AdH Simulation "Go-By"

1) Generate a mesh using good practices of smoothness and aspect ratio
2) Generate bc and hotstart file
3) Determine the time step size by convergence
4) Determine parameters for adaption and check for convergence

AdH Boundary Condition File "Go-By"

Using a Text File for BC generation

1) You have a mesh with materials designated and nodes renumbered
2) Use a previous bc file as a guide or the template in the manual (pg 12)
3) Begin by specifying your operation parameters
   - OP SW2 — 2 Dimensional Shallow Water
   - OP INC 40 — # memory allocation increment
   - OP TRN 0 — # transport equations
   - OP BLK 1 — blocks per processor (should always be 1 on the PC)
   - OP PRE 1 — pre-conditioner selection (you should always start with 1)
4) Give your iteration and convergence parameters
   - IP NIT 5 — max # non-linear iterations per group
   - IP MIT 80 — max # linear iterations
   - IP NTL 0.001 — non-linear tolerances
5) Give your global material properties (this is where your unit designation is determined)
   - MP MUC 1 — Manning's units constant (1 for SI, 1.486 for English)
   - MP G 9.81 — gravity
   - MP MU 0.000001 — kinematic viscosity
   - MP RHO 1000.0 — density
   - MP DTL 0.001 — wetting/drying limits (depth unit, defaults to 0 if not included)
6) Give your material specific material properties (material id is given first)
   - MP EEV 1 0.5 2 — estimated eddy viscosity (or MP EVS - eddy viscosity [MP DF required when OP TRN>0]) (three options for computation)
   - MP ML 1 0 — max # refinement levels (0 = no adaption, 2^{ML}) (should always be 0 for initial simulations)
   - MP SRT 1 1 — error tolerance for refinement (should always be 1 for initial simulations) (MP TRT required when OP TRN>0)
7) Define your strings (material, edge, node, mid) (all strings are included in a single string number listing)
   - MTS 1 1 — material strings (material id from mesh, string #)

- EGS 100 101 2 — edge strings (node 1, node 2, string #) (along a mesh edge)
- NDS 103 3 — node strings (node, string #)
- MDS 104 105 4 — mid strings (node1, node 2, string #) (internal to the mesh)

8) Define your time series (these are included for any parameters that could vary in time even if you are simulating a nonchanging condition: inflows, tailwaters, concentrations, timesteps, output) (all time series are included in a single series number listing) (your number of time series will depend on the problem) (time series should extend to the final time or greater; less will cause AdH to stop at the final time of the time series)
   - XY1 1 3 0 0 — xy-series (1 data value for each time, series #, number of points In the series, units of the time, output units if this series defines the output)
     0.0 2.34
     100.0 1.58
     999.0 -0.75
   - XYC 2 10.0 0.0 1 — wind series coordinates (series #, x & y coordinate, option)
   - XY2 2 2 0 0 — xyy-series (2 data values for each time, same format as XY1, currently only used for wind data, same series # as the XYC)
     0.0 2.34 1.0
     100.0 1.58 2.0
   - OS 3 2 0 — Auto-build output time series (series #, number of points in the series, output units; follows by lines of start time, end time, increment, units of the times)
     0.0 100.0 1.0 0
     100.0 500.0 10.0 0

9) Give your boundary conditions (these include bed or sidewall frictions, inflows, tailwaters, off materials, etc.) (Natural Boundary-NB for edges or materials and Dirichlet Boundary-DB for nodes) (many options available…see manual pg 44 - 58)
   - FR MNG 1 0.02 — Manning's friction on string 1 with a value of 0.02 (if not given, defaults to 0, options for vegetative roughness and equivalent roughness height)
   - NB DIS 1 1 — discharge boundary condition on string 1 defined by time series 1
   - NB OTW 3 2 — tailwater boundary condition on string 3 defined by time series 2
   - OFF 4 — turn off string 4 (no computations made for these elements)

10) Give your time controls (start time, end time, time step options)
    - TC T0 0.0 — start at time 0.0
    - TC TF 100.0 2 — final time (time, units: 0=sec, 1=min, 2=hr, 3=day, 4=wk)(should be equal to or less than the final time of the time series)
    - TC IDT 3 — Time series number for incremental time steps (options also available for steady state timestepping - STD and automatic timestep find – ATF…see manual pg 59)

11) Give output and print controls (for data output and screen output)
    - OC 2 — XY time series that defines the output control (or you have an

OS time series card)

- PC LVL 1 — screen output level (optional, default is zero)
- PC ADP — print adapted meshes (optional, only useful when adapting)

12) END — End card closes the bc file
13) ALWAYS CONSULT THE MANUAL AND/OR QUICK REFERENCE FOR AVAILABLE BC OPTIONS!!!

Using SMS for BC generation

1) Set the model units and projection first
2) Generate your mesh with material designations and renumber
3) Set up the Model Control
    a. Model Parameters:
        i. operation cards (keep defaults in most cases)
        ii. include wind stations if using (you define the station locations in a map file that gets dragged into the AdH mesh section of the data tree
        iii. include density effects if modeling salinity or temperature mixing
        iv. include bendway correction (vorticity – used for meandering channels)
    b. Iterations:
        i. iteration and convergence parameters
        ii. defaults for parameters is generally ok but you may wish to remove the criteria for both convergence options
        iii. you always want to reduce the timestep if the tolerance is not satisfied
    c. Time:
        i. dynamic, steady state, and automatic time step options
        ii. define the time series for the time step control
        iii. when you set the steady state option, you will lose any time series information already entered
    d. Output:
        i. Specify the start time, end time, and increment for saving data
        ii. Must select "add" and populate the window
        iii. Choose if you want to output adapted meshes (only useful when adapting)
    e. Global Material Properties:
        i. MU, G, RHO set to values appropriate for the projection units
        ii. Include DTL card if tolerance should be greater than zero

f. Transport Constituents:
    i. Add new generic or sediment transport constituents here
    ii. If a sediment is added, bed layer options will become available on the global material properties tab
g. Consolidation:
    i. Only used for cohesive bed characteristics
h. Advanced:
    i. Linear iteration count (generally leave as default)
    ii. Always stop the solution if tolerance is not satisfied…this will cut the timestep

4) Set up the Material Properties for each material
    a. Properties: Eddy viscosity, bed friction, coriolis, off card
    b. Boundary Condition: add rain or evaporation if included
    c. Refinement and Transport: turn on refinement and give tolerance if adapting
        i. If transport is included, transport refinement parameters are available
5) Define the node string where your boundary condition will be applied (SMS calls all strings node strings regardless of the AdH terminology)
6) Assign the boundary condition
    a. Select the node string
        i. right click; select Boundary Condition – Assign
        ii. or Assign BC under the ADH menu
    b. choose the boundary condition type
        i. Flow is applied to AdH edge strings
        ii. Pressure is applied to AdH node strings
    c. Define the time series curve
        i. Select new and provide a name
        ii. Choose the units for the data
        iii. Enter or paste in the data for the time series defining the boundary condition
        iv. Set a roughness if one is necessary (such as a sidewall)
        v. Include the flux computation across the string if desired
7) Save the boundary condition file separately or Save ADH to get all input files

AdH Hot Start File "Go-By"

Using SMS's Data Calculator

For a constant water surface elevation

a. Open the mesh in SMS
b. Go to the data calculator
c. Compute water surface elevation – elevation
d. Name "ioh"
e. Compute
f. Export the ioh dataset as an ascii file, name *filename.hot*

Using SMS's AdH Hotstart File Creator

a. Under AdH, select "Hotstart Initial Conditions"
b. Choose the depth condition option
    i. Constant water surface elevation
    ii. Constant depth
    iii. Defined from loaded data set
c. Choose the velocity condition option (optional)
    i. Constant x and y values
    ii. Defined from loaded data set
d. Save the hotstart file individually or save ADH to save all input files

Using Results from a Previous Simulation (adh_hot.exe)

a. Previous simulation is complete with depth and velocity results available at the desired time
b. Generate the input file for the adh_hot.exe utility code
    i. Time to pull from the data files, new filename (first line)
    ii. Filename to pull the data from, name of the data set (ioh for depth, iov for velocity, 1 line per data file, see manual pg 81 for others)
    iii. Example:

```
10000.0   riprap2d.hot_new
riprap2d_dep.dat    ioh
riprap2d_ovl.dat     iov
```

c. Run the adh_hot.exe utility (adh_hot.exe *inputfilename*)

AdH Running "Go-By"

Running AdH inside SMS

1) Generate the mesh (.3dm)
2) Generate the boundary conditions (.bc)
3) Generate the initial conditions (.hot)
4) Save ADH to get the .sim file
5) Under the ADH menu, select Run ADH
    a. Pre_adh will run first, the output is shown
    b. Select Run ADH in the lower left
    c. AdH is run with the output shown on the screen
    d. No screen output is saved until the run is complete and the user chooses to save the output (lower right)

Running AdH with a Batch Script

1) Generate the mesh (.3dm)
2) Generate the boundary conditions (.bc)
3) Generate the initial conditions (.hot)
4) All three files reside in the same location
5) Generate your batch script (if already created with a .bat extension, right click and select edit, *Inlet* is the root filename in this example)

```
title ADH
"../../bin/pre_adh.exe" Inlet > zout.pre_adh 2>&1
"../../bin/adh.exe" Inlet > zout.adh 2>&1
```

    a. Give the path to the pre_AdH executable, the root filename, store screen output to the listed file, put standard error and output in the same file, and run in background
    b. Give the path to the AdH executable, the root filename, store screen output to the listed file, put standard error and output in the same file, and run in background

6) Double click the batch script to execute

7) The screen output files can be viewed in many text editors and notepad while running

Running AdH from the Command Prompt

1) Generate the mesh (.3dm)
2) Generate the boundary conditions (.bc)
3) Generate the initial conditions (.hot)
4) All three files reside in the same location
5) Open the command prompt and move to the location of the input files
6) Give the path to the pre_AdH executable followed by the root file name.
    a. If the output should be saved, direct it into a file
    b. You can use the same information as in the batch script option
7) Give the path to the AdH executable followed by the root file name.
    a. If the output should be saved, direct it into a file
    b. You can use the same information as in the batch script option
8) The screen output files (if being saved) can be viewed in many text editors and notepad while running

Tips

1) Remember to make a new hotstart file any time nodes have been added or removed, bathymetry has changed, or time series have changed for your starting time.
2) When attempting to post-process multiple runs in SMS, you may need to re-sample the datasets so that their time intervals are consistent. This is done in the data toolbox under the time options.
3) Always check your pre_adh output to ensure that you have no errors…people make this mistake and end up running with old *.adh files and can't figure out why nothing has changed!
4) The order of the bc file does not matter. Spacing is only important on the cards with two character strings. These cards are read by 5 characters…meaning that gravity (MP G) will need two spaces after the *G* before giving the value else you will get an error in pre_adh.