Lock Approach Modeling –suggestions

March 14, 2012

This paper will layout how I would approach run of the river lock approach. This would also apply to any river model in which a tailwater elevation is applied downstream and a flow rate upstream. This means that at least at the boundaries the river is subcritical. As I learn more about your problems I will modify this paper to reflect what appears to be the best approach.

The entries on each subject will be kept brief, but an "explanation section" will follow that will mirror these entries and give the reasoning behind each.

1. Developing the mesh
    a. The mesh should be smooth with gradual transition in size of the elements
    b. The edges of the river in which you expect to wet and dry
        i. Should be a separate material type-this allows you to turn off adaption there, and change the eddy viscosity type.
        ii. Since we aren't particularly interested in the results near the edge of the river we can expand the size of the elements somewhat at this river boundary. This will make wetting-drying easier
    c. If we have the elevation data from surveys then it would be good to extend the mesh up to just above the highest pool elevation you will want to run. We will want to make maybe a couple of material types that correspond to the region between two elevations out in the near edge of the river.
2. Creating a hotstart file
    a. I'm assuming that you are starting with a flat water surface.
    b. Select an elevation that matches your tailwater, or perhaps slightly higher in elevation (0.01m). Create a file that gives you the depths (water surface elevation – bed elevation).
    c. Using Excel, or the data calculator, change all negative depths to something like -0.001.
    d. Use this as your hotstart file.
3. Boundary condition file
    a. Steady-state option
        i. Let's try using the steady-state boundary condition. We need to get fast turn around and this is a way to get it. Rich has had problems with this for really high velocity conditions. But folks solving problems like these have found it to be very helpful. This is the TC STD entry. On the same entry of TC STD you give it a starting time step size and a maximum time step. To get a good estimate of the starting time step take the length of a typical element and divide by the speed of a free surface wave. $time\_step\_initial \approx \Delta x/(gh)^{1/2}$ the maximum time step is something that you will have to find by trial. I would start with something really large like 10,000 seconds. If the model runs okay for a while and then fails as the time steps get bigger, then use the maximum as what was running

successfully. Also set the IP NIT to 1. This uses a single iteration. Also make your inflow values start at 0.0 and ramp up to the value you want in say 120 seconds.

b. Dynamic Option-Once you get a successful steady-state run you can then make a dynamic run by using the results to create a hotstart file to make this run.
    i. Since we will have wetting-drying and perhaps using ported guardwalls simulated with lids, we are doing to be solving problems that are non-smooth. These don't converge well using the Newton method. This means it is better to have fewer nonlinear iterations (use IP NIT of say 3 or so) and smaller time steps.
    ii. The TC ATF is a way to have the model select its own time step. This can be quite beneficial in speed.

c. General
    i. You will need to use the MP DTL option. Probably with a value of about 0.1 m (or less).
    ii. You'll want to use option 1 of the EEV for the eddy viscosity. Option 2 is more nonlinear and tends to behave worse. For the material describing the shoreline that wets and dries you have want to use a fixed eddy viscosity via the EVS entry. You can estimate this value using the manual.
    iii. I would set 0 levels of adaption in the material that wets and dries.
    iv. Take advantage of the fraction capability for each material type. This is given by MP FRC *material# value* where the material number is obvious and the value is the fraction of the element below which the element is dropped from the calculations. For the wet/dry areas probably try 0.2 to start.

Explanations

When we are using ADH for these problems, we are solving differential equations that represent conservation of mass, and conservation of momentum in two dimensions. The equations are nonlinear. This means that we have parameters, like velocity and depth, that are unknown and they are multiplied (or divided, raised to powers). For example, if you consider the conservation of mass equation we are looking at gradients in the unit flux u*h. This is the x component of velocity times the depth. We don't know either of these (velocity or depth), but need to calculate them. In order to solve for these we linearize the problem and iterate to get it to converge at each time step. These are the nonlinear iterations you set in ADH. So there is a progression in which we take a time step and then make several iterations to converge. When we do this we are time accurate. We have a dynamic solution that is time accurate.

We can also run steady-state calculations in which we don't get time accurate solutions along the way, but the final steady-state is okay. This is the steady-state option. This is why we take only one iteration per time step. The method also chooses its own time step within the bounds that you give it. It typically progresses from the starting relatively small time step and grows this until you are convinced it has reached steady state. ADH can't really tell if you've reached steady-state. You have to give it a large enough simulation time that you are sure it is at steady-state. You can look at the fluxes of the inflow and outflow as well as the depths and determine a) do the fluxes match and b) are the solutions unchanged from time step to time step.

Now consider resolution in the shallows near the edge of the mesh.