1. *Explain why the values are not constant along the boundary (relate to the definition of a Type I boundary). Explain the shapes of the flow distributions and why they are not the same for the left (inflow) and right (outflow) boundaries.*

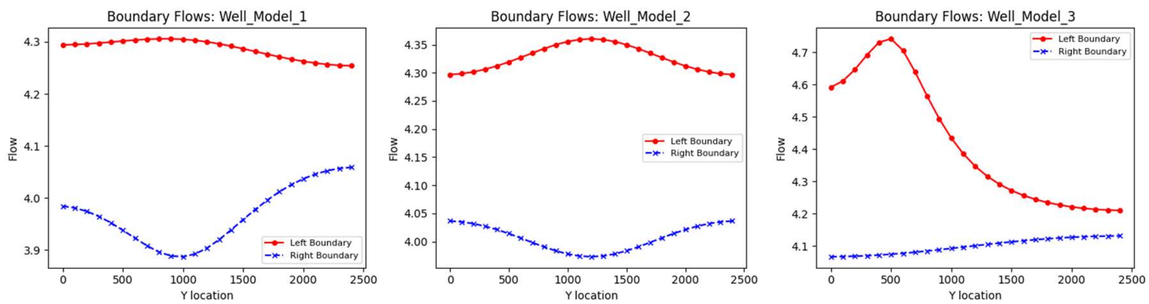


Figure 1: From left to right; Well_Model_1, well position [0,10,15]; Well_Model_2, well position [0,12,12]; Well_Model_3, well position [0,5,5] (positions in MODFLOW coordinates).

The values are not constant along the boundaries of each model for flow due to the capture by the well cell (a negative flux, Type 2 boundary condition.) I visualize this as a mass balance equation for the model (amount of water into the system is the amount of water out of the system (with reduced systems like our current model, we do not account storage within the system.) The MODFLOW {model}.list file shows this with a perfectly balanced flow in = flow out, with the well seen as a contribution to flow out (negative flux out).

The Type 1 boundary conditions along all borders of the model are the h_{left} = 20 m, h_{right} = 10 m, with the top and bottom boundaries having no flow (I guess these could be Type 2 boundaries, with the flux in/out equal to 0).

UPDATE: The flow varies along the boundaries due to two factors: (1) the no-flow boundaries present at the top/bottom boundaries, & (2) the distance the cells are relative to the well. The closer the well is to the boundaries, the more of an influence we see by the well; this is due to the wells zone of capture, which will vary dependent upon the rate at which the well is withdrawing water from the aquifer. When the water is immediately adjacent to the no-flow boundaries, the water prefers to move parallel to the boundaries.

2. *How do you interpret the flow along this transect? Hint, also look at the flow along a transect just upgradient from the well [:{row-1}].*

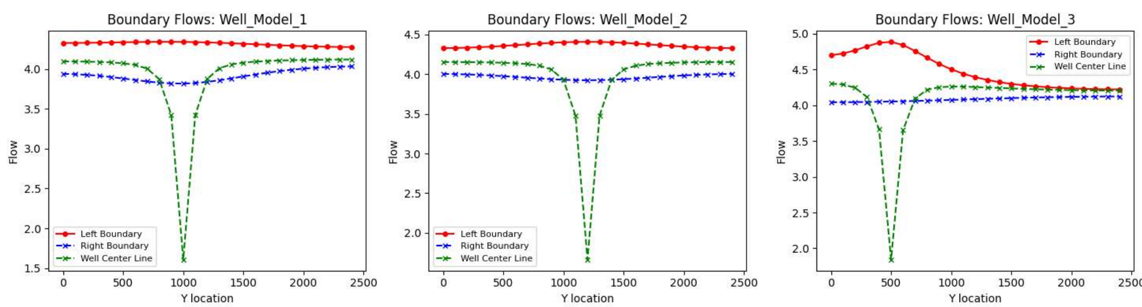
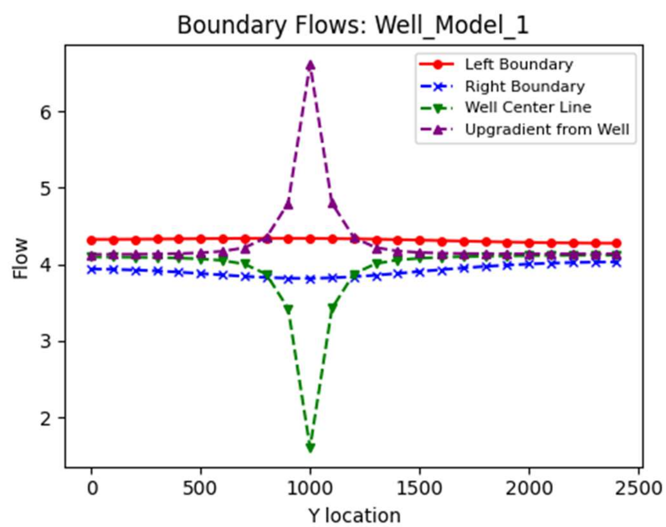


Figure 2: Center line flow transects overlain on the prior flow graphs for all three models; each "Well Center Line" goes straight through the column on which the well is situated within the flopy/MODFLOW model.

I interpret the flow along these transects as showing the sink/capture that the well brings to the model; at these points, overall flow is reduced due to a negative flux within the model at the well coordinates.

Immediately upgradient of the well cells, this green line is inverted, showing a distinct increase in flow near the well location:



This change immediately upgradient also implies the capture action of the well, showing how it pulls in more water immediately upgradient of the well, while retarding the flow downgradient from the well (well-screens are omnidirectional, with water being pulled into the well approximately orthogonal to the surface of the casing, under "perfect" conditions). I would imagine if I set $h_{left} = h_{right}$, the flow around the well would be like the actual well position flow.

UPDATE: Flow immediately down-gradient of the well is still oriented from the left to right, though the magnitude is smaller than upgradient of the well. This magnitude reduction is indicative of capture by the well, though not at a rate significant enough to completely stall flow (this may not be the case if the domain grid was set to be larger, say 100x100 cells or greater; at this point, you may see some flow towards the well in cells immediately adjacent to the well cell if the negative flux out of the system via the cell is strong enough.)

3. *Describe how water flows through the domain. To aid in your description, draw a line through all of the flow vectors that terminate in the well. This approximates the capture zone of the well. Use this to refine your description of the flow system, being as specific as possible about where water that ends up being extracted by the well originates on the inflow boundary.*

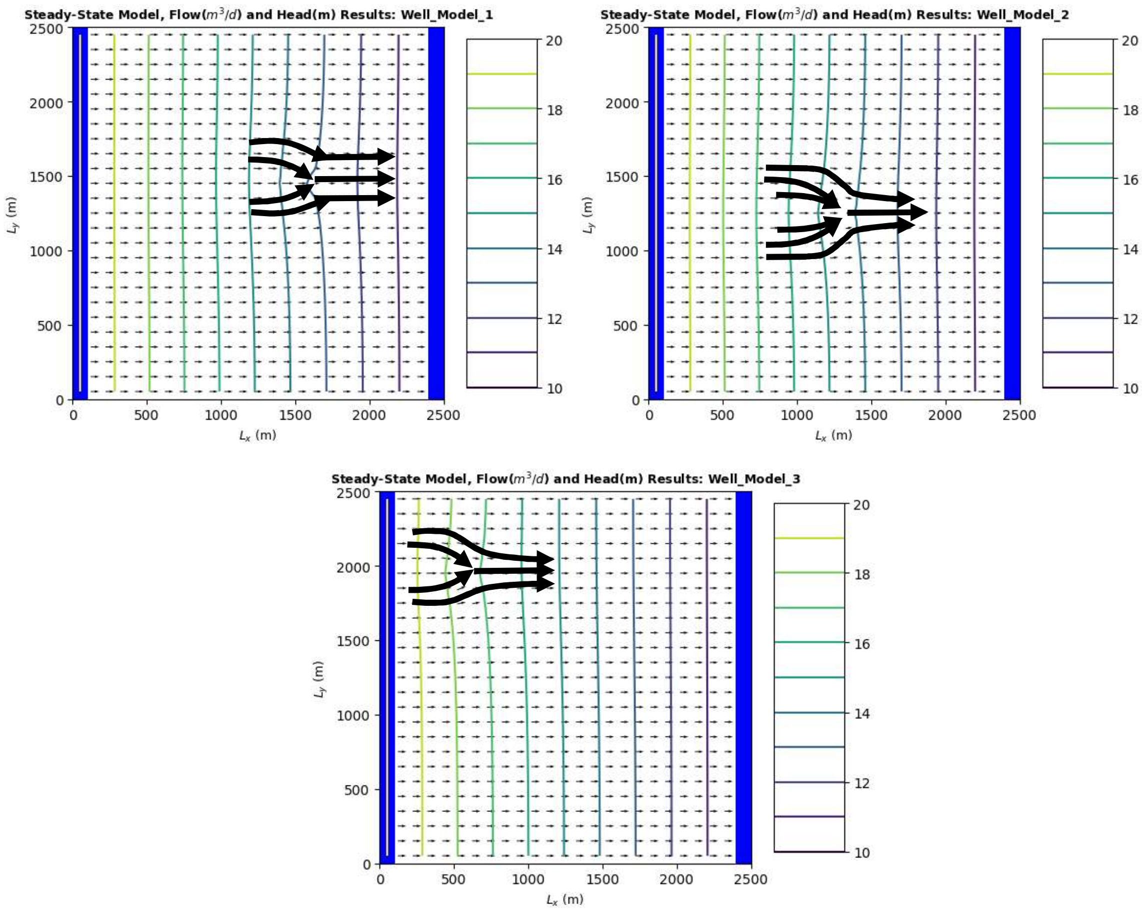


Figure 3: Flownets with equipotential lines overlain for all 3 well locations; Initial, Centered, and Off-centered models. Black arrows indicate relative flow path of a particle released on the left boundary around the well (inferred from flow vectors). Uses initial - 8 m/d flux at the well location.

Flow through the system is relatively linear, almost 1-dimensional in nature, at the initial conditions for well draw (q_{well} = -8 m/d) for the majority of the model domain; however, in the cells immediately surrounding the well (in a 2-3 cell radius around well) are "drawn" towards the negative flux, Type 2 boundary set by the well draw. Flow vectors upgradient in the same row as the well tend to have a larger magnitude upgradient, and lower magnitude down gradient, indicating there is capture by the well.

UPDATE: While the arrows only look at flow into and out of the well cell (plus a few close to the cell), it doesn't illustrate the well capture zone. The capture zone is likely closer to the drawdown plan view graphs below, and can extend to the boundaries dependent on: (1) distance from the boundary, & (2) the rate at which the well is pumped. If the well remains at the initial conditions (q_{well} = -8 m/d), the zone of capture only reaches so far from the well coordinates; at higher withdrawal, the zone extends further from the well (especially noticeable above a withdrawal of 20 m/d).

4. *Look at the plan view drawdown plot. Why aren't the drawdown contours circles? Either explain why this is correct, or fix the plot.*

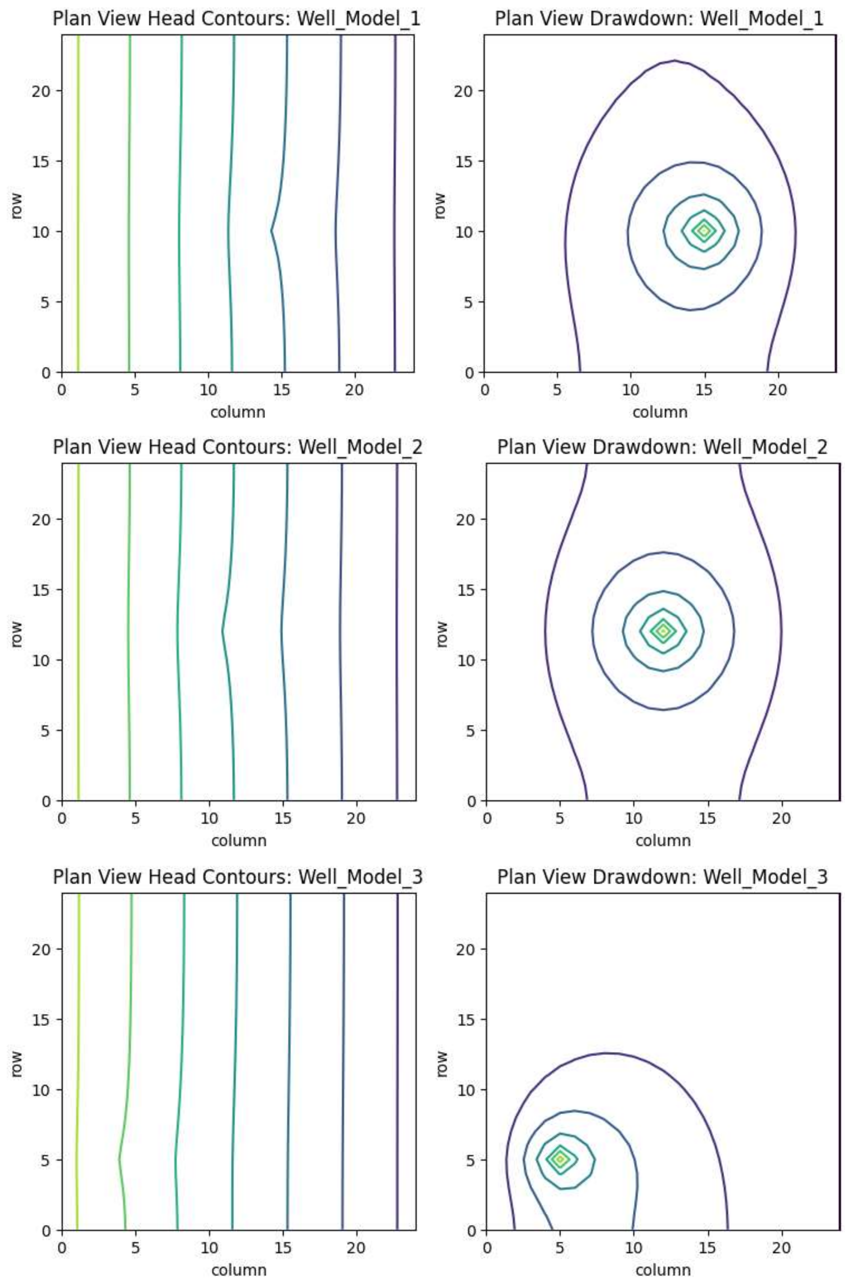


Figure 4: Head contours and the plan view drawdown of all 3 model conditions.

UPDATE: The well drawdown contours are not perfect circles due to the no-flow boundary conditions along the top/bottom of the model domain; the equipotential lines must be perpendicular to the no-flow boundary where they intersect the boundary. The existing head gradient in the model also causes some of the skewing/stretching seen within the models. The well-centered model illustrates the skew/stretch due to the head gradient and the no-flow boundaries (mostly no-flow boundaries influencing the curves). I'm a little unsure as to how I'd test this the head gradient influence with a model still (in fact I think I remember you saying that it really shouldn't affect the drawdown curves).

5. ***Be sure to include the drawdown plot in your discussion - compare this plot to the equipotentials and flow vectors. Something is not right about how the well location is shown. Fix it and explain what was wrong!!***

The above graphs (from 4 and 5) show that plotting via a combo of matplotlib/numpy modules result in graph positions flipped from the flopy modules graphing method. This is due to the way Python and MODFLOW count rows; Python counts row from the bottom up in this case, whereas MODFLOW starts from the top down (e.g.: The row at 2500 meters on the y-axis would be row 1 in MODFLOW, but would be row 24 within Python.)