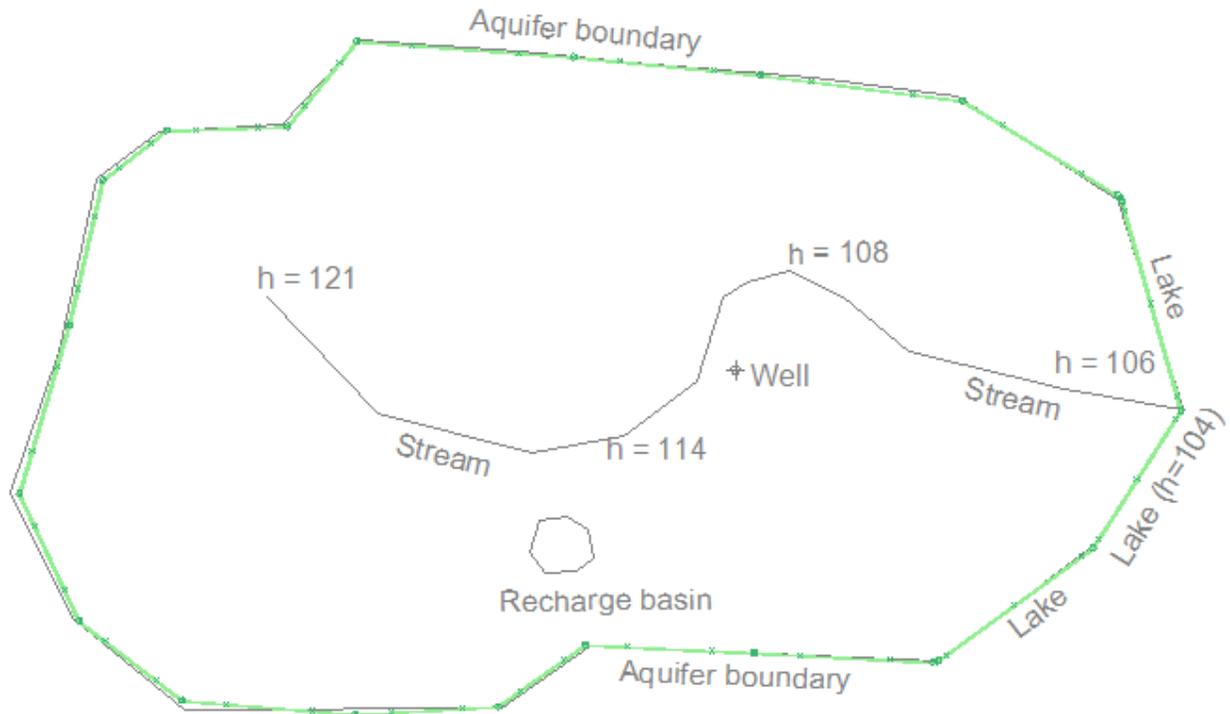
Click on the *Data* tab to display the data table for normal flux-specified line boundaries.

Click on the *Edit* button in the *Coordinates* column of the row for this boundary to show the coordinates window and then paste (*Ctrl-V*) the digitized coordinates in, then press *OK* to save and close this window.

Save your model again using *File/Save* or by typing the keyboard shortcut *Ctrl-S*.

Plotting the Elements and Examining Details

Make a plot of the model elements that have been input so far by selecting *Make Plot/Model Elements Only*, then selecting Level 1 and pressing OK. The resulting plot will look as follows, with the entered line elements shown green.



If you move the cursor over a line boundary, it turns black and a pop-up text window defines the boundary. If you hover over the small circles at vertexes, it displays the boundary condition specifics at that vertex.

Graphically Editing a Line Boundary

Please see this [video on how to graphically edit well and line boundary coordinates](#).

Select the outer impermeable “outer” aquifer boundary and experiment with moving, adding, and deleting vertexes in the line boundary, then select *Make Plot/Model Elements Only* to see the altered line boundary.

Adding River Line Boundaries

We will add three river line boundaries to represent three reaches of the stream shown in the basemap. We will start with the lowest reach.

Select *Model Input/Line Boundaries/River*, and edit the table so it looks like this:

Data: river lines

Label	Domain	Parame	Dries_up	Stage_st	Stage_end	Conductance_st	Conductance_er	Base_resisting_layer_sta	Base_resisting_layer_en	Coordinates
lower	all	3	<input type="checkbox"/>	106	108	0.3	0.3	104	106	Edit
			<input type="checkbox"/>							Edit

River boundaries are always internal to a domain. By not checking *Dries_up*, this river reach will maintain its stage and leak to/from the domain, even if the domain heads drop below stage. The starting and ending

stages correspond to the vertices labeled “h=106” and “h=108” on the basemap. See the User Guide (*Help*) for more information on this type of boundary.

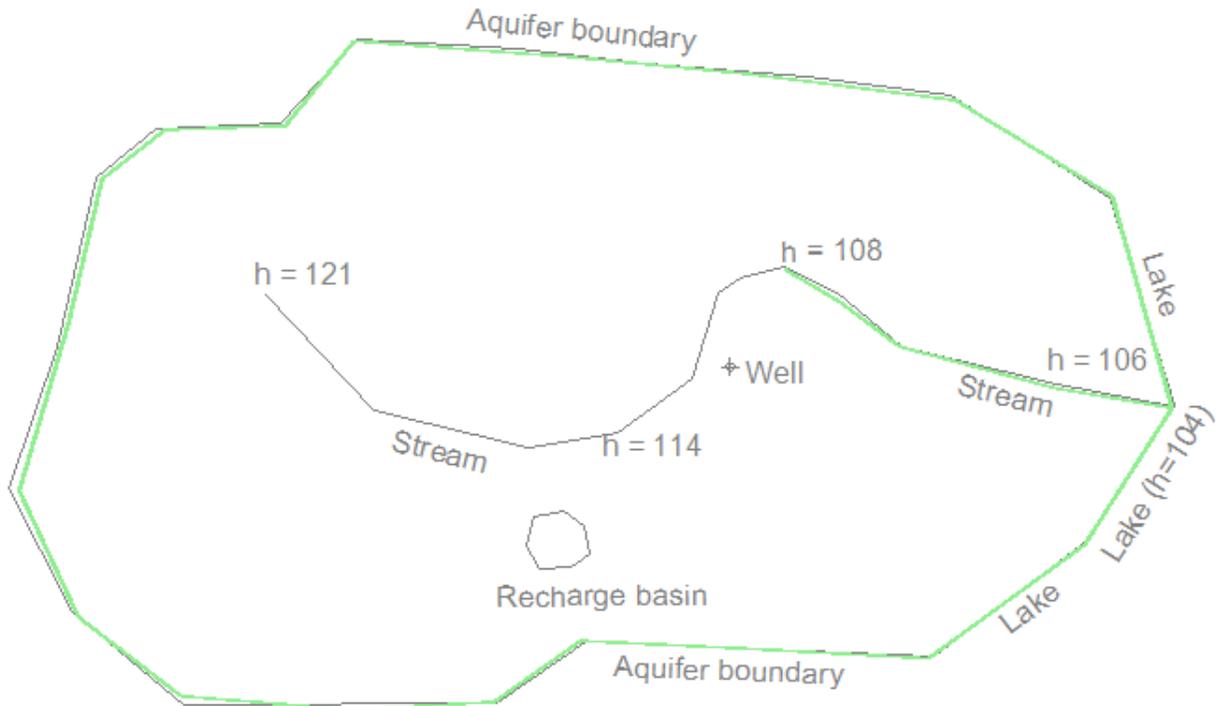
Click on the *Plot* tab.

Right-click anywhere on the plot area and select *Digitize/Polyline* from the right-click menu. Digitize the lower (east) river reach, starting at the east end where it meets the lake boundary, and ending at the vertex labeled “h=108”.

Click on the *Data* tab to display the data table for river line boundaries.

Click on the *Edit* button in the *Coordinates* column to show the coordinates window and then paste (*Ctrl-V*) the digitized coordinates in, then press OK to save and close this window.

Select *Make Plot/Model Elements Only* to make a plot that should look like this:



Repeat these steps to create the input for the middle (“h=108” to “h=114”) and upper (“h=114” to “h=121”) reaches of the stream, digitizing and entering the coordinates as you did with the lower reach:

Data: river lines

Label	Dom	Para	Dries_up	Stage_st	Stage_er	Conductance_st	Conductance_er	Base_resisting_layer_st	Base_resisting_layer_er	Coordinates
lower	all	3	<input checked="" type="checkbox"/>	106	108	0.3	0.3	104	106	Edit
middle	all	3	<input type="checkbox"/>	108	114	0.3	0.3	106	112	Edit
upper	all	3	<input type="checkbox"/>	114	121	0.3	0.3	112	119	Edit
			<input type="checkbox"/>							Edit

Save your model again using *File/Save* or by typing the keyboard shortcut *Ctrl-S*.

Adding the Interdomain Boundary

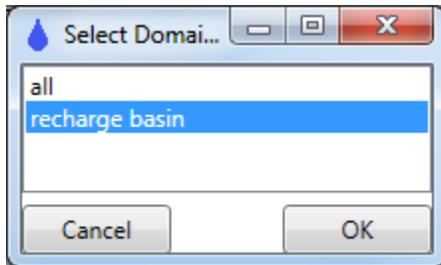
We will now add the interdomain boundary that defines the limits of the “recharge basin” domain. Select *Model Input/Line Boundaries/Interdomain*, and edit the table so it looks like this:

Data: inter-domain lines

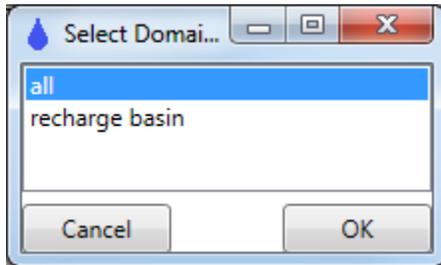
Label	Domains_left	Domains_right	Parameters_per_line	Coordinates
ID	Select	Select	4	Edit
	Select	Select		Edit

With 4 *Parameters_per_line*, heads will match perfectly at 4 points on each line segment, and discharges across the boundary will match perfectly for 4 subsegments of each segment. See the User Guide (*Help*) for more information on this type of boundary.

Next, click the *Select* button in the *Domains_left* column to select the domains that will be to the left of this boundary. Then select the recharge basin domain like below and click *OK*.



Now do the same for *Domains_right*, selecting the “all” domain as below.



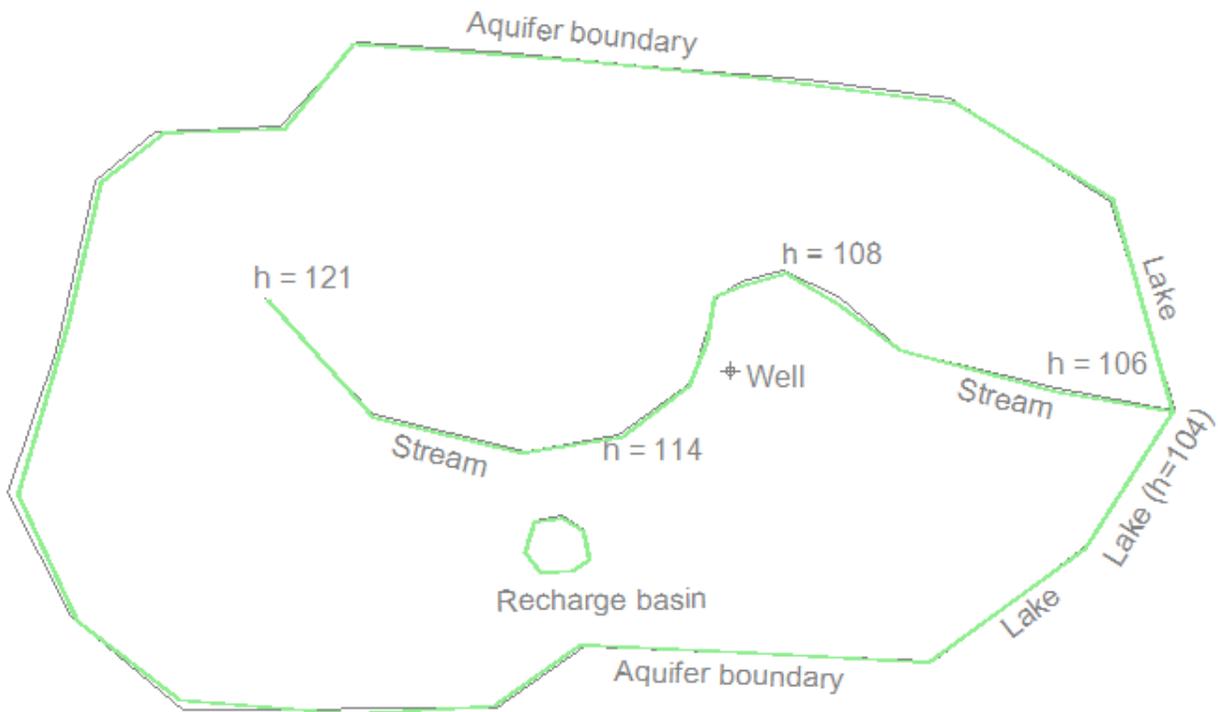
Click on the *Plot* tab.

Right-click anywhere on the plot area and select *Digitize/Polyline, Measure Distance* from the right-click menu. Digitize the recharge area boundary going in a counterclockwise direction, starting and ending at the same point. Imagine walking along this boundary on the ground in the same counterclockwise direction as you digitized; the “recharge basin” domain would be on your left and the “all” domain would be on your right.

Click on the *Data* tab to display the data table for interdomain line boundaries.

Click on the *Edit* button in the *Coordinates* column to show the coordinates window and then paste (*Ctrl-V*) the recently-digitized coordinates in, then press *OK* to save and close the coordinates window.

Select *Make Plot/Model Elements Only* to make a plot that should look like this:



Adding the Well

To add the well, we will first digitize the well's location by right-clicking over the plot and then selecting *Digitize/Point*, then move the cursor to the location of the well (zoom in if you want to get a more accurate point) and then left-click. Then select *Model Input/Pumping Wells/Discharge-Specified* and paste the digitized point coordinates into the Coordinates column and complete the other columns as shown below. Make sure to press enter to register the new row in the database, which exposes a new blank row.

Data: discharge-specified wells

Label	Domain	X,Y	Radius	Discharge
W	all	221.927344116928, -52.0001952335555	0.3	-414

The negative sign for the *Discharge* means the well is pumping water out of the aquifer.

If you select *Make Plot/Model Elements Only*, you should now see the green well element over the basemap gray:



Adding Uniform Recharge

Next we will add in a uniform recharge rate over the “all” domain and a higher uniform rate over the “recharge basin” domain. Select *Model Input/Area Source/Sink/Uniform, Domain* and edit the table so it looks like this:

Data: uniform area source/sink, by domain

Domain	Top_flux	Bottom_flux
all	0.0006	0
recharge basin	0.008	0

The *Top_flux* is the recharge rate [length/time] that applies at the top of the domain. You could also specify a uniform rate of leakage into or out of the bottom of the domain. For both the top and bottom fluxes, a positive value means flux into the domain and a negative value means flux out of the domain. In this case, the bottom of each domain is impermeable with zero flux. The recharge rate is substantially higher in the “recharge basin” domain than in the surrounding “all” domain. Uniform area sinks are appropriate in this model, but if there were multiple levels with vertical leakage or significant transient storage fluxes, you would use spatially-variable area sinks (SVAS) instead.

Solution Settings

The model has many boundary conditions that are associated with unknowns that need to be solved for. To do that, AnAqSim builds a system of simultaneous equations and then applies a matrix-solving routine to determine the unknowns. There are several settings that apply to the solve process. Select *Model Input/Solution/Solve Settings*, to bring up some of these settings:

Data: solve settings

Underrelaxation	Maximum_iterations	Starting_heads_source	Almost_dry_fraction	Interface_leakage_option
0.9	20	file	0.01	<input type="checkbox"/>

These are the default settings and work reasonably well in most models. If your model is not converging or taking many iterations to converge, you can adjust some of these inputs to improve convergence. We will not change any of these, but if you want to understand these inputs, see explanations in the User Guide.

Select *Model Input/Solution/Check Settings*, to bring up the settings that determine how accurate a solution must be in order to achieve convergence:

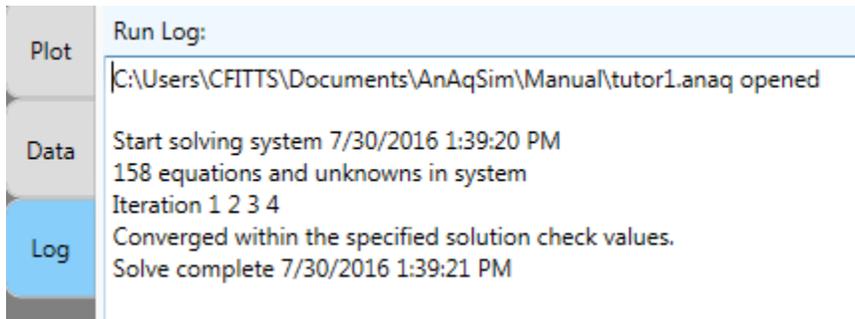
Data: solution check settings

Head_check_tolerance	Q_check_tolerance	Qn_check_tolerance	Extraction_check_tolerance
1e-3	1e-3	1e-4	1e-6

These are default settings that are reasonable if your length unit is feet or meter, and your time unit is days. For example, all head-specified boundary conditions must converge to within 0.001 units [L]. Decreasing these values improves boundary accuracy at control points, but at the expense of increased numbers of iterations to achieve these criteria. Please see the User Guide for definitions of each of these.

Solving the Model

This is accomplished by simply selecting *Solve* in the main menu. When you do that, the *Log* tab is displayed and information about the solve process is written there. In my model, this is what was written (your model may differ depending on how you digitized in your line boundaries).



The system of equations included 158 equations and it took 4 iterations to achieve boundary conditions accurate enough to meet the solution check criteria. The time stamps show that this took only about one second.

Plot Input Settings

Before making a plot, we need to decide what to plot and establish some settings.

What to Plot

Select *Plot Input/What to Plot* and edit the table so it looks like this:

Data: what to plot

Window	Basemap	Basemap_File	Elements	Contours	Pathlines	Vectors	SVAS_polygons	AreaPathline_polygons	Head_Calibration	Analysis_Inputs
	gray	Select	elements w/ details, control points	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>

Leaving the *Window* column blank will default to making a plot of the entire model. The plot will include the basemap (gray), elements with details and control points (green), contours of head (blue), and velocity vectors (red).

Contour Settings

Select *Plot Input/Contour Settings* and edit the table so it looks like this:

Data: contour settings

Surface	Subtract	Points_Evaluated	Increment	Minimum	Maximum
h	no	20000	0.5		

In this case, the surface contoured is head (h). See the User Guide for a description of other surface choices.

The *Subtract* option is for subtracting this surface from a previously saved surface, which is handy for drawdown and other comparison plots. In this case, we will not subtract and just contour heads. The *Points_Evaluated* sets the number of points where head will be evaluated. These points are the basis for drawing contours, and are arrayed on a regular grid. A larger number here makes for smoother, more accurate contours at the cost of more computation. *Increment* is the increment between adjacent contours. If *Increment* is left blank, AnAqSim will set the increment such that about 25 contours are drawn.

Minimum and *Maximum* define the range of contours drawn. If these are left blank, AnAqSim selects minimum and maximum levels that span the range of values among the points evaluated.

Vector Settings

Select *Plot Input/Contour Settings* and edit the table so it looks like this:

Data: vector settings

Vector_Type	Points_Evaluated	Scale_Factor
v	600	1

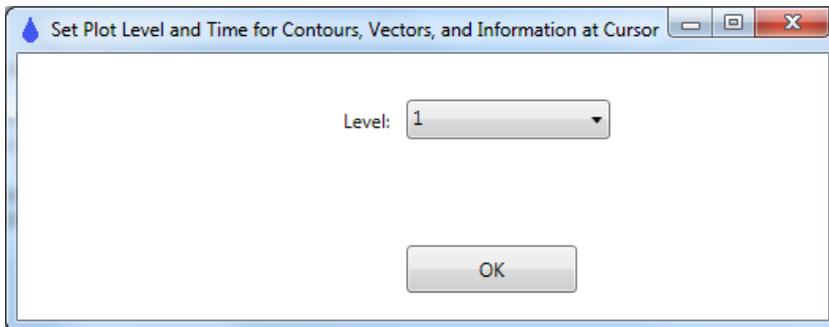
In this case, vectors of average linear velocity (v) are created. Other choices include specific discharge (q) and aquifer discharge (Q), which is specific discharge times saturated thickness.

Points_Evaluated sets the number of points where vectors will be evaluated. A larger number here makes for more, but shorter vectors on the plot. Selecting a larger number here can make the plot too busy.

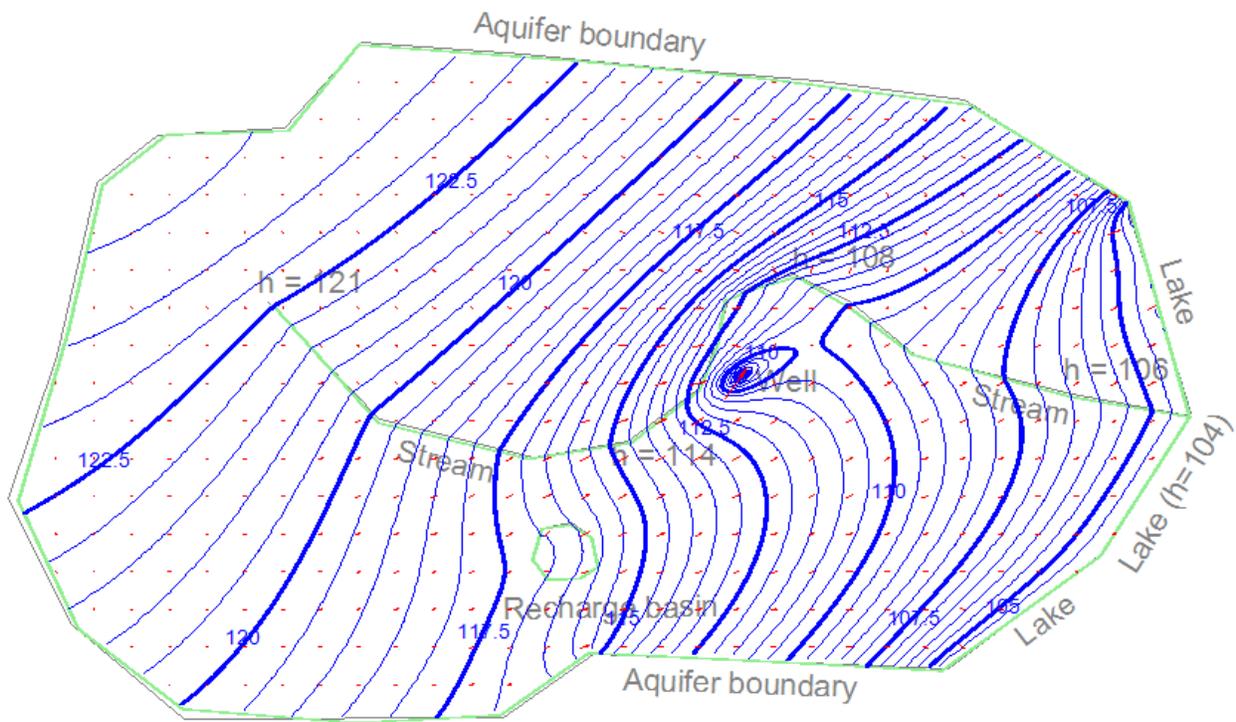
Scale_Factor lets you scale the length of all vectors drawn up or down by entering a larger or smaller number. Try 1.0 here and adjust as needed.

Making a Plot

To make a plot that includes the features you just entered, select *Make Plot/All Selected Features*. This will bring up a dialog where you can select the level (in this model, there is only Level 1):



Press the *OK* button and the plot that that results should look like this:



Data at Cursor Location and Zooming and Panning

When you move the cursor around the plot, you will notice that there is data at the left that is updated as you move the cursor. See this [video about the plot tab](#), which describes the plot menus, zooming, panning, and data at the cursor.

At this point, experiment with changing some of the contour and vector settings. If your mouse has a scroll wheel you can zoom in and out using the scroll wheel while the cursor is over the plot. If you press down on the scroll wheel and drag, the plot will pan with your dragging motion.

Analysis of the Solution

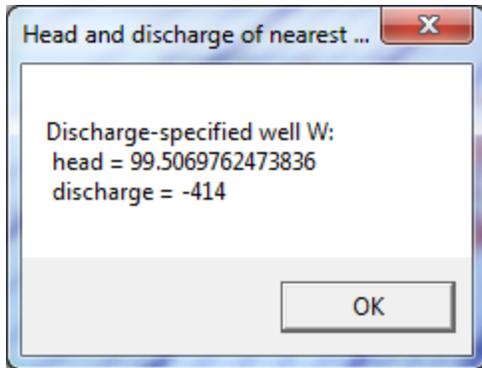
AnAqSim has many tools to analyze the solution and check its accuracy and outputs.

Well Heads and Discharges

There are a couple of ways to check the solution results at wells. One is using the main menu, and the other is using the right-click menu. In this model we have one well and it is a discharge-specified well. To check results with the main menu, select *Analysis/Discharge-Specified Well Heads/Write Heads to Run Log*, which will print the well head to the run log:

Log	Heads at discharge-specified wells: W: 99.50697
-----	--

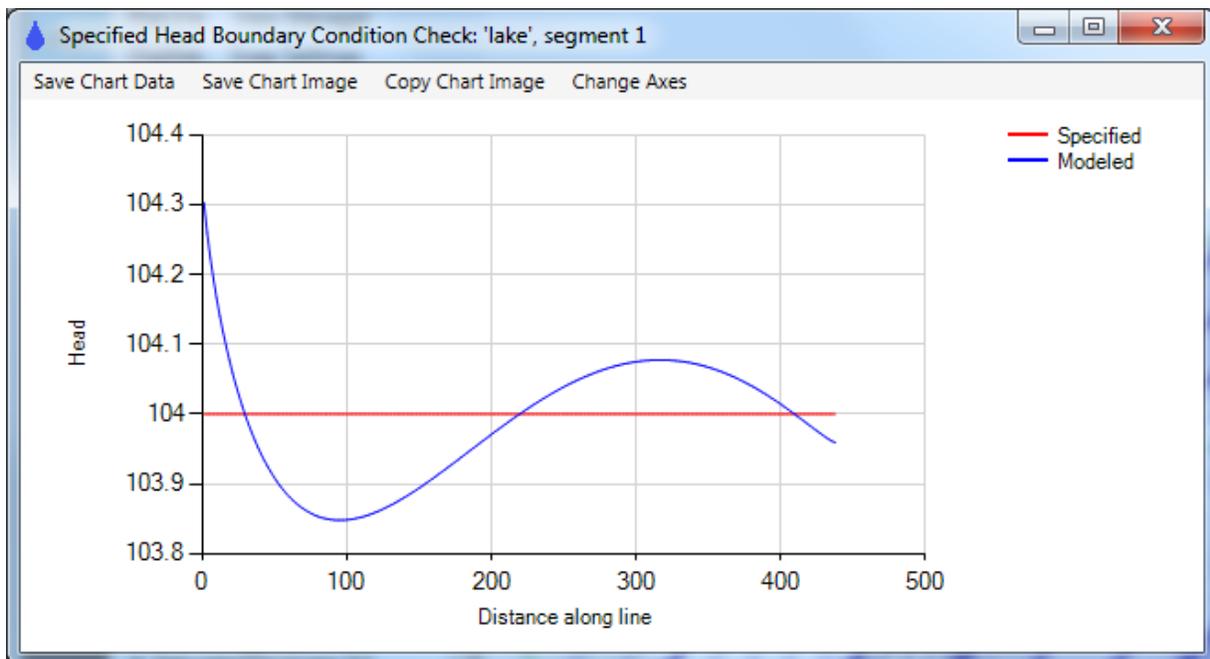
Alternatively, you can select the *Plot* tab, then move the cursor near the well, right-click, and select *Check Nearest Well Head and Discharge*. This will cause a window like this to display with both the head and discharge of the well:



Click *OK* to close this window.

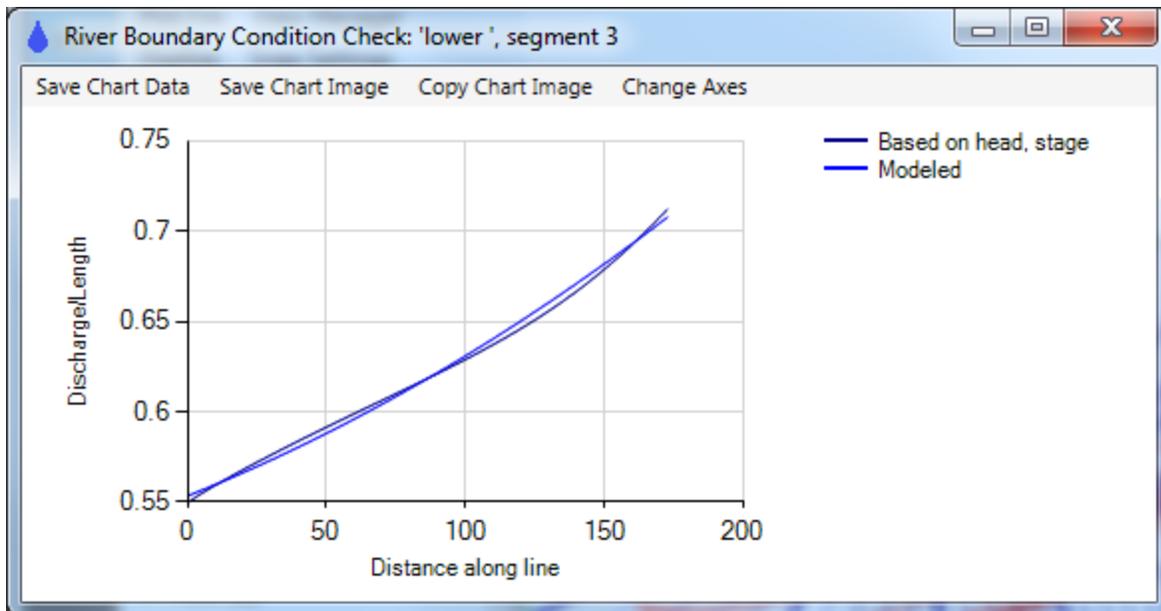
Checking Line Boundary Conditions

These can be checked using the right-click menu. To do this, move the cursor near a line boundary line segment, right-click, and then select *Check Nearest Line Boundary Condition*. This causes a graph to be displayed of the specified boundary condition vs. the modeled boundary condition. If you do this for one of the “Lake” specified head line boundary segments at the east side of the model, the graph will look something like this:



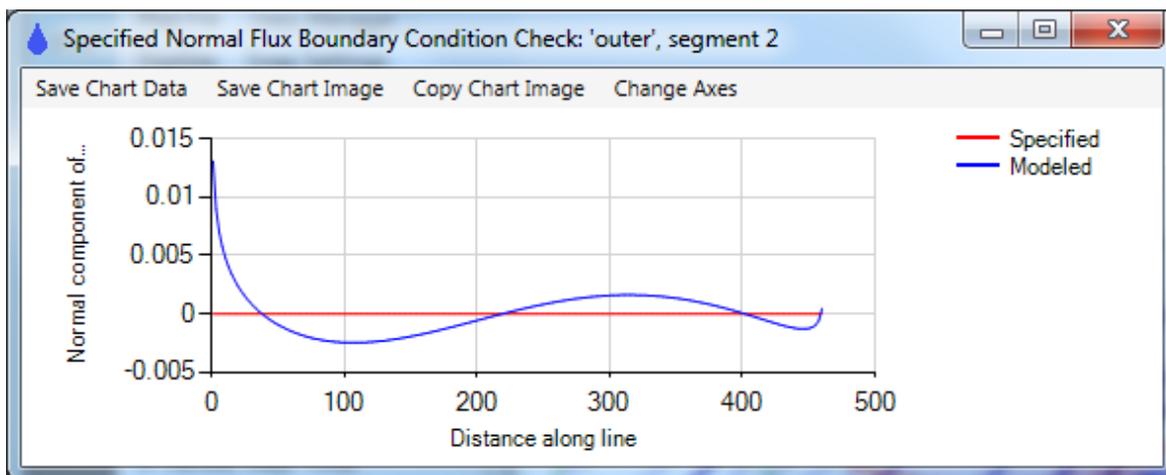
You can see that we specified $h=104.0$ all along this boundary and that the modeled head is close to 104.0 but varies along the segment. The modeled head is exactly 104.0 at three points; these are the 3 control points where equations were written to achieve $h=104.0$. In between these control points, the specified boundary condition is approximated. Because you specified 3 parameters per line for this line boundary, there are 3 control points and head is exactly equal to the specified value at the 3 control points. You could improve the accuracy by increasing the number of parameters per line (more on that in the next section).

Now repeat the line boundary check at other kinds of line boundaries by repeating the sequence of moving the cursor near a line boundary line segment, right-clicking, and selecting *Check Nearest Line Boundary Condition*. If you check one of the river line boundary segments, you should see a plot like this:



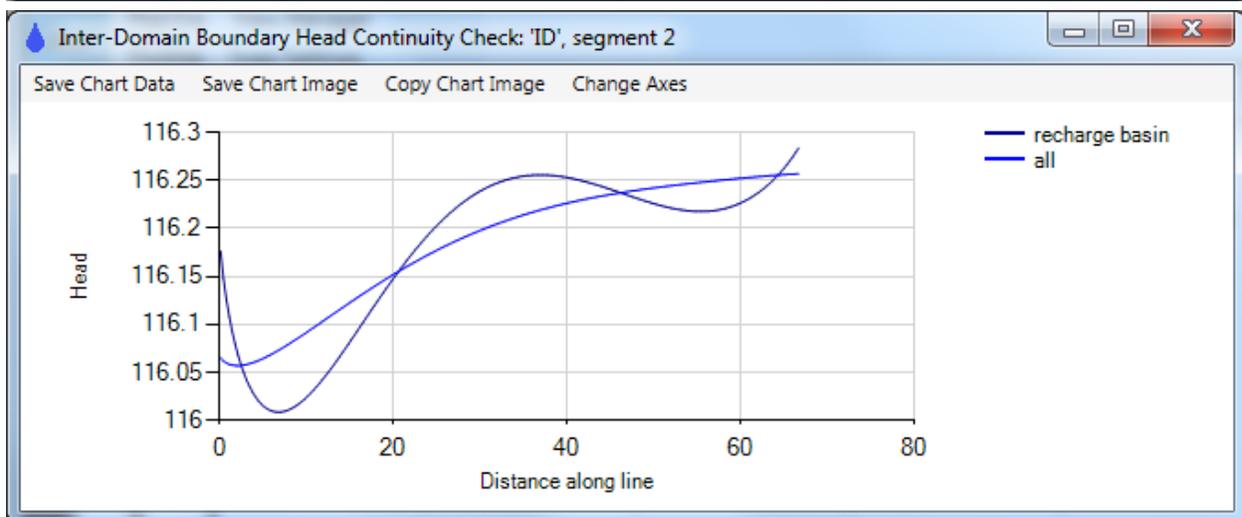
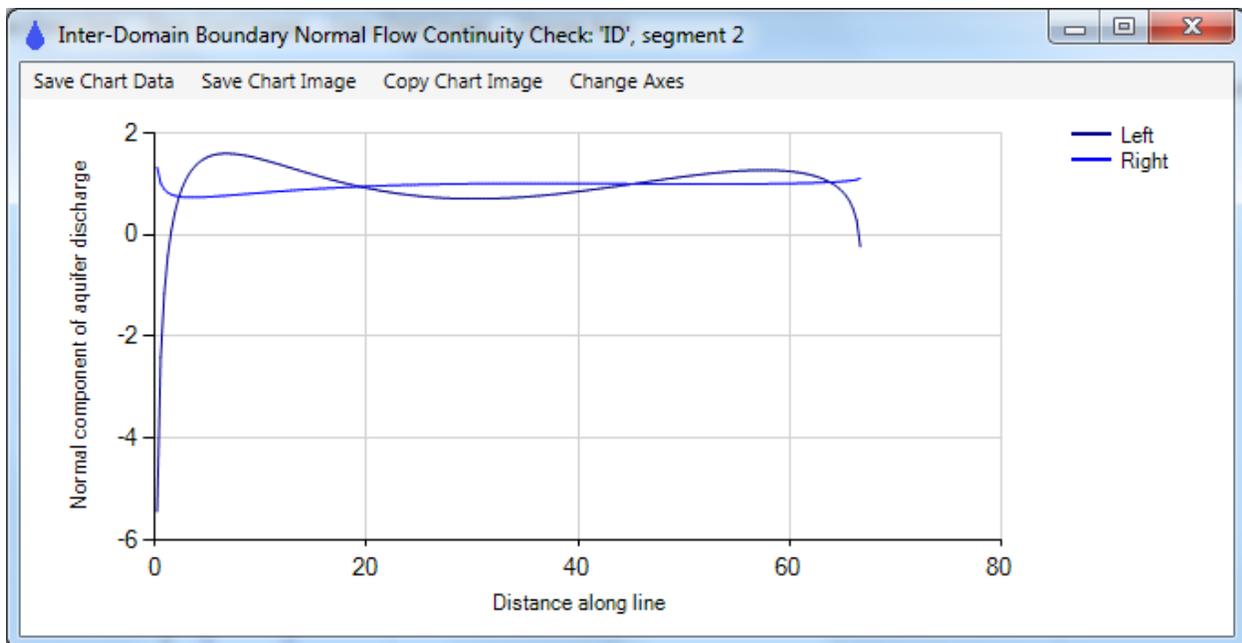
In this case the modeled and desired discharge per length (based on head, stage, and riverbed resistance) are quite close. They are exactly the same at three points, since there were 3 parameters per line and 3 control points.

Checking one of the outer normal flux-specified line boundaries results in a plot like this:



The boundary has a specified normal flux = 0 (impermeable). The specified normal flux is zero over 3 subsegments of this segment, but deviates from 0 along the line. The specified boundary condition is met perfectly over these three subsegments, but is approximated at points along the segment. Specifying a larger number of parameters per line segment will mean more, smaller subsegments over which the condition is met perfectly.

Checking one of the interdomain boundary segments that form the boundary of the “recharge basin” domain results in two plots as follows. At interdomain boundaries, two conditions are imposed: continuity of discharge normal to the boundary, and continuity of head at the boundary.



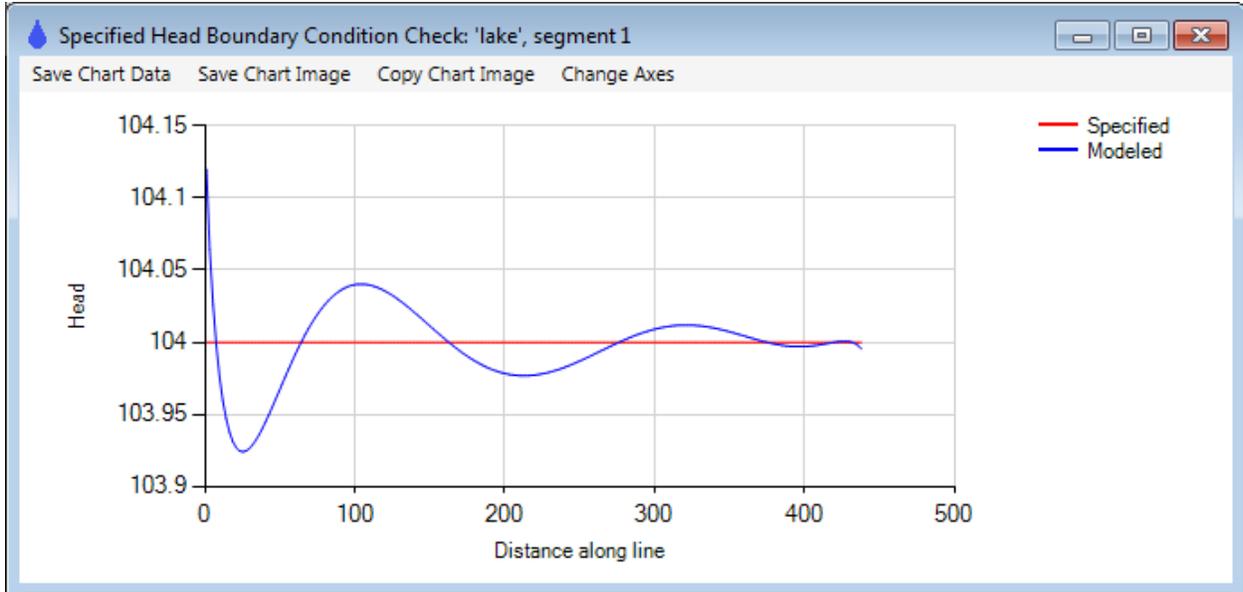
The upper plot compares the normal component of aquifer discharge for the domains to the left (“recharge basin”) and right (“all”) of the boundary. Like the normal-flux specified boundary, this condition is met perfectly over subsegments of the segment. In this case, with 4 parameters per line, there are 4 subsegments. The head continuity condition is met perfectly at 4 control points and is approximate in between.

Changing Line Boundary Accuracy with Parameters per Line

We saw in the previous section that the modeled boundary conditions along line boundaries are approximations and their accuracy is a function of how long the segments are and how many parameters per line segment are specified. To demonstrate this, consider the check of the head-specified line boundary discussed in the previous section. Next we will demonstrate how the accuracy of the modeled boundary conditions changes when you increase the number of parameters per line.

Select *Model Input/Line Boundaries/Head-Specified* from the main menu. Edit the lake line boundary so that it has 6 *Parameters_per_line* instead of 3. Make sure you press enter after changing the value to 6, to

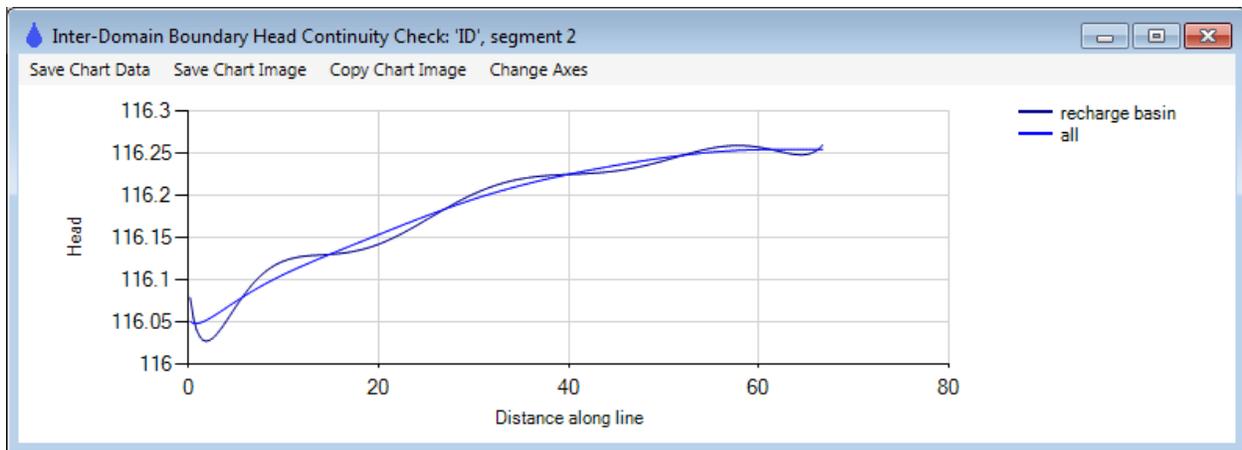
register the change with the underlying database. Press Solve to solve the system with this change, and note that the number of equations is greater, now that there are more equations per line segment on this boundary. Select *Make Plot/All Selected Features*, then move the cursor near the same “Lake” line boundary segment as before, right-click and select *Check Nearest Line Boundary Condition*, which will create a plot like this:



Compare this with the [check of the same boundary segment, but with only 3 parameters per line](#). Now, instead of meeting the $h=104.0$ condition perfectly at 3 points, it is met perfectly at 6 points, and the magnitude of deviations from 104.0 are smaller.

If you experiment with the interdomain line boundary in the same way, changing from 4 to 8 parameters per line, you will see similar improvements in accuracy. Here are plots for 8 parameters per line to compare with [those in the previous section](#) with 4 parameters per line:

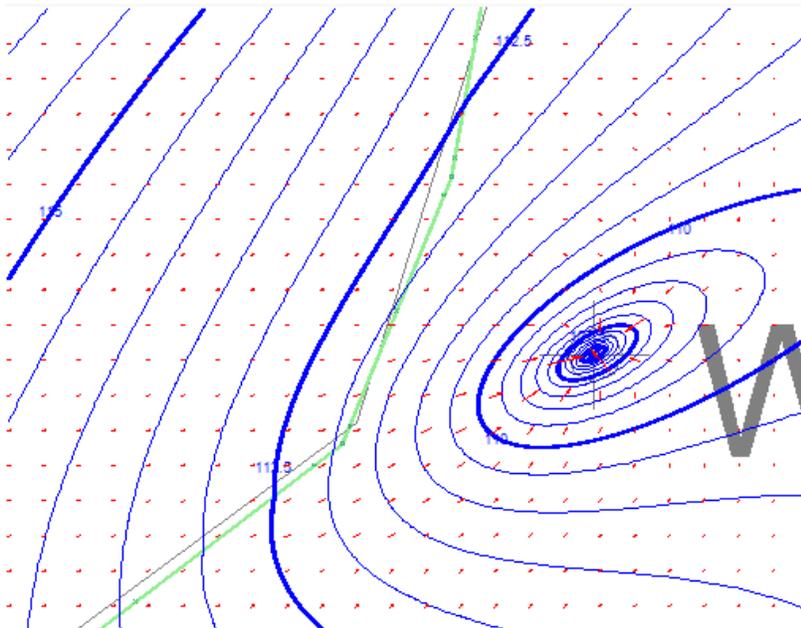




At this point, save your model again.

Making a Plot in a Smaller Window

It is often useful to make close-up plots of certain areas in a model. To do this, first use the zoom/pan features (right-click or use the plot menu at the left) to zoom in on a smaller area of the model. Then right-click over the plot and select *Set Plot Window to Current View*. Now when you select *Make Plot/All Selected Features*, you will see a plot of that close-up area, something like this, although you probably zoomed to a different window:



Adding Pathlines to the Plot

Now, instead of showing vectors, we will show pathlines in the plot. To do that, select *Plot Input/What to Plot* and uncheck the vectors column and check the pathline column, then press enter to register the change. While you are there, clear the entry in the *Window* column so that subsequent plots will be of the entire model area:

Data: what to plot

Window	Basemap	Basemap_File	Elements	Contours	Pathlines	Vectors	SVAS_polygons	AreaPathline_polygons	Head_Calibration	Analysis_Inputs
	gray	Select	elements w/ details, control points	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>				

Pathline Settings

Next, adjust the pathline settings that apply to all pathlines by selecting *Plot Input/Pathline Settings* and edit the row of data like this:

Data: pathline settings

Step_size	Time_markers	Time_marker_increment	Total_time	Start_time	Freeze_transient	Capture_constrain	Capture_wells	Capture_lines
0.005	<input checked="" type="checkbox"/>	200	20000	0.0	<input type="checkbox"/>	<input type="checkbox"/>	Select	Select

Pathlines are traced in small, straight segments, each with a length equal to *Step_size* times the largest (x or y) dimension of the plot window. As pathlines are traced, the travel time is computed, and if *Time_markers* is checked, there will be arrows drawn at time intervals on the pathline equal to *Time_marker_increment*. The pathlines will be traced until they exit the model, exit the window, or reach the *Total_time* of travel. *Start_time* is not used in steady models, but is used in transient models, where pathlines trace through the transient flow field. By entering a very large *Total_time*, pathlines can be traced to their full extent. The User Guide explains the other choices under pathline settings, but we don't need those now.

Well Pathlines

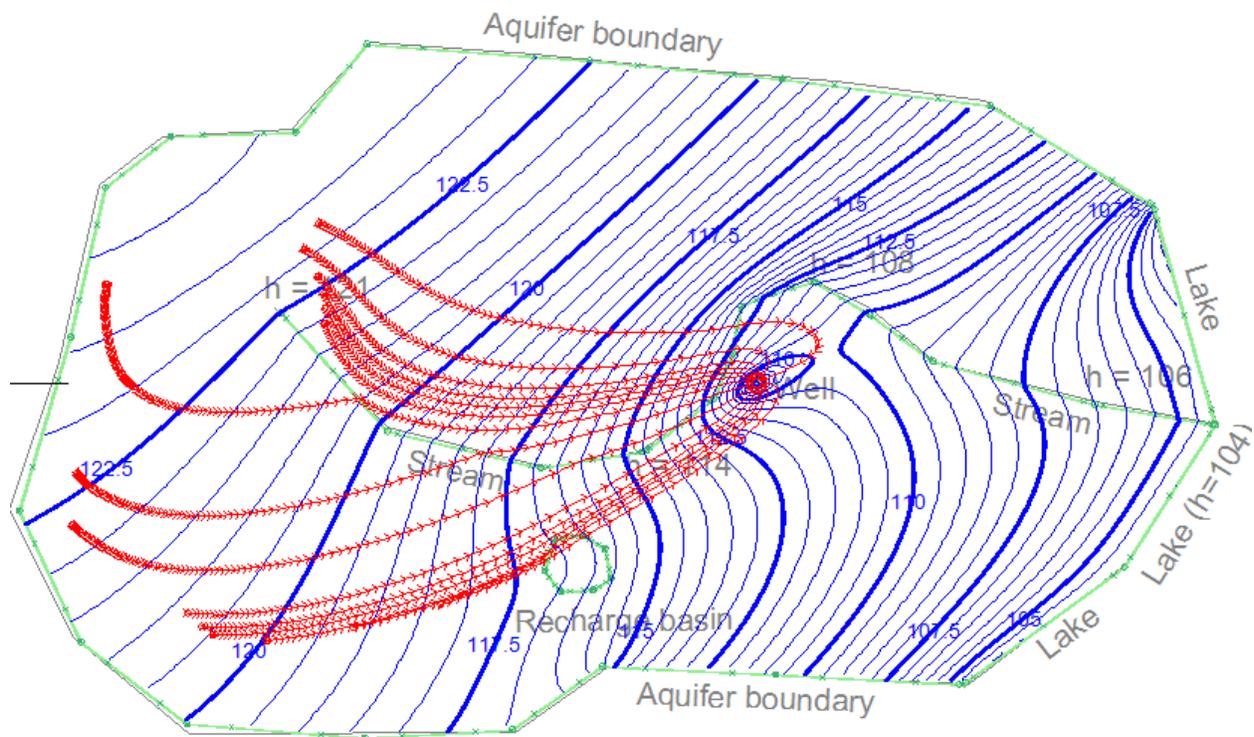
Pathlines are commonly used to show the region of water captured by a well, what is commonly called the *capture zone*. Creating pathlines that converge on a well is easy. Select *Plot Input/Pathlines/Well* and create a row in the table like this:

Data: well pathlines

Well Label	Number	Start_level	Show
W	15	1.8	<input checked="" type="checkbox"/>
			<input type="checkbox"/>

These pathlines will emanate from well W (the only well in the model), and head upstream since the well is extracting water. For injecting wells, pathlines trace downstream. The *Start_level* = 1.8 means that pathlines will start 80% of the way from the top of level 1 towards the bottom of level 1. If you had entered *Start_level* = 1.2, the pathlines would enter the well 20% of the way down from the top to the bottom of the saturated thickness of level 1. The *Show* button allows you to display certain pathlines or not display them. There will be 15 pathlines entering the well equally spaced around the well, just outside the well radius.

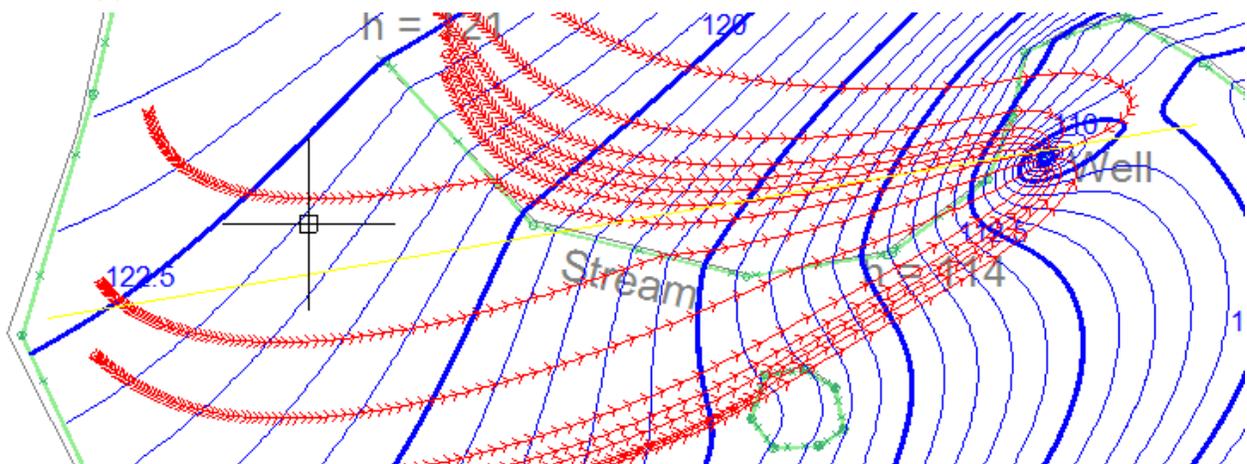
Select *Make Plot/All Selected Features* to make a plot with these pathlines and pathline settings, which should look like this:



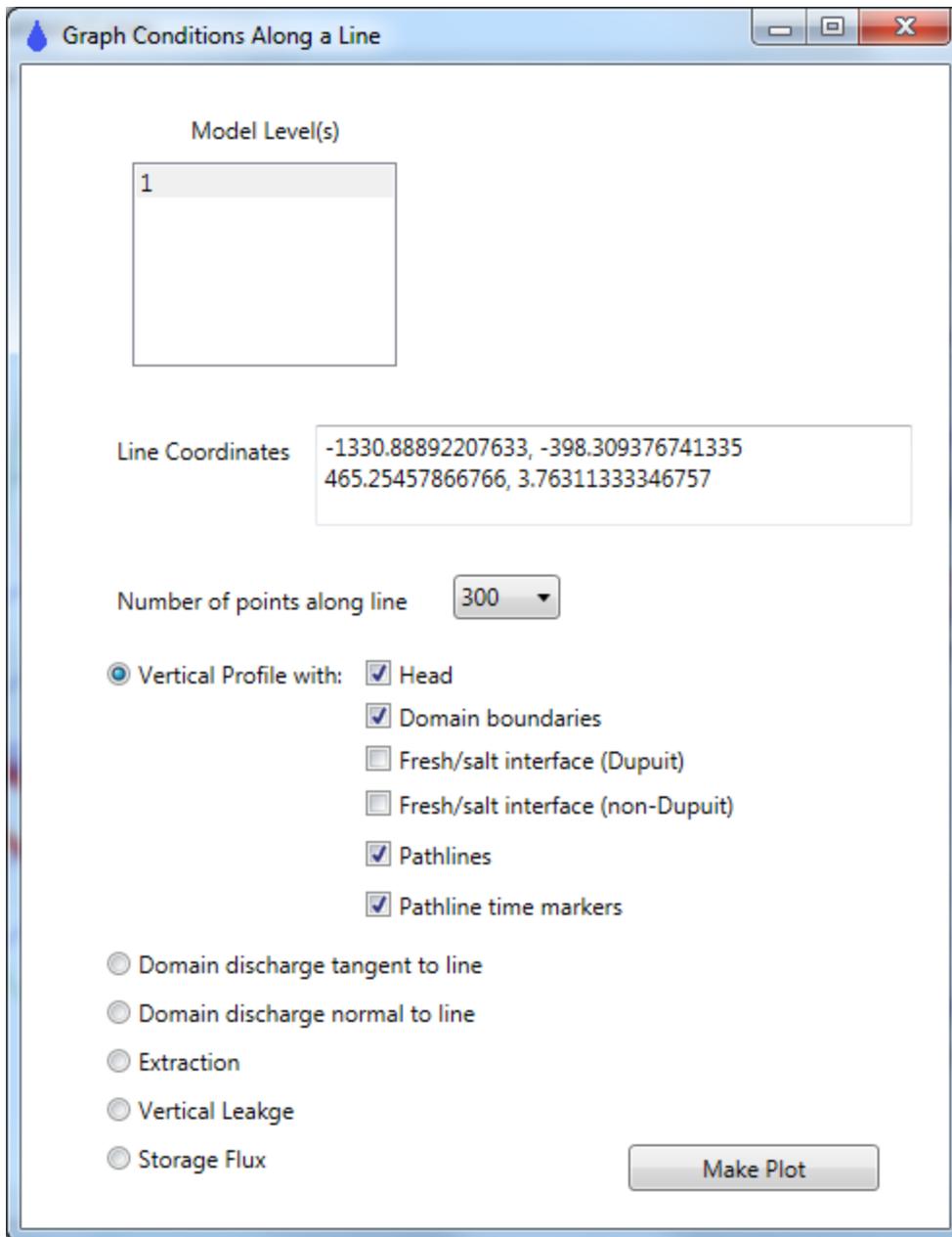
If you move the cursor over the pathline arrows, you will see information about the elapsed time and elevation of the pathline. If you move to the upstream end of the pathlines, you will see that they end at a red circle. Hover over the circle to see information about the end of the pathline; in this case, the pathlines end at the water table. These “end” points are actually starting points – water that enters the well 80% of the way down from the top of the saturated thickness originated at the water table at these locations marked by the small circles.

Vertical Profile with Pathlines

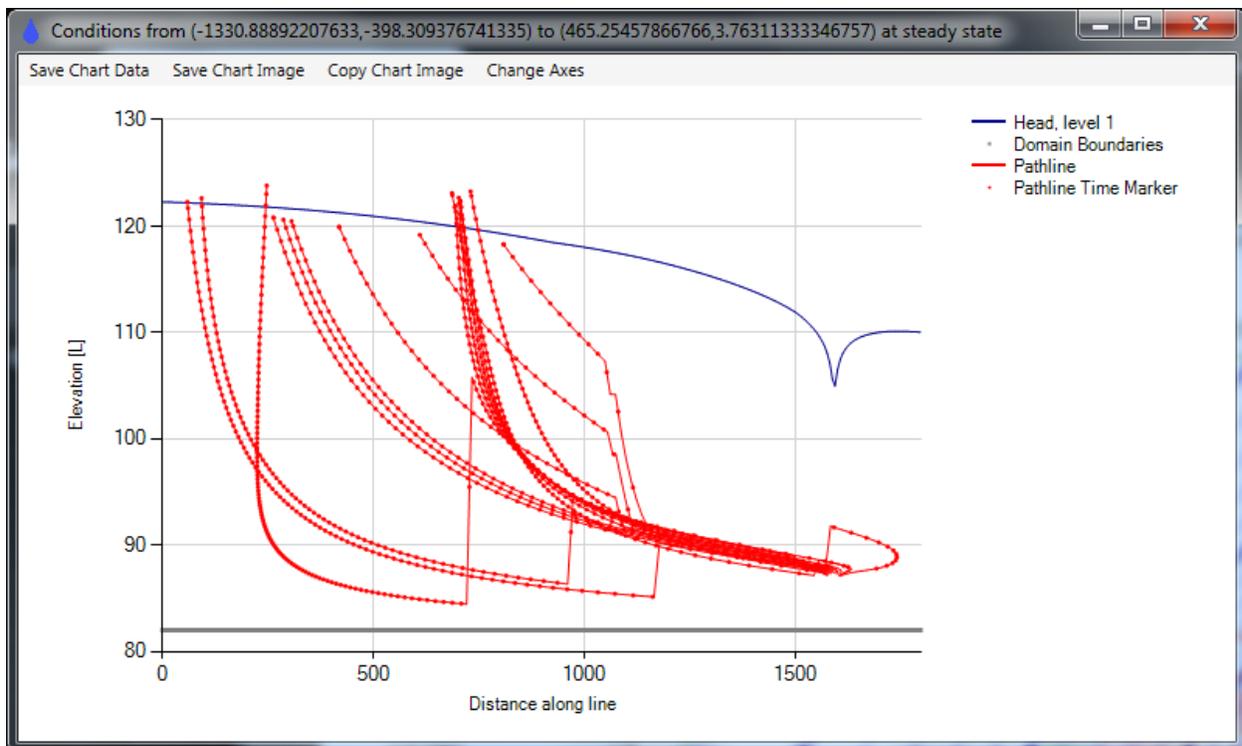
Next, we will create a vertical profile that shows how these pathlines migrated in the vertical direction. Right-click and select *Digitize/Polyline*, then digitize a line something like the yellow line shown in the following plot:



This will be the line that pathlines will be projected onto in the profile plot. Select *Analysis/Graph Conditions Along a Line* and then edit the dialog so it looks like this (you can paste in the digitized line coordinates):



This will make a profile along the line that shows head, domain boundaries, pathlines, and pathline time markers projected onto the line. Press the *Make Plot* button to generate a profile like this:



The pathlines originate at the water table at various elevations near 120. Some pathlines appear to originate above or below the water table because these pathlines are projected on to the line from locations significantly to the side of the line. The head profile is exactly on the line. Note that some pathlines travel through the recharge basin at a distance of about 1100 on the x axis; these pathlines plunge steeply due to the high recharge rate in the recharge basin. Each time marker dot is drawn at the location of an arrow on the map-view plot. Although the model is two-dimensional (one level), pathlines are traced in three dimensions using mass balance, a method first outlined by [Strack \(1984\)](#).

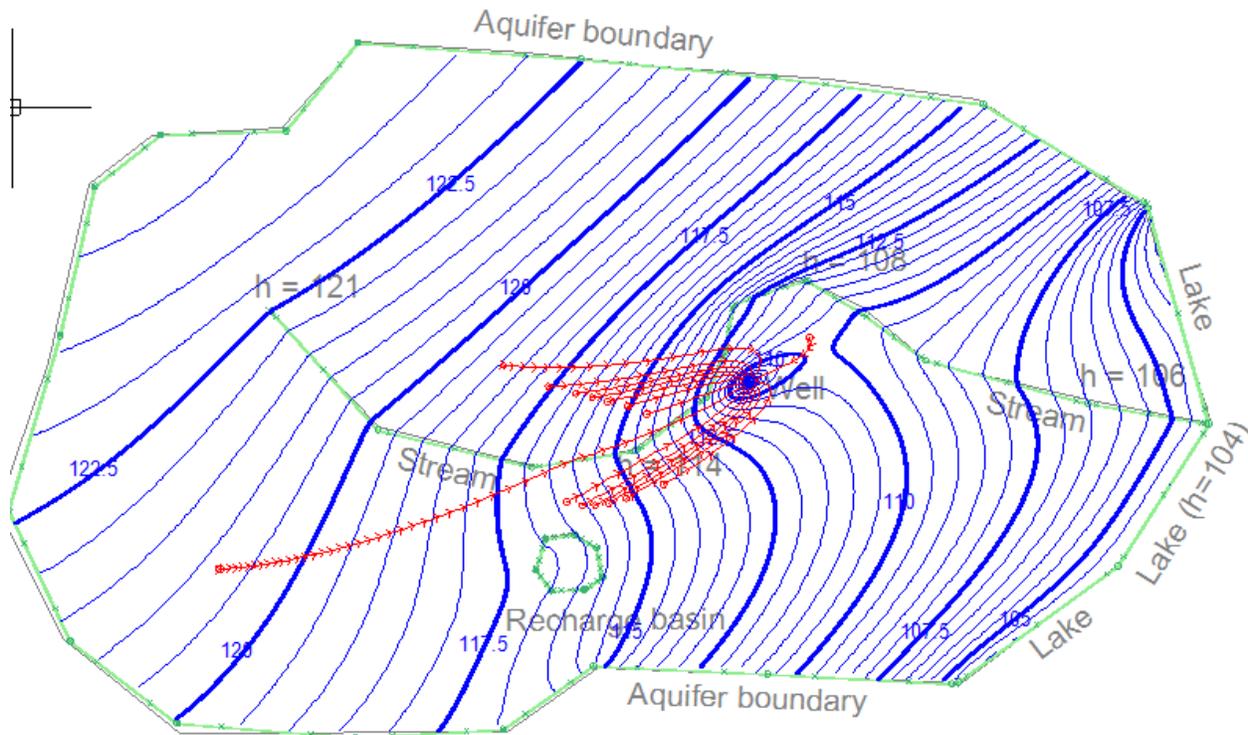
Also note that pathlines jump vertically where they cross under a discharging line boundary such as the river in this model, in a manner governed by mass balance. Pathlines are pulled upward by a line boundary extracting water from the aquifer (e.g. gaining river reach) and are pushed downward by a line boundary injecting water into the aquifer (e.g. losing river reach). The lowest pathline at about $x=700$ is pulled up from about elevation 84 to about elevation 106 where it crosses under a river boundary. This pathline continues on and enters the well with the others 80% of the way from the water table to the base of the domain. The pathline that curls into the well from the east rises abruptly at about $x=1600$ where it passes under the river.

Now, make a similar plot and profile, but for pathlines that enter the well at shallower depth, just 20% of the way down from the top of the saturated thickness. Do this by editing *Plot Input/Pathlines/Well* so the table looks like this:

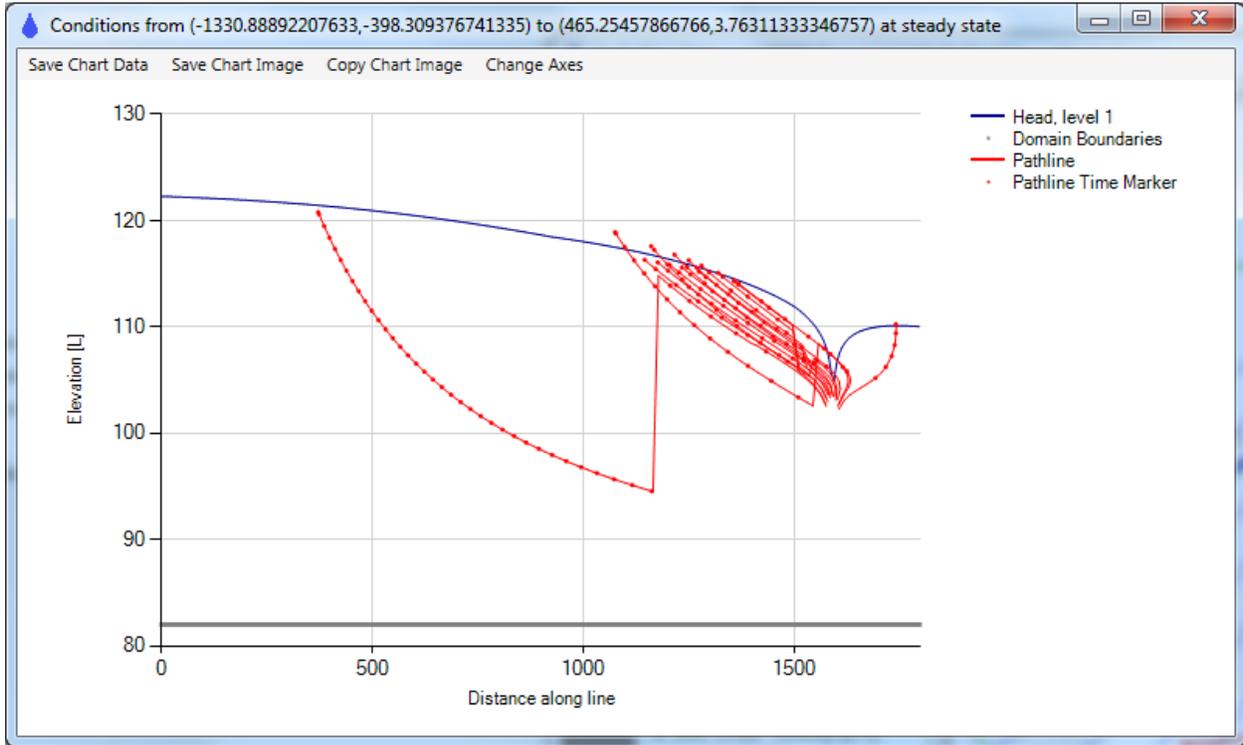
Data: well pathlines

Well Label	Number	Start_level	Show
W	15	1.2	<input checked="" type="checkbox"/>
			<input type="checkbox"/>

Select *Make Plot/All Selected Features* to make a plot, which should look like this:



And then select *Analysis/Graph Conditions Along a Line* and press the *Make Plot* button to generate a profile:

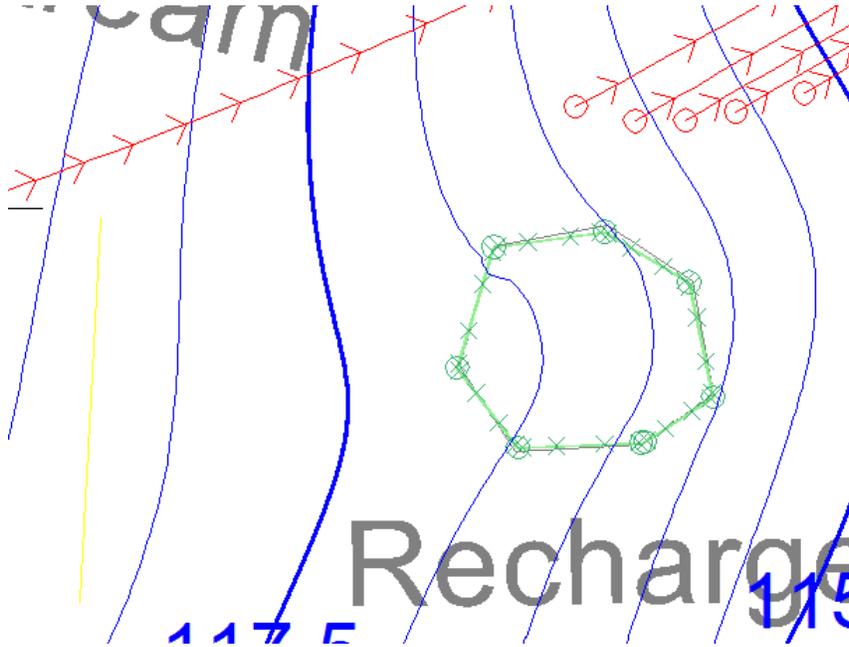


These pathlines all enter the well 20% of the way from the water table to the base, and they enter the water table much closer to the well. The one long pathline crosses under the river at about $x=1200$, where it is pulled upward from about elevation 94 to elevation 114. Other pathlines cross under the river closer to the

well, with some being pulled upward slightly and others being pushed downward slightly, since in this area the river discharge/length is small and varies from positive to negative.

Line Pathlines

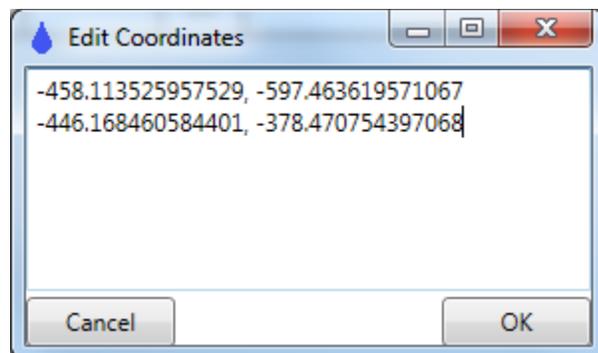
Pathlines are often useful for tracing flow from a known source or spot. This can be done with single pathlines, with pathlines spread out along a line, and with pathlines spread out over an area. We will create some line pathlines that start just upstream of the recharge basin. First, right-click and select *Digitize/Polyline* and digitize a line about like the yellow line shown below:



Then select *Plot Input/Pathlines/Line* and edit the table so it looks like the following, and press the Edit button and paste in the just-digitized line coordinates and then press *OK*.

Data: line pathlines

Label	Direction	Coordinates	Number	Start_level	Show
upstream of recharge	downstream	Edit	10	1.0	<input checked="" type="checkbox"/>
		Edit			<input type="checkbox"/>

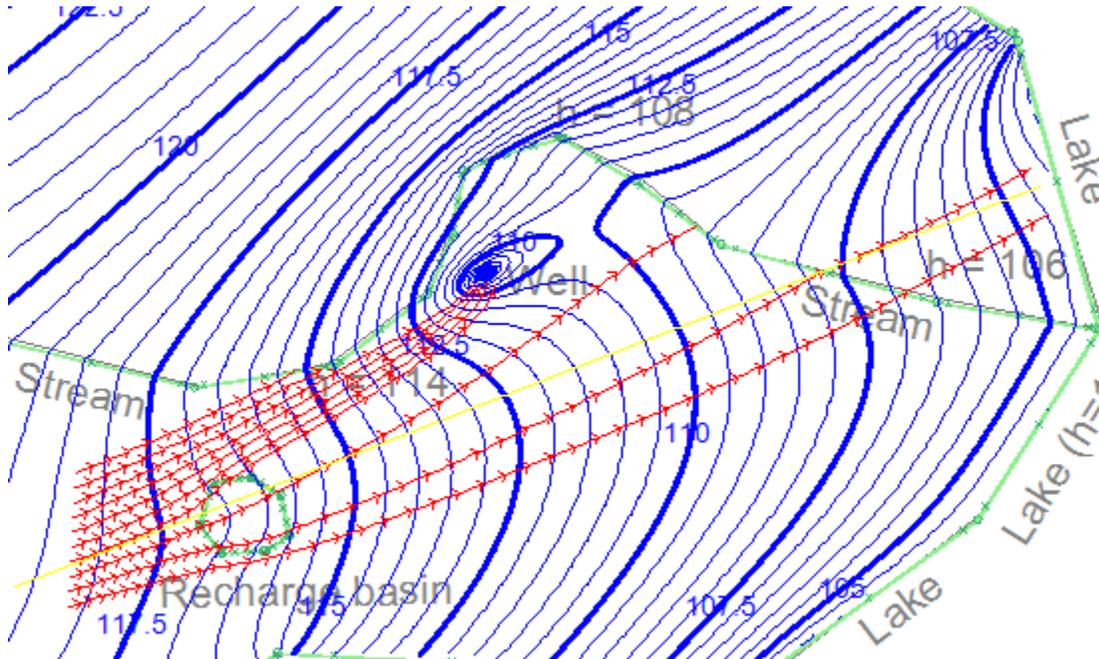


Now turn off the well pathlines we had been looking at by selecting *Plot Input/Pathlines/Well* and unchecking the *Show* column:

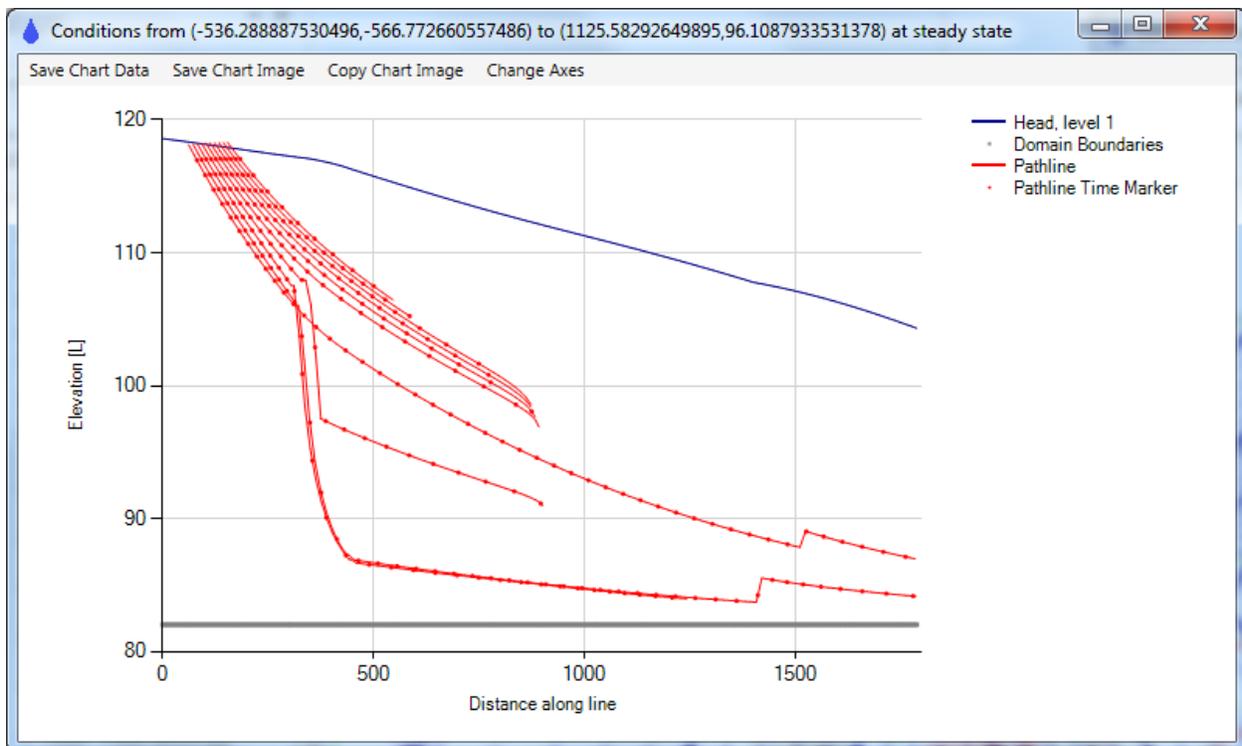
Data: well pathlines

Well Label	Number	Start_level	Show
W	15	1.2	<input type="checkbox"/>
			<input type="checkbox"/>

Select *Make Plot/All Selected Features* to make a plot, which should look like this:



Then digitize a line like the yellow one above and select *Analysis/Graph Conditions Along a Line*, paste in these line coordinates, and press the *Make Plot* button to generate a profile:



In this case, 3 pathlines enter the river, 5 pathlines enter the well, and 2 pathlines pass under the river and enter the lake. Three pathlines go beneath the recharge basin (around $x=300$ to 400) and are pushed steeply downward by the high recharge rate. A couple pathlines cross under the stream near $x=1500$ and are pulled upward some.

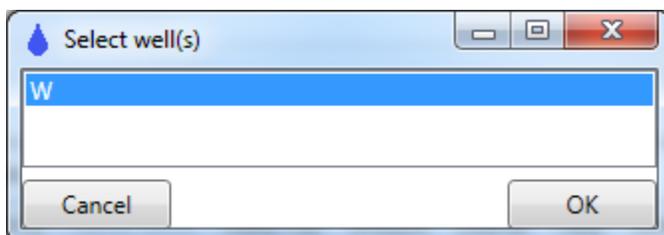
Capture Constraints on Pathlines

A new feature in release 2017-1 allows you to display only the pathlines that are captured by selected wells or internal line boundaries. You can test this new feature by editing *Plot Input/Pathline Settings* so *Capture_constrain* is checked:

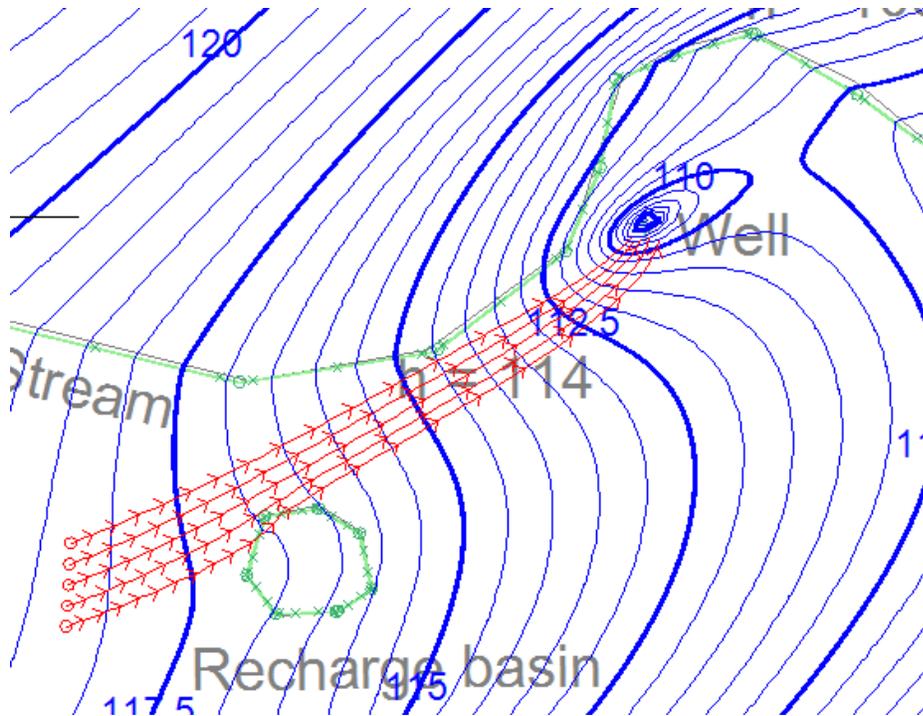
Data: pathline settings

Step_size	Time_markers	Time_marker_increment	Total_time	Start_time	Freeze_transient	Capture_constrain	Capture_wells	Capture_lines
0.005	<input checked="" type="checkbox"/>	200	30000	0.0	<input type="checkbox"/>	<input checked="" type="checkbox"/>	Select	Select

Then click *Select* under *Capture_wells*, and select the one well:



If you then make a plot, the 5 pathlines that enter the well are drawn, but the other pathlines are not:



This feature is quite useful for mapping out an area that ultimately flows to a well, drain, or river reach. A common application is defining the zone of contribution to a water supply well, using *Plot Input/Pathlines/Area Pathlines* to define pathline starting points at the water table, spread over an area.

Wrap up and Next Steps

This concludes the first tutorial. If you want to see the input file that was generated by this tutorial or any of the tutorials, you can find the files in the same Documentation directory where the basemap was located:

- Program Files/Fitts Geosolutions/AnAqSim/Documentation/tutor1.anaq (64-bit AnAqSim)
- Program Files (x86)/Fitts Geosolutions/AnAqSim/Documentation/ tutor1.anaq (32-bit AnAqSim or AnAqSimEDU)

To explore multi-level (3D) modeling with AnAqSim, please see the second tutorial, which edits this single-level model to create a multi-level zone near the river and pumping well. The third tutorial takes this multi-level model and makes it transient.