

Gillian Noonan

## HW 2 Challenge and Discussion Questions, Box Model- Hand-Built MODFLOW

### Challenge:

MODFLOW files have been supplied that model steady state saturated flow through a 3D system. The layers are flat lying. The top and bottom faces and two opposite vertical boundaries are no-flow. Each of the other two faces has a constant head applied to each.

The model is set up for a homogeneous medium. You will have to change the K distribution to replicate the Excel exercise, but for horizontal flow rather than vertical. You can do this using a text editor to change values in the .bcf file.

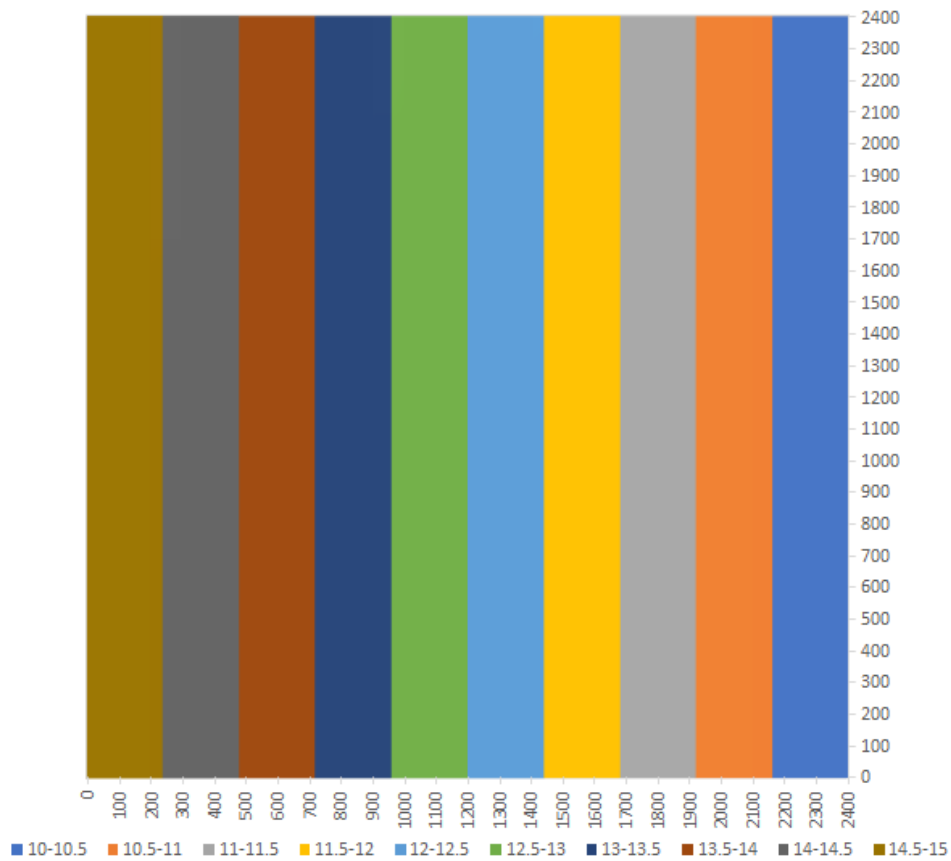
### Noonan - Notes

- The Block-Centered Flow package (.bcf) is used to specify properties controlling flow between cells. It lists hydraulic conductivity distribution for your model cells.
- Basics process flow - Run MODFLOW for given case (homogeneous), look in list file and take head values out, make contour plot of hydraulic head distribution. Then will change K values in .bcf, re-run MODFLOW, take out list file head values again and repeat for third model.

### The Process and the Key Figures:

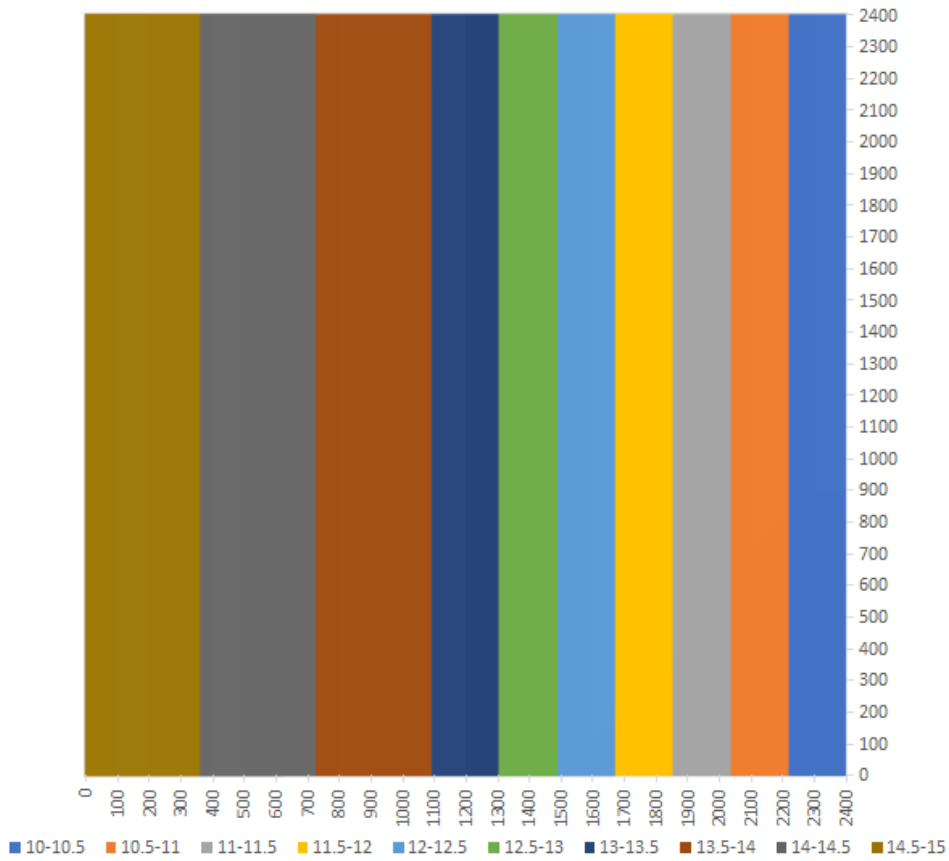
The initial test run using the files provided by Ty ran successfully and very quickly in MODFLOW. Excel was used to produce the head distribution across the model as shown below. The K value for the whole area is 1m/d.

Homogenous Case - Model 1 - Head Distribution



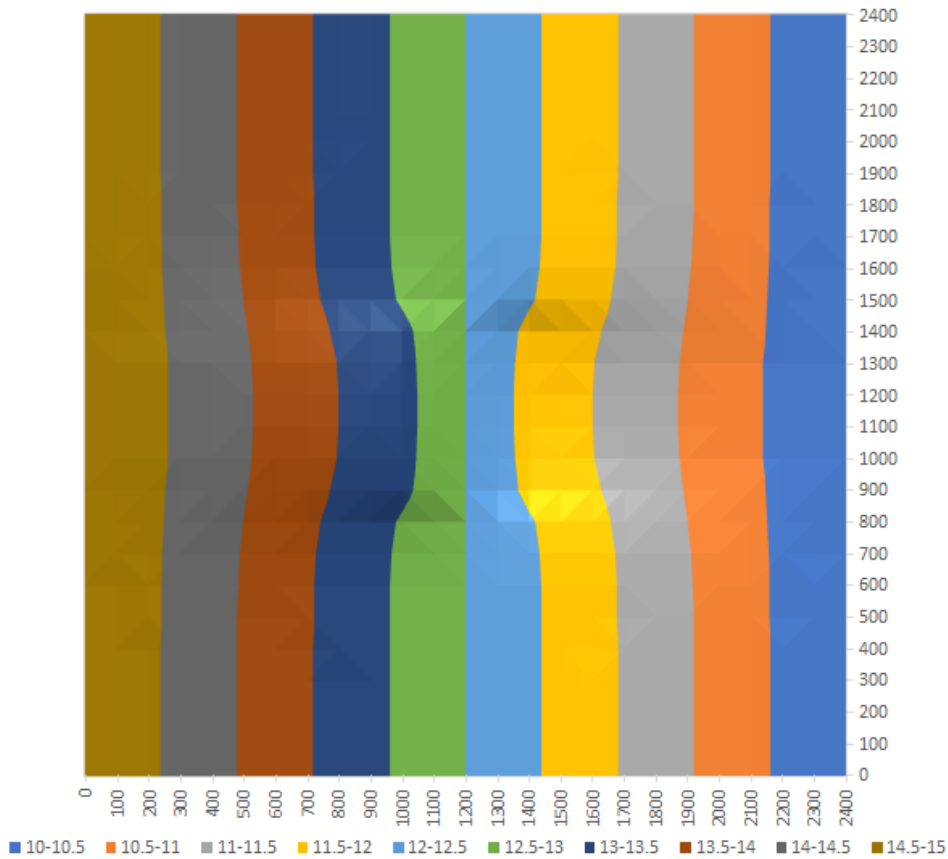
I then started trying to edit the .bcf file to mimic a heterogenous section. I gave it two layers changing across the profile in series (for horizontal flow) which produced the following results (after much trial and error and fixing of text input file formatting). The left side retains the 1m/d K value, while the right side is now assigned a lower hydraulic conductivity of 0.5m/d. It is clear that the head is changing as it enters the lower k zone and the gradient is becoming steeper, based on the narrowing of the vertical head lines.

## Heterogenous Case - Model 2 - Head Distribution



I then edited the .bcf file to add a low K inclusion in the center of the model and re-ran. The center zone is 0.5m/d and the surrounding material is 1m/d. The results produced are below. The now show a perturbation in the vertical head distribution where the contours are no longer regularly spaced across the profile, but bend near this low flow zone.

Inclusion Case - Model 3 - Head Distribution



## Noonan - Challenge Response

1) Show, based on the flux with horizontal distance from a constant head boundary, that the model is steady state. Repeat this for a homogeneous and for a heterogeneous column for which zones of different K are placed in series with the direction of flow. Note that the best way to do this is to take the values from the .list file into Excel, combine them with the K values from the .bcf file, and calculate the flux at each point. Keep in mind that heads are calculated at the center of a cell (a node) and the K values are defined over each cell.

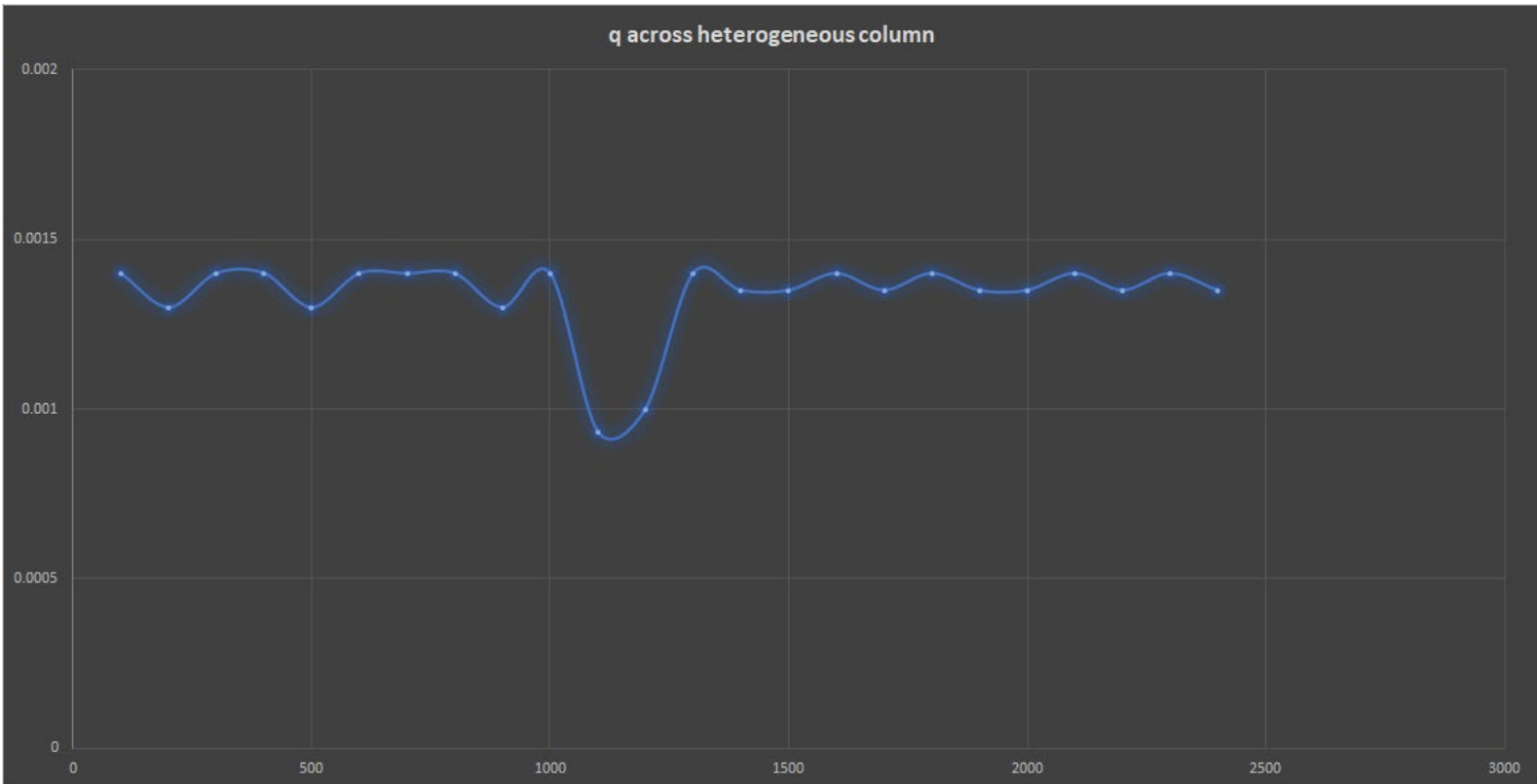
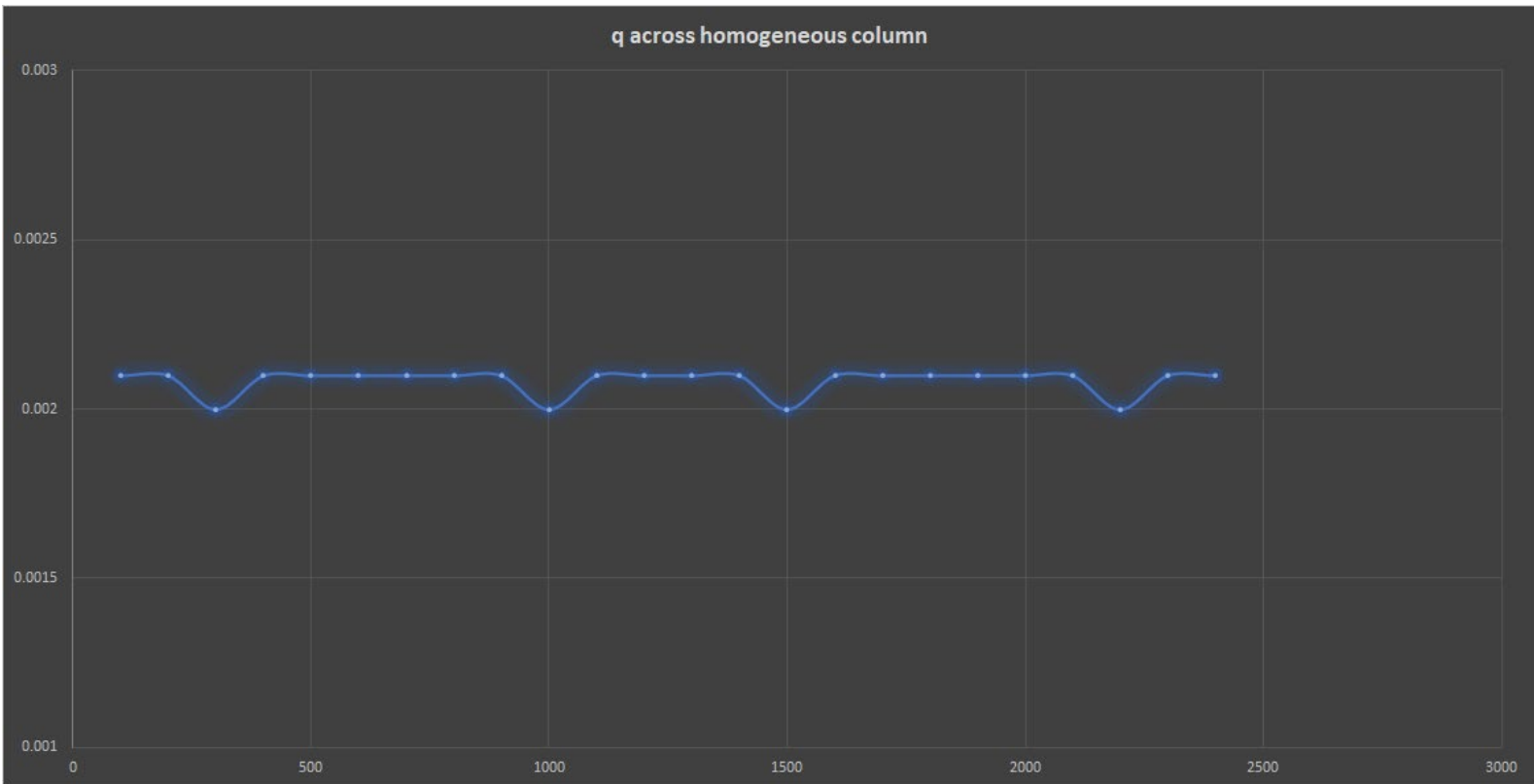
Answer: I followed the instructions above to take out the necessary data from the list and bcf files and then performed a calculation across the rows and columns to find 'q' using the equation:  $q = -K * (dh/dl)$ .

What I would expect for steady state is that the q would be constant across the profile horizontally as storage/flux should not change from point to point by definition. HOWEVER, my initial heterogeneous plot did not show this but had a shift where the transition occurred between zones.

Updated: Turns out the problem was not entirely as I thought - It's true that I needed to use a different way of calculating q to account for the different calculation locations across the cells for K and head - to reflect the harmonic mean K vs. the straight K value. HOWEVER, when making that one change, it was still shifted. So I went back in detail through my excel sheet and found that I actually was referencing some old data there where I was playing around with MODFLOW and I had changed the zone on the right to 10m instead of 0.5m. Problem solved! For reference, the updated formula I used for K was the one indicated by Ty in our breakout which is:

$$K_h = (L_1 + L_2) / (L_1/K_1 + L_2/K_2), \text{ which simplifies to, } 2 / (1/K_1 + 1/K_2)$$

Then substituting this K into the above listed equation for  $q = -K (dh/dl)$

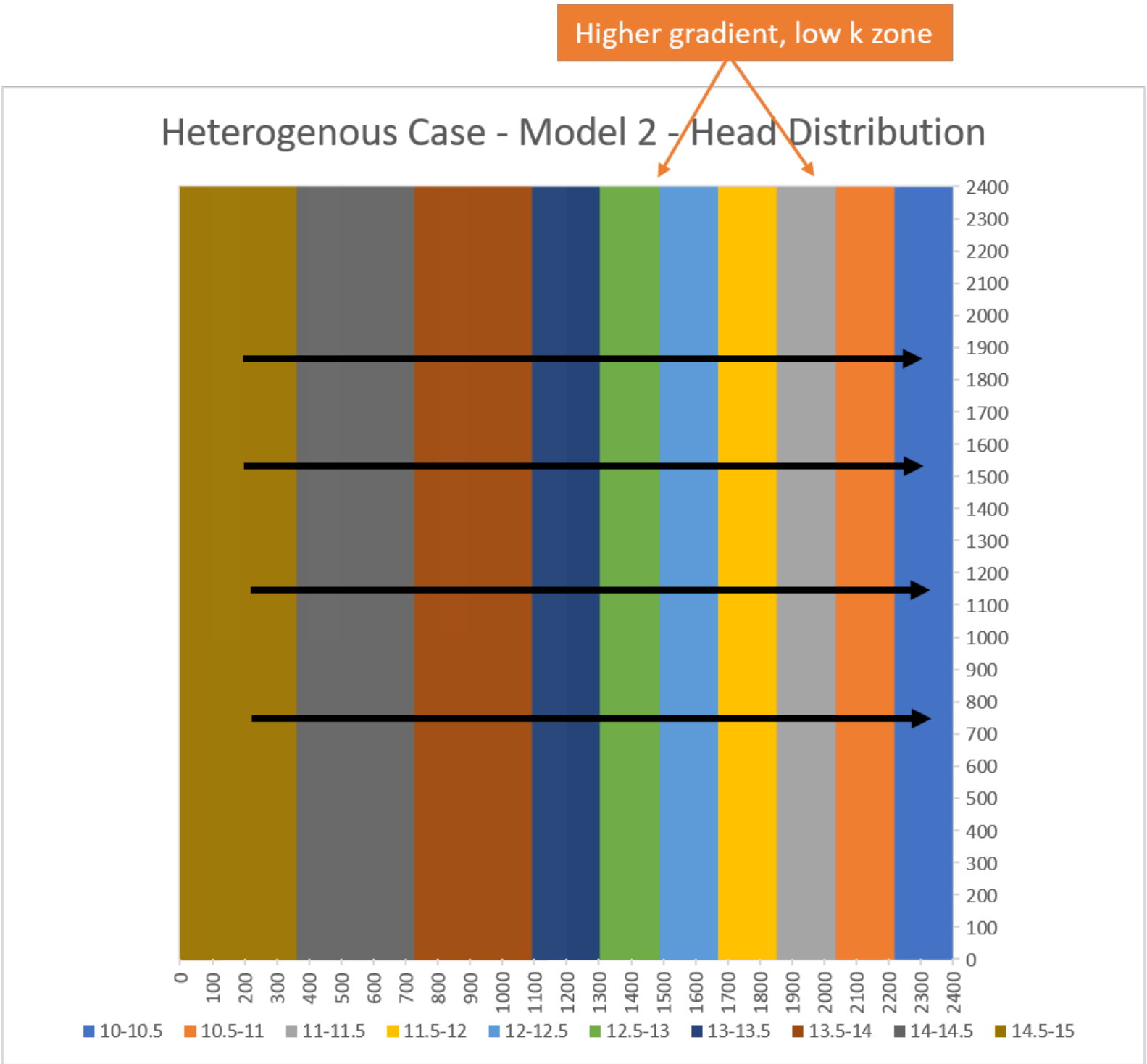


2) Show the steady state head contour in plan view for the heterogeneous (zones in series) condition. Use this plot to defend a contention that flow is 1D. Then, drawing on your Excel assignment, use the results to explain WHY the equivalent hydraulic conductivity,  $K_{eq}$ , is closer to the lower of the two  $K$  values.

Answer: 1D flow proof has to do with the flow paths - they are in a straight line across the page as they follow perpendicular to the head contours which are all vertically straight up and down (see below). So the flow is 1D across the page, as in all in one direction.

The  $K_{eq}$  value will be closer to the lower of the two  $k$  values for the system because we are forcing the water through the low  $k$  zone where it will spend a lot more energy. The system is most sensitive where most energy is lost - this is also can be related by the higher gradient observed in the head contours through this section.

- Key Concepts:**
- From Darcy: we know that we have a certain amount of energy to dispel through a system. Here we are requiring same flow through whole system due to steady state conditions.
  - $K_{eq}$  is controlled by the layer where more energy is used to move through that layer.
  - The energy is related to the square of the gradient. Therefore the energy loss is larger in the low  $k$  medium, which has a higher gradient.
  - In our model, we are forcing the energy through the low  $K$  area where it is losing a lot of energy.

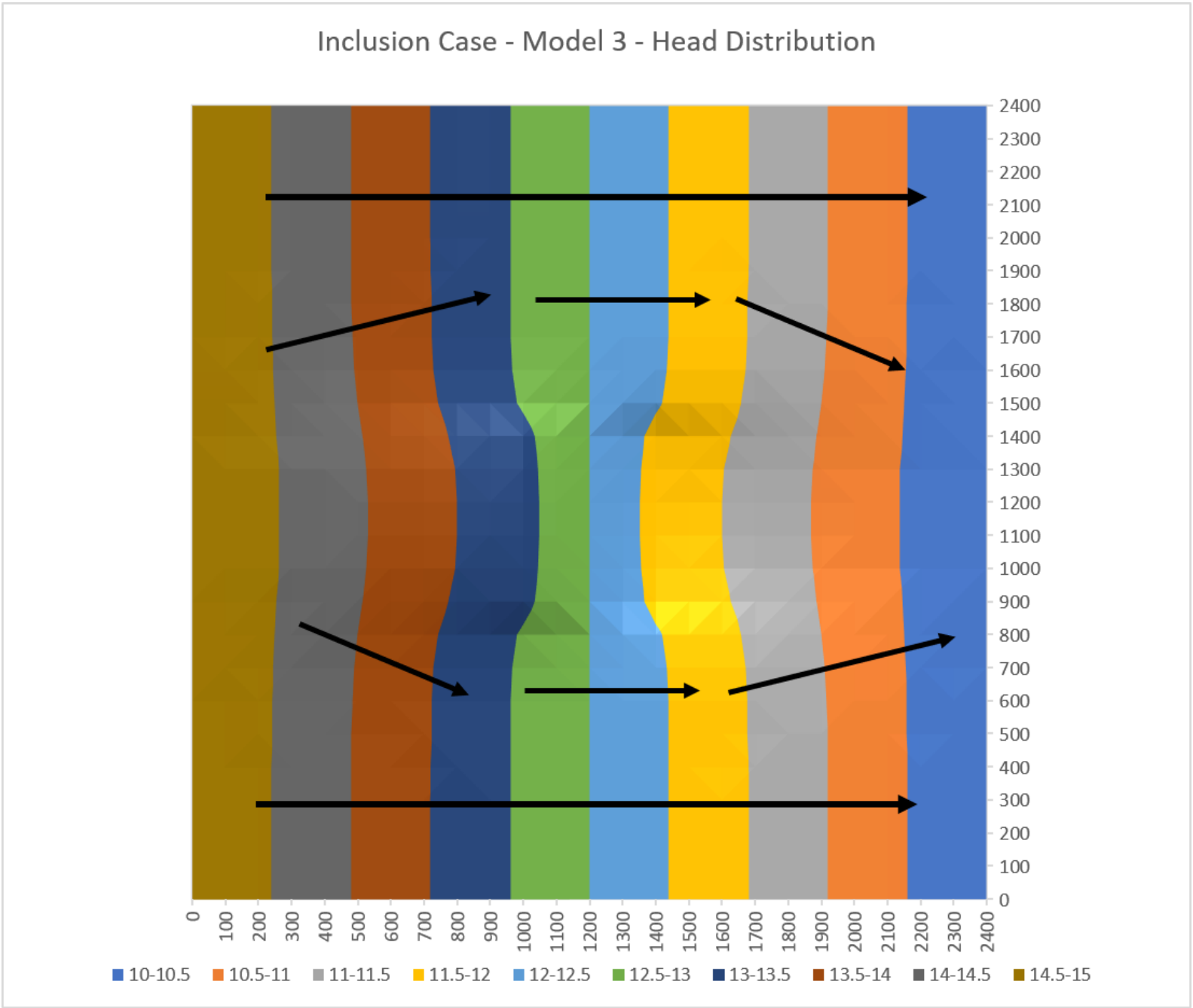


3) Build a model based on a homogeneous domain with a square region of lower  $K$  in the middle of the domain. What can you learn based on your explanation of what controls the effective  $K$  for a 1D flow system now that you are applying it to a 2D system? What do you think the  $K_{eq}$  of this entire system would be compared to the high and low  $K$  values? Explain why it is much more difficult to develop a direct solution for this 2D system

than it was for a 1D system (including the zones placed in series).

Answer: Now flow lines will bend to go around the low flow zone so this is no longer a 1D problem because flow is going in more than one direction. For the low K distribution in series for Model 2, the flow lines had to go through the low K zone - and stayed parallel because they had no other choice and could not be "lazy" and go around it. For the inclusion low k zone of Model 3, the water is trying to stay in the high energy zones - it now has a choice to go around - so it will take the road of least resistance. Based on these diverging flow lines, now we need to consider multiple directional movement of flow with a range of gradients, which makes the direct solution much more difficult.

Given that the water no longer has to flow through the low inclusion zone and can go around, I would expect this zone to have a lesser effect on the  $K_{eq}$  - the water is retaining its energy by avoiding this area. So the  $K_{eq}$  would NOT be closer to the lower K value in this case.



#### Discussion Points

In addition to The Challenge, start thinking about the following ideas:

What is MODFLOW?

Initial Thoughts: I took a bunch of notes in the "Questions" doc on this one in the MODFLOW page. It is a modular flow program, initially built using Fortran. Takes inputs, runs model, and outputs results. Groundwater flow code is its basic function, "what it does best". Runs on command line, note: mf2005.exe must be in folder with your files, text input is very particular in its formatting.

What is a MODFLOW?

Initial Thoughts: I don't understand what this is asking that is different than the above question.

What is a MODFLOW package?

Initial Thoughts: Each package sets up some aspect of the model that you are running.

Packages ([http://inside.mines.edu/~epoeter/583CSM/DOC4\\_MODFLOW2005-TM6A16.pdf](http://inside.mines.edu/~epoeter/583CSM/DOC4_MODFLOW2005-TM6A16.pdf)) There are two types of hydrologic packages. The first type is the internal flow package, which simulates flow between adjacent cells. The second type is the stress package, which simulates an individual kind of stress (such as rivers, wells, and recharge).

What is a MODFLOW input package?

Initial Thoughts: Basic package example: helps define locations of active, inactive, and specified head cells; the head stored in inactive cells; the initial heads in all cells.

What is meant by model dimensionality?

Initial Thoughts: 1D and 2D flow example - Can have a 2D system but flow can be entirely 1D still.

Why might parameters be defined in zones?

Initial Thoughts: To reflect the heterogeneity changes/differences (assumed) of the area to be modeled. For example, changing geological units of differing hydraulic conductivity.

If you want to establish purely horizontal flow, what (specifically) should be defined as constant along the constant head boundaries?

Initial Thoughts: no flow? I really have no idea.

What is an equipotential?

Initial Thoughts: It is where a value is the same all along a certain location/contour line.

Assuming that the medium is isotropic (why?) and that flow is horizontal, how can you track the path of a water particle through the domain?

Initial Thoughts: Isotropic means the same in all directions. You can track the water particle by looking at the head distribution (equipotential lines) - water particle will flow from high energy to lower energy (high head to low head), perpendicular to those lines.

For steady state conditions, there are equivalent Type I and Type II boundary conditions.

What would the Type II boundary condition be that would result in the same equipotentials for the first model?

Initial Thoughts: Type II is Gradient or flux? Assuming "first model" references the homogeneous starting model given in the challenge? It must be constant of 1 maybe since the K throughout was 1?

What is the value of the constant flux?

Initial Thoughts: ??

What about the second model?

Initial Thoughts: ??

What are the values of the constant flux on the left and right boundaries?



Initial Thoughts: ??

What is fundamentally different about the equivalent Type II boundary for the third model compared to the first two?

Initial Thoughts: ??