FINITE DIFFERENCE SPREADSHEETS OF YOUR MODELS #1:

Always indicate units.

Units always need to be consistent in xls and modflow xyzWKQ etc

Cell size can easily be adjusted to make a reasonable size for your site. The cell size only applies to plan view (b is full thickness in vertical)

You may need to have reasonable values for other parameters before changing cell size or calculations could blow up

You can turn calculation to manual while loading information

Always calculate mass balance. Think of the water in and out like your check book you should not have any trouble getting this correct. Include flow for every active face around every constant head cell AND all other inputs and outputs.

GW in/out flow occurs only to/from constant heads not between them or between other cells.

HDF conductance L W K Thk is the FEATURE NOT the CELL

FINITE DIFFERENCE SPREADSHEETS OF YOUR MODELS cont:

Recharge is net flux to water table, NOT precipitation.

Remember the logical observations we made in class last week. Without specified flow we cannot get heads higher than highest or lower than lowest boundary head, and changing homogeneous K will not change the gradient.

Recharge, pumping, HDFs etc are irrelevant on Constant Head CH cells. They will not influence the system and should not be included in mass balance calculations.

If you use multiple layers you need to allow flow between them in the FD equations AND include all flows to/from all layers in the mass balance

These sheets used a steady state formula, so there is NO storage.

TRY OUT YOUR STRESS! You went to the effort to make the model, now USE it!

Always sit back and make sure everything makes sense. Heads and flows should be a reasonable magnitude. Flow patterns should make sense to you.

DUE TODAY

Files showing progress toward Assignment #5 Steady State MODFLOW MODEL

modflow files for your class project zippped in a file named ASSGN5_LASTNAME

bat nam dis bas6 lpf mult zone rch riv sip and your output lst file which may reflect errors .. that is OK

RENAME bat file to something like .bat.txt if your email will not send it

If one of more of those files is not needed for your system, you need not create one, just note that. If you find there are features in your system that have not been represented by those files, please let me know what they are. It would be most helpful if you would submit a scan of the map you are working from to create your modflow model. This is what you did for assignment 2b. It will help me to figure out what you are doing with your files and whether they may be a problem.

SIP a solver package

dis
bas6
lpf
mult
zone
rch
riv
sip
oc

Build sip package: find instructions in the manual or on-line guide

What are the solver options?

Try 50 iterations with a tolerance=1.0

If the mass balance is poor decrease tol and if necessary increase # iterations (maybe tol 0.0000001 and maxiter 10000)

It should converge and mass balance but may not be calibrated yet

New file on class web page at MODFLOW FILES #1-revised-sip (Feb 9)

Try to run MODFLOW double click *.bat

PRE-POST PROCESSING OPTIONS

http://water.usgs.gov/nrp/gwsoftware/modflow.html

POST PROCESSING

USGS ModelViewer
USGS GWChart
USGS ZoneBudget
USGS MODPATH

PRE PROCESSING

HUF

PRE&POST PROCESSING

GUIs
Out-dated: USGS MF-GUI (needs ArgusONE)
NOW!! ModelMuse
Ground Water Vistas (GWV)
Ground Water Modeling System (GMS)
Visual MODFLOW

Model Viewer (a wrdapp contouring program)

In the Geology Application Desktop Folder or through Start Programs
File > New
MODFLOW 2000/2005
Browse to your name file OK OK
Show > Solid
Show > ColorBar
Manipulate object in space (click and drag any corner)
Toolbox > Geometry – scale z dimension to x2 – manipulate – return to 1x
Action > view from direction > +z to get back
Show > Isosurfaces and manipulate, then go to –y and to +z
Toolbox > Isosurface – choose min max #lines to suit the problem
Toolbox > colorbar – choose blue for min, red for max Values
Show > Vectors – Toolbox Vectors - Experiment
Show > Axes Symbol
Show > Model features (select and add)
Show > Time
Show > GridShell
Show > GridLines
Now, visualize hydraulic conductivity (K)
Action > View From Direction and choose -y
Show > Solid
Toolbox > Data and choose K along rows
Toolbox > Solid - select Blocky
Toolbox > Crop - reduce max for Z and do the same from +z direction
experiment a bit

This is a quick start. Read documentation for more.

GWChart (a wrdapp graphing program)

In the Geology Application Desktop Folder or through Start Programs

Chart Type > Water Budgets
Read Data From: MODFLOW
File Open your lst file
Select Plot Bar Chart
Turn ON ALL Ins and Outs, Turn OFF in-out & % discrepancy

All this is more interesting with
more complex and especially transient simulations
We will use it much more effectively later on those exercises

We will do more with this later.

This is a quick start. Read documentation for more.

ZoneBudget (a wrdapp water budget program)

In windows explorer under your working directory
make a new folder called: ep-zonbudget
Find ZoneBudget documentation ofr90392.pdf zonbud3.pdf on computer (wrdapp)
Make a batch file in your working directory
_zbat.bat
C:\wrdapp\Zonbud.3_01\Bin\zonbud.exe
Pause

Follow the prompts

To listing file type:	zonbud.output
Cell by cell flow file:	ep.cbc
Title in the listing file:	example
Name of zone file or interactive:	interactive <CR>
Choose the entire model all layers, rows and columns for ONE zone	
Choose the option:	A

Do not waste time doing that again, put the responses in a response file
perhaps call it zonebud.responses
Edit _zbat.bat to say:
C:\wrdapp\Zonbud.3_01\Bin\zonbud.exe < zonebud.responses

Files on class web page at ZoneBudgetFiles (Feb 9)

Read documentation for more.

Instead of Interactive use a zone file named rchzones.dat

Determine budget for recharge zones using a zone file

See ZoneBudget Manual for defining zones with a zone file

Manuals are in the wrdapp doc folders

```
 2 3 10
internal          1              (10i2)        0
 0 3 3 3 2 2 0 0 1 0
 0 3 3 3 2 2 0 0 1 0
 0 3 3 3 2 2 0 0 1 0
internal          1              (10i2)        0
 0 3 3 3 2 2 0 0 1 0
 0 3 3 3 2 2 0 0 1 0
 0 3 3 3 2 2 0 0 1 0
```

Automate by typing the responses in a file named zone.in.rch

```
zb.recharge
ep.cbc
recharge zones
rchzones.dat
A
```

Then execute with a batch file _zbat.rch.bat

```
C:\wrdapp\ZONBUD.3_0\BIN\zonbud.exe < zone.in.rch
Pause
```

Review results in zb.recharge Do they make sense?

This will be more useful for more complex and transient models

Files on class web page at ZoneBudgetFiles (Feb 9)

ZoneBudget

View output in GWChart

ChartType > Water Budgets

Open and browse to zb.recharge

Note you need to choose all files because we did not use the standard extension

This is a very simple model so the results are few

Choose plot bar chart for one time step and zone

View for each zone

Identify components of the budget and confirm they make sense

Turn off the components and check % discrepancy by zone

Notice you could create many complex zones for various analyses

MODPATH (a wrdapp particle tracking program)

Find Mpath.5_0 documentation on your computer disk
MODPATH1-ofr94464.pdf
note the quick start guide beginning at 4-3 & input instructions in appendix A

Batch files are provided with the code C:\wrdapp\Mpath.5_0\Setup
and are used directly here

Three modpath input files are required for our project today:
nam main locations

View the input instructions along with the input files already created to understand what is being accomplished

We must enter new information about porosity
to calculate particle travel time
Also we need to specify
starting locations for particles and times when we want reports

Files on class web page at MODPATHFiles (Feb 9)

MODPATH (a wrdapp particle tracking program)

Like ZoneBudget you can run MODPATH interactively, responding to prompts
MODPATH will create a response file as you do this
mpath.rsp

RUN mpath5.bat
You will need a mpath.nam file & locations file see manual & class page

1st time respond, thereafter mpath.rsp (see my rsp on class page)
Note the date and times of files that are created and view them
Description of items is in the manual

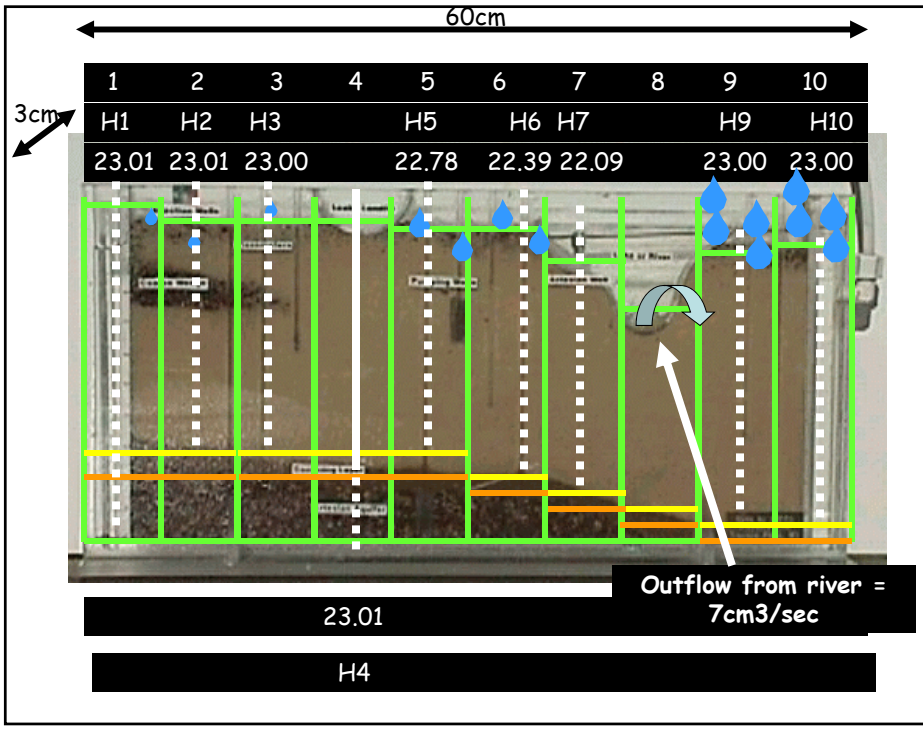
RUN mplot5.bat
Enter mplot.rsp
Note the date and times of files that are created and view them

Files on class web page at MODPATHFiles (Feb 9)

This is a quick start. Read documentation for more.

What do the heads tell you about the model?
What are the flows at your reservoirs?
What do the flows tell you about your model?
Does this set up make sense?
How could you improve it?

CALIBRATE



Try a bit of manual calibration. Consider altering:

rch (lower/redistribute rates)

riv (increase Kriv)

lpf (increase K1)

Head Observations 95% confidence is 0.1 cm

				HEAD		
1	x= 1.5	y=1.5	multilayer	23.01 cm	column 1	coff 0
2	x= 7.5	y=1.5	layer 1	23.01 cm	column 2	coff 0
3	x=13.5	y=1.5	layer 1	23.00 cm	column 3	coff 0
4	x=19.5	y=1.5	layer 2	23.00 cm	column 4	coff 0
5	x=25.5	y=1.5	layer 1	22.78 cm	column 5	coff 0
6	x=33.0	y=1.5	layer 1	22.39 cm	column 6	coff 0.25
7	x=37.5	y=1.75	layer 1	22.08 cm	column 7	coff -0.25
8	x=49.5	y=1.5	layer 1	22.99 cm	column 9	coff 0
9	x=55.5	y=1.5	layer 1	23.01 cm	column 10	coff 0

Flow Observation 95% confidence +/-5%

Outflow from river = 7.0cm^3/sec

Using observation packages could help

hob package

rvob package

Instructions for both in the manual &
On-line guide under observation process

ALSO ADD A PVAL FILE

Remember when you add a package
Include in the name file

Files on class web page Observation-ParameterFiles (Feb 9)

"TRUE" PARAMETERS

K1	4.8252817903337 27 E-1
K2	1.000000000000000 E+1
KVA	1.000000000000000 E+0
KVCB	7.111995260943306 E-4
rch1	3.009375386922079 E-1
rch2	2.777381769353287 E-2
rch3	1.080147516268200 E-2
Kriv	2.925696922556294 E-2

REVISED "TRUE" PARAMETERS

K1	4.8295441617986568 E-01
K2	1.000000000000000 E+01
KVA	1.000000000000000 E+00
KVCB	5.7583124979263326 E-04
rch1	3.0107019869053658 E-01
rch2	2.7846585806703041 E-02
rch3	1.0708881967146309 E-02
Kriv	8.7751713381414714 E-02

WEL well package

Remember when you add a package: Include in the name file

Build wel package: find instructions in the manual or on-line guide

What is a reasonable rate?
What is unusual about adding a well to this system?

Try to run MODFLOW double click *.bat
Do results make sense?
Find the changes
How has the pumping affected the boundaries?
Try a different pumping rate and evaluate the response

MNW2 well package

find instructions in the on-line guide

Multi-node wells

Flow can be Drawdown Limited

MNW Dynamically distributes flow between nodes under pumping, recharging, or unpumped conditions

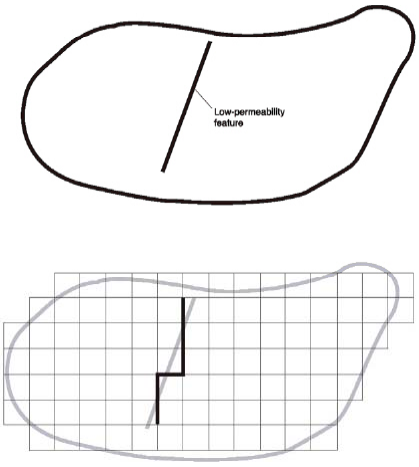
Can track mixes of water

Simulated wellbore flow can be compared with measured wellbore flow

HFB Hydrologic Flow Barrier package

find instructions in the on-line guide

Use to represent features too thin to be represented by a reasonably sized grid cell



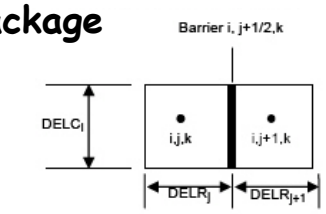
HFB Hydrologic Flow Barrier package

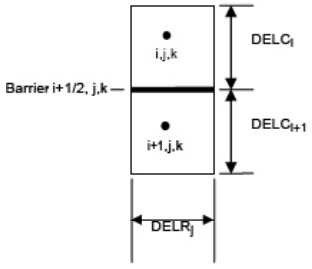
Input:

PARNAM PARTYP Parval NLST
Layer IROW1 ICOL1 IROW2 ICOL2 Factor

$$CR_{i,j+1/2,k} = \frac{CR^{Original}_{i,j+1/2,k}\, C_{Barrier}}{CR^{Original}_{i,j+1/2,k} + C_{Barrier}}$$

$$C_{Barrier} = \frac{K_{Barrier}\Delta v_{Barrier} DELC_i}{L_{Barrier}}$$





$K_{Barrier}$ is hydraulic conductivity of the barrier,
$\Delta v_{Barrier}$ is vertical thickness of the barrier, which is the average saturated thickness of the two cells
$L_{Barrier}$ is the distance across the barrier in the flow direction.

HUF Hydrologic Unit Flow package

find instructions in the on-line guide

Geologic units are assigned top, bottom, and properties

These units may be more or less numerous than the modflow model layers and their tops and bottom need not coincide with layers, but the spatial distribution has the same frequency as the grid centers

Equivalent properties are assigned to the grid based on how it corresponds to the geology

Substantial misreprentations can occur if model grid and geohydrologic unit dips differ

This is ameliorated if the geohydrologic units are thick relative to the grid layers

Output Control Options – see OC in on-line guide

e.g. in oc file:
Add to get more significant figures for heads in the list file:
HEAD PRINT FORMAT 2

Or to get any format you would like to write as a text rather than binary file in the hds file try for example:
HEAD SAVE FORMAT (9F15.7) LABEL

This must be accompanied by a name file change such that
data(binary) 21 hds
Is changed to
data 21 ep.hds

and you will get the following format:

```
1    1   1.000000E+00   1.000000E+00            HEAD     9     5     1 (9F15.7)
   24.0599213     24.0599613     23.6493073     22.9445572     22.2021313     20.5678501     18.7800140     21.5068359     23.4369640
   24.0599213     24.0599613     23.6493073     22.9445572     22.2021313     20.5678501     18.7800140     21.5068359     23.4369640
   24.0599213     24.0599613     23.6493073     22.9445572     22.2021313     20.5678501     18.7800140     21.5068359     23.4369640
   24.0599213     24.0599613     23.6493073     22.9445572     22.2021313     20.5678501     18.7800140     21.5068359     23.4369640
   24.0599213     24.0599613     23.6493073     22.9445572     22.2021313     20.5678501     18.7800140     21.5068359     23.4369640
   1    1   1.000000E+00   1.000000E+00            HEAD     9     5     2 (9F15.7)
   22.9623165     22.9623127     22.9623070     22.9622993     22.9622917     22.9622841    999.0000000    999.0000000    999.0000000
   22.9623165     22.9623127     22.9623070     22.9622993     22.9622917     22.9622860    999.0000000    999.0000000    999.0000000
   22.9623165     22.9623127     22.9623070     22.9622993     22.9622917     22.9622860    999.0000000    999.0000000    999.0000000
   22.9623165     22.9623127     22.9623070     22.9622993     22.9622917     22.9622860    999.0000000    999.0000000    999.0000000
   22.9623165     22.9623127     22.9623070     22.9622993     22.9622917     22.9622860    999.0000000    999.0000000    999.0000000
```

See Solver Options in On-line Guide

Try a different solver, PCG package

In change solver to pcg

EXECUTE MODFLOW

Do you need to change the name file?

EVALUATE RESULTS

Compare the heads and flows
Compare the progress of the solution
How do they differ?
Did the model improve?

File on class web page PCGFile (Feb 9)

What Package might you need for your model?

Stream flow routing?
Lake?
There are many possibilities

See the
On-line Guide
and
USGS software web pages for
more information on those possibilities

DUE NEXT WEEK

to get credit for Assgn#5 (which is not due until Feb 23)

Submit MODFLOW files covered today (Feb 9) but for your project and your grid

If it does all you expect tell me that. If you are having problems, tell me specifically how you know you are having a problem and I will offer suggestions.