Guidelines for Solving Two Dimensional Shallow Water Problems with the ADaptive Hydraulics (ADH) Modeling System

R.C. Berger
J.N. Tate [1]

(Note: This manual has been updated to be used with ADH rev #5939 dated January 2009)

Contents

List of Tables

List of Figures

Abstract:

Guidelines are presented for using the US Army Corps of Engineers (USACE) ADH modeling software to model two dimensional shallow water problems. ADH can be used in conjunction with the Department of Defense Groundwater Modeling System (GMS) and the Surface Water Modeling System (SMS); examples in this manual did use these systems. Other pre- and post-processors are available for grid generation and visualization and can be used with ADH with some modification of the files.

This two dimensional modeling module of ADH is intended to be of general use and as such examples are given for supercritical flow channels, rivers, and even currents caused by the wave radiation stresses along a coastal shoreline.

Introduction

The ADH Model (ADH) is a software package that can describe both saturated and unsaturated groundwater, overland flow, 3D Navier-Stokes, and 3D Shallow Water problems in addition to the 2D shallow water module described herein. The model is designed to work in conjunction with the DoD Groundwater Modeling System (GMS). The GMS (GMS) and the Surface Water Modeling System (SMS) are modeling packages for building models, running simulations, and visualizing results. For further information regarding the GMS or SMS, contact the USACE Research and Development Center, Waterways Experiment Station Site or visit the website at http://chl.erdc.usace.army.mil and select the software link.

An example of the use of ADH for shallow water is Pool 8 of the Mississippi River . This detailed mesh was run using ADH, giving the velocity distribution seen here in Fig. 1.1. Also shown is the overland head distribution in the flowfield, Fig. 1.2.

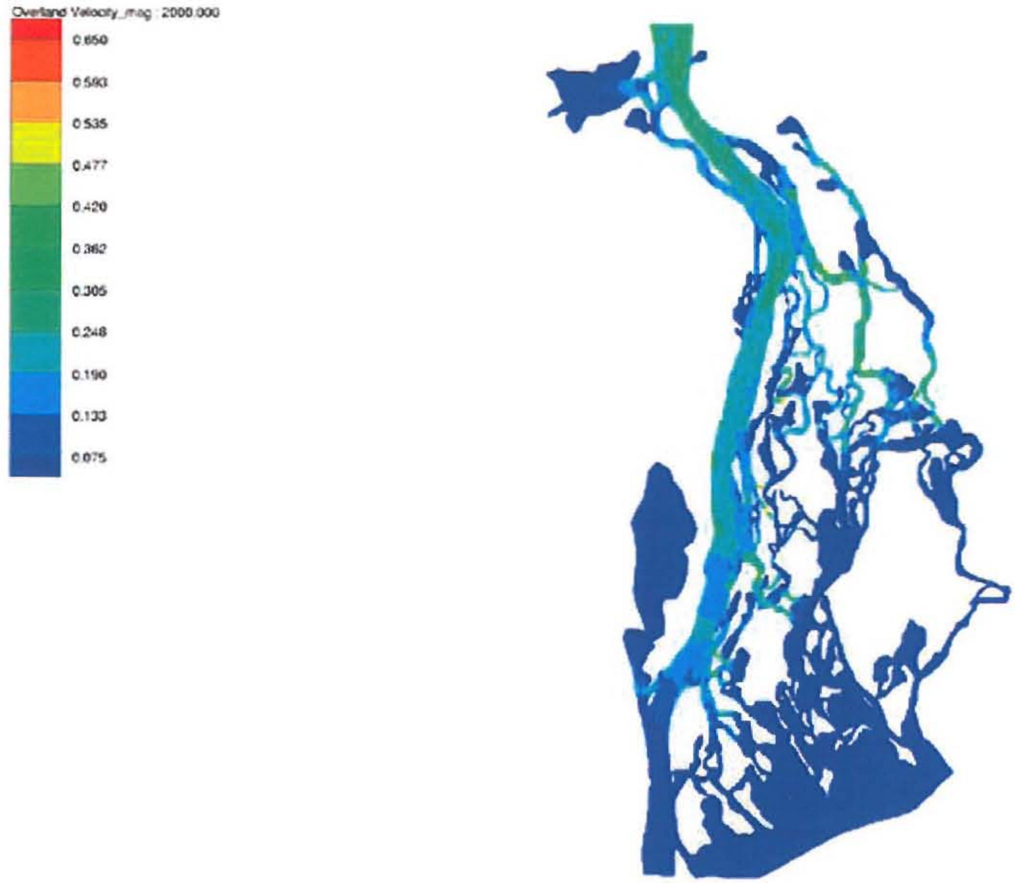

Figure 1.1: Velocity for Pool 8, Mississippi River .

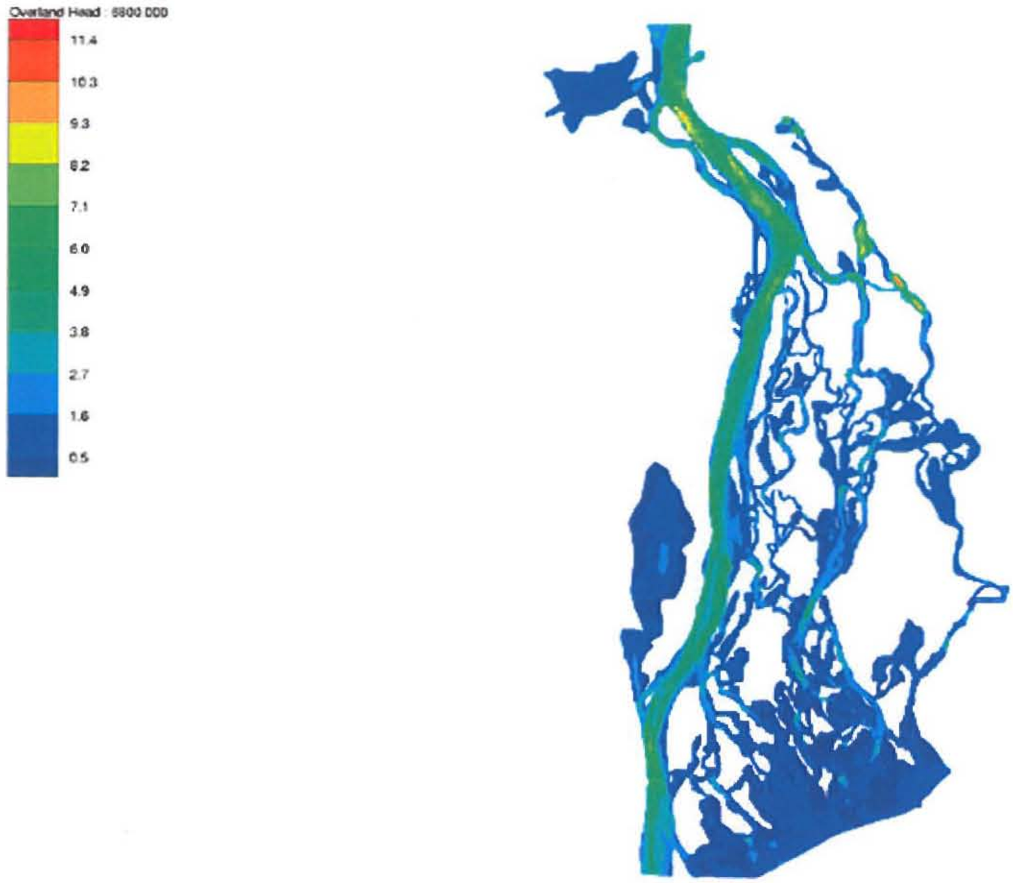

Figure 1.2: Overland depth for Pool 8, Mississippi River .

Three files are needed to run a model in ADH. These files are the mesh file, the boundary conditions file, and the hot start file. The mesh file must be constructed first and can be generated directly with the GMS (2D or 3D) or SMS (2D). Once a mesh file has been constructed, the boundary conditions for the problem and operating parameters for ADH must be specified in the boundary condition file. The hot start file is then generated to establish the initial conditions of the problem.

Once the three required files have been created, pre_adh is run and it creates the necessary input file for ADH. Then the ADH model is run with the command

```
adh filename
```

where *filename* is the root of the model's filenames, i.e. for a model named `pl8_adh` the following three files would be required `pl8_adh.3dm`, `pl8_adh.hot` and `pl8_adh.bc`. All three files must have the same *filename* as their root followed by one of three suffixes. After the model is run, the GMS or SMS can be used to visualize the results.

A Note on Units:

ADH is designed so that the user can specify the unit system to use. However, all parameters must be consistent in that they are all given in English units or SI units and not mixed. The geometry file, boundary condition file, and hotstart file must all be given in the same unit system. There is no card that directly specifies the units being used. Rather ADH uses the values given and calculates with them. If any equations internal to ADH are unit specific, the density or gravity terms are used to decipher which system is being used. This manual will give unit specifications where necessary in dimensional form. **The exception to allowing the user to determine the unit system is when sediment transport is being simulated.** These equations are based on empirical relationships of SI units and require all calculations to be made in SI units. This means that all of the input files must also be given in SI units. **Sediment transport must be in SI units.**

Sign Convention:

The sign convention in ADH is the standard Cartesian coordinate system and flow into the control volume is positive.

Mesh Files

ADH can run two dimensional shallow water flow in two manners. The first is to simply create a 2D mesh. The second is for the shallow water flow to be run on the surface of a 3D mesh. This facilitates running in conjunction with groundwater problems or as a part of a 3D shallow water simulation. Mesh files can be generated quickly and efficiently using the GMS or SMS. Mesh files, *filename*`.3dm`, follow the GMS 3-D mesh format. ADH uses only tetrahedral elements, although the GMS supports tetrahedral, pyramid, wedge and hexahedral elements in 3D. In 2D ADH uses triangles while GMS and SMS support triangles and quadrilaterals. ADH uses the mesh files generated with the GMS directly, without any modifications. GMS mesh files are designated with a .3dm extension. SMS mesh files are designated with a .2dm extension, which the user must simply change to .3dm. Simple rectilinear domains are convenient starting points for more complicated models. For full details on how to produce mesh files using the GMS or SMS, refer to the appropriate reference manual and tutorials which can be found at http://chl.erdc.usace.army.mil.

Boundary Condition Files

The boundary condition file contains many pieces of information necessary to perform simulations with ADH including specified water surface elevation and velocity boundary conditions; Natural and outflow boundary conditions; time step data; output controls; adaptivity controls; error tolerances; and maximum levels of mesh refinement.

Error tolerances are specified in the boundary condition file. Mesh refinement is governed by the specified error tolerance in the boundary condition file. In addition, the maximum levels of refinement must be specified by material type, giving the user additional control over the mesh adaptation.

A boundary condition file, *filename*`.bc`, for the ADH code contains a series of one line control cards. Cards are single line entries and cannot be wrapped across lines. The cards fall into eight basic categories: operational parameters, iteration parameters, material properties, boundary strings, solution controls/boundary conditions, time controls, output controls, and series. Operation Parameters control the operation of the code, the reserved memory space, type of problem being modeled, and the solver preconditioning arrangement. Iteration parameters control the iterative methods employed by the model. Material Properties define the flow and transport constants for each material in the model. Boundary conditions are set using string array cards defining the interior and surface boundaries of the problem, including node and face boundaries. Solution Controls specify the initial/boundary conditions. Time controls specify the time steps used to run the model. Output controls define the times at which the output is printed and Series cards are used to define various parameters. The different cards and their categories are shown in Table 1.1.

Table 1.1: Cards

Operation Parameters	OP SW2	2D Shallow Water
	OP INC	Incremental memory
	OP TRN	Transport Quantities
	OP BLK	Blocks per processor
	OP PRE	Preconditioner
	OP BT	Enable Vessel Movement
	OP TEM	Enable Second Order Temporal Terms
	OP TPG	Petrov-Galerkin Coefficient
	OP BTS	Enable Vessel Entrainment
Iteration Parameters	IP NIT	Non-Linear Iterations
	IP NTL	Non-Linear Tolerance
	IP ITL	Increment Tolerance
	IP MIT	Maximum Linear Iterations
	IP FNI	Forced Non-Linear Iterations
	IP FLI	Forced Linear Iterations

	IP RTL	Runga-Kutta tolereance for reactive constituents
	IP SST	Quasi-Unsteady Tolerance
Constituents	CN CON	Any Contituent
	CN CLA	Clay and/or Silt Sediment
	CN SND	Sand Sediment
	CN VOR	Vorticity
	CN SAL	Salinity
	CN TMP	Temperature
Materials	MP EVS	Eddy Viscosity
	MP EEV	Calculated Eddy Viscosity
	MP MU	Kinematic Molecular Viscosity
	MP G	Gravitational Acceleration
	MP MUC	Manning's units constant
	MP RHO	Density
	MP COR	Coriolis Latitude
	MP DTL	Wetting/drying limits
	MP ML	Maximum Mesh Refinement
	MP SRT	Mesh Refinement Tolerance
	MP DF	Molecular Diffusion
(Quantities for Transport)	MP TRT	Transport Solution Error Tolerance

	MP NBL	Number of Bed Layers
	MP SBA	Bed layer applied to all nodes
	MP SBN	Bed layer applied to selected nodes
	MP SBM	Bed layer applied by material
	MP CBA	Cohesive sediment applied by layer
	MP CBN	Cohesive sediment applied to selected nodes
	MP CBM	Cohesive sediment applied by material
Boundary Strings	NDS	Node String
	FCS	Face String
	MTS	Material String
	EGS	Edge String
	MDS	Mid String
Time Series	XY1	X-Y Series Cards
	XY2	X-Y-Y Series Cards
	XYT	Curve Fit Tolerance
	XYC	Wind Station Coordinates
Friction Controls	FR MNG	Manning's N Roughness
	FR ERH	Equivalent Roughness Height
	FR SAV	Submerged Aquatic Vegetation
	FR URV	Un-submerged Rigid Vegetation
Solution Controls	DB OVL	Dirichlet - Velocity

	DB OVH	Dirichlet - Velocity and Depth
	DB TRN	Dirichlet - Transport
	DB LDE	Dirichlet - Stationary lid elevation
	DB LDH	Dirichlet - Depth of water under stationary lid
	DB LID	Dirichlet - Floating stationary lid assignment
	NB DIS	Natural - Total Discharge
	NB OVL	Natural - Flow
	NB OTW	Natural - Tailwater elevation
	NB TRN	Natural - Transport
	OB OF	Outflow Boundary
	EQ TRN	Equilibrium Transport Boundary
	OFF	Deactivate String
Time Controls	TC T0	Start Time
	TC IDT	Time Series
	TC NDP	Non-Adaptive Time Control
	TC IAC	Adaptive Time Control turned on
	TC TF	Final Time
	TC SDI	Sediment transport increment for time step
	TC STD	Steady State solution
	TC	Quasi-Unsteady solution

	STH	
Output Controls	OC	Output Control Series
	OS	Auto-build Output Series
	FLX	Flow Output
	PC ADP	Adapted Mesh Print Control
	END	Signifies the end of the BC file

Each card consists of at least one character string identifying the type of card. It may then contain further character fields and/or numeric data fields. There are two important points to note about this file. First, the leading 6 columns are reserved for character field keywords ONLY. All numeric data MUST start in column 7 or later. As an example, consider the line of input

```
MP MU 1
```

This input would result in two character fields being read. One would have the value "MP" and the other, the value "MU" as intended. The following incorrect line

```
MP MU1
```

would result in fields containing the values "MP" and "MU1". It is important to note that the parser cannot handle lines more than 150 characters wide. An input file template is provided in Table A.12.

Operation Parameters

Each Operation Parameter card consists of two character fields and may contain one numeric field. Operational parameter cards are identified by an "OP" in the first field. OP cards control the type of system being modeled. An OP SW2 card is used to specify 2D Shallow Water flow modeling.

The code allocates memory as needed during the run to store the additional elements and nodes created during the refinement process. The memory is allocated in blocks. The size of the block is specified by the user on the OP INC card. If the specified number is too small, the program will continually seek additional memory, slowing the run time of the program. Alternately, if the number is too large, the program will require excess memory not needed to run the code.

The preconditioner for the linear solver and the manner in which it is implemented are specified by the

OP PRE and the OP BLK cards.

```
OP SW2
OP INC 200
OP PRE 2
OP BLK 10
```

The first card specifies the preconditioner. The integer can be 0, 1, 2, or 3 for various preconditioning schemes. The second card defines how many blocks per processor are to be used in the preconditioner. These are subdividing the problem to perform a direct solve on each block and the total group of all blocks can be used to perform a coarse grid solve. Which of these options is used is specified by the OP PRE choice. In this case, the 2 indicates two-level Additive Schwarz preconditioning using 10 blocks per processor.

After finding a flow solution, an associated transport problem can be solved. The number of transported quantities is given on an OP TRN card. The OP TRN card is a required input card. If the problem does not involve transport, zero (0) quantities are specified on the OP TRN card. In addition, if transport equations are not being modeled, no transport properties or boundary conditions may be specified. An error message will be displayed if transport properties are included in the input file but no transport quantities have been specified. The format of the OP cards is found in Table A.1. The following card specifies one transported quantity.

```
OP TRN 1
```

The OP TEM card is included to enable second order temporal terms when solving the time derivatives so that numerical dissipation is reduced. The numerical scheme is a form in which the user can choose between the 1st or 2nd order schemes or even a fractional amount of each with the use of the variable tau_temporal. This variable, the alpha term below, is controlled via the OP TEM card and is defaulted to 0 if not included in the boundary conditions file. The final form of the temporal scheme is given by:

$$\frac{dh}{dt} \approx \alpha \frac{\left(\frac{3}{2}h^{n+1} - \frac{1}{2}h^{n}\right) - \left(\frac{3}{2}h^{n} - \frac{1}{2}h^{n-1}\right)}{dt} + (1-\alpha)\frac{h^{n+1} - h^{n}}{dt} \quad (1.1)$$

The following card specifies to use only the second order temporal terms since the variable is given a value of 1.

```
OP TEM 1
```

A detailed description of this approach is given at the end of this document.

Iteration Parameters

There are three iteration parameter cards that must be specified by the user. Iteration parameter cards are identified by an "IP" in the first field. An `IP NIT` card specifies the maximum number of non-linear iterations. An `IP NTL` card specifies the convergence tolerance for the non-linear iterations. An `IP MIT` card specifies the maximum number of linear iterations for each non-linear iteration. At the maximum number of iterations specified on the `IP NIT` or `IP MIT` cards, if the convergence is not sufficient ADH will reduce the timestep size and continue the calculations. Another option is available for each of these cards. They function like the two previous cards but if the maximum iteration count is reached the calculations are accepted and ADH proceeds. The `IP FNI` card then is for the non-linear iteration maximum and `IP FLI` is for the linear iteration maximum. The `IP ITL` card is an optional card that allows convergence to be determined by the change in the velocity, depth and concentration solutions. By default this tolerance is zero. The format of these cards is specified in Table A.2.

```
IP NIT 5
IP NTL 1.0E-3
IP MIT 500
```

Materials

Material Property cards are identified by the designation "MP". There will be a set of cards for each material type in the model. Each group contains a set of refinement control cards. Refinement may be adjusted to independently follow the error in the flowfield and transport equations. For four of these cards, the first two fields contain character strings, specifying the type of card (MP) and the specific parameter (ML, SRT, EVS, etc.). The third field is an integer field containing the material number to which the values apply, (*mat#*). The remainder of these cards are applied throughout the problem and do not include a material number. These are MU, G, RHO , MUC, and DTL. The formats for these cards are listed in Tables A.4 and A.6.

Flow Parameters

Four cards can be used to specify flow parameters: kinematic eddy viscosity (EVS), estimated or calculated eddy viscosity (EEV), kinematic molecular viscosity (MU), and density (RHO). The acceleration due to gravity is defined as Length/Time^2 (G). The Manning's units constant (MUC) is used to keep the units proper for shear stress calculation. This is 1.0 for SI units and 1.486 in English units. The density for 2-D shallow water problems is also set such that units remain consistent when wind data is used to influence the flow. The density (RHO) should be set to values corresponding to units of Mass/Length^3 (kg/m^3 or slugs/ft^3); for water, $1000.0 kg/m^3$ for SI units and $1.94 slugs/ft^3$ for English units. The 2-D shallow water equations also include the coriolis force due to the earth's rotation. The COR card requires the material number and the latitude in decimal degrees for each material. Most of these parameters are obvious but some explanation of the kinematic eddy viscosity and estimated eddy viscosity is warranted.

Kinematic Eddy Viscosity, EVS

The eddy viscosity is representative of the turbulence generated in the spreading of momentum that is smaller than can be represented by the grid resolution. Kinematic eddy viscosity has units of $Length^2/Time$ and is related to the flow itself. The molecular viscosity on the other hand is a fluid property.

The kinematic eddy viscosity is expressed as a tensor in the following form:

$$\begin{matrix} EV_{xx} & EV_{xy} \\ EV_{xy} & EV_{yy} \end{matrix}$$

The three values of the tensor are entered on the `MP EVS` card in the following order: EV_{xx}, EV_{yy}, EV_{xy}. If the hydraulic conductivity is independent of the direction of measurement, the formation is termed *isotropic*. In the isotropic case, $EV_{xx} = EV_{yy}$ and $EV_{xy} = 0$. Another option is to set all terms in the tensor equal to 0 and declare the total viscosity through the `MP MU` card.

Estimated Eddy Viscosity, EEV

The estimated eddy viscosity is used as a means to calculate the eddy viscosity needed within the model as it runs. If the `EEV` card is used in place of the `EVS` card, the user will give only a weighting factor or coefficient on the following equation and the components of the eddy viscosity are then determined.

$$\begin{aligned} EV &= \frac{coef * h_{avg}^{5/6} * n * |v_{avg}| * \sqrt{8 * g}}{C} \\ EV_{xx} &= EV + viscosity \\ EV_{xy} &= EV \\ EV_{yy} &= EV \end{aligned} \tag{1.2}$$

The terms for the above equation are listed below.

> It is coef (this varies between 0.1 and 1.0)
> h_avg^(5/6) that is the average depth in the element raised to the 5/6 power
> n is mannings n
> magnitude of the element average velocity, v_avg
> times square root of 8 time g (gravitational acceleration)
> divided by the units constant, C, 1.0 for meters, and 1.486 for feet.

Eddy viscosity must be given for every material with either an EEV or EVS card. Both cannot be used for one material, but both can be used in a single model.

Mesh Refinement

An MP ML card is used to specify the maximum levels of mesh refinement, or the total number of times that an original element may be split within a material type. Refinement can be turned off in a material by specifying zero (0) as the maximum level of refinement. When refinement is on, the solution error tolerance is given on the SRT card. If the solution error on an element exceeds the refine error tolerance given on the SRT card, the element is split. A separate error tolerance is specified for the transport equations and is set on the TRT card when transport quantities are included in the model. This card is only a tolerance, however. The material must be set to allow refinement in order for any adaption to occur. The error for hydrodynamics is calculated as

$$coefficient = \int_e \left(\frac{\partial h}{\partial t} + u * \frac{\partial h}{\partial x} + v * \frac{\partial h}{\partial y} + h * \frac{\partial u}{\partial x} + h * \frac{\partial v}{\partial y} \right)^2$$
$$Error = \sqrt{coefficient} * Area_{element} \tag{1.3}$$

and this value is compared to the number given on the SRT and TRT card. The error value given in the output data file is the ratio of the calculated error to the specified tolerance. By observing the *_err.dat file produced, an appropriate error tolerance for adaption can be determined. It is useful to run a short time with the SRT card set to one (1) in order to get an idea of what this value should be for a particular model run. When including transport constituents, the error tolerance for hydrodynamics and sediment should be determined separately. This can be accomplished by setting the transport tolerance to a large value (100000) so that these errors essentially get ignored while running with the hydrodynamic tolerance set to one. This simulation will provide an error file useful for determining how the SRT card should be set. Then reverse the tolerances in order to determine where to set the TRT card. The unrefine tolerance is currently set within the code as 10% of the refine tolerance for both flow conditions and transport conditions. When the grid solution error improves, the elements are recombined.

```
MP ML 1 5
MP SRT 1 100
```

Different material types can have different levels of refinement. Some experimentation with the error tolerance is usually necessary to gain the desired level of refinement. The adapted meshes can be output during the simulation by including a PC ADP card. By including this card, the mesh and associated solution files will be saved at the timestep intervals specified on the output control card. The output files will be named like so: "filename.3dm-timestep#.0", "filename.dep-timestep#.0", "filename.ovl-timestep#.0" which is a geometry file for each timestep, the depths for each timestep, and the velocities for each timestep.

Constituents

To add any transported constituent, the OP TRN card should be set for the number of quantities being included. Every transported constituent will also require the necessary transport constituent properties discussed below.

The constituent cards are provided so that specific sediment types can be accounted for as well as other transported quantities. There is a separate card for each sediment type. Sand is specified on a card by itself. Clay and silt can be specified with the same card. Vorticity, temperature, and salinity can also be transported as constituents. Other constituent types can be categorized as a general constituent and use the CON card. Also included on the CN cards are the constituent ID number, the reference concentration, and other sediment specific parameters.

```
CN CON 3 10.0 0. 0.  0. any
CN SND  2 0.4 0.00044  2.65 0.3  sand
CN CLA # # # # # # # # # clay or silt
```

The first number indicates the constituent number. The next number for all cases is the reference concentration. The reference concentration, like all concentration of suspended sediment in ADH, is in mass per unit mass multiplied by 1.E+6. That is, it is micromass per unit mass or parts per million. Transport will be described in more detail below. The formats for these cards are given in Table A.3.

Vorticity Transport-Bendway Correction

A method for correcting 2-dimensional models for the 3-dimensional effects of vorticity around bends has been included in ADH. This is done voluntarily with the VOR card. The vorticity is included as a transport constituent due to its constituent like behavior as it moves with the model and must therefore be included in the OP TRN card. The bendway correction is enabled by including CN VOR in the boundary condition file. This card is followed by the constituent id number, a normalization factor, A_s term, and D_s term. The A_s and D_s terms are empirical coefficients determined by integrating against measured values. ADH used default values of $A_s = 5.0$ and $D_s = 0.5$ which will be set automatically if these terms are input as 0.0 in the boundary condition file.

```
CN VOR 1 1.0 0.0 0.0
```

Baroclinic Transport

ADH typically computes independent of density, meaning that the solutions are not affected by density gradients. However, salinity and temperature affect density and must be simulated as such when modeling salinity or temperature. ADH will automatically invoke the baroclinic effects so that these transport constituents can be model appropriately. Here's an example.

```
CN SAL 1 35. 0.0  0.0  0.0
```

Sediment Transport

ADH allows the user to calculate the transport of cohesionless sediment (sand), cohesive sediment (clay and silt), and mixed sediments (i.e. sand, clay, and silt). The model is capable of running multiple grain sizes. The sediment is transported as suspended load and bed load. As suspended load, each grain class is transported as a moving constituent. So in making a sediment transport calculation, one must determine how many grain classes are going to be modeled. One will have this number of constituents plus any other non-sediment constituents being routed. In other words, one might be routing 3 grain classes of sediment and 1 other constituent so the card describing this is "OP TRN 4".

Cohesionless Sediment (Sand)

The characteristics of the cohesionless sediment are its grain diameter, its specific gravity, and its porosity. These are supplied via the "CN SND" cards. The "SND" stands for "sand". Here's an example.

```
CN SND 1 20.0 1.E-4 2.65 0.3
CN SND 2 7.27 5.E-4 2.65 0.3
CN SND 3 4.23 1.E-3 2.65 0.3
```

Beginning after the "CN SND" are the constituent number, the reference concentration for this grain class, grain diameter, the specific gravity, and the porosity. The transport numbers must be in ascending order by grain diameter. The reference concentration like all sediment concentrations given in ADH are in units of micromass per unit mass. ADH actually operates on the dimensionless units but the input/output is easier if we multiply this by 1.E+6. ADH sediment transport is also based on metric units, so grain diameter must be given in meters.

Next we need to describe the bed layer structure. We must describe each layer beginning with the lowest layer and then working our way up to the bed surface. These can be given by individual node, by material type, or for all nodes at once. In fact, the model writes over any prior designation so one could specify all first, followed by a few material types, followed by individual nodes.

As an example for two layers we might have the following.

```
MP NBL 2
MP SBA 1  0.5  0.0  0.4  0.6
MP SBA 2  0.5  0.4  0.3  0.3
```

The "MP NBL" card tells ADH to expect 2 bed layers to be defined with the "MP SBA" cards. "MP SBA" means that the sediment bed description will be used for all nodes. The next number is the layer number, followed by the layer thickness. Layer 1 is the deepest (or bottom) layer. The next 3 numbers are the fractions of each of the sediments shown in the "CN SND" cards. There are two layers shown here. Layer 2 is the top layer and its distribution is more weighted toward the smaller grain diameter sand. We could now designate by material and/or node and these nodes would be modified to reflect this difference.

The implementation of boundary conditions is handled precisely the same as any other constituent. An additional equilibrium transport boundary condition is available for cohesionless sediment transport. When this condition is specified, the concentration that is required for a state of equilibrium at that location is applied in suspension. An equilibrium condition is one in which no sediment would erode or deposit. This condition is specified with an "EQ TRN" card followed by the node string number where it is to be applied, the constituent number for the grain being applied, and an initial concentration value. If no sediment is included initially, then this value should be set to zero.

Cohesive Sediment (clay and/or silt)

Cohesive sediment behaves differently than cohesionless sediment. Cohesionless sediment tends to erode grain by grain; hence the erosional and depositional properties of cohesionless sediment can be defined purely as grain properties. However, the cohesive properties of silts and clays result in erosion behavior that is more generally a property of the condition of the consolidating bed than a property of the individual grains. Hence, the user must specify the erosion characteristics of the cohesive bed. In ADH, these erosion characteristics are governed by the following equation:

$$F_E = M\left(\frac{\tau}{\tau_c} - 1\right)^n \tag{1.4}$$

Where F_E is the erosion flux, M is the erosion rate constant, n is the erosion rate exponent, and τ_c is the critical shear stress for erosion.

The erosional characteristics of *newly deposited* sediment are given in the ?CN CLA? card (The "CLA" stands for clay, although both clay and silt are specified with this card.) This card also gives the grain properties and settling properties of the individual cohesive sediment grain classes. These parameters given on the "CN CLA" card are the grain diameter, the specific gravity, the bulk density, the critical shear stress for erosion (τ_c), the erosion rate constant (M), the critical shear stress for deposition, and the settling velocity. Note that the erosion rate exponent (n) is assumed to be equal to 1 for newly deposited sediment (this is consistent with the observation of Parthenaides).

The complete list of entries on the "CN CLA" card is given as follows. Beginning after the "CN CLA" are the constituent number, the reference concentration for this grain class, grain diameter, the specific gravity, the bulk density, the critical shear stress for erosion, the erosion rate constant, the critical shear stress for deposition, and the settling velocity. The transport numbers must be in ascending order by grain diameter. Here's an example:

```
CN CLA  1  1.0  0.000001  2.65  1200.0  0.014  0.00016  0.01   0.00006
CN CLA  2  1.0  0.00001   2.65  1400.0  0.02   0.00018  0.015  0.00016
```

The reference concentration like all sediment concentrations given in ADH are in units of micromass per unit mass.

The grain class distributions in the bed are defined in the same way as they are defined for the cohesionless case (i.e. with "MP NBL" and "MP SBA" cards). In addition to these, cohesive sediment beds require a "MP CBA" card for each bed layer, to define the cohesive properties of the layers. The inputs to the "MP CBA" cards include the following: layer number, bulk density, the critical shear stress for erosion (τ_c), the erosion rate constant (M), and the erosion rate exponent (n). Here's an example for

2 bed layers:

```
MP CBA 1 2200.0  0.1    0.00018   3.0
MP CBA 2 2000.0  0.08   0.00016   2.0
```

"EQ TRN" cards cannot be used for cohesive sediments. The concept of an equilibrium concentration has no physical meaning for cohesive sediments, since in general simultaneous erosion and deposition does not occur.

Mixed Sediments (sand, silt and clay)

Mixed sediment beds behave as either cohesive or cohesionless beds, depending on the total fraction of cohesive material in the bed. In ADH, this fraction has been set at 0.1. So, at each time step, the model calculates the fraction of cohesive sediment present in the active layer at each node. If, at a given node, that fraction is greater than 0.1, then the bed at that node behaves as a cohesive bed for that time step.

If both "CN SND" cards and "CN CLA" cards are given, then the transport numbers for the "CN CLA" cards must be greater than those of the sand cards. That is, all sand should be listed in increasing size order, and then all clays and silts should be listed in increasing size order. Here?s an example of 2 sands, 1 clay, and 1 silt being given in the same file.

```
CN SND 1 1.0 0.0001    2.65  0.3
CN SND 2 1.0 0.001     2.65  0.3
CN CLA 3 1.0 0.000001  2.65  1200.0  0.01  0.00016326  0.01  0.00006
CN CLA 4 1.0 0.00001   2.65  1400.0  0.01  0.00016326  0.01  0.00016
```

The additional grain and bed layer: the solid boundary

The sediment transport model will automatically add an extra grain and bottom-most bed layer in order to ensure a solid boundary that cannot erode. The extra grain (labeled with the largest constituent number) is defined as 10% larger than the largest grain included by the modeler and assigned a large critical shear stress for erosion so that it cannot enter the suspended load. The extra bed layer (labeled as bed layer 1 since it is on the bottom) is set to 5.0 meters in thickness and composed only of the largest grain size, as defined by the added grain. The model output will include solution files for the added grain and bed layer and these must also be included when any hot-starting is performed. However, the boundary condition file should not be modified for these additional variables when hot-starting - they are accounted for when Pre_ADH is run.

Transport Constituent Properties

When the model contains transported quantities, those constituents must be described within the boundary condition file. The diffusion and refinement tolerance must all be specified as given in table 1.1. The structure for each of these cards is given in A.5. These quantities must be given for each material type and constituent; meaning that if the model has two materials and two constituents, there will be four sets of constituent properties. The molecular diffusion is set on the "MP DF" card, but if an estimated eddy viscosity (EEV) card is used then this value will over-ride the diffusion set on the "MP DF" card for diffusion of transported material.

Boundary Strings

For most problems, boundary data includes Dirichlet data on the domain and flux data (Natural) through a region of the domain surface. Each of these boundary conditions is applied to a "string" of element nodes or faces. Each component of a string is input on a card that specifies the string type and node or element face it contains. There are four types of boundary strings: node, face, edge, and material. Complete strings are input on multiple cards with one node, face or edge per card. The string numbers must not include any gaps...if there is a string 1 and a string 3, there must also be a string 2. Cards may be input in any order and cards for different node strings may be interspersed. See Table A.7 for details on the format of these cards.

Node Strings

Dirichlet data are specified on node strings. These can be made up of boundary and/or interior nodes as the problem requires. The identifier for this card is `NDS`. On each card, the node number is followed by a string number (*string#*).

Face Strings

These are used to designate a boundary for a 3D mesh. The identifier for this card is `FCS`. The card lists the identifier, element number, element face number (*face#*),and then the string number. Currently, mixed or Robin boundary conditions are not supported.

Edge Strings

Natural or flux data are specified across edge strings. They can be also used to identify a wall, i.e. solid boundary. The identifier for this card is `EGS`. The card lists the identifier, two node numbers that comprise an element, and then the string number.

Mid Strings

Flow outputs internal to the domain are determined across mid-strings. They must begin and end on a mesh boundary and are created specifically for flow output. The `MDS` card has the same format as the `EGS` card.

Material Strings

This is used to designate a group of elements for Natural or flux data. They identify a surface area. The identifier for this card is `MTS`. The card lists the identifier, the material number, and the string number.

```
NDS 1037 2
FCS 1008 1 1
EGS 1601 1603 3
MTS 1 4
```

All strings are included in the calculations by default. You can turn off a string if it will not be included in shallow water computations by using the `OFF` card followed by the string number. This allows the modeler to add or remove sections of the domain without having to generate a new mesh.

Time Series

The data that is to be applied at or along a boundary string is specified in a time series. The time series are just as they sound - a value for a given time. The series may be used to define how the flow changes with time, the change in the boundary depth over time, the change in the timestep as the model runs, and even the time at which data is output for review. These series must be sequential in time and the data values will be linearly interpolated for times falling between two specified values. There are several different time series options that will be discussed. All time series, regardless of the type, should be numbered sequentially. You will not have a time series one for both XY1, XY2, and OS but rather a time series one, time series two, and time series three. Also to note, the final time of the series must be equal to or greater than the final time of the simulation so that ADH knows the values to define the boundary conditions.

X-Y Series

The first time series is a basic XY time series. These series begin with `XY1` to signify that they are the start of a time series. The XY1 is followed by the number of the series, the number of points (time, value) that are to follow, the units of which the times will be read in and the units of which the times will be output (if it is the output series). The unit specifications are as follows: 0 = seconds, 1 = minutes, 2 = hours, 3 = days, 4 = weeks, 5 = months and 6 = years. Below this `XY1` line are the points making up the series. These series will be referenced in the solution controls section of the boundary condition file. Another card which may be of interest is the `XYT` card. This card is used to tell ADH the tolerance for the interpolation of the time series values. The card is followed by the series number it corresponds to and the tolerance value. This card is not necessary and can be used at the user's discretion. Table A.8 gives the structure for these cards.

```
XY1   1   2   0   0
0.0       1.0
2000.0    1.0

XY1   2   8   0   0
100.      0.0
200.      0.0
400.      0.0
600.      0.0
800.      0.0
1200.     0.0
1600.     0.0
2000.     0.0

XYT   3  1.0
XY1   3   4   0   0
0.0       20.0
100.0     40.0
200.0     100.0
2000.0    100.0
```

Wind Series

When applying wind data to the model run, the wind stresses for each of the wind stations are specified as a time series. The coordinates of the wind station are given using an XYC card followed by the time series number containing the data for that station and then the x and y coordinate of the station. The wind stresses are then given using an XY2 card followed by the series number and the number of points to follow. The XY2 card tells the code that there will be two values to read for each time given. These values are the x-component of the wind stress and the y-component of the wind stress.

```
XYC  4   5.0 15.0
XY2  4  3
0.0       0.1   0.1
100.0   0.25  0.2
2000.0 0.25  0.2
```

The wind stress in the x and y directions is determined from the wind speed and direction by the following formulation according to Wu. This calculation will give the wind shear stress in units of grams per centimeter per seconds squared, so there will need to be a conversion applied so that the wind data is input in consistent units with all other data.

$$
\begin{aligned}
T_{wx} &= \rho_a * C * (W^2 \cos\theta) \\
T_{wy} &= \rho_a * C * (W^2 \sin\theta)
\end{aligned}
\tag{1.5}
$$

Where ρ_a = air density, W = wind speed in cm/s at 10 meter height, θ = wind direction measured counter-clockwise from East. C is a wind stress coefficient defined as follows:

$$C = \begin{cases} \dfrac{1.25}{(W/100)^{0.2}} *(0.001), & \text{for } W < 100 \\ \left(\dfrac{W}{100}\right)^{0.5} *(0.001)/2, & \text{for } 100 < W < 1500 \\ 0.0026 & \text{for } W > 1500. \end{cases} \quad (1.6)$$

Friction Controls

Friction controls are used to compute estimated values of the bed friction induced by several types of bed roughness conditions. These represent many of the roughness types that are encountered in typical riverine and estuarine environments.

Bed Shear Stress

This method is used to compute a shear stress coefficient for use in computing the bottom shear stress resulting from a steady or (quasi-steady) current field. The formulation given here is derived from a modified form of the classic logarithmic velocity profile. This modified profile was physically justified by Christensen (1972). The traditional profile yields a velocity of -∞ at the bed, whereas the modified profile forces the velocity to 0 at the bed. To specify bed roughness using Manning's values, use `FR MNG` followed by the string number and roughness value.

```
FR MNG  1   0.006
FR MNG  2   0.008
FR MNG  3   0.010
```

Here we are specifying the friction on strings 1, 2 and 3.

Bed roughness can also be specified as an equivalent roughness height by using the FR ERH card. The value given for this height is an average height of the roughness particles found on the bed over a given area and is specified in the units of the model.

Submerged Aquatic Vegetation

This method is used to compute a shear stress coefficient for use in computing the bottom shear stress resulting from a steady (or quasi-steady) current field over a bed consisting of submerged aquatic vegetation (SAV). To specify bed roughness using submerged aquatic vegetation values, use FR SAV followed by the string number and undeflected stem height. The formulation is from Christensen (1985) with average vegetation characteristics taken from Jacobs and Wang (2003).

Unsubmerged Rigid Vegetation

This method is used to compute a shear stress coefficient for use in computing the bottom shear stress resulting from a steady (or quasi-steady) current through rigid, unsubmerged vegetation. Some examples of this might include flow through mangrove stands, through *phragmites* in coastal wetlands, or through trees and other obstructions in coastal storm surge flooding. To specify bed roughness using unsubmerged rigid vegetation values, use `FR URV` followed by the string number, the roughness height, average stem diameter and average stem density. This roughness height overwrites the one specified on the FR ERH (equivalent roughness height) card. The formulation is taken from Walton and Christensen (1980) and it includes both the form drag induced by flow through the obstructions, and the skin drag induced by flow over the bed.

Solution Controls

Solution development is controlled through the specification of the initial and boundary conditions and the time step parameters.

Dirichlet boundary conditions are specified on a `DB` card, and Natural data on a `NB` card. The following sections discuss how to impose boundary conditions for the various boundary configurations; sidewalls, inflows, and outflows. Details of all of the boundary condition options are given in Table A.9

Friction

Bed

The friction parameter is the constant used to calculate the shear stress at the bed or walls. The formulation for the shear stress on the bed is as follows:

$$\tau = 1/2\, \rho C_f\, ||\mathbf{u}||\, \mathbf{u} \tag{1.7}$$

where C_f is determined by the method used on the `FR` card.

Wall

The sidewall friction is specified with a skin friction coefficient such that the formulation for the shear stress on the wall is defined as:

$$\tau = 1/2\, \rho C_f\, ||\mathbf{u}||\, \mathbf{u} \tag{1.8}$$

Sidewalls

A sidewall can have friction or be frictionless. It can have no flow passing through it, or one can impose a flow in or out of the wall.

Frictionless walls with no through flow are very easy to invoke. You do this by not including this wall in the boundary conditions at all. This means you don't need to create the `EGS` string, which can be time consuming. Also you don't have to include a friction card with the associated roughness.

Walls with either or both friction and through flow require that an edge string (`EGS`) be developed that forms the one-dimensional (1D) elements of this sidewall. Also the the roughness or skin friction must be input. One must also specify a boundary condition for this string via the `NB OVL` or the `NB PRS` line. An example would look like the following table.

```
EGS 101 102 2
EGS 102 103 2
EGS 103 106 2
EGS 106  99 2
FR MNG 2 0.006
NB OVL 2 6
```

The `EGS` lines create four 1D elements along a boundary. The first two numbers of each line are node numbers that form each element. The third number on each line is the string number identifying this group of elements. In this case it is string number 2. The `FR MNG` line states that string number 2 has a skin friction coefficient of 0.006. This line is followed by the boundary condition of `NB OVL` which states that there is flow through this wall, string number 2, and that the specifier for the flow is given in series number 6. These boundary condition cards will contain either a flow or pressure specifier in the second field to signify if they apply to the flow or pressure declarations.

Upstream Boundary

Typical subcritical inflow is to use a node string across the entrance and use a dirichlet boundary condition to specify the velocity components. This boundary condition card, `DB OVL`, is followed by the string number and then the series number for the x-component and the series number for the y-component of velocity. Here's an example.

```
NDS 101 3
NDS 102 3
NDS 103 3
NDS 106 3

DB OVL 3 1 2
```

String 3 is defined as including nodes 101, 102, 103, and 106. Dirichlet boundary conditions are applied for the velocity components of string 3. The x-component is defined by the time series number 1 and the y-component is defined by time series number 2. Other flow options are the natural boundary flow per unit width (`NB OVL`) or the total flow (`NB DIS`) along an edge string.

Supercritical inflow requires that both components of velocity and the depth be defined. This is much as the same example except now one uses the `DB OVH` card to define all three items. Here's the example.

```
NDS 221 4
NDS 232 4
NDS 223 4
NDS 126 4

DB OVH 4 1 2 5
```

Here the node string is number 4. The dirichlet boundary condition for this is for supercritical flow. The x-component of velocity is given by series number 1, the y-component by series number 2, and the depth by series number 5.

Downstream Boundary

This boundary is either going to be subcritical or supercritical. If no flow will be leaving, treat this as a sidewall boundary of some sort.

Subcritical outflow requires the tailwater elevation to be specified. To do this one delineates an `EGS` string of nodes forming 1D elements and then a `NB OTW` card is used to link a particular time series to this string. See the following example.

```
EGS  623  624  5
EGS  624  627  5
EGS  627  629  5
NB OTW  5  6
```

This group of 1D elements composes string number 5. By not specifying a friction card, this string will have a roughness (to tangential flow) of 0.0 . The tailwater elevations for this string (see the `NB OTW` card) are found in time series number 6.

A supercritical downstream boundary requires no boundary condition mathematically. However, the model needs to know that this flow is to be able to pass out of the model. To do this we define this string by a `OB OF` card. Here's the example.

```
EGS  623  624  5
EGS  624  627  5
EGS  627  629  5
OB OF 5
```

This says that string number 5 is an outflow boundary and no boundary condition is specified.

Rain or Evaporation

The 2D planview extent of the model can have evaporation or rain coming into or out of the model. To do this the 2D faces must be linked to a string number. Then we can apply a `NB OVL` boundary condition to specify the flow crossing the boundary surface. Each element has a material number associated with it. Many elements may have the same material number. Usually these material numbers are chosen to represent a particular feature or *material* character. We use these material numbers to identify all the elements that will be grouped into a string. Here's the example.

```
MTS 1 6
MTS 2 6
FR MNG 6 0.006
NB OVL 6 7
```

This example says that material types 1 and 2 (which represent many elements) are grouped together and represented by string number 6. String 6 will have a friction coefficient of 0.006. The inflow/outflow rate per unit area is specified on time series number 7. The third field of the `DB` and `NB` cards specifies the node or edge string, respectively. The fourth and fifth fields of the `DB OVL` card specifies the x- and y-component velocity time series, while the fourth field of the `NB OVL` card specifies the unit flow rate time series.

Time Controls

Evolution of the solution is determined by a group of five cards with the Time Control specifier `TC`. The start time is specified on a `TC TO` card. The `TC IDT` references a time series in the third field that will control the time steps. The final time, at which the run will terminate, is specified on a `TC TF` card. The final time does not have to correspond with the largest value in the time series. The units for the start and end time controls can be individually set using a certain number after the times (0 = seconds; 1 = minutes; 2 = hours; 3 = days; 4 = weeks; 5 = months; 6 = years). Units designation is purely optional, meaning that if you don't include numbers after the starting and final times, they will be read in as seconds.

For adaptive time control, a `TC ADP` card is used. Adaptive time control allows the model to refine the time steps from those specified by the user in the time series. The model may choose finer time steps than those specified but will not adapt the time steps to a larger time step value.

A `TC NDP` card is used for non-adaptive time steps. In this case, the code will only refine the time steps for stability purposes, not for accuracy. Time steps will be reduced if the model fails to converge for the current time step.

```
TC TO 0.0
TC IDT 5
TC TF 2000.0
TC ADP
TC NDP
```

Steady State

The steady state option in ADH is invoked using the `TC STD` card. For steady state, ADH utilizes pseudo-transient continuation. This technique time steps its way to the steady state solution. To reduce computation time, the time step, Δt, is determined using the switched evolution relaxation (SER) strategy where

$$\Delta t^{n+1} = \Delta t^{n} \frac{\| R^{n-1} \|}{\| R^{n} \|} \tag{1.9}$$

R^n refers to the spatial residual for time step n. The initial time step is the first entry on the `TC STD` card. The simulation will perform two time steps with the initial time step size, then the above equation is used to determine each time step thereafter. (It should be noted that the inputted time step is the minimum time step allowed) The simulation will end when one of two flags occur. If the simulation reaches the final time specified on the `TC TF` card the simulation will stop or if the spatial residual becomes less than the residual value specified by the `IP NTL` card. When either of these conditions is satisfied, the simulation will be completed.

This option can be very useful to obtain an initial condition set to be used as the model's hot start file. **The steady state option should NOT be used when the model includes constituent transport, fish passage, vorticity, or any other transport method. This option can ONLY be used for hydrodynamics.**

Sediment Time Steps

When modeling sediment transport, it is often necessary to have a smaller time increment than for hydrodynamic solutions. This is achieved with the `TC SDI` card, which sets the number of sediment time steps to perform for each hydrodynamic time step. If the model does not converge and the time step is cut, the sediment time step will also be cut, but done in such a way that the sediment will reduce its number of steps so that extremely small time increments are not taken. If it is desired to have the sediment take the same time step as specified with the `TC IDT` card, this card should be set to one (1).

Quasi-Unsteady

The quasi-unsteady option in ADH is invoked using the `TC STH` card. This option is intended to be used when running multiple steady state flow conditions in lieu of running a complete hydrograph, especially when attempting to simulate sediment transport. This option will solve the hydrodynamics for the current steady state condition then time step through the sediment calculations. The next hydrodynamic solution is found when the sediment calculations have time stepped to the time associated with the new hydrodynamic step. The time series defining the quasi-unsteady boundary condition is specified on the `TC STH` card along with the maximum number of iterations to allow when attempting to reach steady state and the initial time step for the steady state calculation since this time increment should start small. The sediment calculations will progress according to the time increment given by the `TC IDT` card so the sediment time increment, `TC SDI` card, should be set to one (1). The `IP SST` card is used to specify the tolerance for quasi-unsteady convergence, or the convergence tolerance for the hydrodynamics. This allows the hydro convergence to differ from the tolerance set for the transport convergence. If the SST card is not included, then the tolerance defaults to the value given on the NTL card so that the hydro and sediment converge to the same tolerance.

Output Control

An OC card causes the solution to be printed at startup and at each specified time step. The OC card references a series that controls the output data. These are output as data set files. If using the auto-build feature for the output time series, the OC card is not required.

An END statement is used at the end of the boundary conditions file. The code will read the boundary conditions file through to the END statement. Any information in the boundary conditions file after the END statement will not be read as input to the run. Reference Table A.11 for a full description of the above cards.

```
OC 3
END
```

Another output option is a calculation of flow across strings. For two-dimensional shallow water models (SW2), the calculated flow across edge-strings (EGS) and mid-strings (MDS) is output at every timestep. Strings are selected for flow output with the FLX card, followed by the string number. The strings can be existing edge-strings (EGS) used for boundary condition input or mid-strings (MDS) created specifically for flow output (if the string exists for a boundary condition, do not recreate it, you cannot have duplicate string segments in ADH). The MDS card has the same format as the EGS card. Strings used for flow output must begin and end on a mesh boundary and cannot have common elements shared between them. String segments must also be input in the same direction along the string (direction being from the first node to the second node in the segment), and positive flow will be from right to left when looking in that direction. Values will be output into the *filename*_tlfx file.

Auto Build

The auto-build feature enables an output series to be automatically created during runtime, given certain parameters. This eliminates the need to manually type out each time step desired to be in the solution files. In order to activate this feature, simply use an OS card followed by the series number, number of time segments used to build the output series, and the master time output units. A time segment is composed of four parameters: start time, end time, progression interval, then input units (the output units are determined by the master time output units). The unit specifications are as follows: 0 = seconds, 1 = minutes, 2 = hours, 3 = days, 4 = weeks, 5 = months and 6 = years. The inclusion of the auto-build feature makes the OC card unnecessary, as this feature automatically designates the time series as the output series. An example is shown below:

```
OS  2  4  0
0.0 77.0  5.0  0
1.5  6.0  2.0  1
7.0  9.0  2.0  1
0.5 20.0  6.0  2
```

The auto-build time series above is equivalent to the time series, given in seconds, below.

```
XY1  2  28  0
0.0      0.0
5.0      0.0
10.0     0.0
15.0     0.0
20.0     0.0
25.0     0.0
30.0     0.0
35.0     0.0
40.0     0.0
45.0     0.0
50.0     0.0
55.0     0.0
60.0     0.0
65.0     0.0
70.0     0.0
75.0     0.0
77.0     0.0
90.0     0.0
210.0    0.0
330.0    0.0
360.0    0.0
420.0    0.0
540.0    0.0
1800.0   0.0
23400.0  0.0
45000.0  0.0
66600.0  0.0
72000.0  0.0

OC 2
```

The `OS` card is still a time series card and it is numbered in the same list as the X-Y series. Therefore, if you have an XY1 series with an ID number of 1, if you were to use an OS card next, it will be OS with an ID number of 2.

Hot Start File

The hot start 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 problem. These data are specified in the "hot-start" file. The GMS and SMS provide a simple interface for entering field data. This data is entered into a scatter point data file and interpolated to the problem mesh.

A hot start file is a required part of a model. For ADH, initial water depths must be given, although some can be zero (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 hot start file can be generated with the GMS or SMS. A standard GMS 3-D mesh data set format is used in the hot start 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 hot start 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. 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 TO` card. To create a hot start 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 hot start file.

Wetting and Drying

ADH has the ability to allow areas of the mesh to become dry and then wet again as the flow varies over time. The shallow water equations are gradually turned on and off as the water level rises and falls within a specifed range. The wetting and drying levels are given in the boundary condition file with the "MP DTL" card in units of Length.

```
MP DTL 0.2 0.4
```

The first value given is the lower level and the second the upper level. When the water level is below the lower limit, the node is dry and when above the upper limit, it is wet. However, as the water level falls within the upper and lower limits, a factor is determined to restrict the governing equations. The model does not have to begin completely wet and as time progresses it will wet and dry as necessary. This card is not required, but the limits will default to 0.0 if no other values are given.

Vessel Movement Library

ADH has the ability to simulate the presence of vessels moving within a waterway. This is accomplished by calculating a pressure field, which applies a draft equal to that of the modeled vessel. The vessel characteristics are specified in a boat definition file which will be read by Pre_ADH if the `OP BT` card is given in the boundary condition file. Also, bed shear stresses due to vessel entrainment will be calculated and included as output upon inclusion of the `OP BTS` card. Use of this card requires inclusion of an additional card in the boat definition file (`PROP`).The Vessel Movement Library is discussed further in Appendix C. **NOTE: There can be no blank lines in the boat definition file.**

Stationary Lid in the Flow

If a vessel is moving in the waterway then the ?Boat? library (Vessel Movement Library) can move this pressure throughout the domain to represent the long-wave impacts on the waterway. If, however, a pressure field is stationary then it shouldn?t be necessary to define a boat path and speed. For this case and the case in which a lid is prescribed in the flow we have developed another approach.

This method is implemented by selecting all the nodes that are to comprise the lid or pressure field and assigning them to a node string. This node string will then be assigned the lid elevation with a `DB LDE` card, the depth of the water with a `DB LDH` card, or the pressure (in terms of draft) that is desired with a `DB LID` card. None of these parameters affect the friction that is applied. Also, since the depth or elevation is enforced via a penalty it will not be exact.

Sample Problem

This chapter develops setting up a problem using ADH by demonstrating the development of the boundary condion file for a simple rectangular flume, Fig. 2.1, composed of triangular elements. The flume is 1000 m long and 50 m wide. The model slopes downward in the positive x-direction with a slope of 0.0009. The upstream (left end) bed elevation is 0.0 m. The downstream (right end) bed elevation is -0.9 m. The problem solved is for flow imposed on the upstream face with a velocity of 1.0 meter per second in the positive x direction. Flow exits directly downstream. The sidewalls have an imposed drag. The downstream tailwater elevation is imposed at an elevation of 0.1 m. Just for the sake of an example the mesh is composed of two material types. The right half of the grid is material 2 and the left half is material 1 (see Fig. 2.2).

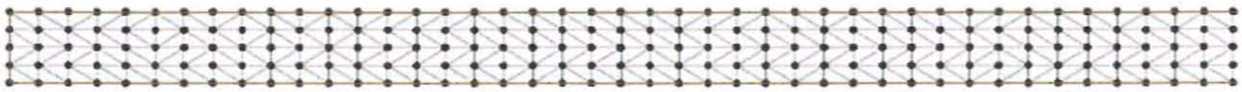

Figure 2.1: Rectangular Flume Mesh.

Figure 2.2: Rectangular Flume Materials.

Geometry File

A portion of the geometry file is shown here. It is composed of triangular elements denoted by the `E3T` card. The term "MESH2D" starts the file. Each element is "E3T Element-# 3-Node-#'s Material ID". This particular example has 392 triangular elements. This list is followed by the list of nodal coordinates. These are "ND Node-# X, Y, and Z coordinates".

```
MESH2D
E3T 1 2 1 6 1
E3T 2 7 2 6 1
E3T 3 8 2 7 1
E3T 4 8 3 2 1
E3T 5 4 3 8 1
E3T 6 9 4 8 1
E3T 7 5 4 9 1
E3T 8 10 5 9 1
E3T 9 7 6 11 1
E3T 10 12 7 11 1
. . . . . .
. . . . . .
. . . . . .
E3T 381 239 238 243 2
E3T 382 244 239 243 2
E3T 383 240 239 244 2
E3T 384 245 240 244 2
E3T 385 242 241 246 2
E3T 386 247 242 246 2
E3T 387 248 242 247 2
E3T 388 248 243 242 2
E3T 389 244 243 248 2
E3T 390 249 244 248 2
E3T 391 245 244 249 2
E3T 392 250 245 249 2
ND 1 0.00000000e+00 0.00000000e+00 0.00000000e+00
ND 2 0.00000000e+00 1.25000000e+01 0.00000000e+00
ND 3 0.00000000e+00 2.50000000e+01 0.00000000e+00
ND 4 0.00000000e+00 3.75000000e+01 0.00000000e+00
ND 5 0.00000000e+00 5.00000000e+01 0.00000000e+00
ND 6 2.04081633e+01 0.00000000e+00 -1.83673469e-02
ND 7 2.04081633e+01 1.25000000e+01 -1.83673469e-02
. . . . .
. . . . .
. . . . .
ND 241 9.79591837e+02 0.00000000e+00 -8.81632653e-01
ND 242 9.79591837e+02 1.25000000e+01 -8.81632653e-01
ND 243 9.79591837e+02 2.50000000e+01 -8.816326531-01
ND 244 9.79591837e+02 3.75000000e+01 -8.81632653e-01
ND 245 9.79591837e+02 5.00000000e+01 -8.81632653e-01
ND 246 1.00000000e+03 0.00000000e+00 -9.00000000e-01
ND 247 1.00000000e+03 1.25000000e+01 -9.00000000e-01
ND 248 1.00000000e+03 2.50000000e+01 -9.00000000e-01
ND 249 1.00000000e+03 3.75000000e+01 -9.00000000e-01
ND 250 1.00000000e+03 5.00000000e+01 -9.00000000e-01
```

Hotstart File

A hotstart file is given below. The first part is the initial estimate of depth. In this case they are all set to 1.0 . If either depths and/or velocities are not in the hotstart file their values are assumed to be zero. These hotstart files can easily be developed from output from previous runs by simply copying the timestep that you want to use from the output file. The "NAME" row then is modified to match the hotstart designation. This is the depth part of the hotstart file, as indicated by the name "ioh".

```
DATASET
OBJTYPE "mesh2d"
BEGSCL
ND 250
NC 392
NAME ioh
TS 0 0.00000000e+00
1.0
1.0
1.0
1.0
1.0
1.0
1.0
1.0
1.0
1.0
1.0
.
.
.
1.0
1.0
1.0
ENDDS
```

The velocity portion ("iov") of the hotstart file then is:

```
DATASET
OBJTYPE "mesh2d"
BEGVEC
ND 250
NC 392
NAME iov
TS 0 0.00000000e+00
1.0 0.0 0.0
1.0 0.0 0.0
1.0 0.0 0.0
1.0 0.0 0.0
1.0 0.0 0.0
1.0 0.0 0.0
1.0 0.0 0.0
1.0 0.0 0.0
1.0 0.0 0.0
1.0 0.0 0.0
1.0 0.0 0.0
.   .   .
.   .   .
.   .   .
1.0 0.0 0.0
1.0 0.0 0.0
1.0 0.0 0.0
ENDDS
```

Boundary Condition File

Next is shown the boundary condition file. The first portion of the file contains the "OP", "IP", and "MP" cards. The "OP" cards designate that this is a two-dimensional Shallow Water run, that memory is added in blocks of 4000, no transport constituents, there are to be 17 blocks per processor, and the preconditioner uses the two-level additive Schwarz approach. The "IP" cards indicate that the maximum number of linear iterations allowed are 200, the maximum non-linear iterations is 8, and the tolerance on the non-linear iterations is 1.0E-5. The "MP" cards designate the other properties of the fluid for this particular problem.

```
OP SW2
OP INC 4000
OP TRN 0
OP PRE 2
OP BLK 17

IP IT 200
IP NIT 8
IP NTL 1.0E-5

MP ML 1 0
MP COR 1 0
MP SRT 1 10
MP EVS 1 0.010 0.010 0.010
MP ML 2 0
MP COR 2 0
MP SRT 2 10
MP EVS 2 0.010 0.010 0.010
MP MUC 1.0
MP MU 0.0
MP G 9.80
MP RHO 1000.0
```

The "strings" are designated by "NDS" for node strings, "EGS" for edge strings, and "MTS" for material strings. Material 1 and 2 make up strings 1 and 6, respectively. Strings 2, 3, and 4 are edges that compose the upstream, sidewalls, and downstream edges of the domain, respectively. Note that both the left and right sidewalls have been designated as the same string. They can be separate strings. String 5 is a list of the upstream boundary nodes.

```
!UPSTREAM INFLOW
NDS 1 5
NDS 2 5
NDS 3 5
NDS 4 5
NDS 5 5

!UPSTREAM EDGE
EGS 1 2 2
EGS 2 3 2
EGS 3 4 2
EGS 4 5 2

!RIGHT SIDEWALL
EGS 1 6 3
EGS 6 11 3
EGS 11 16 3
EGS 16 21 3
EGS 21 26 3
EGS 26 31 3
EGS 31 36 3
EGS 36 41 3
EGS 41 47 3
EGS 47 51 3
EGS 51 56 3
EGS 56 61 3
EGS 61 66 3
EGS 66 71 3
EGS 71 76 3
EGS 76 81 3
EGS 81 86 3
EGS 86 92 3
EGS 92 98 3
EGS 98 101 3
EGS 101 106 3
EGS 106 111 3
EGS 111 116 3
EGS 116 121 3
EGS 121 126 3
EGS 126 131 3
EGS 131 136 3
EGS 136 142 3
EGS 142 146 3
EGS 146 151 3
EGS 151 156 3
EGS 156 161 3
EGS 161 166 3
EGS 166 172 3
```

The XY-series are shown next. These `XY1` cards are followed by the series number, the number of points in the series, and two unit specifiers. The first value in the series is the independent timestep and the second number is the dependent variable describing the condition (depth, velocity, etc.). The series are referenced in the boundary condition cards. Note that series 4 includes the `XY1` card but just before it is the `XYT` card. This `XYT` card simply tells ADH the tolerance for which values will be interpolated in this series. This means that it will only cut the time step to the point that it has an accuracy of 1.0 in interpolating this series.

```
XY1   1  2  0  0
0.0     0.0
2000.0  0.0

XY1   2  2  0  0
0.0     1.0
2000.0  1.0

XY1   3  8  0  0
100.       0.0
200.       0.0
400.       0.0
600.       0.0
800.       0.0
1200.      0.0
1600.      0.0
2000.      0.0

XYT   4 1.0
XY1   4  4  0  0
0.0      20.0
100.0    40.0
200.0    100.0
2000.0   100.0

XY1   5  2  0  0
0.0     0.1
2000.0  0.1
```

The following lines represent the applied boundary conditions. The first 6 lines containing `FR MNG`, all have Manning's □of 0.03. Strings 1,3, and 6 have roughness that is non-zero. Strings 1 and 6 are the 2D elements associated with material types 1 and 2. String 3 is the sidewall of the flume. The next 3 lines are Neumann boundary conditions. These are the `NB OVL` cards. Each is followed by a string number, in this case the 3 strings are 1,3, and 6. The final number represents which series is to be used to get the flux into or out of this string. All of them are associated with series number 1. `XY1` series 1 has 0.0 throughout all time. So there is no flux into or out of each of these strings.

The next boundary condition card `NB OTW 4 5` signifies that string 4 will have a tailwater elevation specified. And that elevation is given on series # 5. If we check that series we see that the elevation specified there has a value of 0.1 for the entire simulation.

The next two cards deal with the upstream boundary condition. The first, given by the `DB OVL 5 2 1` card, says that we intend to enforce a velocity Dirichlet boundary condition on string # 5. These are the upstream nodes. The values of the x and y-components of velocity are given on series 2 and 1, respectively. The next card is `OB OF 2`, which tells ADH to process this string so that flow is allowed to pass through it.

```
FR MNG  1 0.03
FR MNG  2 0.03
FR MNG  3 0.03
FR MNG  4 0.03
FR MNG  6 0.03

NB OVL   1  1
NB OVL   3  1
NB OVL   6  1
NB OTW   4  5
DB OVL   5  2  1
OB OF    2
```

The `OC` card designates series 3 to contain the list of timesteps to be output. Series 3 requests that output be generated at times 100.0, 200.0, 400.0, 600.0, 800.0, 1200.0, ... seconds. `TC T0` and `TC TF` designate the starting and ending times to be 0.0 and 1200.0 respectively. The timestep size is setup by the `TC IDT` card. The timestep size is set by series 4, in this case. Series 4 then indicates that a time step size progresses from 20.0 seconds to 100.0 seconds throughout the run.

```
OC 3

TC T0 0.0
TC IDT 4
TC TF 1200.0
TC NDP

END
```

Running ADH

ADH is run after a setup program called PRE_ADH. PRE_ADH reads the input files and checks that they are somewhat consistent. It also checks the geometry to make sure that the elements are properly formed. The actual command to run PRE_ADH on this sample problem is:

```
pre_adh uniform
```

At this point the program itself can be run. Depending upon the flags set during compilation the program can be run with or without "MPI". If the compiler options do not include "-D_MESSG -D_MPI" then running ADH is:

```
adh uniform
```

This will create files called "uniform_ovl.dat" ,"uniform_dep.dat", and "uniform_err.dat", for velocities, depths, and errors for adaption, respectively. These can be directly imported into GMS or SMS.

If instead the compiler flags are set to run MPI, the run command is as follows:

```
mpirun -np 2 adh uniform
```

In this case, we have requested two processors. The output from this particular example command is identical to the previous: an individual file for depths, velocities, and error values. When including transport quantities, output files will be generated for the concentration of each constituent. If using the sediment transport option, bed displacements will be given along with the concentrations for each sediment.

These results are shown in the next three figures. Fig. 2.3 shows velocity results, Fig. 2.4 shows overland head results, and Fig. 2.5 shows the error used for adapting at time 1200.0 seconds.

Figure 2.3: Velocity Results.

Figure 2.4: Head Results.

Figure 2.5: Error Results.

Modifying the Boundary Conditions

The previous example can be modified easily in the boundary condition (`.bc`) file. The maximum timestep can be increased or decreased by changing the `TC TF` card. Be certain to modify the `XY1` series cards if necessary so that they also reflect the change in run length. The Manning's "n", or roughness, can be manipulated in the `FR MNG` card for each edge or material.

Supercritical Flow

The following change will generate supercritical flow and a hydraulic jump will be observed. To properly pose a supercritical inflow boundary, we need to specify velocity and depth. For subcritical inflow we only need velocity specified. Suppose the Froude number is 2. The corresponding horizontal velocity is 3.333 m/s and the corresponding tailwater height is 0.3 m. Open the uniform.bc file and change the XY1 series 2 card to 3.333 since it corresponds to the x-velocity component at all timesteps. Then add an XY1 series 6 card to equal 0.3 at all timesteps. It will represent the tailwater head. Also replace the DB OVL card with a DB OVH card making series 6 the depth for node string 5. Re-run PRE_ADH and ADH with the modified boundary condition file. The results should show a downward jump as seen in Fig. 2.6 and Fig. 2.7. Since the refinement level (MP ML) is set to zero meaning no adaption is taking place, the error files will not be shown for this example.

Figure 2.6: Velocity Results for Supercritcal Flow.

Figure 2.7: Overland Head Results for Supercritcal Flow.

Rain and Evaporation

As noted previously, rain and evaporation can be modeled with ADH. Since the example mesh is divided into two material types, rain will be applied on material 1 and evaporation on material 2. This can be done easily by modifying the XY1 series cards in the boundary condition file. Using the original .bc file, add an XY1 series 6 card so that it equals 0.0001 at all timesteps. Also add an XY1 series 7 card that is -0.0001 at all timesteps. Keep in mind that the volume raining in or evaporating out cannot greatly exceed the volume of the flume. Now modify the NB OVL card so that string 1 (material 1) uses series 6 and string 6 (material 2) uses series 7. The velocity results of this case at 1200 seconds can be seen in Fig. 2.8 and the overland head results in Fig. 2.9.

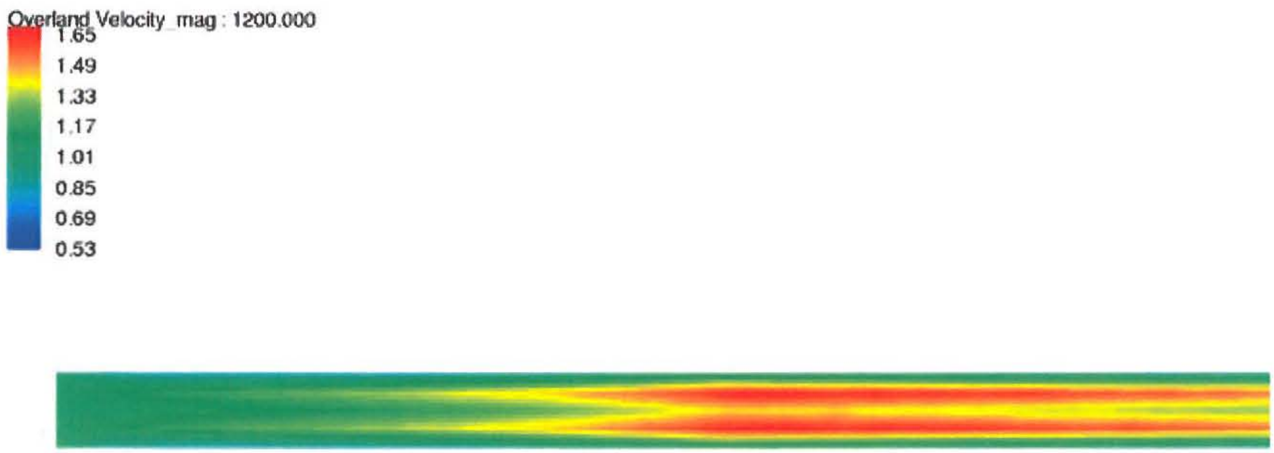

Figure 2.8: Velocity Results for Rain and Evaporation.

Figure 2.9: Overland Head Results for Rain and Evaporation.

This example was divided by material type and the symmetry of the rain and evaporation can be seen. Typically, rain and evaporation are varied over time for long runs. This is done by simply adding more points in the XY1 series card that describes the conditions at different timesteps. Remember that rain is assumed positive and evaporation is assumed negative.

Transport Quantities

Concentration

Transport quantities and concentrations can be modeled in ADH with a few additions to the boundary condition and hotstart files. The TRN card at the beginning of the boundary condition file gives the number of transport items being modeled. Change this value to one (1) for this example. The attributes for the constituent must then be given for each material type. The values necessary for this example with a reference concentration of one are given below. Also, change the flow conditions back to those given initially so that rain, evaporation, or supercritical flow are not occurring.

```
MP DF   1  1  0.02
MP TRT  1  1  0.1

MP DF   2  1  0.02
MP TRT  2  1  0.1

! Reference Constituent: Constituent 1, Concentration
CN CON 1 1 0 0 0
```

The hot start file must also include the initial concentrations at the necessary nodes. To apply an initial concentration cloud having a maximum concentration of one near the left boundary, add the following lines to the hotstart file:

```
DATASET
OBJTYPE "mesh2d"
BEGSCL
ND 250
NC 392
NAME icon 1
TS 0 0.00000000e+00
0
0
0
0
0
0
0
0
0
0
0
0
0.25
0
0
0
0.25
0.5
0.25
0
0.25
0.5
1.0
0.5
0.25
0
0.25
0.5
0.25
0
0
0
0.25
0
.
.
.
0
```

There should be 250 entries so that there is one for each node. The same process would be continued for multiple transport quantities, changing the initial values and the concentration number in the "NAME" field.

Run Pre_ADH and ADH in the same manner as stated previously. The overland velocity, overland head, and error output files will be generated along with a constituent file for each transport quantity. The additional output files will be named in the same form, with the number of the quantity included.

```
uniform_con1.dat
```

Upon viewing the results of the run (Figures 2.10- 2.11), the concentration is initially a tight cloud. As the flow pushes it downstream, the cloud begins to spread out, due in part to numerical approximation. With ADH, however, this error can be minimized by applying the mesh adaption around the concentration.

Figure 2.10: Concentration Results at Time = 0 sec.

Figure 2.11: Concentration Results at Time = 4800 sec.

Mesh Adaption

The unique aspect of ADH is its ability to adapt the mesh in areas where more resolution is needed and then unrefine when necessary. This process is done by normalizing the results so that an error quantity is determined for each element. If this error exceeds the tolerance set by the user, the element is then split. In order to apply adaption, the refinement level for each material must be greater than zero. This value sets a limit on the number of times an element can be divided.

To invoke adaption around the concentration cloud in this example, change the level of refinement from 0 to 4 for each material. The lines should now look like this:

```
MP ML 1 4
...
MP ML 2 4
```

The next step is to set the refinement tolerance for the adaption. This is done on the `TRT` card found in the section describing the concentration. Set this value to 0.1 for this example. In order to ensure that the only adaption occurring is around the concentration, the shallow water refinement tolerance, the `SRT` card, can be set to a large value.

```
MP TRT 1 1 0.1
...
MP TRT 2 1 0.1
```

```
MP SRT 1 500
...
MP SRT 2 500
```

Run Pre_ADH and ADH again with the new boundary condition file. The screen output will show the number of original nodes and the number of new nodes when adaption takes place. If running on multiple processors, the problem will be repartitioned if one processor has 10% more nodes than another. This step allows for better time management by not making one processor work harder than another. The solutions are found at all nodes and then interpolated back to the original nodes, so the user never sees the adapted mesh and the visual results in SMS or GMS may not appear to be improved. However, the results are more accurate due to the smaller elements in the problem areas and the increased number of nodes at which the equations are solved. An example of the adaption taking place in a similar problem can be seen in Figure 2.12.

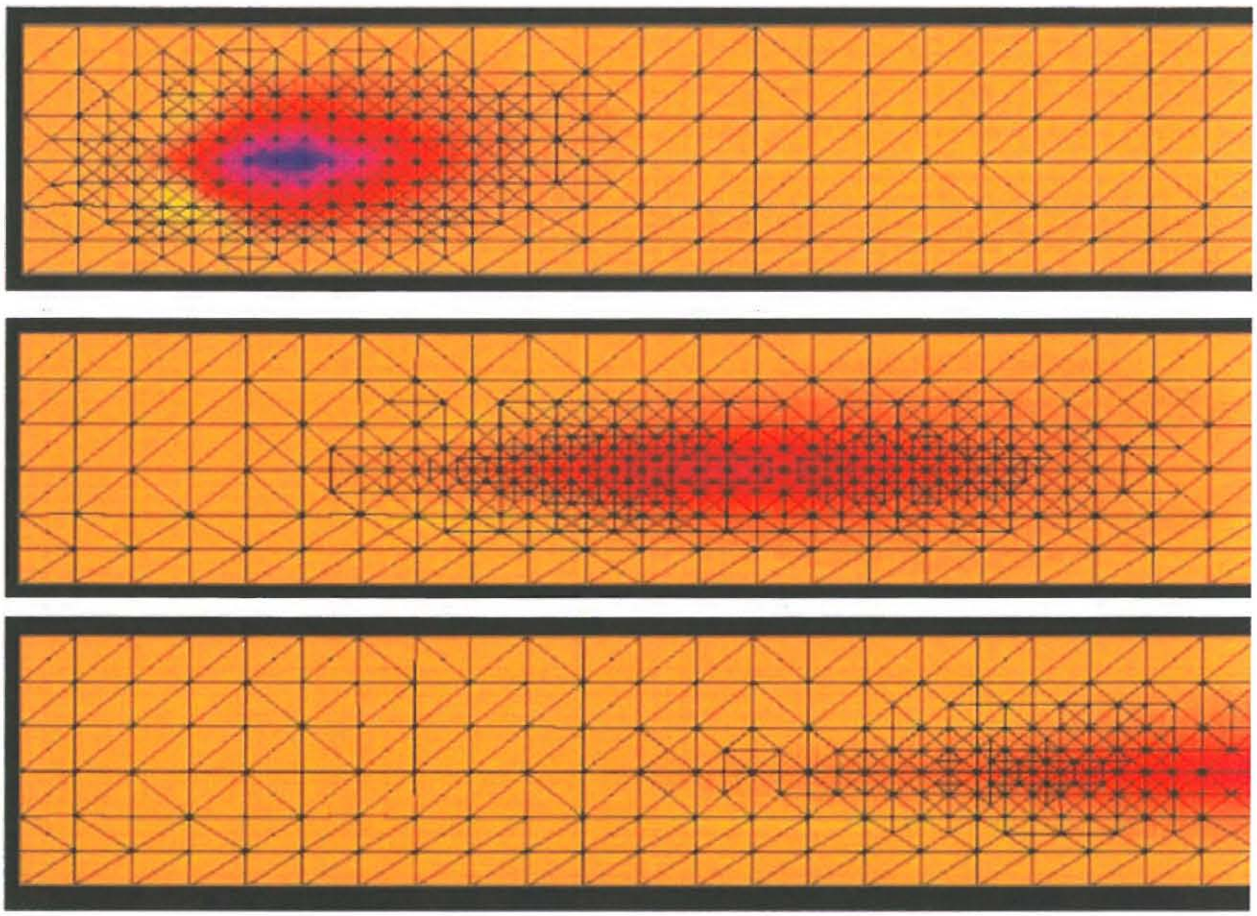

Figure 2.12: Example of Mesh Adapting Around a Concentration Cloud.`

Two-Dimensional Shallow Water Equations,

Finite Element Formulation

The 2D shallow water equations can be written in conservative form as

$$\frac{\partial \mathbf{Q}}{\partial t} + \frac{\partial \mathbf{F_x}}{\partial x} + \frac{\partial \mathbf{F_y}}{\partial y} + \mathbf{H} = 0 \tag{3.1}$$

where

$$\mathbf{Q} = \begin{Bmatrix} h \\ uh \\ vh \end{Bmatrix} \tag{3.2}$$

$$\mathbf{F_x} = \begin{Bmatrix} uh \\ u^2h + \frac{1}{2}gh^2 - \frac{h}{\rho}\sigma_{xx} \\ uvh - \frac{h}{\rho}\sigma_{yx} \end{Bmatrix} \tag{3.3}$$

$$\mathbf{F_y} = \begin{Bmatrix} vh \\ uvh - \frac{h}{\rho}\sigma_{xy} \\ v^2h + \frac{1}{2}gh^2 - \frac{h}{\rho}\sigma_{yy} \end{Bmatrix} \tag{3.4}$$

and

$$\mathbf{H} = \begin{Bmatrix} 0 \\ gh\frac{\partial Z_0}{\partial x} + ghS_x \\ gh\frac{\partial Z_0}{\partial y} + ghS_y \end{Bmatrix} \tag{3.5}$$

σ represents the Reynolds' stresses due to the turbulence plus the molecular stresses.

$$\sigma_{xx} = 2\rho\nu\frac{\partial u}{\partial x} \tag{3.6}$$

$$\sigma_{xy} = \sigma_{yx} = \rho\nu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) \tag{3.7}$$

$$\sigma_{yy} = 2\rho\nu\frac{\partial v}{\partial y} \tag{3.8}$$

S represents the friction slope, which can be calculated in two forms:

$$S_x = \left\{ \begin{array}{c} u\frac{n^2\sqrt{u^2+v^2}}{C_0^2\sqrt[3]{h}} \\ \frac{C_f}{gh}u\sqrt{u^2+v^2} \end{array} \right\} \tag{3.9}$$

and

$$S_y = \left\{ \begin{array}{c} v\frac{n^2\sqrt{u^2+v^2}}{C_0^2\sqrt[3]{h}} \\ \frac{C_f}{gh}v\sqrt{u^2+v^2} \end{array} \right\} \tag{3.10}$$

where C_f = coefficient of friction; n = Manning's roughness coefficient; and C_0 = a dimensional conversion coefficient (1 for SI units, 1.486 for U.S. customary units).

For the Finite Element formulation, first use the linear Lagrange basis functions that are C^0, i.e., the functions are continuous, so that:

$$\mathbf{Q}(x,y,t) \approx \sum_j \Phi_j(x,y)\mathbf{Q}_j(t).$$

The test function is similar to that of Berger and Stockstill (1995). In this method, consider the shallow water equations in non-conservative form:

$$\mathbf{M}\frac{\partial q}{\partial t} + \mathbf{A}\frac{\partial q}{\partial x} + \mathbf{B}\frac{\partial q}{\partial y} + \mathbf{h} = 0 \tag{3.11}$$

$$\mathbf{q} = \left\{ \begin{array}{c} h \\ u \\ v \end{array} \right\} \tag{3.12}$$

$$\mathbf{M} = \left\{ \begin{array}{ccc} 1 & 0 & 0 \\ 0 & h & 0 \\ 0 & 0 & h \end{array} \right\} \tag{3.13}$$

$$\mathbf{A} = \left\{ \begin{array}{ccc} u & h & 0 \\ c^2 & uh & 0 \\ 0 & 0 & uh \end{array} \right\} \tag{3.14}$$

$$\mathbf{B} = \left\{ \begin{array}{ccc} v & 0 & h \\ 0 & vh & 0 \\ c^2 & 0 & vh \end{array} \right\} \tag{3.15}$$

and

$$C = (gh)^{\frac{1}{2}} \tag{3.16}$$

The shallow-water equations are the basis from which the following test function is developed.

$$\psi_i^T = \phi_i^T + \alpha(\Delta x \frac{\partial \phi_i^T}{\partial x} \mathbf{M P^{-1} \Lambda_x P M^{-1}} + \Delta y \frac{\partial \phi_i^T}{\partial y} \mathbf{M R^{-1} \Lambda_y R M^{-1}}) \tag{3.17}$$

or

$$\psi_i^T = \phi_i^T + \varphi_i^T \tag{3.18}$$

where,

$\alpha$is a coefficient between 0 and 0.5,

$$\Delta x = 2[(\frac{\partial x}{\partial \xi})^2 + (\frac{\partial x}{\partial \eta})^2]^{\frac{1}{2}}, \tag{3.19}$$

$$\Delta y = 2[(\frac{\partial y}{\partial \xi})^2 + (\frac{\partial y}{\partial \eta})^2]^{\frac{1}{2}}, \tag{3.20}$$

(ξ, $\eta$are the local variables that have values between 0 and 1)

$$\mathbf{P} = \left\{ \begin{array}{ccc} 0 & 0 & 1 \\ 1 & -\frac{c}{g} & 0 \\ 1 & \frac{c}{g} & 0 \end{array} \right\}, \tag{3.21}$$

$$\mathbf{R} = \left\{ \begin{array}{ccc} 0 & 1 & 0 \\ 1 & 0 & -\frac{c}{g} \\ 1 & 0 & \frac{c}{g} \end{array} \right\}, \tag{3.22}$$

$$\mathbf{\Lambda_x} = \tag{3.23}$$

$$\frac{1}{a}\left\{\begin{matrix} u & 0 & 0 \\ 0 & u-c & 0 \\ 0 & 0 & u+c \end{matrix}\right\},$$

$$\mathbf{A}_y = \frac{1}{a}\left\{\begin{matrix} v & 0 & 0 \\ 0 & v-c & 0 \\ 0 & 0 & v+c \end{matrix}\right\}, \qquad (3.24)$$

and

$$a = (u^2+v^2+c^2)^{\frac{1}{2}}. \qquad (3.25)$$

The weak form finite element approximation is then

$$\sum_e \left[\int_{\Omega_e} (\psi_i^T \frac{\partial \mathbf{Q}}{\partial t} - \frac{\partial \phi_i^T}{\partial x}\mathbf{F_x} - \frac{\partial \phi_i^T}{\partial y}\mathbf{F_y} + \varphi_i^T(\frac{\partial \mathbf{F_x}}{\partial x} + \frac{\partial \mathbf{F_y}}{\partial y}) + \psi_i^T \mathbf{H})\, d\Omega_e + \oint_{\Gamma_e} \phi_i^T (\mathbf{F_x} n_x + \mathbf{F_y} n$$

where the subscript e identifies a particular element; and $(n_x, n_y) = \mathbf{n}$ is the unit vector outward and normal to the boundary Γ_e.

Constructing Model Files

This appendix gives an overview of the three files needed to run ADH. Details and explanations of each of the input parameters is given in the body of the report and in the sample problems. Three files are required for running ADH: the 2-D Mesh File, the Boundary Conditions File, and the Hot Start File. The generation of each of these files and the components of the files are described in the following sections.

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.

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. The following tables contain all of the possible boundary condition file cards and a description of their input.

Table A.1: Operation Parameter Cards

2D Shallow Water Problems			
Field	Type	Value	Description
1	char	OP	Card type.
2	char	SW2	Specifies 2-D Shallow Water Problem
Incremental Memory			
Field	Type	Value	Description
1	char	OP	Card type.
2	char	INC	Parameter.
3	int	> 0	Incremental Memory Allocation
Transport Equations			
Field	Type	Value	Description
1	char	OP	Card type.
2	char	TRN	Parameter.
3	int	≥ 0	Total number of transported materials.
Block Specification for Pre-conditioner			
Field	Type	Value	Description
1	char	OP	Card type.
2	char	BLK	Parameter.
3	int	> 0	Number of blocks per processor, used to pe pre-conditioning
Pre-conditioner Selection			
Field	Type	Value	Description
1	char	OP	Card type.
2	char	PRE	Parameter.
3	int	$3 \geq \# \geq 0$	Prec_value
			0 No pre-conditioning
			1 one level Additive Schwarz pre-condition

			2 two level Additive Schwarz pre-condition
			3 two level Hybrid pre-conditioning
Vessel Movement Library Inclusion			
Field	Type	Value	Description
1	char	OP	Card type.
2	char	BT	Parameter.
Second Order Temporal Term			
Field	Type	Value	Description
1	char	OP	Card type.
2	char	TEM	Parameter.
3	real	1 ≥# ≥0	Coefficient for the second order temporal sc
Petrov-Galerkin Coefficient			
Field	Type	Value	Description
1	char	OP	Card type.
2	char	TPG	Parameter.
3	real	0.5 ≥# ≥0	Coefficient for the Petrov-Galerkin equatioi
Vessel Stress Effects Inclusion			
Field	Type	Value	Description
1	char	OP	Card type.
2	char	BTS	Parameter.

Table A.2: Iteration Parameter Cards

Non-linear Iterations (option 1)			
Field	Type	Value	Description
1	char	IP	Card type.
2	char	NIT	Parameter
			Number of non-linear itera

3	int	≥1	per time step, if at NIT the tolerance is not satisfied A[illegible] will reduce the time step a[illegible] recalculate.

Non-linear Iterations (option 2)			
Field	Type	Value	Description
1	char	IP	Card type.
2	char	FNI	Parameter.
3	int	≥1	Number of non-linear itera[illegible] per time step, even if at FN[illegible] tolerance is not satisfied th[illegible] ADH will accept the soluti[illegible] proceed to the next time st[illegible]

Non-linear Tolerance			
Field	Type	Value	Description
1	char	IP	Card type.
2	char	NTL	Parameter.
3	real	≥0	Tolerance for Non-Linear Equations

Increment Tolerance			
Field	Type	Value	Description
1	char	IP	Card type.
2	char	ITL	Parameter.
3	real	≥0	Tolerance for maximum ch[illegible] in the velocity and depth solutions.

Linear Iterations (option 1)			
Field	Type	Value	Description
1	char	IP	Card type.
2	char	MIT	Parameter.
			Maximum number of linea[illegible] iterations per non-linear ite[illegible]

3	int	≥1	by the interative solver. If t internal linear tolerance (0. * NTL) is not met at MIT t solution stops.

Linear Iterations (option 2)			
Field	Type	Value	Description
1	char	IP	Card type.
2	char	FLI	Parameter.
3	int	≥1	Maximum number of linea iterations by the iterative s If the internal tolerance (0. NTL) is not met at FLI the solution will proceed to the nonlinear iteration.

Runge-Kutta Tolerance for Reactive Constituents			
Field	Type	Value	Description
1	char	IP	Card type.
2	char	RTL	Parameter.

Quasi-Unsteady Tolerance			
Field	Type	Value	Description
1	char	IP	Card type.
2	char	SST	Parameter.
3	int	>0	Tolerance for quasi-unstead hydrodynamic convergence

Table A.3: Constituent Cards

Any Constituent			
Field	Type	Value	Description
1	char	CN	Card type.
2	char	CON	Parameter.
3	int	≥1	The constituent ID number

4	real	> 0	Characteristic concentratio
5	real	> 0	Placeholder.
6	real	> 0	Placeholder.
7	real	> 0	Placeholder.
Clay and/or Silt Sediment			
Field	Type	Value	Description
1	char	CN	Card type.
2	char	CLA	Parameter.
3	int	≥1	The constituent ID number
4	real	> 0	Characteristic concentratio
5	real	> 0	Grain diameter.
6	real	> 1	Specific gravity.
7	real	> 0	Bulk density.
8	real	> 0	Critical shear for erosion.
9	real	> 0	Erosion rate constant.
10	real	> 0	Critical shear for depositio
11	real	> 0	Settling velocity.
Sand Sediment			
Field	Type	Value	Description
1	char	CN	Card type.
2	char	SND	Parameter.
3	int	≥1	The constituent ID number
4	real	> 0	Characteristic concentratio
5	real	> 0	Grain diameter.
6	real	> 1	Specific gravity.
7	real	> 0	Grain porosity.
Vorticity			
Field	Type	Value	Description
1	char	CN	Card type.

2	char	VOR	Parameter.
3	int	≥1	The constituent ID number
4	real	> 0	Normalization factor.
5	real	≥0	A_s term, default is 0.0 whi A_s = 5.0.
6	real	≥0	D_s term, default is 0.0 whi D_s = 0.5.

Salinity Transport (Baroclinic)			
Field	Type	Value	Description
1	char	CN	Card type.
2	char	SAL	Parameter.
3	int	≥1	The constituent ID number.
4	real	> 0	Reference concentr
5	real	≥0	Placeholder.
6	real	≥0	Placeholder.
7	real	≥0	Placeholder.

Temperature Transport (Baroclinic)			
Field	Type	Value	Description
1	char	CN	Card type.
2	char	TMP	Parameter.
3	int	≥1	The constituent ID number.
4	real	> 0	Reference concentr
5	real	≥0	Placeholder.
6	real	≥0	Placeholder.
7	real	≥0	Placeholder.

Table A.4: Material Property Cards

Eddy Viscosity			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	EVS	Parameter.
3	int	≥ 1	Material type ID number.
4	real	> 0	E_{xx}
5	real	> 0	E_{yy}
6	real	> 0	E_{xy}

Estimated Eddy Viscosity			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	EEV	Parameter.
3	int	≥ 1	Material type ID number.
4	real	$0.1 \leq$ coef ≤ 1	Coefficient.

Viscosity			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	MU	Parameter.
3	real	≥ 0	Uniform background viscosity (kinematic molecular viscosity

Coriolis			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	COR	Parameter.
3	int	≥ 1	Material type ID number.
4	real	-90 $\geq$# $\geq$90	Latitude.

Gravitational Acceleration			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	G	Parameter.
3	real	≥0	Value of gravity induced acceleration.

Density			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	RHO	Parameter.
3	real	≥0	Density.

Manning's Units Constant			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	MUC	Parameter.
3	real	≥0.0	Coefficient. (1.486 for English units, 1.0 f standard)

Wetting/Drying Limits			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	DTL	Parameter.
3	real	≥0.0	Depth below which the model considered dry, default is 0.0.
4	real	≥0.0	Depth above which the model completely wet, default is 0.0.

Bed Layers			
Field	Type	Value	Description
1	char	MP	Card type.

2	char	NBL	Parameter.
3	int	≥0	Number of bed layers for sedi transport (layer number begin the bottom-most layer).

Sediment Applied to Selected Nodes			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	SBN	Parameter.
3	int	> 0	Bed layer ID number.
4	int	> 0	The node number from which start.
5	int	> 0	The node number at which to
6	real	≥0.0	The bed layer thickness.
7	real	≥0.0	The distribution for the first sediment.
8	real	≥0.0	The distribution for the secon sediment.
#	real	≥0.0	The distribution for the final sediment.

Sediment Applied to All Nodes			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	SBA	Parameter.
3	int	> 0	Bed layer ID number.
4	real	≥0.0	The bed layer thickness.
5	real	≥0.0	The distribution for the first sediment.
6	real	≥0.0	The distribution for the secon sediment.
#	real	≥0.0	The distribution for the final sediment.

Sediment Applied by Material			

Field	Type	Value	Description
1	char	MP	Card type.
2	char	SBM	Parameter.
3	int	> 0	Bed layer ID number.
4	int	> 0	Material type ID number.
5	real	≥0.0	The bed layer thickness.
6	real	≥0.0	The distribution for the first sediment.
7	real	≥0.0	The distribution for the seconc sediment.
#	real	≥0.0	The distribution for the final sediment.
Cohesive Sediment Applied to Selected Nodes			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	CBN	Parameter.
3	int	> 0	Bed layer ID number.
4	int	> 0	The node number from which start.
5	int	> 0	The node number at which to
6	real	≥0.0	The bulk density.
7	real	≥0.0	The critical shear stress for er
8	real	≥0.0	The erosion rate constant.
9	real	≥0.0	The erosion rate exponent.
Cohesive Sediment Applied by Material			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	CBM	Parameter.
3	int	> 0	Bed layer ID number.
4	int	> 0	Material type ID number.
5	real	≥0.0	The bulk density.

6	real	≥ 0.0	The critical shear stress for er(
7	real	≥ 0.0	The erosion rate constant.
8	real	≥ 0.0	The erosion rate exponent.
Cohesive Sediment Applied by Layer			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	CBA	Parameter.
3	int	> 0	Bed layer ID number.
4	real	≥ 0.0	The bulk density.
5	real	≥ 0.0	The critical shear stress for er(
6	real	≥ 0.0	The erosion rate constant.
7	real	≥ 0.0	The erosion rate exponent.

Table A.5: Constituent Properties			
Molecular Diffusion Rate			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	DF	Parameter.
3	int	≥ 1	Material type ID number.
4	int	> 0	Constituent ID number.
5	real	≥ 0.0	Molecular Diffusion Rate.

Table A.6: Material Meshing Control Cards

Refinement Levels			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	ML	Parameter.
3	int	≥ 1	Material type ID number.

4	int	≥0	Maximum number of refinement levels.
Flow Refinement Tolerances			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	SRT	Parameter.
3	int	≥1	Material type ID number.
4	real	≥0	Error tolerance for the refinemer terms.
Transport Constituent Refinement Tolerance			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	TRT	Parameter.
3	int	≥1	Material type ID number.
4	int	≥1	Constituent ID number.
5	real	≥0	Error tolerance for refining elem

Table A.7: String Structures

Node Strings			
Field	Type	Value	Description
1	char	NDS	Card type.
2	int	≥1	ID number of a node with a Dirichlet conc
3	int	≥1	String ID number.
Edge Strings			
Field	Type	Value	Description
1	char	EGS	Card type.
2	int	≥1	ID number of the first node of an edge ele
3	int	≥1	ID number of the second node of an edge element.

4	int	≥1	String ID number.
Mid Strings			
Field	Type	Value	Description
1	char	MDS	Card type.
2	int	≥1	ID number of the first node of an edge ele
3	int	≥1	ID number of the second node of an edge element.
4	int	≥1	String ID number.
Face Strings			
Field	Type	Value	Description
1	char	FCS	Card type.
2	int	≥1	Element number.
3	int	≥1	ID number of a face.
4	int	≥1	String ID number.
Material Strings			
Field	Type	Value	Description
1	char	MTS	Card type.
2	int	≥1	Material type ID number.
3	int	≥1	String ID number.

Table A.8: Series Structures

X-Y Series			
Field	Type	Value	Description
1	char	XY1	Card type.
2	int	> 0	ID number of the series.
3	int	> 0	Number of points in the sei
			Input units. (0 = seconds;

4	int	> 0	1 = minutes; 2 = hours; 3 = days; 4 = weeks; 5 = months; 6 = years)
5	int	> 0	Output units. (0 = seconds; 1 = minutes; 2 = hours; 3 = days; 4 = weeks; 5 = months; 6 = years)

X-Y-Y Series			
Field	Type	Value	Description
1	char	XY2	Card type.
2	int	> 0	ID number of the series.
3	int	> 0	Number of points in the ser

Curve Fit Tolerance			
Field	Type	Value	Description
1	char	XYT	Card type.
2	int	≥1	ID number of the series to it is associated.
3	int	≥1	The tolerance to use when adapting.

Wind Station Coordinate			
Field	Type	Value	Description
1	char	XYC	Card type.
2	int	≥1	ID number of the series to it is associated.
3	real	#	X coordinate of the wind st
4	real	#	Y coordinate of the wind st

Table A.9: Initial and Boundary Conditions

Dirichlet - Velocity			
Field	Type	Value	Description
1	char	DB	Card type.
2	char	OVL	Parameter.
3	int	≥1	String ID number (node).
4	int	≥1	Series ID number for x-velocity component.
5	int	≥1	Series ID number for y-velocity component.

Dirichlet - Velocity and Depth			
Field	Type	Value	Description
1	char	DB	Card type.
2	char	OVH	Parameter.
3	int	≥1	String ID number (node).
4	int	≥1	Series ID number for x-velocity component.
5	int	≥1	Series ID number for y-velocity component.
6	int	≥1	Series ID number for the depth.

Dirichlet - Transport			
Field	Type	Value	Description
1	char	DB	Card type.
2	char	TRN	Parameter.
3	int	≥1	String ID number (node).
4	int	≥1	Constituent ID number.
5	int	≥1	Series ID number that contains the constituent concentration in parts per million.

Dirichlet - Stationary Lid Elevation			
Field	Type	Value	Description
1	char	DB	Card type.
2	char	LDE	Parameter.
3	int	≥1	String ID number (node).
4	int	≥1	Series ID number that contains the elevation to be implemented.

Dirichlet - Depth of Water under Stationary Lid			
Field	Type	Value	Description
1	char	DB	Card type.
2	char	LDH	Parameter.
3	int	≥1	String ID number (node).
4	int	≥1	Series ID number that contains the deptl

Dirichlet - Floating Stationary Object			
Field	Type	Value	Description
1	char	DB	Card type.
2	char	LID	Parameter.
3	int	≥1	String ID number (node).
4	int	≥1	Series ID number that contains the draft the lid.

Natural Boundary Condition - Total Discharge			
Field	Type	Value	Description
1	char	NB	Card type.
2	char	DIS	Parameter.
3	int	≥1	String ID number (edge).
4	int	≥1	Series ID number containing the total discharge across the string. Positive in.

Natural Boundary Condition - Flow			
Field	Type	Value	Description
1	char	NB	Card type.
2	char	OVL	Parameter.
3	int	≥1	String ID number (edge or face).
4	int	≥1	Series ID number containing the flow d For face strings the series values represe the flow per unit area (positive in). For edge strings the series values represent t flow per unit length (positive in).

Natural Boundary Condition - Water Surface Elevation			
Field	Type	Value	Description
1	char	NB	Card type.
2	char	OTW	Parameter.
3	int	≥1	String ID number (edge).
4	int	≥1	Series ID number that contains the time series of the water surface elevation.

Natural Boundary Condition - Transport			
Field	Type	Value	Description
1	char	NB	Card type.
2	char	TRN	Parameter.
3	int	≥1	String ID number (edge).
4	int	≥1	Constituent ID number.
5	int	≥1	Series ID number that contains the constituent concentration in parts per million.

Equilibrium Transport Boundary Condition			
Field	Type	Value	Description

1	char	EQ	Card type.
2	char	TRN	Parameter.
3	int	≥1	String ID number (node).
4	int	≥1	Constituent ID number.
5	real	≥0	Initial concentration in parts per million
Outflow Boundary			
Field	Type	Value	Description
1	char	OB	Card type.
2	char	OF	Parameter.
3	int	≥1	String ID number (edge).
Dirichlet Free-Surface Pressure			
Field	Type	Value	Description
1	char	DB	Card type.
2	char	FRS	Parameter.
3	int	≥1	String ID number, identifies the free surface nodes.
4	int	≥1	Series ID number containing the pressu
Dirichlet Displacement Condition			
Field	Type	Value	Description
1	char	DB	Card type.
2	char	MVS	Parameter.
3	int	≥1	String ID number.
4	int	≥1	Series ID number containing the displacement history.

Table A.10: Friction Control Cards

Manning's N Roughness			
Field	Type	Value	Description
1	char	FR	Card type.
2	char	MNG	Parameter.
3	int	> 0	String ID number.
4	real	≥ 0.0	Roughness.

Equivalent Roughness Height			
Field	Type	Value	Description
1	char	FR	Card type.
2	char	ERH	Parameter.
3	int	> 0	String ID number.
4	real	≥ 0.0	Roughness height.

Submerged Aquatic Vegetation			
Field	Type	Value	Description
1	char	FR	Card type.
2	char	SAV	Parameter.
3	int	> 0	String ID number.
4	real	≥ 0.0	Undeflected stem height.

Un-Submerged Rigid Vegetation			
1	char	FR	Card type.
2	char	URV	Parameter.
3	int	> 0	String ID number.
4	real	> 0.0	Roughness Height.
5	real	> 0.0	Average stem diameter.
6	real	> 0.0	Average stem density.

Table A.11: Solution Control Cards

Starting Time			
Field	Type	Value	Description
1	char	TC	Card type.
2	char	T0	Parameter.
3	real	#	Starting time of the model.
4	real	#	Units (optional). (0 = secor 1 = minutes; 2 = hours; 3 = days; 4 = weeks; 5 = months; 6 = years)

Final Time			
Field	Type	Value	Description
1	char	TC	Card type.
2	char	TF	Parameter.
3	real	#	Ending time of the model.
4	real	#	Units (optional). (0 = secor 1 = minutes; 2 = hours; 3 = days; 4 = weeks; 5 = months; 6 = years)

Time Step Size			
Field	Type	Value	Description
1	char	TC	Card type.
2	char	IDT	Parameter.
3	int	> 0	Series ID number containii length of timestep (Δt).

Sediment Time Steps			
Field	Type	Value	Description
1	char	TC	Card type.

2	char	SDI	Parameter.
3	int	> 0	Number of time steps to so sediment within each hydr step.

Steady State Simulation			
Field	Type	Value	Description
1	char	TC	Card type.
2	char	STD	Steady state option turned omit to run dynamic simul
3	int	> 0	Initial time step.

Quasi-Unsteady Simulation			
Field	Type	Value	Description
1	char	TC	Card type.
2	char	STH	Parameter.
3	int	> 0	Series ID number containi steady state hydrodynamic condition.
4	int	> 0	Maximum number of itera for steady state solve.
5	real	> 0	Initial time step size for ste state calculation.

Adaptive Time Control On (Interpolation Accuracy Checking)			
Field	Type	Value	Description
1	char	TC	Card type.
2	char	IAC	Adaptive time control turn omit to turn off.

Non-Adaptive Time Steps			
Field	Type	Value	Description
1	char	TC	Card type.
2	char	NDP	Non-Adaptive time contro turned on - omit to turn of

Output			
Field	Type	Value	Description
1	char	OC	Parameter
2	int	> 0	Series ID number that cont the time steps to be output.

Auto-build Output Series			
Field	Type	Value	Description
1	char	OS	Card type.
2	int	> 0	ID number of the series.
3	int	> 0	Number of points in the se
4	int	> 0	Output units. (0 = seconds; 1 = minutes; 2 = hours; 3 = days; 4 = weeks; 5 = months; 6 = years)

Flow Output			
Field	Type	Value	Description
1	char	FLX	Parameter.
2	int	> 0	String ID number for the n string or edge string for wh flow is to be output.

Deactivate String			
Field	Type	Value	Description
1	char	OFF	Card type.
2	int	> 0	String ID number.

Adapted Mesh Printing			

Field	Type	Value	Description
1	char	PC	Card type.
2	char	ADP	Adaptive mesh printing tu on - omit to turn off.

Stopping the Model			
Field	Type	Value	Description.
1	char	END	Close the model.

Table A.12: ADH input file template.

```
OP SW2 2 Dimensional Shallow Water
OP INC #memory_allocation_increment
OP TRN #transport_equations
OP BLK #blocks_per_processor
OP MVS Moving grid option
OP PRE pre-conditioner_selection
OP THT time-weighting. θ
IP IT max_linear_iterations_per_non-linear_iteration
IP NIT max_non-linear_iterations_per_time_step
IP NTL non-linear_tolerance
MP EVS mat# EVxx EVyy EVzz EVxy EVxz EVyz
MP ML mat# max_levels_refinement
MP SRT mat# error_tolerance_for_refinement
MP FUT mat# error_tolerance_for_unrefinement
MP COR mat# latitude_for_the_problem_location
MP MU background_viscosity
MP G gravity
MP RHO fluid_density
MP TMN coefficient_gls_momentum
MP TCN coefficient_gls_continuity
MP U0 reference_velocity
MP R0 reference_density
MP L0 reference_length
MP MU0 reference_viscosity
NDS node# string#
FCS element# face# string#
EGS node# node# string#
DB OVL string# series# series# series# Dirichlet_bc velocity
DB PRS string# series# Dirichlet_bc pressure
NB OVL string# Neumann_bc
DB FRS string# series# Free-surface Dirichlet_bc pressure
DB MVS string# series# Dirichlet_bc displacement
OF string# Outflow_bc
TC T0 initial_time
TC TF final_time
TC IDT series# timestep_size
TC NDP no time adaption
OC series# output
END
```

Troubleshooting

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

- Compile ADH by untarring and running a "make" program...ensure that the correct environment box is set. This can be found in the machines directory for ADH. For example:
  - to run on a t3e system - setenv BOX t3e
  - to run on an origin 3000 system - setenv BOX o3k

- 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:

  1. The percent area change between adjacent elements should not exceed 50%.
  2. Typically the minimum for interior angles is 20 degrees and the maximum is 130 degrees.
  3. No more than eight elements should join at one node.
  4. 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 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.

- ADH outputs a velocity file and an overland head file with a `.dat` extension. These files can be opened and viewed in SMS or GMS without any further post-processing.

- ADH will output the initial values for overland head 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 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.

Vessel Movement Library

ADH has the capability to simulate the effects of a moving vessel on the hydrodynamics of a model. This is done using a pressure field which applies a draft equal to that of the modeled vessel. All vessel characteristics are defined in the boat definition file. An example of such a file is given below.

```
BOAT 1
FDEF 1 3 185.0 -60.0 0.0
DRFT 1 10.0
BLEN 1 30.0
BWID 1 10.0
PBOW 1 0.1
PSTR 1 0.1
CBOW 1 0.95
CSTR 1 0.95
SDEF 1 1 0 135.0 -60.0 5.0
SDEF 1 2 0 125.0 -50.0 5.0
SDEF 1 3 0 125.0 -1.0 8.0
ENDD 0 0
```

This file describes a 30 X 10 boat with a draft of 10.0 moving linearly for one segment, turning right in another segment, and them linearly again for the third and final segment. During linear travel, the vessel moves in the direction of the path. However, during curved segments, the vessel travels tangent to the path. The direction of turn is given as +1.0 for left turns and -1.0 for right turns. This convention follows the "right hand rule" for cartesian coordinate systems. The velocity of the vessel is given at in the field definition and at the end of each segment. Uniform acceleration is assumed throughout the length of the segment. The cards in the above file will be explained in detail in C.1.

NOTE: There can be no blank lines in the boat definition file.

Example Problem

Create a simple square mesh whose sides are 100 meters in length. In order to correspond to the given boat file, the corners should have coordinates of (85.23, -100.6), (185.23, -100.6), (185.23, -0.6), and (85.23, -0.6). The mesh should consist of 676 nodes and 1250 elements, 26 nodes per side, and be numbered along the left edge from left to right. The elevation is -20 meters everywhere.

The hotstart file contains depths only and are set such that the initial water level is zero. In other words, the initial depths are 20 meters.

The boundary condition file is set for 2D shallow water with a Manning's "n" of 0.03. The only boundary string specified is a material string including all of the elements. To this string is applied a Neumann Velocity boundary with a flow of zero. The model is set up to take one second timesteps and complete at 40.0 seconds. The entire boundary condition file is given below.

```
OP SW2
OP INC 4000
OP TRN 0
OP PRE 2
OP BLK 17
OP BT

IP MIT 200
IP NIT 8
IP NTL 1.0E-5

MP ML  1 4
MP MUC 1.0
MP EEV 1 0.50
MP SRT 1 .5
MP COR 1 0
MP G   9.80
MP MU 1.0E-6
MP RHO 1000.0

MTS 1 1
```

```
XY1   1  40   0  0  0  0
1.00    0.0
2.00    0.0
3.00    0.0
4.00    0.0
5.00    0.0
 .       .
 .       .
 .       .
35.00   0.0
36.00   0.0
37.00   0.0
38.00   0.0
39.00   0.0
40.00   0.0

XYT   2   100
XY1   2     2  0  0  0  0
0.0     1.0
40.0    1.0

XY1   3    2  0  0  0  0
0.0     0.0
40.0    0.0

FR MNG   1  0.03

NB OVL   1   3

OC  1

TC  T0    0.0
TC  IDT    2
TC  TF   40.0
TC  NDP
END
```

Once all of the files - the boundary condition file, the hotstart file, the geometry file, and the the boat file - have been completed and named appropriately, Pre_ADH and ADH can be run as previously instructed and the results will show the effects of the boat on the depths and velocities as it moves within the model. See figures 6.1 and 6.2.

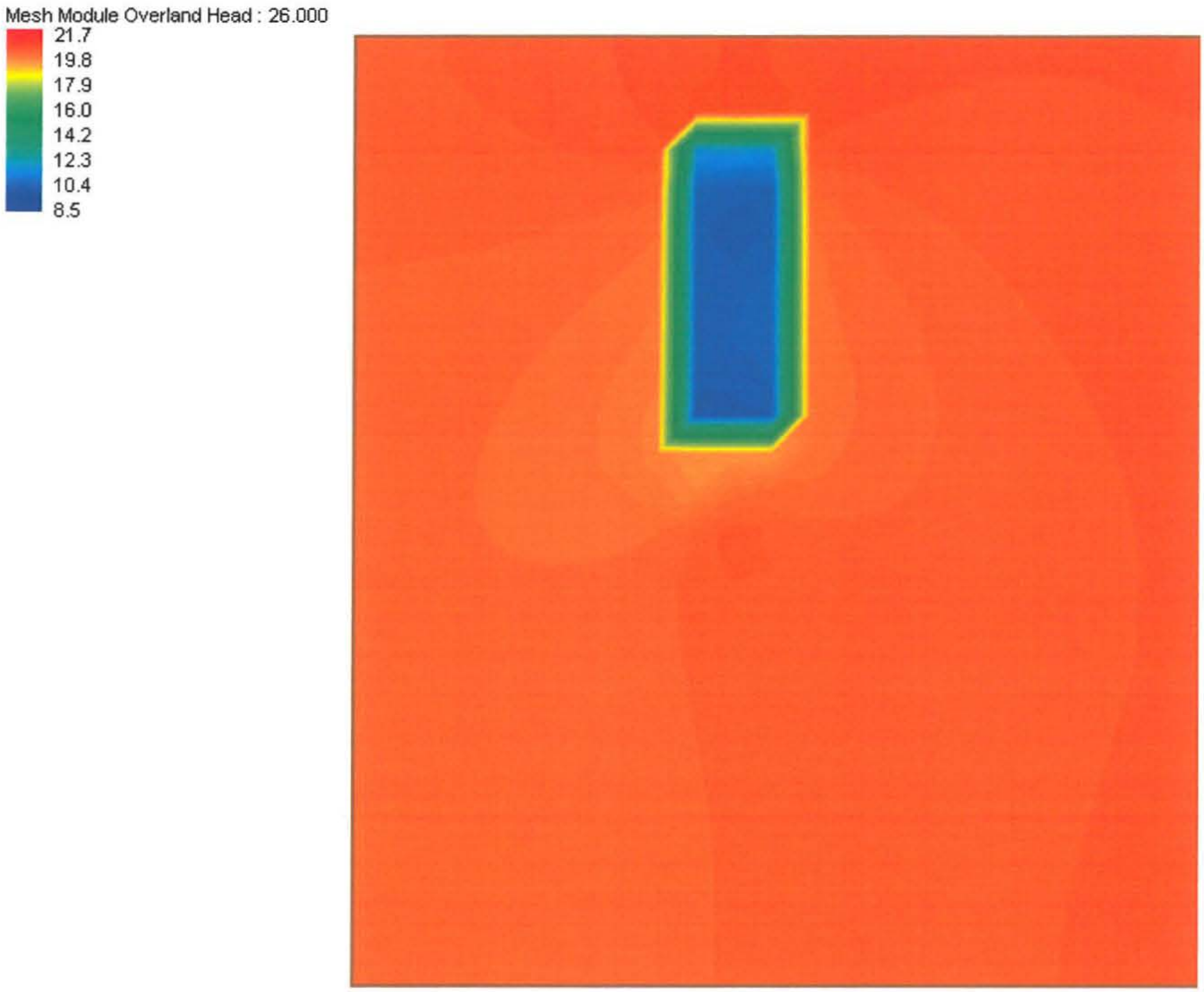

Figure 6.1: Overland head contours for the vessel movement example.

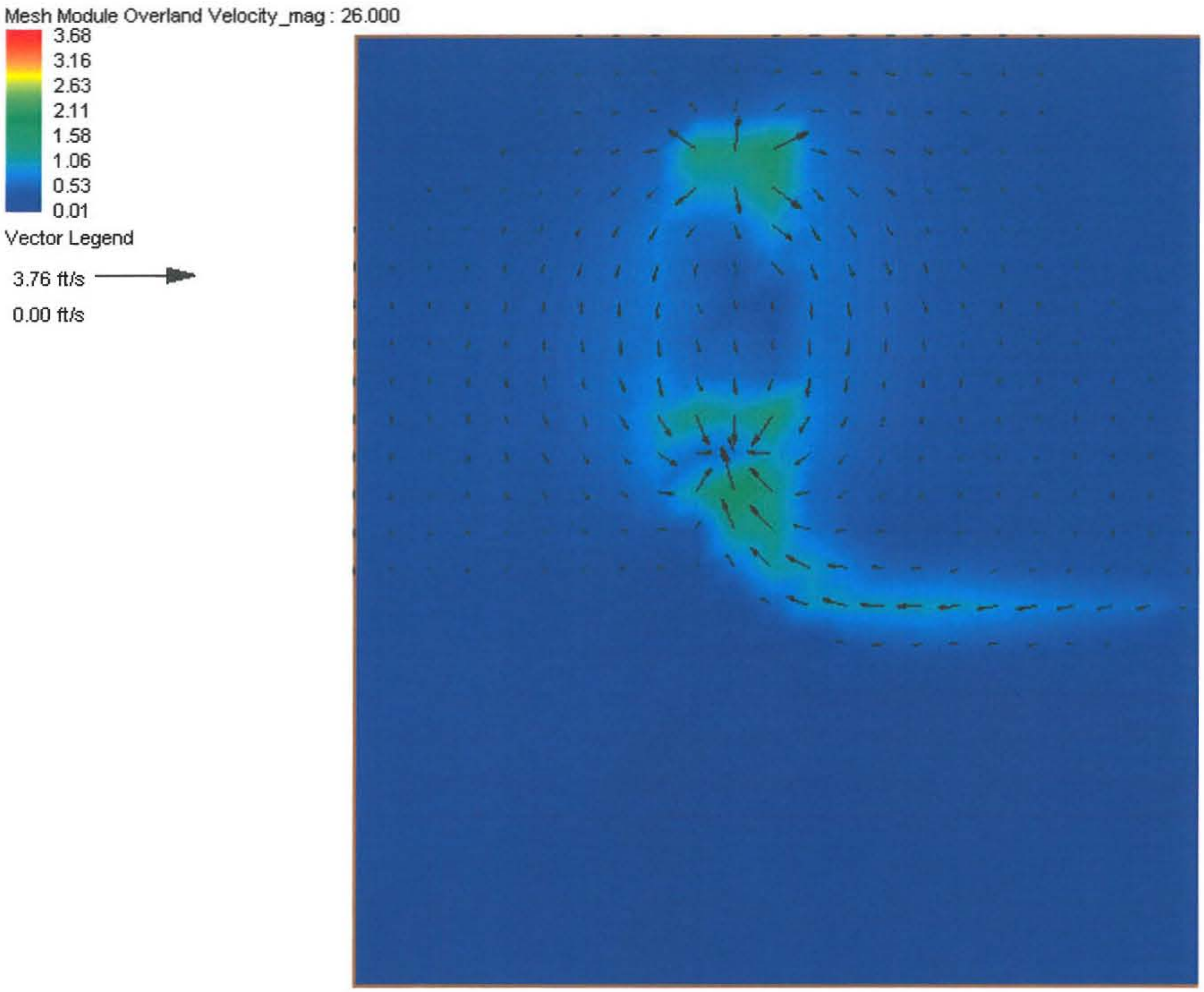

Figure 6.2: Overland velocity vectors and contours for the vessel movement example.

Table C.1: Boat Definitions

Number of Vessels			
Field	Type	Value	Description
1	char	BOAT	Card type.
2	int	> 0	Number of vessels to be modeled.
Sailing Line and Initial Vessel Position			
Field	Type	Value	Description

1	char	FDEF	Card type.
2	int	> 0	Vessel number sailing line defined.
3	int	> 0	Number of segments in sail line.
4	real	#	x-coordinate of the vessel's position.
5	real	#	y-coordinate of the vessel's position.
6	real	#	Initial velocity magnitude c vessel.

Vessel Draft			
Field	Type	Value	Description
1	char	DRFT	Card type.
2	int	> 0	Vessel number.
3	real	#	Vessel draft.

Vessel Length			
Field	Type	Value	Description
1	char	BLEN	Card type.
2	int	> 0	Vessel number.
3	real	> 0	Vessel length.

Vessel Width			
Field	Type	Value	Description
1	char	BWID	Card type.
2	int	> 0	Vessel number.
3	real	> 0	Vessel width.

Bow to Length Ratio			
Field	Type	Value	Description
1	char	PBOW	Card type.
2	int	> 0	Vessel number.

3	real	> 0	Ratio of bow length to the v length.

Stern to Length Ratio			
Field	Type	Value	Description
1	char	PSTR	Card type.
2	int	> 0	Vessel number.
3	real	> 0	Ratio of stern length to the length.

Fraction of Draft Applied to PBOW			
Field	Type	Value	Description
1	char	CBOW	Card type.
2	int	> 0	Vessel number.
3	real	> 0	Bow draft ratio.

Fraction of Draft Applied to PSTR			
Field	Type	Value	Description
1	char	CSTR	Card type.
2	int	> 0	Vessel number.
3	real	> 0	Stern draft ratio.

Propeller Parameters			
Field	Type	Value	Description
1	char	PROP	Card type.
2	int	> 0	Vessel number.
3	int	1 or 2	Propeller type; (1=open wh 2=Kort nozzle).
4	real	> 0	Propeller diameter.
5	real	> 0	Distance between the cente the propellers.
6	real	> 0	Tow boat length.
7	real	> 0	Distance from the propeller the stern of the tow boat.

Sailing Line Segment Definition			
Field	Type	Value	Description
1	char	SDEF	Card type.
2	int	> 0	Vessel number.
3	int	> 0	Sailing line segment numbe
4	int	0 or 1	Type of segment; (0=line;1
5	real	#	x-coordinate of segment en
6	real	#	y-coordinate of segment en
7	real	#	Vessel velocity at segment
8	real	#	If segment is an arc, the x-coordinate of the arc center
9	real	#	If segment is an arc, the y-coordinate of the arc center
10	real	?1.0	If segment is an arc, the dir of turn (+1.0 for left turns a 1.0 for right turns).

End of File			
Field	Type	Value	Description
1	char	ENDD 0 0	Card type.

Second Order Temporal Terms

Introduction

Recently tests were performed which analyzed the capabilities of the Adaptive Hydraulic Model (ADH) (Chapman 2005). It is widely known that 1st order backward schemes are dissipative. This often is not a problem in a forced system with a relatively long wave. But this test involves up to 55 cycles of refection with no forcing; and the 1st order scheme was not up to the task. It should be said that all numerical methods have some form of numerical dissipation which is often used to stabilize the scheme. However for best results in terms of accuracy, the numerical dissipation is reduced as much as possible. In an effort to decrease the numerical dissipation in ADH, it was decided that the option of 2nd order accurate temporal terms should be available to the user as well as the 1st order accurate temporal terms that were used during the tests performed by Chapman. That is to say, terms in the form:

$$\frac{dh}{dt} \approx \frac{h^{n+1} - h^{n}}{dt}$$

would now be replaced by approximations in the form:

$$\frac{dh}{dt} \approx \frac{\left(\frac{3}{2}h^{n+1} - \frac{1}{2}h^{n}\right) - \left(\frac{3}{2}h^{n} - \frac{1}{2}h^{n-1}\right)}{dt}$$

The increase in accuracy of these terms has much improved the previous numerical dissipation issues. This is shown using a Slosh test that was performed in the original report by Chapman (2005) and then addressed here using the improved 2nd order accurate temporal scheme.

The numerical scheme was further enhanced by developing a form in which the user could choose between the two schemes or even a fractional amount of each with the use of the variable tau_temporal. The variable tau_temporal is controlled via the OP TEM card in ADH. The final form of the temporal scheme is given by:

$$\frac{dh}{dt} \approx \alpha \frac{\left(\frac{3}{2}h^{n+1} - \frac{1}{2}h^{n}\right) - \left(\frac{3}{2}h^{n} - \frac{1}{2}h^{n-1}\right)}{dt} + (1-\alpha)\frac{h^{n+1} - h^{n}}{dt}$$

where α is tau_temporal. The possible values for α range from 0 to 1.0. Therefore, when α is input by the user as zero, the scheme will be the original first order accurate scheme. However, when α has a value of 1.0, then the resulting scheme is second order accurate.

Example Problem

Slosh Test

The computational grid shown in figure 7.1a was used to perform the slosh test on ADH. The rectangular mesh is 1000m long and 50m wide. Each cross section consists of five grid nodes which are approximately 12.5m apart. There are 50 grid nodes in the streamwise direction, which are approximately 20.4m apart. The simulations were performed without friction in order to investigate the full effect of the dissipative tendencies of the model. The Manning?s n value was therefore approximately zero. The initial water surface elevation increased linearly from 4.9920m upstream to 5.0080m downstream (see figure 7.1b). Consequently the magnitude of the displacement at both ends is 0.008 in the negative and positive directions respectively. As ADH is used to model the system, the fluid appears to ?slosh? back and forth.

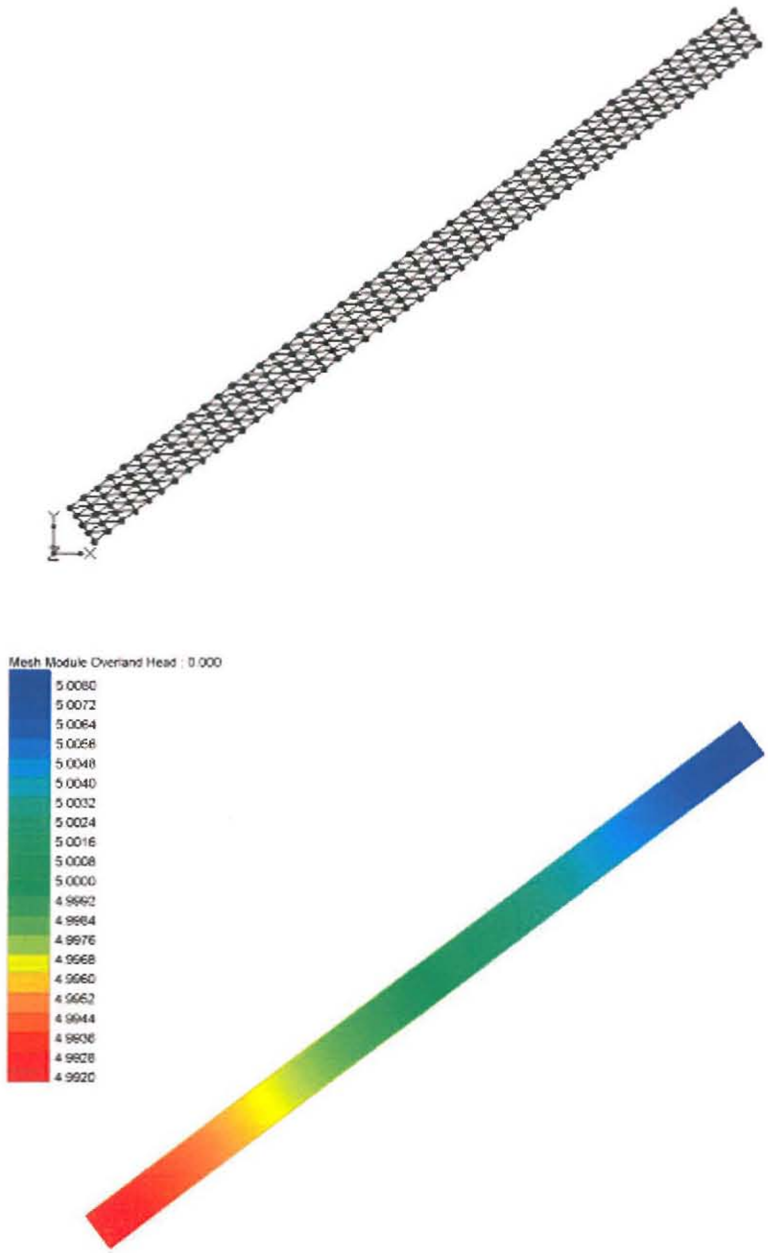

Figure 7.1: Second Order Transport a) Computational Grid. b) Initial Water Surface Elevation for Slosh test

In order to simulate the same conditions of the previous report, an 8 second time step was used in the first test. Seen in figure 7.2 are the results from Chapman (2005). The x-axis gives the grid node and the y-axis the displacement from the still water horizontal position of 5.0m. As time increases, the surface displacement decreases until finally settling at a value of zero within 30minutes. In a truly nondissipative system, the surface displacement should continue to rise and fall for an infinite amount of time. As no numerical technique is completely lacking in dissipation, it is expected that any model undergoing this test will display some dissipative tendencies, most likely settling at zero at extremely long times. However the purpose of this test is to show the obvious decrease in artificial dissipation and therefore increase in accuracy provided by the new 2nd order temporal scheme.

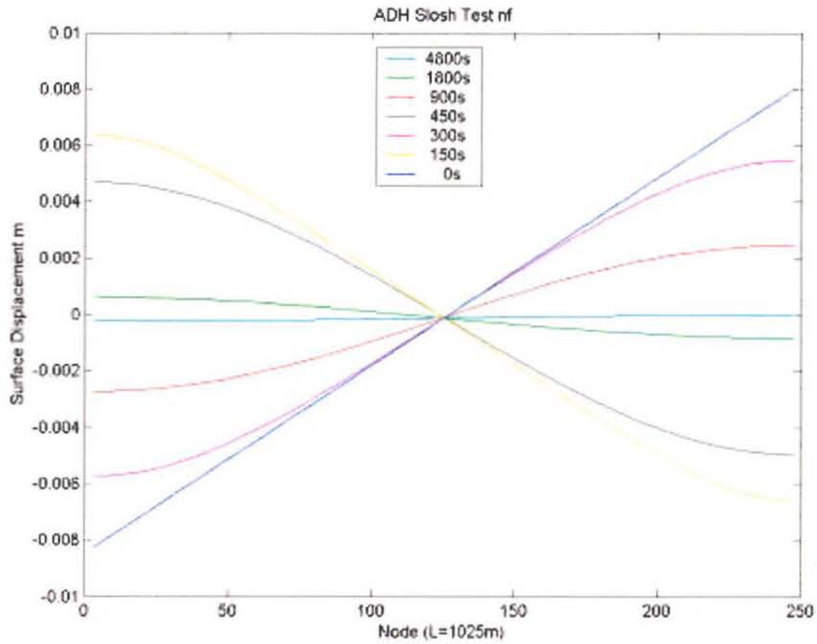

Figure 7.2: ADH slosh test performed by Chapman (2005) with 8 second time step.

In figure 7.3 the slosh test is shown with results from the 2nd order scheme. As in figure 7.2, the x-axis gives the grid node and the y-axis the displacement from the still water horizontal position of 5.0m. The apparent step-like characteristic of the lines show that across the channel cross-section the water surface elevation remains constant. From this figure it is possible to see that there is a definite decrease in the dissipative effects of the scheme. However this concept is perhaps better visualized in figure 7.4 which shows a comparison of the two different schemes. In this figure a time series is shown of the surface displacement at the first grid node only. Initially, as expected, the surface displacement is -0.008 for both schemes because this is at the furthest upstream position. As time progresses, the displacement level continues to rise and fall as the water sloshes back and forth for the 2nd order scheme. However the 1st order scheme quickly drops to zero.

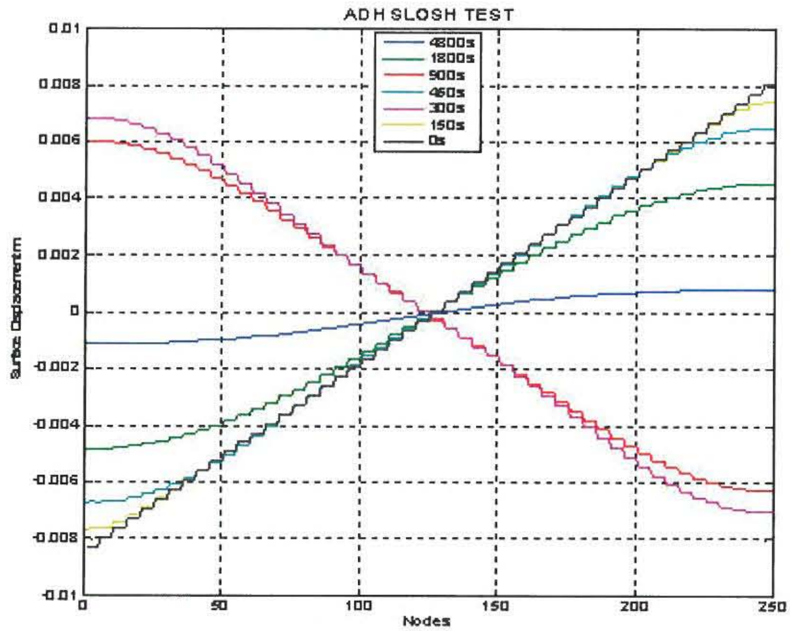

Figure 7.3: Slosh test using 2nd order accurate temporal terms.

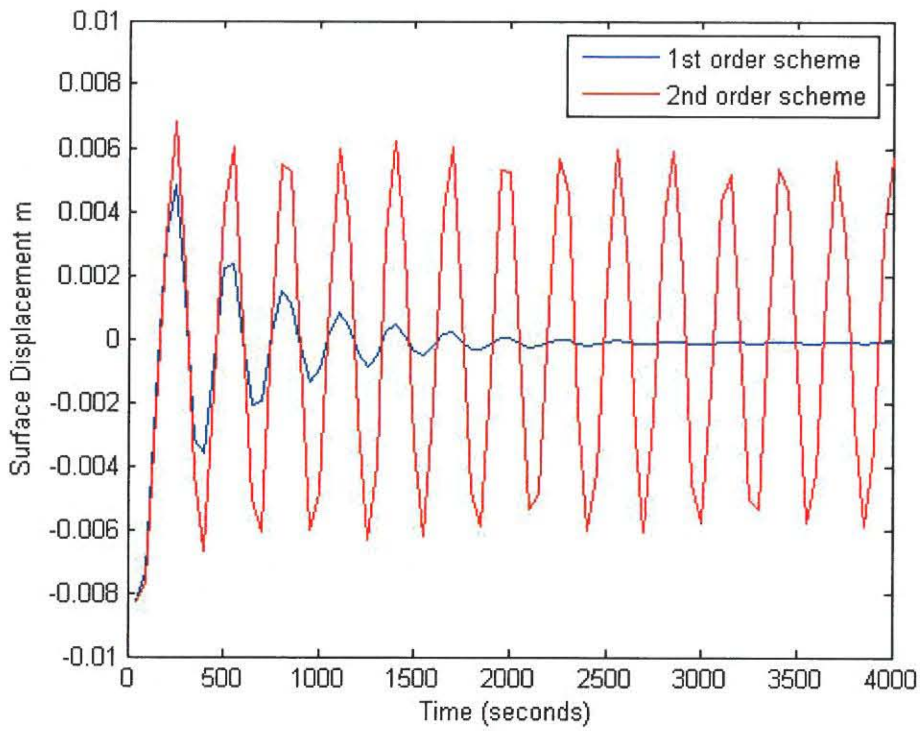

Figure 7.4: Comparison of surface displacement obtained using 1st and 2nd order accurate temporal

schemes.

Convergence Test

In addition to the dissipation test, a convergence test (see figure 7.5) was performed by Chapman (2005), which showed that when using relatively small time steps of 0.25 and 0.125 seconds, the resulting solutions did not converge. Figure 7.6 shows the solutions obtained when using the 2^{nd} order accurate temporal scheme. The solutions appear to have converged to one time step independent solution.

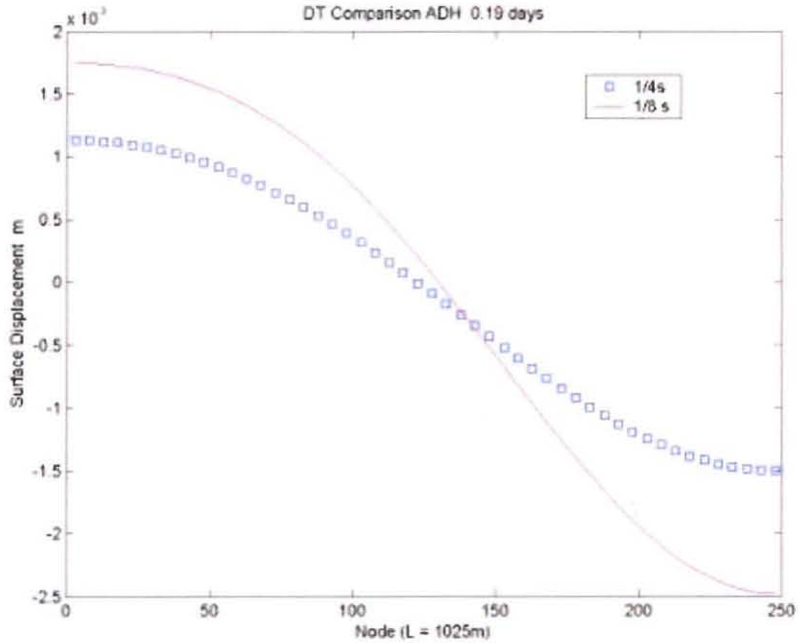

Figure 7.5: ADH time step comparison at 0.19 days into the simulation using 1^{st} order accurate temporal scheme performed by Chapman(2005).

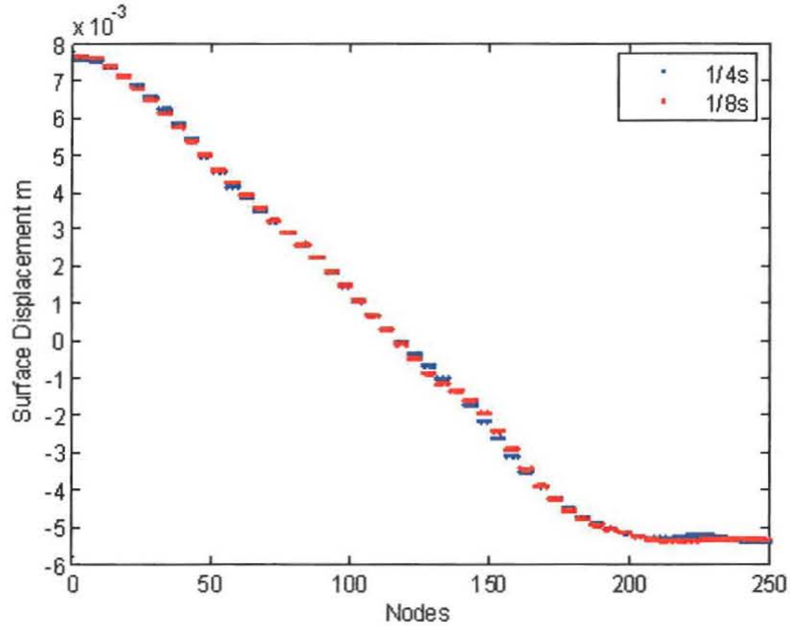

Figure 7.6: ADH time step comparison at 0.19 days into the simulation using 2nd order accurate temporal scheme.

Output Filename Convention (*.dat)

***_dep.dat** overland head (scalar, depth)

***_ovl.dat** overland velocity (vector)

***_con#.dat** constituent concentration, # = constituent number (scalar, parts per million for sediment)

***_dpl.dat** bed displacement (scalar, meters)

***_alt.dat** active layer thickness (scalar, meters)

***_ald.dat** active layer distribution (scalar, one column for each grain size ? finest to coarsest left to right)

***_blt#.dat** bed layer thickness, # = layer number (scalar, meters, 1 is the bottom-most layer)

***_bld#.dat** bed layer distribution, # = layer number (scalar, 1 is the bottom-most layer, one column for each grain size ? finest to coarsest left to right)

***_cbp#.dat** cohesive bed property, # = layer number (scalar, 1 is the bottom-most layer, one column for each grain size ? finest to coarsest left to right)

***_bsh.dat** bed shear stress magnitude (scalar)

***_bedload.dat** bedload (vector)

For a 3 constituent simulation of 2 grains and 2 bed layers?(remember the added grain and layer!)

***_dep.dat** (ioh)

Depth value

***_ovl.dat** (iov)

X_vel, Y_vel, Z_vel

***_con1.dat** (ic 1)

Concentration 1

***_con2.dat** (ic 2)

Concentration 2

***_con3.dat** (ic 3)

Concentration 3

***_con4.dat** (ic 4)

Concentration 4

***_dpl.dat** (id)

Displacement

***_alt.dat** (ialt)

Active layer thickness

***_ald.dat** (iald)

Ald-grain1, Ald-grain2, Ald-grain3

***_blt1.dat** (iblt 1)

Bed layer thickness

***_blt2.dat** (iblt 2)

Bed layer thickness

***_blt3.dat** (iblt 3)

Bed layer thickness

***_bld1.dat** (ibld 1)

Bld-grain1, Bld-grain2, Bld-grain3

***_bld2.dat** (ibld 2)

Bld-grain1, Bld-grain2, Bld-grain3

***_bld3.dat** (ibld 3)

Bld-grain1, Bld-grain2, Bld-grain3

***_bsh.dat**

Bed shear magnitude

***_bedload.dat**

Bedload_X, Bedload_Y

***_conflx**

concentration flux for each constituent is now included when the FLX card is used with a transport simulation

References

Christensen, B. A., (1972) "Incipient Motion on Cohesionless Channel Banks", *Sedimentation*, H.W. Shen, editor, Fort Collins , CO.

Christensen, B. A., (1995) "Open Channel and Sheet Flow Over Flexible Roughness" Proceedings, 21st IAHR Congress, Melbourne, Australia, 19-23 August.

Jacobs, Jennifer M., and Wang, Min-Hui (2003) "Atmospheric Momentum Roughness Applied to Stage-Discharge Relationships in Flood Plains" ASCE Journal of Hydrologic Engineering, March/April (2003)8:2(99).

Walton R., and Christensen, B. A. (1980). "Friction factors in storm surges over inland areas" *J. Waterw. Port, Coastal Ocean Div., Am. Soc. Civ. Eng.*, 106(2), 261-271.

Footnotes

1

USAE Research and Development Center

Last modified Apr-09