ADAPTIVE HYDRAULICS

A TWO-DIMENSIONAL MODELING SYSTEM
DEVELOPED BY THE COASTAL AND HYDRAULICS LABORATORY
ENGINEER RESEARCH AND DEVELOPMENT CENTER

A PRODUCT OF THE SYSTEM-WIDE WATER RESOURCES PROGRAM

QUICK GLANCE:
Constructing A Model

Three files are required for running ADH: the **2-D mesh** file, the **boundary conditions** file and the **hotstart** file.

1. 2-D Mesh Files

The three dimensional mesh files needed for ADH are generated completely within the GMS or SMS. Once the mesh has been generated in the GMS or SMS, the file will be used in ADH without modification. The filename given to the mesh file, having an extension *.3dm*, will serve as the root name for all ADH input files. Details on mesh generation can be found in the example problems contained within this text or in the GMS and SMS reference manuals.

2. Hotstart File

The **hotstart** file, *filename*.hot, is used to specify two types of model data: initial conditions and scale factors. Initial conditions for hydrodynamics consist of the depths and velocity components.

Field data are often available and used as a starting point for many problems. The field data can be specified as the initial conditions used in the flow and transport equations for a specific probleminterface for entering field data. This data is entered into a scatter point data file and interpolated to the . These data are specified in the **hotstart** file. The GMS and SMS provide a simple problem mesh.

For ADH, initial water depths must be given, although some can be zero or negative (dry). All other variables can start at zero and therefore be omitted from the hot start file. If data types are not specified default values of zero will be supplied.

A set of predefined dataset names is used to declare data types, shown below:

ioh or IOH Initial Depth
iov or IOV Initial Velocity
icon or ICON Initial Concentration
id or ID Initial Displacement (sediment calculations)

The datasets used for the **hotstart** file can be generated with the GMS or SMS. A standard GMS 3-D mesh data set format is used in the **hotstart** file. The files contain a specific heading, the timestep for the values to follow, and the depths or X, Y, Z velocity components. No node or element

numbers are given as the values are listed; there is simply the correct number of lines to match sequentially with the number of nodes.

Multiple data sets are exported from the GMS or SMS and copied to the **hotstart** file in any order. If a dataset is not supplied for one or more of the parameters, ADH will assign default values to all the cells for that parameter. Default initial conditions assume a value of 0.0.

Typically initial velocities of zero are fine, but zero depth everywhere will create problems and is never a good method for starting a problem.

When hotstarting, ADH reads the values in the hot start file and assigns them at the start time specified in the boundary conditions file. The time in the **hotstart** file is ignored by ADH.

For consistency, however, it is recommended that the time in the hot start file —located on the **TS** line — match the start time in the boundary conditions file, located on the **TC T0** card. To create a **hotstart** file from a previous run, the ASCII data list for each variable at the desired time should be taken from the output files and combined to create the new **hotstart** file.

3. Boundary Conditions

The boundary conditions file contains a series of cards that represent the operation controls, iteration parameters, material properties, boundary strings, solution controls, time controls, and output controls.

Starting a model

The basic steps for preparing and running ADH are given in this section along with several checks to possibly prevent solver errors and delays.

Prepare the mesh: Prepare the mesh file (.3dm). This can be done with SMS or GMS. Be certain to check the mesh quality and modify the mesh where necessary. In SMS and GMS, the mesh quality can be turned on under the display options dialog box. Nodes can be selected and moved to better the quality. Also check for and delete disjoint nodes, or nodes not connected to any elements. These can be found under the nodes menu in the mesh environment of SMS. For good meshes that run best in the solver:

- The percent area change between adjacent elements should not exceed 50%.
- Typically, the minimum for interior angles is 20 degrees and the maximum is 130 degrees.

- No more than eight elements should join at one node.
- Try to avoid large gradients due to slope in bed elevation.

The bed elevations can be included after the initial mesh is developed by mapping a scatter data set of the bed elevations to the existing mesh. Be certain to check the box labeled map elevations in order to have them included in the mesh geometry file.

Prepare the boundary condition file (.bc): For ease in inputting large sets of data, spreadsheet software may be used, but avoid tabs and other control characters. Save the file as a text file so that formatting characters are eliminated. If ftp software is used to move the text files, transfer using ascii mode rather than binary to avoid excess characters attached to the end of each line.

Prepare the hotstart file (.hot): The solver does not like to begin with values of zero for depth. The best choice is to begin with the initial water surface at some level. If a tide is being included, then begin with the water surface at the initial tide level. This step can be done in SMS with the Data Calculator to set the water surface level and exporting the data set that it generates to a text editor. These values should be equal to the desired water surface elevation minus the bed elevation so that the depth at each node is calculated regardless of the signs on your elevations. Make certain to change the "NAME'' to ioh in the text file. No difficulties have been found from starting the velocities at zero; therefore, if zero velocity is appropriate for your problem, the velocity portion of the hotstart file may be omitted.

Run two files: Run Pre_ADH first, then run ADH. ADH runs on multiple processors, so consult your system administrators for the correct method of running interactively or submitting jobs. The preconditioner and number of blocks per processor, BLK card in the boundary condition file, can be modified to determine best performance for an individual problem.

What is output: ADH outputs a velocity file and a depth file with a .dat extension. These files can be opened and viewed in SMS or GMS without any further post-processing.

Creating filmloops: ADH will output the initial values for depth and velocity at the model's start time. Therefore, if this time is included in your output control series, two sets of data at this time will be in your data files. This is not a problem for ADH, but when creating filmloops in SMS, duplicate times will generate errors and may cause SMS to close.

Sediment transport: Sediment transport simulation require the use of SI units. If converting input files from English to SI units, be certain to convert the the geometry file, hotstart file, and boundary condition file. The geometry file can be easily converted within SMS. The hotstart file can be converted with the help of the data calculator is SMS or a spreadsheet application. The boundary condition file will need to be corrected for any cards containing length units. These cards include the eddy viscosity (EVS), wetting/drying limits (DTL), density (RHO), gravity (G), Manning's units constant (MUC), and any XY1 series describing flows and elevations.