ADAPTIVE HYDRAULICS

A TWO-DIMENSIONAL MODELING SYSTEM
DEVELOPED BY THE COASTAL AND HYDRAULICS LABORATORY
ENGINEER RESEARCH AND DEVELOPMENT CENTER
A PRODUCT OF THE SYSTEM-WIDE WATER RESOURCES PROGRAM

USERS MANUAL

Guidelines for Solving Two-Dimensional Shallow Water Problems
with the Adaptive Hydraulics Modeling System

AdH Version 4.201

R.C. Berger | J.N. Tate | G.L. Brown | G. Savant

Last modified September 7, 2012

Table of Contents

Abstract/Introduction **5**
Shallow water example **6**
Sign convention/A note on units **8**
Files needed to run AdH **8**
1. Mesh file 8
2. Hotstart file 9
3. Boundary Condition file 10
Boundary Condition file: Control cards 10
Using control cards 11
Input file template **12**
Control card categories **13**
Operation parameters 13
Iteration parameters 13
Material properties 14
Boundary strings 15
Constituent properties 15
Time series 15
Friction controls 15
Sediment Process controls 16
Output controls 16
Solution controls 16
Time controls 17
Operation parameters **18**
Simulating vessels in a waterway 19
Operation parameter cards 19
Iteration parameters **22**
Iteration parameter cards 22
Material properties **25**
Specifying flow parameters 25
Constant Eddy Viscosity (EVS) 25
Estimated Eddy Viscosity (EEV) 26
Mesh refinement 27
Wetting and drying 29
Wind Specification 29
Wind Attenuation 30
Material property cards 30
Material meshing cards 33
Constituents **34**
Vorticity transport-bendway correction 34
Baroclinic transport (Salinity and temperature) 34
Constituent cards 35
Boundary strings **37**

Node strings 37
Edge strings 37
Mid strings 37
Material strings 37
Ice strings 37
Boundary string cards 38
Time series 40
X-Y series 40
Wind series 40
DSS Time Series Input 43
Time series cards 43
Friction controls 45
Bed shear stress 45
Submerged aquatic vegetation 45
Unsubmerged rigid vegetation 45
Ice friction 46
Friction control cards 46
Solution controls 49
Friction 49
Bed 49
Sidewalls 49
Upstream boundary 50
Downstream boundary 50
Rain or evaporation 51
Stage discharge 52
Stationary lid in the flow 52
Structures library 52
Solution control cards 54
Time controls 60
Steady state 60
Automatic Time Step Determination 61
Sediment time steps 61
Quasi-unsteady 61
DSS Input Time Control 62
Time control cards 62
Output control 65
Auto-build output series 65
Standard screen output 67
DSS Output 67
MEO Output 67
Output control cards 68
Sediment transport 71
Cohesionless sediment (Sand) 71
Cohesive sediment (Sand and/or silt) 72

Mixed sediments (Sand, silt or clay) 73
Sediment output files 74
Transport constituent properties 74
Material property cards 74
Constituent cards 78
Sediment process cards 79
Solution control cards 80
Running AdH 81
Output filename conventions 82
Two-dimensional shallow water equations, finite element formulation 85
Precision of Conservation in AdH 91
Troubleshooting 93
AdH-DSS Instructions 96
References 97

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) or 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. These examples can be obtained in the supplemental material provided with the AdH code.

Introduction

The Adaptive Hydraulics (AdH) Model is a modeling 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 here.

It can be used in a serial or multiprocessor mode on personal computers (currently serial only), UNIX, Silicon Graphics and CRAY operating systems. The uniqueness of AdH is its ability to dynamically refine the domain mesh in areas where more resolution is needed at certain times due to changes in the flow conditions.

AdH can simulate the transport of conservative constituents, such as dye clouds, as well as sediment transport that is coupled to bed and hydrodynamic changes. The ability of AdH to allow the domain to wet and dry as flow conditions or tides change is suitable for shallow marsh environments, beach slopes, floodplains and the like.

AdH is set up such that supercritical and subcritical flows can be represented at the model boundaries as well as internal to the system. It has the ability to simulate vessel transport as well as bridge decks and culvert entrances as pressure field applications.

This tool is being developed at the Coastal and Hydraulics Laboratory (CHL) and has been used to model sediment transport in sections of the Mississippi River, tidal conditions in southern California, and vessel traffic in the Houston Ship Channel. The model is designed to work in conjunction with the DoD Surface Water Modeling System (SMS). The Surface Water Modeling System (SMS) and the Groundwater Modeling System (GMS) 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.

Shallow water example

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 depth distribution in the flow field, Fig. 1.2.

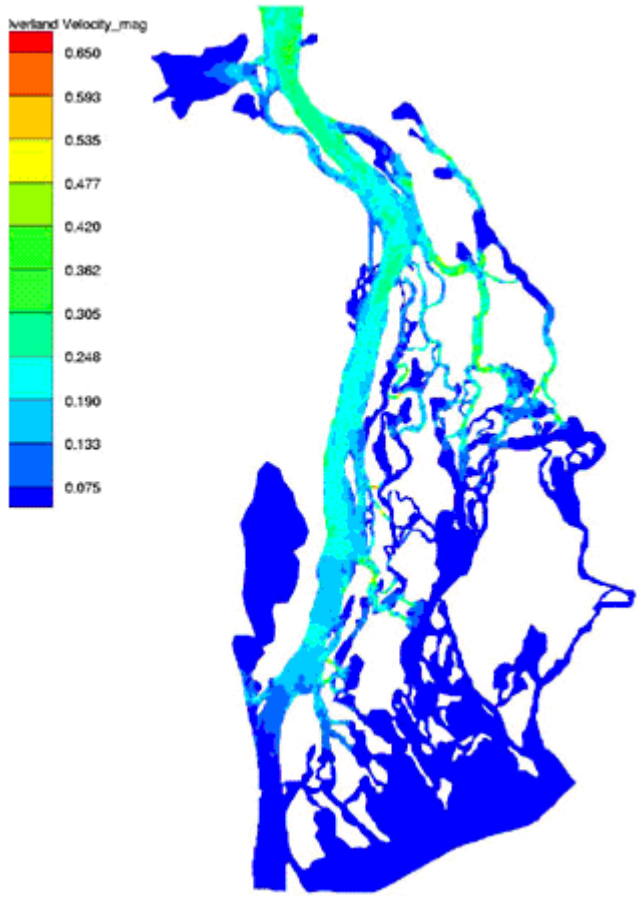


Figure 1.1: Velocity for Pool 8, Mississippi River

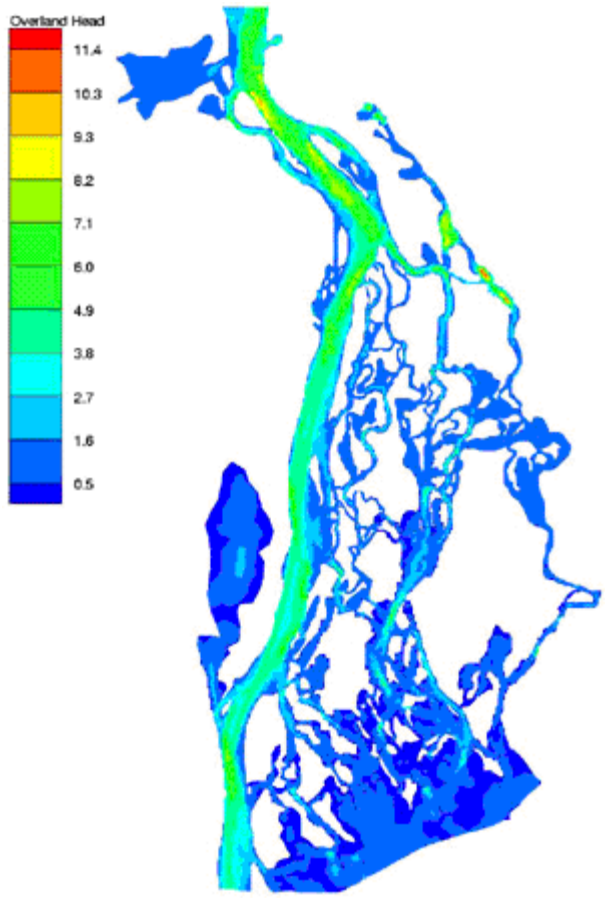


Figure 1.2: Depth for Pool 8, Mississippi River

Files needed to run AdH

Three files are needed to run a model in AdH — the mesh file, the boundary conditions file and the hotstart file.

The **mesh file** must be constructed first and can be generated directly with 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 **hotstart** 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. The commands are:

pre_adh filename
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, GMS or SMS can be used to visualize the results.

Sign Convention

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

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.**

1. Mesh file

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 GMS or SMS. Mesh files, *filename*.3dm, follow the GMS 3-D mesh format. AdH uses only linear tetrahedral elements, although GMS supports tetrahedral, pyramid, wedge and hexahedral elements in 3D.

In 2D, AdH uses linear triangles while GMS and SMS support triangles and quadrilaterals. AdH uses the mesh files generated with GMS or SMS 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. However, if the mesh files are created using the AdH module of SMS, the appropriate .3dm extension is assigned. Simple rectilinear domains are convenient starting points for more complicated models.

For full details on how to produce mesh files using GMS or SMS, refer to the appropriate reference manual and tutorials, which can be found at http://chl.erdc.usace.army.mil.

2. Hotstart file

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

Field data is 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. This data is specified in the **hotstart** file. GMS or SMS provides a simple interface for entering field data. This data is entered into a scatter point data file and interpolated to the problem mesh.

A **hotstart** file is a required part of a model. For AdH, initial water depths must be given, although some can be zero or negative (dry). All other variables can start at zero and, therefore, be omitted from the hotstart 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. An example of these is shown below (additional dataset names are provided near the end of this document):

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

The datasets used for the **hotstart** file can be generated with GMS or SMS. A standard GMS 3-D mesh data set format is used in the **hotstart** file. The files contain a specific heading, the timestep for the values to follow, and the depths or X, Y, Z velocity components. No node or element numbers are given as the values are listed; there is simply the correct number of lines to match sequentially with the number of nodes.

Multiple data sets are exported from GMS or SMS and copied to the **hotstart** file in any order. If a dataset is not supplied for one or more of the parameters, AdH will assign default values to all the nodes 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 hotstart file and assigns them at the start time specified in the boundary conditions file. The time in the **hotstart** file is ignored by AdH. For consistency, however, it is recommended that the time in the hotstart file — located on the **TS** line — match the start time in the boundary conditions file, located on the **TC T0** card. To create a **hotstart** file from a previous run, the ASCII data list for each variable at the desired time should be taken from the output files and combined to create the new **hotstart** file.

3. Boundary Condition file

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
- Timestep data
- Output controls
- Adaptivity controls
- Model operation controls
- Transport controls.

Details of the options available for the boundary conditions file and therefore the type of computations that AdH can perform are in the following sections. These options will be expanded as the AdH model continues its development and new needs arise.

Boundary Condition file: Control cards

A boundary condition file, *filename*.bc, for the AdH code contains a series of one-line control cards. There is no particular order that these cards must be provided, so the user can develop a method that is convenient. Cards are single line entries and cannot be wrapped across lines.

The cards fall into ten basic categories:

- Operational parameters
- Iteration parameters
- Material properties
- Boundary strings
- Solution controls/boundary conditions
- Constituent properties
- Friction properties
- Time controls
- Output controls
- Time 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 and the level of convergence.

Material properties define how the model is divided up spatially in order to define properties differently in various locations.

Boundary strings 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 as well as the model's start and end time.

Output controls define the times at which the output is printed.

Time series define how the data used to define various aspects of the model changes with time.

Constituent properties define transport parameters for sediment and other transported quantities.

Friction controls define the method and parameters for roughness properties.

Using control cards

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 six 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.

Comments can be included in the Boundary Condition file by preceding the comment with and exclamation point (!).

The following input file template gives the required cards for any hydrodynamic simulation using AdH.

Input file template

```
! Operation parameters
OP SW2     2 Dimensional Shallow Water
OP INC     # memory allocation increment
OP BLK     blocks per processor
OP TRN     # transport equations
OP PRE     pre-conditioner selection
! Iteration parameters
IP NIT     max # non-linear iterations per group
IP MIT     max # linear iterations
IP NTL     non-linear tolerances
! Material properties
MP MUC     Manning's units constant
MP G       gravity
MP MU      kinematic viscosity
MP RHO     density
MP DTL     wetting/drying limits (defaults to 0 if not included)
MP EEV     estimated eddy viscosity
or
MP EVS     eddy viscosity (MP DF required when OP TRN>0)
MP ML      max # refinement levels
MP SRT     error tolerance for refinement (MP TRT required when OP TRN>0)
! Strings  (see manual for optional strings depending on specific problem)
MTS        material strings
! Time Series
XY1        xy-series
! Boundary conditions (boundary condition according to specific problem)
! Friction controls (several options available)
! Time controls
TC T0      starting time of model
TC TF      extending time of model
TF IDT     xy-series number giving timestep size
! Output controls
OC         timestep series defining the output (requires corresponding xy-series)
or
OS         auto-build output series (if used, omit OC with corresponding xy-series)
END        stops the model
```

Control card categories

The cards and their categories are:

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 BTS	Enable Vessel Entrainment
OP TEM	Enable Second Order Temporal Terms
OP TPG	Petrov-Galerkin Coefficient
OP NF2	Velocity Gradients

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 tolerance for reactive constituents
IP SST	Quasi-Unsteady Tolerance

Material properties

MP EVS	Constant Eddy Viscosity
MP EEV	Estimated 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 Limit
MP ML	Maximum Mesh Refinement
MP SRT	Mesh Refinement Tolerance
MP DF	Turbulent Diffusion (Transport Constituent Property)
MP WAT	Wind Attenuation
MP TRT	Transport Refinement Tolerance (Transport Constituent Property)
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 bed sediment applied by layer
MP CBN	Cohesive bed sediment applied to selected nodes
MP CBM	Cohesive bed sediment applied by material
MP NCP	Number of consolidated layers
MP CPA	Consolidation properties applied by layer
MP CPN	Consolidation properties applied to selected nodes
MP CPM	Consolidation properties applied by material
MP NDM	Turn off sediment bed displacement activated by material
MP FRC	Wet/dry fraction of an element for computation
MP WND	Wind specification by material

Boundary strings

NDS	Node String
EGS	Edge String
MDS	Mid String
MTS	Material String
INS	Ice Node String

Constituent properties

CN CON	Any Constituent
CN CLA	Cohesive Sediment (Clay and/or Silt)
CN SND	Cohesionless Sediment (Sand)
CN VOR	Vorticity Transport - Bendway Correction
CN SAL	Salinity (Baroclinic Transport)
CN TMP	Temperature (Baroclinic Transport)

Time series

XY1	X-Y Series Cards
XY2	X-Y-Y Series Cards
XYC	Wind Station Coordinates
DSS	DSS formatted time series input data

Friction controls

FR MNG	Manning's N Roughness
FR ERH	Equivalent Roughness Height
FR SAV	Submerged Aquatic Vegetation
FR URV	Un-submerged Rigid Vegetation
FR ICE	Ice Thickness

FR IRH	Ice Roughness Height
FR BRH	Bed Roughness Height

Sediment Process controls

SP NSE	Noncohesive Suspended Sediment Entrainment
SP NBE	Noncohesive Bedload Sediment Entrainment
SP HID	Noncohesive Hiding Factor
SP CSV	Cohesive Settling Velocity
SP WWS	Bed Shear Stress due to Wind waves

Output controls

OC	Output Control Series
OS	Auto-build Output Series
FLX	Flow Output
PC ADP	Adapted Mesh Print Control
PC ELM	Numerical Fish Surrogate Output
PC LVL	Screen Output Level
PC HOT	Hotstart File Print Control
PC DSS	Output data in DSS format
PC MEO	Mass Error Output
END	Signifies the end of the BC file

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
DB BCH	Dirichlet – Breach Elevation
NB DIS	Natural - Total Discharge
NB OVL	Natural - Flow
NB OTW	Natural - Tailwater elevation
NB TRN	Natural – Transport
NB SDR	Stage - Discharge Boundary
NB SPL	Spillway Boundary
OB OF	Outflow Boundary
EQ TRN	Equilibrium Sand Transport Boundary
OFF	Deactivate String
OUT	Outflow Edge String
WER	Number of Weirs
WRS	Weir Parameters
FLP	Number of Flap Gates
FGT	Flap Gate Parameters

Time controls

TC T0	Start Time
TC IDT	Time Step Series
TC TF	Final Time
TC ATF	Auto Time Step Find
TC SDI	Sediment transport increment for time step
TC STD	Steady State solution
TC STH	Quasi-Unsteady solution
TC DS0	DSS start time
TC DSF	DSS final time
TC FIN	File input (currently only used for wind fields, **WND**)

OPERATION PARAMETERS

Each operation parameter card consists of two character fields and may contain one numeric field. An **OP** in the first field identifies operational parameter cards. **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 **OP PRE** and the **OP BLK** cards specify the preconditioner for the linear solver and the manner in which it is implemented.

```
OP SW2
OP  INC   40
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. **A good starting option is PRE = 1 and BLK = 1.**

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 following card specifies one transported quantity:

```
OP TRN 1
```

The **OP TEM** card is an optional card that can be 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 first or second 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:

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

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

OP TEM 1

The **OP NF2** operational parameter activates the computation of 2D Shallow water gradients using a Superconvergent Patch Recovery (SPR) method (*Zeinkiewicz and Zhu, 2005)*. When this card is included, two additional output files will be generated that contain the velocity gradients with time. The **OP TPG** card can be included to inform AdH of the coefficient to be utilized in Streamwise Upwind Petrov-Galerkin (SUPG) stabilization for convection dominated flows.

Simulating vessels in a waterway

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**).

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

For reference, there is a technical report titled “Modeling vessel-generated currents and bed shear stresses” and a technical note titled “Considerations for modeling vessel-generated currents and bed shear stresses” for the vessel effects modeling study located on the AdH publications page.

Operation parameter cards

OP SW2

2D SHALLOW WATER PROBLEMS

Field	Type	Value	Description
1	char	OP	Card type
2	char	SW2	Specifies 2-D shallow water problem

OP INC

INCREMENTAL MEMORY

Field	Type	Value	Description
1	char	OP	Card type
2	char	INC	Parameter
3	int	> 0	Incremental memory allocation

OP TRN

TRANSPORT EQUATIONS

Field	Type	Value	Description
1	char	OP	Card type
2	char	TRN	Parameter
3	int	≥ 0	Total number of transported materials

OP BLK

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 perform pre-conditioning

OP NF2

SW2 GRADIENTS

Field	Type	Value	Description
1	char	OP	Card type
2	char	NF2	Parameter (Optional card)

OP PRE

PRE-CONDITIONER SELECTION

Field	Type	Value	Description
1	char	OP	Card type
2	char	PRE	Parameter
3	int	3 ≥ # ≥ 0	Prec_value 0 No pre-conditioning 1 one level Additive Schwarz pre-conditioning 2 two level Additive Schwarz pre-conditioning 3 two level Hybrid pre-conditioning

OP BT

VESSEL MOVEMENT LIBRARY INCLUSION (ENABLE VESSEL MOVEMENT)

Field	Type	Value	Description
1	char	OP	Card type
2	char	BT	Parameter

OP BTS

ENABLE VESSEL ENTRAINMENT

Field	Type	Value	Description
1	char	OP	Card type
2	char	BTS	Parameter

OP TEM

SECOND ORDER TEMPORAL TERM

Field	Type	Value	Description
1	char	OP	Card type
2	char	TEM	Parameter
3	real	$1 \geq \# \geq 0$	Coefficient for the second order temporal scheme

OP TPG

PETROV-GALERKIN COEFFICIENT

Field	Type	Value	Description
1	char	OP	Card type
2	char	TPG	Parameter
3	real	$0.5 \geq \# \geq 0$	Coefficient for the Petrov-Galerkin equation (optional input card)

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 time step size of ¼ of the previous 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 additional card that allows convergence to be determined by the change in the velocity, depth and concentration solutions. By default this tolerance is zero. The user may use either the **IP NTL** or **IP ITL** card to determine convergence; however, if both cards are used then convergence must be met for both. The **IP RTL** parameter specifies a change tolerance against which a 4th order Runge-Kutta (RK) solution is compared. If the RK solution change is greater than the specified tolerance, RK internally reduced the time step and repeats the computation until the solution change is less than or equal to the tolerance (*Numerical Recipes in C, 1992)* . **IP RTL** is an optional card. The **IP SST** card is used to specify the tolerance for quasi-unsteady convergence so that it can differ from the tolerance set for the transport convergence. More details on the quasi-unsteady method can be found under Time Controls.

IP NIT 5
IP NTL 1.0E-3
IP MIT 100

Iteration parameter cards

IP NIT

NON-LINEAR ITERATIONS (OPTION 1)

Field	Type	Value	Description
1	char	IP	Card type
2	char	NIT	Parameter
3	int	≥ 1	Number of non-linear iterations per time step, if at NIT the tolerance is not satisfied AdH will reduce the time step and recalculate

IP FNI

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 iterations per time step, even if at FNI the tolerance is not satisfied then AdH will accept the solution and proceed to the next time step

IP NTL

NON-LINEAR TOLERANCE

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

IP ITL

INCREMENT TOLERANCE

Field	Type	Value	Description
1	char	IP	Card type
2	char	ITL	Parameter
3	real	≥ 0	Tolerance for maximum change in the velocity and depth solutions

IP MIT

LINEAR ITERATIONS (OPTION 1)

Field	Type	Value	Description
1	char	IP	Card type
2	char	MIT	Parameter
3	int	≥ 1	Maximum number of linear iterations per non-linear iteration by the interative solver. If the internal linear tolerance (0.0001* NTL) is not met at MIT the solution stops and the time step size is reduced.

IP FLI

LINEAR ITERATIONS (OPTION 2)

Field	Type	Value	Description
1	char	IP	Card type
2	char	FLI	Parameter
3	int	≥ 1	Maximum number of linear iterations by the iterative solver. If the internal tolerance (0.0001 * NTL) is not met at FLI the solution will proceed to the next nonlinear iteration.

IP RTL

RUNGE-KUTTA TOLERANCE FOR REACTIVE CONSTITUENTS

Field	Type	Value	Description
1	char	IP	Card type
2	char	RTL	Parameter

IP SST

QUASI-UNSTEADY TOLERANCE

Field	Type	Value	Description
1	char	IP	Card type
2	char	SST	Parameter
3	real	> 0	Tolerance for quasi-unsteady hydrodynamic convergence

MATERIAL PROPERTIES

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 several 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#*). However, a few of these cards are applied throughout the entire problem and do not include a material number. These are **MU**, **G**, **RHO**, **MUC**, and **DTL**.

Specifying 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.

Constant 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.

The estimated eddy viscosity is given by one of 3 methods. The first method is a purely isotropic treatment of the eddy viscosity. The second method consists of both isotropic and anisotropic terms. The third method is the Smagorinsky (1963) method which is a common eddy viscosity method for rapidly changing velocity directions. The user determines which method they wish to employ as part of the input to the EEV card.

Method 1 is an isotropic estimate of the eddy viscosity, as given in Rodi (1984). The equation is given as follows:

$$\varepsilon_{EVI} = 2K\sqrt{C_d}hu$$

Method 2 is a means of estimating the eddy viscosity with 2 separate terms: : an isotropic term that accounts for turbulent mixing (EVI), and an additional anisotropic term in the direction of flow that accounts for streamwise dispersion (SDA). The equations for each of these are given as follows (Note: the equation for turbulent mixing (EVI) is taken from Webel and Schatzmann(1984). The anisotropic term is derived from basic hydraulic principles):

$$\varepsilon_{EVI} = 0.92K\sqrt{C_d}hu$$

$$\varepsilon_{SDA} = 1.3\sqrt{C_d}hu$$

The terms for the above equations are listed below:

K = a user-defined scaling coefficient (default = 0.5)
C_d = the drag coefficient, as determined by the bed friction
h = the water depth
u = the depth averaged velocity
NOTE for these equations, a minimum value of u is specified, as follows:

$$u_{MIN} = 0.1\sqrt{gh}$$

Note that if vorticity transport is active, an anisotropic dispersion term normal to the direction of flow is included in the calculations that accounts for transverse dispersion.

$$\varepsilon_{TDA} = 0.5\sqrt{C_d}\,hu_{T.MAX}$$

Where:

$u_{T.MAX}$ = the maximum transverse velocity (taken from the vorticity calculations)

Method 3 uses the Smagorinsky formulation to compute the eddy viscosity. This option utilizes the square-root of the area of the element as the length scale, L, and a user specified coefficient, c. . The algorithm is given below.

$$v_t = CL\left[\left(\frac{\partial u}{\partial x} - \frac{\partial v}{\partial y}\right)^2 + \left(\frac{\partial u}{\partial y} - \frac{\partial v}{\partial x}\right)^2\right]^{0.5}$$

Lilly (1967) analytically derived the value of C to be between 0.16 and 0.20.

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. Note that, if the **EEV** card is used, the **DF** values for turbulent diffusion of constituent calculations will override the **EEV** values in wet/dry elements. Vorticity induced dispersion is always active when vorticity transport is active.

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. The number of times an element can be divided is 2^{ML}. So when setting the **ML** value to 4, a single element can be divided into a maximum of 16 elements.

When refinement is on, the refinement error tolerance is given on the **SRT** card. If the solution error on an element exceeds the refinement 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:

$$coefficient = \int_e \left(\frac{\partial h}{\partial t} + u\frac{\partial h}{\partial x} + v\frac{\partial h}{\partial y} + h\frac{\partial u}{\partial x} + h\frac{\partial v}{\partial y}\right)^2$$

$$Error = \sqrt{coefficient} * Area_{element}$$

and this value is compared to the number given on the **SRT** card. The error value given in the output data file is the ratio of the calculated error to the specified tolerance. By observing the *project_name*_err.dat file produced, an appropriate error tolerance for adaption can be determined.

The error for transport calculations is:

$$coefficient = \int_e \left(h * \frac{\partial c}{\partial t} + uh \frac{\partial c}{\partial x} + ch \frac{\partial u}{\partial x} + vh \frac{\partial c}{\partial y} + c \frac{\partial v}{\partial y} \right)$$

$$Error = \sqrt{coefficient} * Area_{element}$$

In models with transport, the larger of the flow refinement or the transport refinement will determine each element's value in the *project_name*_err.dat file. However, the hydrodynamic and transport errors will be stored separately in files labeled as such: *project_name*_err_hydro.dat and *project_name*_err_con#.dat.

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, the **TRT** card, to a value of one (1) as well so that both sets of errors are stored accordingly and can be referenced in order to determine the appropriate refinement tolerance for each parameter and material. The combined error file, *project_name*_err.dat, contains values that are normalized by the tolerance values and can be useful to determine how the refinement is adjusting the mesh. The individual error files for hydrodynamics and transport are not normalized but as the mesh is refined, these values should reduce in areas where refinement is occurring.

The unrefine tolerance is currently set within the code as 10 percent of the refine tolerance for both flow conditions and transport conditions. When the grid solution error improves, the elements are recombined, although no coarser than the original mesh. The solutions saved, although calculated on a finer mesh, are displayed on the original mesh.

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 time step 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 time step, the depths for each time step, and the velocities for each time step.

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 limit specified on the **DTL** card does NOT represent a depth below or above which the node is dry or wet, rather the value describes parameters that control the shock capturing and stability parameters applied within AdH. Nodes in AdH are considered dry when the depth at the node falls below 0.0, and wet when the depth is greater than 0.0 . The wetting and drying level is given in the boundary condition file with the **MP DTL** card in units of *Length*.

MP DTL 0.01

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.

A wetting/drying fraction can be included in the boundary conditions file to help speed computation time by telling AdH to remove elements from the calculations. The values on the **MP FRC** card are the material id number and the fraction of the element area that is wet such that when the wet area drops equal to or below this fraction, the element will be considered dry and no longer in the computations.

MP FRC 1 0.1

A value of 0.1 means that when the element's wet area has dropped to only 10 percent, and 90 percent of the area is dry, the equations will no longer be solved for this element. When 10 percent or more of the element's area is wet the element will remain in the computations. The fraction is set by material so the user can vary this limit throughout the model domain. The default value is 0.05 such that when the FRC card is not included elements are removed from computations when 5 percent of the element is dry. This fraction can sacrifice mass conservation, so the fraction should be set to zero when testing conservation.

Wind Specification

Wind can be applied over the model domain multiple ways and will be discussed in more detail in a later section of this manual. However, alternative formulations for different materials within AdH can be used by including the MP WIND STR control card and specifying different wind stress computations for different areas of the domain. The default calculation of wind shear utilizes the Wu transformation. The following example tells AdH to use two different wind stress calculations for the two different material types. The Wu

transformation is specified by “1” and the Teeter method by “2”. If wind stresses are supplied to the model then a “0” transformation option can be specified meaning that no computations will be performed to convert the supplied data.

MP WND STR 1 2
MP WND STR 2 1

(Note: the MP WND 2 1 is redundant, since the default calculation for material 2 is Wu).

Wind Attenuation

Wind is applied over the entire domain based on time series data to be described later. However, a wind attenuation card is available to modify the wind stress by material. This is a means to account for structures or vegetation affecting the wind that are not included in the model’s definition. The wind attenuation card, **MP WAT**, references a material number and gives a fraction between 0 and 1. The default is 1.0 such that the wind is applied fully. As the fraction decreases, less of the wind stress is applied for the specified material region.

Material property cards

MP EVS

CONSTANT 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}

MP EEV

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 ≤ coef ≤ 1	Coefficient (default = 0.5)
5	int	1, 2, or 3	1 for isotropic (legacy) formulation, 2 for anisotropic formulation, 3 for Smagorinsky.

MP MU

KINEMATIC MOLECULAR VISCOSITY

Field	Type	Value	Description
1	char	MP	Card type
2	char	MU	Parameter
3	real	≥ 0	Uniform background viscosity (kinematic molecular Viscosity, units L^2/T)

MP COR

CORIOLIS LATITUDE

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

MP G

GRAVITATIONAL ACCELERATION

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

MP RHO

DENSITY

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

MP MUC

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 for SI standard)

MP DTL

WETTING/DRYING LIMITS

Field	Type	Value	Description
1	char	MP	Card type
2	char	DTL	Parameter
3	real	≥ 0.0	Depth below which stability parameters are included, default is 0.0

MP FRC

WETTING/DRYING FRACTION

Field	Type	Value	Description
1	char	MP	Card type
2	char	FRC	Parameter
3	int	≥ 1	Material type ID number
4	real	≥ 0.0	Fraction of element area that is wet at which the element is removed from the computations, default is 0.5

MP DF

TURBULENT 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	Turbulent diffusion rate

MP WND

WIND STRESS

Field	Type	Value	Description
1	char	MP	Card type
2	char	WND	Parameter
3	char	STR	Parameter
3	int	≥ 1	Material type ID number
4	real	0,1,2	Wind transform (0 = no transform, 1 = Wu, and 2 = Teeter)

MP WAT

WIND ATTENUATION

Field	Type	Value	Description
1	char	MP	Card type
2	char	WAT	Parameter
3	int	≥ 1	Material type ID number
4	real	0 ≤ #coef ≤ 1	Fraction applied to wind stress (default is 1.0)

Material meshing cards

MP ML

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

MP SRT

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 refinement terms

MP TRT

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 refinement terms

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   1   10.0                                                    generic
CN SND   2   0.4  0.00044  2.65  0.3                                 sand
CN CLA   3   1.0  0.00001  2.65  1400.0  0.02  0.00018  0.015  0.00016  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. If uncertain about how to set this parameter, it should be set equal to 1.0. Transport will be described in more detail in a later section. The results for each constituent are saved according to the user specified options in a file entitled *filename_con#.dat* where # is the constituent ID number.

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. The development of the method is given by Bernard (1992). Vorticity is activated voluntarily with the **VOR** card. The vorticity is included as a transport constituent due to its constituent like behavior as it moves within the model and must therefore be included in the **OP TRN** card.

Including **CN VOR** in the boundary condition file enables the bendway correction. 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 semi-empirical coefficients determined by comparisons against measured values. AdH uses 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 (Salinity and temperature)

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 when **SAL** or **TMP** cards are used.

Here is an example of salinity transport using a reference concentration of 35.

CN SAL 1 35.0

Constituent cards

CN CON

ANY GENERIC CONSERVATIVE 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 concentration

CN VOR

VORTICITY TRANSPORT - BENDWAY CORRECTION

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 which sets $A_s = 5.0$
6	real	≥ 0	D_s term, default is 0.0 which sets $D_s = 0.5$

CN SAL

SALINITY (BAROCLINIC TRANSPORT)

Field	Type	Value	Description
1	char	CN	Card type
2	char	SAL	Parameter
3	int	≥ 1	The constituent ID number
4	real	> 0	Reference concentration

CN TMP

TEMPERATURE (BAROCLINIC TRANSPORT)

Field	Type	Value	Description
1	char	CN	Card type
2	char	TMP	Parameter
3	int	≥ 1	The constituent ID number
4	real	> 0	Reference concentration

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 five types of boundary strings:

- Node
- Edge
- Mid
- Material
- Ice

Complete strings are input on multiple cards with one node, edge, or material per card. Cards may be input in any order and cards for different node strings may be interspersed. The string number will group the items together such that the same boundary conditions will be applied to all items on a specified string.

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#*).

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 edge, and then the string number.

Mid strings

Flow outputs (flux computations) 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 and requires that elements exist on both sides of the string.

Material strings

These strings are 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 from the mesh file and the string number. The string number does not have to be identical to the mesh material number.

Ice strings

ADH includes a friction option that is appropriate to account for the effects on the flow due to stationary ice on the water surface. The ice covered area is applied using an imposed pressure field and can be defined by material or by a circular area. To define the ice covered area by a circle, an ice string must be defined. The **INS** card is followed by the x

and y coordinates for the center of the circle, the radius of the circle, and then the string number.

```
NDS     1037     2
EGS     1601     1603     3
MTS     1        4
INS     100.0  50.0  10.0  5
```

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.

Boundary string cards

NDS

NODE STRINGS

Field	Type	Value	Description
1	char	NDS	Card type
2	int	≥ 1	ID number of a node with a Dirichlet condition
3	int	≥ 1	String ID number

EGS

EDGE STRINGS

Field	Type	Value	Description
1	char	EGS	Card type
2	int	≥ 1	ID number of the first node of an edge element
3	int	≥ 1	ID number of the second node of an edge element
4	int	≥ 1	String ID number

MDS

MID STRINGS

Field	Type	Value	Description
1	char	MDS	Card type
2	int	≥ 1	ID number of the first node of an edge element
3	int	≥ 1	ID number of the second node of an edge element
4	int	≥ 1	String ID number

MTS

MATERIAL STRINGS

Field	Type	Value	Description
1	char	MTS	Card type
2	int	≥ 1	Material type ID number
3	int	≥ 1	String ID number

INS

ICE STRINGS

Field	Type	Value	Description
1	char	INS	Card type
2	real	#	X coordinate for center
3	real	#	Y coordinate for center
4	real	≥ 0	Radius
5	real	≥ 1	String ID number

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. Therefore the time increment on the time series does not have to match the time step being taken by the model. 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**, **DSS,** and **OS** (details given in the output control section) but rather a time series one, time series two, time series three, and time series four.

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.

```
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

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

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, and 4 = weeks. These unit specifications only refer to the time column. They do not impact the data in the additional data columns and these values will always reference time in seconds. 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.

Wind series

Typically, AdH includes wind stresses by user-specification of some number of wind stations within the boundary condition file. When applying wind data to the model run, the wind stresses/velocities 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 the x and y coordinate of the station.

Some situations arise, however, when wind data are given at many locations, even at every finite element node. For such cases, time-series specification within the BC file can be cumbersome. In this scenario, it is better to use a file listing the wind grid and associated

stresses/velocities at each time-snap data are given. AdH will interpolate the wind grid onto the AdH finite element mesh on each read. More specifically, wind data must be supplied via a *.winds file structured as follows:

"project name".winds

```
  np
x(0)  y(0)
x(1)  y(1)
…
t(0) unit_flag
Dx(0)  Dy(0)
Dx(1)  Dy(1)
…
t(1) unit_flag
Dx(0)  Dy(0)
Dx(1)  Dy(1)
…
  ……..
```

In the above file description, np is the total number of wind grid point locations, {x(i),y(i)} are the locations of the i-th wind grid point, t(i) is the wind snap time and {Dx(i),Dy(i)} are winds stresses/velocities at the i-th grid point location. The variable unit_flag is used to specify what units t(i) are given in (i.e. secs=0, mins=1, hours=2, days=3, weeks=4).

For AdH to read this file, an additional control card is used, **TC FIN WND**, so that AdH knows to look for the wind file.

There are two methods that are currently included for wind stress calculations in AdH. The first is the Wu method, which is typically used for deep waters, while the second, the Teeter method, is used for very shallow water. The default calculation of wind shear assumes wind velocities are given and utilizes the Wu transformation. Alternative formulations for different materials within AdH can be used by including the **MP WND STR** control card.

The wind components are then either read by file or provided for individual stations. If supplying the wind at individual stations, the wind terms are 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 at each time given (the x and y-component of the wind term). These values can be in English or SI units and AdH will convert them if necessary for the wind stress coefficient equations to be accurate. An example using both the Wu and Teeter transform (two materials) is as follows such that material 1 will use the Wu method, material 2 will use the Teeter method, the data is located at the coordinate 5, 15, and the values (wind velocities since Wu and Teeter are selected) over time are given in the XY2 series for 0 to 2000 seconds:

```
MP WND STR  1  1
MP WND STR  2  2

XYC   4   5.0    15.0
XY2   4   3   0   0
0.0         0.1     0.1
100.0     0.25    0.2
2000.0    0.25    0.2
```

The Wu and Teeter methods determine the wind stress in the x and y directions from the wind speed and direction by the following formulation.

$$T_{wx} = \rho_a * C * (W^2 \cos\theta)$$
$$T_{wy} = \rho_a * C * (W^2 \sin\theta)$$

Where ρ_a = air density, W = wind speed in L/T (m/s, ft/s) at 10 meter height, Θ = wind direction measured counter-clockwise from East. C is a wind stress coefficient defined according to the method used. The equation below is for the Wu method (Wu, 1982) where W is the magnitude of the wind speed in m/s.

$$C = (0.8 + 0.065 * W) * 0.001$$

The Teeter method for computing wind shear stress gives a wind stress coefficient based on water depth and wind speed. *W* is the wind speed in meters per second and *W** is the wind speed limited to a minimum value of 5 m/s. The depth is given by *h* and *h** is a limited depth set to a minimum value of 2.001 meters. For more specifics on the development of the Teeter method for wind shear, see Teeter 2002.

$$C = \left(\frac{0.4}{16.11 - 0.5 * \ln(h) - 2.48 * \ln(W)} \right)^2 * \left(1.0 - \left(\frac{1.118}{\sqrt{W_*}} \right) \right) * e^{(-0.6(h_* - 2.0))}$$

DSS Time Series Input

AdH can read time series data in DSS format for input as well as save data in this format for output. Many users of AdH also model in the one-dimensional HEC tools. Data from these output can be read by AdH as input when the **DSS** cards are used. To read data from a DSS file, the DSS time series card must be used in the boundary conditions file. This time series is given a reference number just like all other time series data. This card allows the user to specify a DSS file name and a pathname from that file to use. The date in the pathname is ignored, and a time window is specified using two required time control cards: **TC DS0** and **TC DSF**. Parameters used with these two cards are dates and times, readable by the HECLIB Fortran functions, DATJUL and IHM2M, respectively. The DSS time series input is then referenced on a solution control card for discharge or other appropriate boundary condition. An example of the DSS time series card is given here.

DSS 7 BARKLEY / BARKLEY DAM/FLOW-RES OUT/OBSERVED

An example of how to set up the DSS parameters is provided at the end of this manual.

Times series cards

XY1

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 series
4	int	> 0	Input units. (0 = seconds; 1 = minutes; 2 = hours; 3 = days; and 4 = weeks)
5	int	> 0	Output units (0 = seconds; 1 = minutes; 2 = hours; 3 = days; and 4 = weeks)

XY2

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 series
4	int	> 0	Input units. (0 = seconds; 1 = minutes; 2 = hours; 3 = days; and 4 = weeks)
5	int	> 0	Output units (0 = seconds; 1 = minutes; 2 = hours; 3 = days; and 4 = weeks)

Currently, only the data that is to be used for wind series is to be input via the X-Y-Y series.

XYC

WIND STATION COORDINATES

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

DSS

DSS INPUT TIME SERIES DATA

Field	Type	Value	Description
1	char	DSS	Card type
2	int	≥ 1	ID number of the series
3	char	*	Filename (with or without extension)
4	char	*	Full path (spaces are allowed)

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. Note that a value of 0 is not a realistic Manning's 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, the effective bed roughness height, and 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.

Ice Friction

ADH has the ability to account for the effects that stationary ice on the water surface has on the flow below. The ice is applied as a pressure field on the water surface and appropriate functions are applied to determine the correct friction on the flow field. The ice covered area can be defined by material or by ice circles using the **INS** card. In the absence of **INS** card, the string number on the **FR ICE** card is defaulted to ice designation by material number. If the string number specified does not correspond to a material an error will result.

Once a string is defined, it is referenced on three additional cards: **FR ICE** card to give the ice thickness and density, **FR IRH** card to give the ice roughness height, and **FR BRH** card to give the bed roughness height. These cards define the necessary parameters for the friction library to accurately account for this type of roughness element.

Friction control cards

FR MNG

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	Manning's n

FR ERH

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

FR SAV

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	The effective roughness height of the SAV cover (e.g. 0.1 * the undeflected stem height)
5	real	≥ 0.0	Undeflected stem height

FR URV

UN-SUBMERGED RIGID VEGETATION

Field	Type	Value	Description
1	char	FR	Card type
2	char	URV	Parameter
3	int	> 0	String ID number
4	real	> 0.0	Bed Roughness Height (not including the stems)
5	real	> 0.0	Average stem diameter
6	real	> 0.0	Average stem density

FR ICE

ICE THICKNESS

Field	Type	Value	Description
1	char	FR	Card type
2	char	ICE	Parameter
3	int	> 0	String ID number
4	real	> 0.0	Ice thickness
5	real	> 0.0	Ice density
6	int	= 0	0 – stationary
		= 1	1 – moving ice (not implemented yet)

FR IRH

ICE ROUGHNESS

Field	Type	Value	Description
1	char	FR	Card type
2	char	IRH	Parameter
3	int	> 0	String ID number
4	real	> 0.0	Ice roughness height

FR BRH

BED ROUGHNESS HEIGHT

Field	Type	Value	Description
1	char	FR	Card type
2	char	BRH	Parameter
3	int	> 0	String ID number
4	real	> 0.0	Bed roughness height

SOLUTION CONTROLS

Solution development is controlled through the specification of the initial and boundary conditions as well as 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.

Friction

Bed

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

$$\tau = 1/2\rho C_f \parallel u \parallel u$$

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

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 roughness or skin friction must be input. One must also specify a boundary condition for this string via the **NB OVL** or the **NB DIS** 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** that states that there is flow through this wall – string number 2 – and that the specifier for the flow is given in time series number 6.

Upstream boundary

Typical subcritical inflow is to use an edgestring along the entrance and a natural boundary condition to specify flow or a node string across the entrance and a dirichlet boundary condition to specify the velocity components. For the following example, the DB OVL boundary condition 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.

```
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 similar to the previous 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

The downstream 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 of water surface elevation 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 are found in time series number 6.

A supercritical downstream boundary requires no boundary condition mathematically. However, the model needs to know where this flow is to be able to pass out of the model. To do this we define this string and define it with an **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 plan view 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.

As with inflow directions on the boundary, positive values are into the control volume (rain) and negative values are out (evaporation).

Stage Discharge

The **NB SDR** card specifies a stage-discharge boundary dependent on the user specified power relationship in the form:

$$Q = CoefE * [CoefA + CoefC * (WaterSurfaceElevation - CoefB)^{CoefD}]$$

This boundary condition option requires an edge string and relates the total flow to be applied along this edge to a stage elevation.

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.

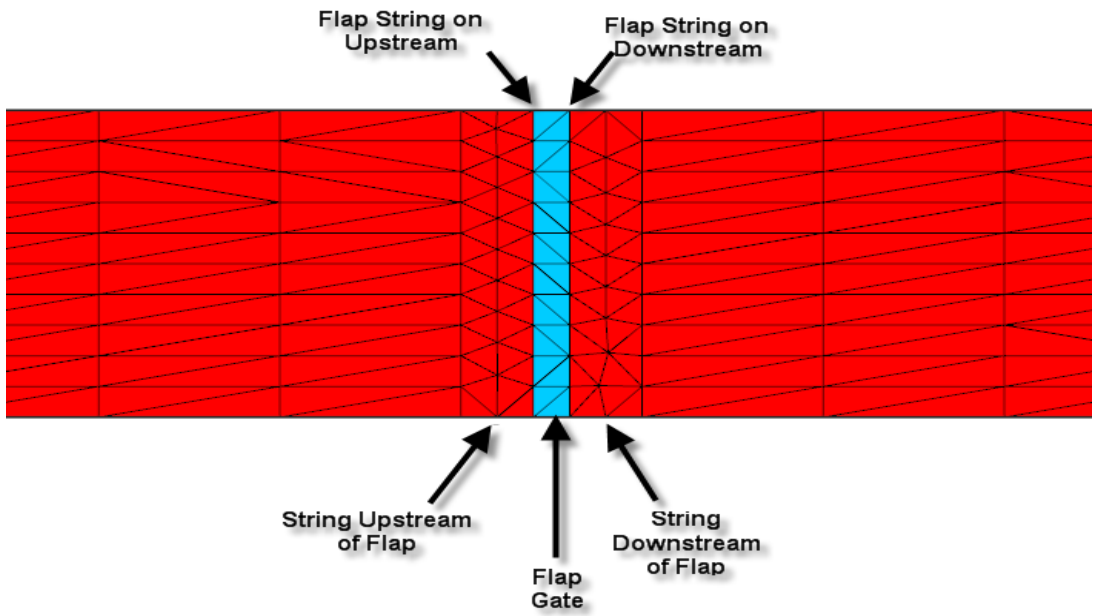
This method is implemented by selecting all the nodes that are to comprise the lid or pressure field and assigning them to a node string. This node string will then be assigned the lid elevation with a **DB LDE** card, the depth of the water with a **DB** L**DH** card, or the pressure (in terms of draft) 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.

Structures Library

AdH now has several boundary conditions to represent structures such as flap gates, weirs, and spillways. The **NB SPL** boundary can be specified when a free outflow is required but the flow is not supercritical at all computation times. The **WRS** card is used in conjunction with the **WER** where the number of weirs being modeled is given and then the definition of the wier is provided. A description of the weir implementation is provided in *Savant and Berger (2009)* and the image below shows the definitions required.

The weir card can also be used to specify a free flowing weir as a model boundary condition. To do this, the downstream reference string is defined as the same string as the upstream reference string and the downstream weir string is defined as the same string as the upstream weir string.

Flap gates are defined similarly to the weirs. The number of gates is given on the **FLP** card and the definition of each gate is given on the **FGT** card. A description of the necessary strings for both flap gates and weirs is given in the image below.



Flap gate flow is computed using a user defined polynomial rating curve in the form:

$$q = [CoefA * \Delta h]^{CoefB} + CoefC * \Delta h + CoefD$$

Where Δh= upstream water surface elevation – downstream water surface elevation

A total of 6 user specified coefficients are available such that a larger polynomial expression can be implemented, but at this time only four are used and the additional values are simply placeholders. The flap gate must be specified by a separate material and then that material must be turned **OFF**. To do this, use the **OFF** card followed by the flap gate string number.

In addition to typical structures that are often included in modeling, a breaching boundary condition is included in AdH. This card, **DB BCH**, is a dirichlet boundary condition such that values are assigned at particular nodes. This card is intended to assist engineers with realistic dam and levee breach simulations by allowing users to input time varying bed elevations to define the breach displacement. This condition does not applying any functions - it simply moves the bed at a node according to the given elevation. An additional output file is supplied when using the BCH card. The *projectname*_belev.dat file

gives the change in the bed elevation throughout the simulation in order to track the breach conditions.

Solution control cards

DB OVL

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

DB OVH

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

DB TRN

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 (units depend on the transport type)

DB LDE

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

DB LDH

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 depth

DB LID

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 of the lid

DB BCH

DIRICHLET – BREACH DISPLACEMENT

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 elevation of the string

NB DIS

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

NB OVL

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 data; for face strings the series values represent the flow per unit area (positive in); for edge strings the series values represent the flow per unit length (positive in)

NB OTW

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

NB TRN

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 (units dependent of the transport type)

NB SDR

STAGE DISCHARGE BOUNDARY

Field	Type	Value	Description
1	char	NB	Card type
2	char	SDR	Parameter
3	int	≥ 1	String ID number
4	dbl	≥ 0	Coefficient A
5	dbl	≥ 0	Coefficient B
6	dbl	≥ 0	Coefficient C
7	dbl	≥ 0	Coefficient D
8	dbl	≥ 0	Coefficient E

NB SPL

NATURAL BOUNDARY CONDITION - SPILLWAY

Field	Type	Value	Description
1	char	NB	Card type
2	char	SPL	Parameter
3	int	≥ 1	String ID number (edge)
4	int	≥ 1	Series ID number that contains the time series of the Percent (%) flow out.

OB OF

OUTFLOW BOUNDARY

Field	Type	Value	Description
1	char	OB	Card type
2	char	OF	Parameter
3	int	≥ 1	String ID number (edge)

OUT

FLOW OUTPUT FROM INSIDE THE GRID

Field	Type	Value	Description
1	char	OUT	Card type
2	int	≥ 1	Outflow edge string
3	int	≥ 1	Inflow edge string
4	int	≥ 1	Series of outflow

WER

NUMBER OF WEIRS

Field	Type	Value	Description
1	char	WER	Card type
2	int	≥ 1	Number of weirs

WRS

WEIR PARAMETERS

Field	Type	Value	Description
1	char	WRS	Card type
2	int	≥ 1	Weir Number
3	int	≥ 1	String upstream of weir
4	int	≥ 1	String downstream of weir
5	int	≥ 1	Weir string on upstream
6	int	≥ 1	Weir string on downstream
7	real	≥ 0	Length of wir
8	real	≥ 0	Weir crest elevation
9	real	≥ 0	Weir height

FLP

NUMBER OF FLAP GATES

Field	Type	Value	Description
1	char	FLP	Card type
2	int	≥ 1	Number of flap gates

FGT

FLAP GATE PARAMETERS

Field	Type	Value	Description
1	char	FGT	Card type
2	int	≥ 1	Flap gate number
3	int	= 1	1 – User specified parameters
		= 2	2 – Automatic computation
4	int	≥ 1	String upstream of flap
5	int	≥ 1	String downstream of flap
6	int	≥ 1	Flap string on the upstream
7	int	≥ 1	Flap string on the downstream
8	real	≥ 0	Coefficient A
9	real	≥ 0	Coefficient B
10	real	≥ 0	Coefficient C
11	real	≥0	Coefficient D
12	real	≥0	Coefficient E
13	real	≥0	Coefficient F
14	real	≥0	Length of flap gate

TIME CONTROLS

Evolution of the solution is determined by a group of cards with the Time Control specifier **TC**. The start time is specified on a **TC T0** card. The **TC IDT** references a time series in the third field that will control the time step size. The time step size is always given in seconds regardless of the units of the time variability of the series. The final time, at which the run will terminate, is specified on a **TC TF** card. The final time does not have to equal the largest value in the time series. The units for the start and end time controls can be individually set using a certain flag after the time values (0 = seconds; 1 = minutes; 2 = hours; 3 = days; and 4 = weeks). 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.

Time steps will be reduced if the model fails to converge for the current time step. The time step is dropped to ¼ the previous time step size with each failure. When a reduced time step size does converge, however, the time step size will double until it reaches the value specified in the appropriate time series. This is the suggested method to use:

TC T0 0.0
TC IDT 5
TC TF 2000.0

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, **DT**, is determined using the switched evolution relaxation (**SER**) strategy where

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

R^n refers to the spatial residual for time step n. The initial time step size is the first entry on the **TC STD** card, followed by the maximum timestep size desired for the simulation. 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. 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.**

Automatic Time Step Determination

This Automatic time step computation option is invoked using the **TC ATF** card. For time step determination AdH utilized the property of the continuity equation that the spatial and temporal terms must sum to '0'. The gradient in the spatial terms is an indication of the temporal gradient and is utilized to determine the time step for the present computation step. The automatic time step calculation is dependent upon the initial time step (user specified), an explicit time step is a good choice and is computed as:

$$\delta_0 = \Delta x / \sqrt{gh}$$

Where:

Δx = mesh element length
g = gravitational acceleration
h = initial water depth.

A detailed description of the automatic time step find option is provided in *Savant et al. (2010)*.

The **TC ATF** can be used for **both hydrodynamics and transport. TC ATF** and **STD can not be utilized at the same time.**

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 maximum allowable sediment time step. If the hydro time step exceeds this, then multiple sediment time steps will be taken.

If the model does not converge and the time step is cut, the sediment time step will also be cut. If it is desired to have the sediment take the same time step as specified with the **TC IDT** card, this card should be omitted.

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 omitted. The **IP SST** card is used to specify the tolerance for quasi-unsteady convergence, or the

convergence tolerance for the hydrodynamics if a different tolerance is desired for the hydrodynamic and sediment computations. 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.

DSS Input Time Control

The DSS input data as specified in the Solution Controls section requires two time control cards to provide the time window of the DSS data to use. Multiple DSS input files can be used but all will use the same time window as specified on the **TC DS0** and **TC DSF** cards which give the start and end dates and times in DSS format.

TC DS0 03-01-1997 1200
TC DSF 03-11-1997 1200

Time control cards

TC T0

STARTING TIME

Field	Type	Value	Description
1	char	TC	Card type
2	char	T0	Parameter
3	real	> 0	Starting time of the model
4	int	#	Units (optional; 0 = seconds, 1 = minutes, 2 = hours, 3 = days, 4 = weeks)

TC TF

FINAL TIME

Field	Type	Value	Description
1	char	TC	Card type
2	char	TF	Parameter
3	real	> 0	Ending time of the model
4	int	#	Units (optional; 0 = seconds, 1 = minutes, 2 = hours, 3 = days, 4 = weeks)

TC IDT

TIME STEP SIZE

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

TC SDI

SEDIMENT TIME STEP CONTROL

Field	Type	Value	Description
1	char	TC	Card type
2	char	SDI	Parameter
3	real	> 0	The maximum allowable sediment time step. If the hydro time step exceeds this, then multiple sediment time steps will be performed

TC STD

STEADY STATE SIMULATION

Field	Type	Value	Description
1	char	TC	Card type
2	char	STD	Steady state option turned on - omit to run dynamic simulation
3	real	> 0	Initial time step
4	real	> 0	Maximum time step

TC STH

QUASI-UNSTEADY SIMULATION

Field	Type	Value	Description
1	char	TC	Card type
2	char	STH	Parameter
3	int	> 0	Series ID number containing the steady state hydrodynamic condition
4	int	> 0	Maximum number of iterations for steady state solve
5	real	> 0	Initial time step size for steady state calculation

TC ATF

AUTO TIME STEP FIND

Field	Type	Value	Description
1	char	TC	Card type
2	char	ATF	Parameter
3	real	> 0	Initial time step size
4	int	> 0	Series ID number containing the maximum time step size

TC DS0

DSS STARTING TIME

Field	Type	Value	Description
1	char	TC	Card type
2	char	DS0	Parameter
3	char	*	Start date in format readable by DATJUL
4	char	*	Start time in format readable by IHM2M

TC DSF

DSS ENDING TIME

Field	Type	Value	Description
1	char	TC	Card type
2	char	DSF	Parameter
3	char	*	End date in format readable by DATJUL
4	char	*	End time in format readable by IHM2M

OUTPUT CONTROL

An **OC** card causes the solution to be printed at startup and at each specified time step. The **OC** card references a time series that controls the output data. The solutions are output as data set files at the solution for the closest time equal to or greater than each time requested on the output control time series. Since AdH can vary the timestep size throughout the simulation, the times requested for output may not match those for which a solution is obtained. This is why the exact time requested may not appear in the solution 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.

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 time step.

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*_tflx file. This file format is a table of values giving time, string #, flux value, and water surface values. It is not an output file read by SMS or GMS. When simulating transport of any kind, the concentration flux is also saved with the inclusion of the FLX card. The results of these computations for all transported constituents are in the *filename*_conflx file.

OC 3
FLX 2
END

Auto-build output series

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 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 **OS** card time output units). The unit specifications are as follows: 0 = seconds, 1 = minutes, 2 = hours, 3 = days, and 4 = weeks. 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   77.90   5.0   0
1.5   6.0     2.0   1
7.0   9.0     2.0   0
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                0.0
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. It has been found that memory limits can be reached when applying a small output increment over a long time period. If pre_AdH fails with a memory limit at this location, modifying the total number of output times in the output series is likely to fix the problem.

All strings are included in the calculations by default. You can turn off a string if it will not be included in the 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.

When including adaption in the simulation it may be desired to see the adapted meshes. To request that the meshes are saved with the associated hydrodynamic solution files, the **PC ADP** card can be included in the boundary condition file. By including this card, the mesh and associated solution files will be saved at the time step 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.

PC ELM card can be included to activate the printout in TecPlot format of numerical fish surrogate information. If PC ELM is turned on the OP NF2 card must be included in the boundary conditions file. Ommission will result in catastrophic failure during ADH runtime.

Standard Screen Output

The standard screen output is in a tab delimited format. The details will be given in a later section on “Running AdH”. However, if the user chooses, additional formats of the standard output can be obtained by including a **PC LVL** card in the boundary condition file. The screen output is always given in seconds.

DSS Output

AdH can store solution data in DSS format for specified node strings. To get a DSS file output from AdH, the user must include the **PC DSS** card and declare a node string that designates points of interest on the input mesh. This card and the node string are linked together, allowing for all data at each node in the node string to be printed to the DSS file at each output time. This creates an AdH output DSS file that shares the project name of the simulation being run.

MEO Output

AdH can output the mass error to the screen. To get the mass error output to the screen from AdH, the user must include the **PC MEO** card and set it to 1. This will enable the screen output of the mass error. By default this output is turned off.

Output control cards

OC

OUTPUT

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

OS

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 series
4	int	> 0	Output units (Units 0 = seconds, 1 = minutes, 2 = hours, 3 = days, 4 = weeks)

FLX

FLOW OUTPUT

Field	Type	Value	Description
1	char	FLX	Parameter
2	int	> 0	String ID number for the mid string or edge string for which flow is to be output

OFF

DEACTIVATE STRING

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

PC ADP

ADAPTED MESH PRINTING

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

PC LVL

SCREEN OUTPUT FORMAT

Field	Type	Value	Description
1	char	PC	Card type
2	char	LVL	Screen output format – level 0 is default
3	int	0, 1, 2	0 gives short column format; 1 gives long column format; 2 gives original AdH format (non-tabular)

PC HOT

HOTSTART PRINT CONTROL OUTPUT

Field	Type	Value	Description
1	char	PC	Card type
2	char	HOT	Hotstart print control turned on - omit to turn off
3	int	> 0	Number of time step increments between saving hot start file

PC ELM

NUMERICAL FISH SURROGATE OUTPUT

Field	Type	Value	Description
1	char	PC	Card type
2	char	ELM	Numerical Fish Surrogate output (optional card)

PC DSS

DSS PRINT CONTROL OUTPUT

Field	Type	Value	Description
1	char	PC	Card type
2	char	DSS	DSS output format from AdH simulation
3	int	> 0	Node string ID where DSS data is desired

PC MEO

MEO PRINT CONTROL OUTPUT

Field	Type	Value	Description
1	char	PC	Card type
2	char	MEO	MEO output from AdH simulation
3	int	0, 1	0 is the default (does not print), 1 prints the mass error output to screen

END

STOPPING THE MODEL

Field	Type	Value	Description
1	char	END	Close the model

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 in a single simulation.

The sediment is transported separately as suspended load and bed 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. SND stands for sand. Here's an example:

```
CN SND  1  1.0  1.E-4  2.65  0.3
CN SND  2  1.0  5.E-4  2.65  0.3
CN SND  3  1.0  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 reference concentration, like all sediment concentrations given in AdH, is in units of micromass per unit mass. AdH actually operates on the dimensionless units but the input/output is easier if multiplied by 1.E+6. AdH sediment transport is also based on metric units, so grain diameter must be given in meters.

Next, describe the bed layer structure. Each layer must be described by beginning with the lowest layer and then working the 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 two 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 three 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. Material and/or

node can now be designated and these nodes would be modified to reflect this difference. Note that it is often desireable to have several layers of zero thickness specified at the bed surface (i.e. the highest layer numbers). These are used by the model as depositional layers, each one storing information from discrete depositional events, and thereby generating new strata in the bed.

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, and the constituent number for the grain being applied.

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 sediment bed than a property of the individual grains. Therefore, the user must specify the erosion characteristics of the cohesive bed. In AdH, these erosion characteristics are governed by the following equation:

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

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 (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.

The parameters given on the **CN CLA** card are:

- Grain diameter
- Specific gravity
- Bulk density
- Critical shear stress for erosion (τ_c)
- Erosion rate constant (M)
- Critical shear stress for deposition
- 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).

```
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 is in units of micromass per unit mass, *ppm*. 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
- Critical shear stress for erosion (τ_c)
- Erosion rate constant (M)
- 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
```

As with sand beds, the cohesive bed can be defined by all nodes (CBA), specific materials (CBM), or specific nodes (CBN). As with the previous bed definitions, the layer numbering begins with the bottom-most layer.

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 bed surface 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. Below is 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
```

Sediment output files

When running the transport functions in AdH, much more data is saved and output than when running just the hydrodynamics. In addition to the water depth, velocity, and residual error files, concentration files are provided for each transported quantity along with solution files for bed displacement, bedload vector, and bed shear stress. The bed layer thickness for each bed layer and the active layer as well as the grain distribution for each bed layer and active layer are saved. Cohesive bed properties are also saved for each bed layer. An additional file is saved containing the sediment mass residual information for use in mass balance computations.

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 the control cards. 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 turbulent diffusion is set on the **MP DF** card, but if an estimated eddy viscosity (EEV) card is used then it will override the diffusion set on the **MP DF** card for diffusion of transported material. The refinement tolerance is set on the **MP TRT** card.

Material property cards

MP NBL

NUMBER OF SEDIMENT BED LAYERS

Field	Type	Value	Description
1	char	MP	Card type
2	char	NBL	Parameter
3	int	≥ 0	Number of bed layers for sediment transport (layer number begins with the bottom-most layer)

MP SBN

SEDIMENT BED INITIALIZATION 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 to start
5	int	> 0	The node number at which to end
6	real	≥ 0.0	The bed layer thickness
7	real	≥ 0.0	The grain class fraction for the first sediment
8	real	≥ 0.0	The grain class fraction for the second sediment
#	real	≥ 0.0	The grain class fraction for the final sediment

MP SBA

SEDIMENT BED INITIALIZATION 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 grain class fraction for the first sediment
6	real	≥ 0.0	The grain class fraction for the second sediment
#	real	≥ 0.0	The grain class fraction for the final sediment.

MP SBM

SEDIMENT BED INITIALIZATION 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 grain class fraction for the first sediment
7	real	≥ 0.0	The grain class fraction for the second sediment
#	real	≥ 0.0	The grain class fraction for the final sediment

MP CBN

COHESIVE BED SEDIMENT PROPERTIES 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 to start
5	int	> 0	The node number at which to end
6	real	≥ 0.0	The bulk density
7	real	≥ 0.0	The critical shear stress for erosion
8	real	≥ 0.0	The erosion rate constant
9	real	≥ 0.0	The erosion rate exponent

MP CBM

COHESIVE BED SEDIMENT PROPERTIES 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 erosion
7	real	≥ 0.0	The erosion rate constant
8	real	≥ 0.0	The erosion rate exponent

MP CBA

COHESIVE BED SEDIMENT PROPERTIES APPLIED TO ALL NODES

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 erosion
6	real	≥ 0.0	The erosion rate constant
7	real	≥ 0.0	The erosion rate exponent

MP NCP

CONSOLIDATION TIME SERIES SPECIFICATION

1	char	MP	Card type
2	char	NCP	Parameter
3	int	≥ 0	Number of time values in the consolidation time series for sediment transport

MP CPN

CONSOLIDATION TIME SERIES PROPERTIES APPLIED TO SELECTED NODES

1	char	MP	Card type
2	char	CPN	Parameter
3	int	> 0	Consolidation time value number
4	int	> 0	The node number from which to start
5	int	> 0	The node number at which to end
6	real	≥ 0.0	Elapsed time since sediment deposition (sec)
7	real	≥ 0.0	The bulk density
8	real	≥ 0.0	The critical shear stress for erosion
9	real	≥ 0.0	The erosion rate constant
10	real	≥ 0.0	The erosion rate exponent

MP CPM

CONSOLIDATION TIME SERIES PROPERTIES APPLIED BY MATERIAL

1	char	MP	Card type
2	char	CPM	Parameter
3	int	> 0	Consolidation time value number
4	int	> 0	Material type ID number
5	real	≥ 0.0	Elapsed time since sediment deposiiton (sec)
6	real	≥ 0.0	The bulk density
7	real	≥ 0.0	The critical shear stress for erosion
8	real	≥ 0.0	The erosion rate constant
9	real	≥ 0.0	The erosion rate exponent

MP CPA

CONSOLIDATION TIME SERIES PROPERTIES APPLIED BY LAYER

1	char	MP	Card type
2	char	CPA	Parameter
3	int	> 0	Consolidation time value number
4	real	≥ 0.0	Elapsed time since sediment deposiiton (sec)
5	real	≥ 0.0	The bulk density
6	real	≥ 0.0	The critical shear stress for erosion
7	real	≥ 0.0	The erosion rate constant
8	real	≥ 0.0	The erosion rate exponent

MP NDM

TURN OFF DISPLACEMENT BY MATERIAL TYPE

1	char	MP	Card type
2	char	NDM	Parameter
3	int	> 0	Material type ID number

CN CLA

COHESIVE SEDIMENT (CLAY AND/OR SILT)

Field	Type	Value	Description
1	char	CN	Card type
2	char	CLA	Parameter
3	int	≥ 1	The constituent ID number
4	real	> 0	Characteristic concentration
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 deposition
11	real	> 0	Free Settling velocity

CN SND

COHESIONLESS SEDIMENT (SAND)

Field	Type	Value	Description
1	char	CN	Card type
2	char	SND	Parameter
3	int	≥ 1	The constituent ID number
4	real	> 0	Characteristic concentration
5	real	> 0	Grain diameter
6	real	> 1	Specific gravity
7	real	> 0	Grain porosity

Sediment Process cards

The sediment process cards are designed to allow the user to select among various methods of describing a specific process. Since some methods require additional parameters, the fields 4 and above are reserved for any parameter that may be required for a specific method.

SP CSV

SEDIMENT PROCESS: COHESIVE SETTLING VELOCITY

Field	Type	Value	Description
1	char	SP	Card type
2	char	CSV	Parameter
3	int	= 0	0 - Free Settling
		= 1	1 - Hwang and Mehta
4…	real	≥ 0	Process specific parameter(s)

SP WWS

SEDIMENT PROCESS: WIND WAVE SHEAR

Field	Type	Value	Description
1	char	SP	Card type
2	char	WWS	Parameter
3	int	= 0	0 – No applied wind-wave stress
		= 1	1 - Grant and Madsen
		= 2	2 - Teeter
4…	real	≥ 0.0	Process specific parameter(s)

SP NSE

SEDIMENT PROCESS: NONCOHESIVE SUSPENDED ENTRAINMENT

Field	Type	Value	Description
1	char	SP	Card type
2	char	NSE	Parameter
3	int	= 0	0 - Garcia-Parker
		= 1	1 - Wright-Parker
		= 2	2 – Van Rijn
4…	real	≥ 0.0	Process specific parameter(s)

SP NBE

SEDIMENT PROCESS: NONCOHESIVE BEDLOAD ENTRAINMENT

Field	Type	Value	Description
1	char	SP	Card type
2	char	NSE	Parameter
3	int	= 0	0 – Van Rijn
		= 1	1 – Meyer Peter Mueller
		= 2	2 – Meyer Peter Mueller with Wong Parker Correction
4…	real	≥ 0.0	Process specific parameter(s)

SP HID

SEDIMENT PROCESS: NONCOHESIVE HIDING FACTOR

Field	Type	Value	Description
1	char	SP	Card type
2	char	NSE	Parameter
3	int	= 0	0 – Karim Holly Yang
		= 1	1 - Egiazaroff
4…	real	≥ 0.0	Process specific parameter(s)

Solution control cards

The solution control cards for sediment boundary conditions include the **NB TRN** and **DB TRN** card that apply to any transport quantity as well as the **EQ TRN** which is described below.

EQ TRN

EQUILIBRIUM SAND 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	int	≥ 0	placeholder

Running AdH

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. The commands are:

pre_adh filename
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.

The standard output for AdH is in a tab delimited format so that it can be manipulated by the user in many different ways. The order of the data in the short column tabular form (**PC LVL** 0) from left to right is time, time step size, percent completion progress, nonlinear iteration number, linear iteration count, node number giving the maximum residual, node number giving the maximum increment norm, node count after adaption, failure flag. The order of the data in the long column tabular form (**PC LVL** 1) from left to right is time, time step size, percent completion progress, nonlinear iteration number, linear iteration count, maximum residual norm, node number giving the maximum residual, x, y, and z-coordinates of this worst node, maximum increment norm, node number giving this maximum increment, x, y, and z-coordinates of this worst node, node count after adaption, failure flag. All time values in the screen output are in seconds. The maximum residual norm is used to determine convergence against the **NTL** value. The maximum increment norm is used to determine convergence against the **ITL** value, if included in the boundary conditions file. If no adaption is taking place, the node count after adaption will not change throughout the run. The failure flag is the **#** symbol and indicates that convergence did not occur and the time step will be cut to ¼ the previous value. This column is left empty in all other instances. For transport iterations, the same information is provided but preceded by a line indicating that the data to follow is from the transport computations.

After the model is run, GMS or SMS can be used to visualize the results. The depth, velocity, and error files are the minimum output files that will be produced in any simulation. Other output files will be generated depending on the options requested in the boundary conditions file.

Here is the complete list of the output files:

Output filename conventions (*.dat)

***_dep.dat**	overland head (scalar, depth)
***_ovl.dat**	overland velocity (vector)
***_err.dat**	normalized residual error for use in setting refinement parameters (scalar)
***err_hydro.dat**	non-normalized residual error for the hydro (scalar)
***_con#.dat**	constituent concentration, # = constituent number (scalar, parts per million for sediment), rouse profile factor, bedload mass per unit area
***err_con#.dat**	non-normalized residual error for the transport constituent (scalar)
***_dpl.dat**	bed displacement (scalar, meters)
***_alt.dat**	active layer thickness (scalar, meters)
***_ald.dat**	active layer distribution (scalar, one column for each grain class)
***_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 class)
*_**cbp#.dat**	cohesive bed property, # = layer number (scalar, 1 is the bottom-most layer, one column for each grain class)
*_**bsh.dat**	bed shear stress magnitude (scalar)
***_smr.dat**	sediment mass residual (kg/m^2)
*_**bedload.dat**	bedload (vector, $kg/m^2/s$)
***_belev.dat**	breach bed elevation (scalar, length)
***_conflx**	concentration flux across a string for each constituent is included when the FLX card is used followed by the string number...this is NOT an SMS file

***_tflx**	hydrodynamic flux across a string is included when the FLX card is used followed by the string number…this is NOT an SMS file
***_ugd**	u-velocity gradients is included when the OP NF2 card is used
***_vgd**	v-velocity gradients is included when the OP NF2 card is used
***_ELAM**	**numerical fish surrogate format output when OP NF2 and PC ELM are specified**

For a 3-constituent simulation (1 non-sediment constituent and 2 sediment constituents) of 2 grains and 3 bed layers, the sediments are constituent 2 and 3, information in parenthesis gives names for the hotstart file.

***_dep.dat** (ioh)	Depth value
***_ovl.dat** (iov)	X_vel, Y_vel, Z_vel (Z_vel = 0 for 2D)
***_err.dat**	Residual error
***_err_hydro.dat**	Hydro residual error
***_con1.dat** (icon 1)	Concentration 1
***_con2.dat** (icon 2)	Concentration 2, Rouse factor*, bedload mass per unit area
***_con3.dat** (icon 3)	Concentration 3, Rouse factor, bedload mass per unit area
***_err_con1.dat**	Transport residual error for transport 1
***_err_con2.dat**	Transport residual error for transport 2
***_err_con3.dat**	Transport residual error for transport 3
***_dpl.dat** (id)	Displacement
***_alt.dat** (ialt)	Active layer thickness
***_ald.dat** (iald)	Ald-grain1, Ald-grain2
***_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
***_bld2.dat** (ibld 2)	Bld-grain1, Bld-grain2

***_bld3.dat** (ibld 3)	Bld-grain1, Bld-grain2
***_bsh.dat**	Bed shear magnitude
***_smr.dat**	Sediment mass residual grain 1, grain 2
***_bedload.dat**	Bedload_X, Bedload_Y

*Rouse factor is the ratio of the near-bed concentration to the depth averaged concentration.

Two-Dimensional Shallow Water Equations, Finite Element Formulation

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

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

Where

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

$$F_x = \begin{Bmatrix} uh \\ u^2 h + \frac{1}{2} g h^2 - h \dfrac{\sigma_{xx}}{\rho} \\ uvh - h \dfrac{\sigma_{yx}}{\rho} \end{Bmatrix}$$

$$F_y = \begin{Bmatrix} vh \\ uvh - h \dfrac{\sigma_{xy}}{\rho} \\ v^2 h + \frac{1}{2} g h^2 - h \dfrac{\sigma_{yy}}{\rho} \end{Bmatrix}$$

and

$$H = \begin{pmatrix} 0 \\ gh\dfrac{\partial Z_D}{\partial_x} + ghS_x \\ gh\dfrac{\partial Z_D}{\partial_y} + ghS_y \end{pmatrix}$$

where:

ρ	=	fluid density
g	=	gravitational acceleration
Z_D	=	bottom elevation
h	=	flow depth
u	=	x-component of velocity
v	=	y-component of velocity

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

$$\sigma_{xx} = 2\rho v \frac{\partial u}{\partial x}$$

$$\sigma_{xx} = \sigma_{yx} = \rho v \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)$$

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

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

$$S_x = \begin{pmatrix} u\dfrac{n^2\sqrt{u^2+v^2}}{C_D^2\sqrt[3]{h}} \\ \dfrac{C_f}{gh}u\sqrt{u^2+v^2} \end{pmatrix}$$

and

$$S_y = \begin{pmatrix} v\dfrac{n^2\sqrt{u^2+v^2}}{C_D^2\sqrt[3]{h}} \\ \dfrac{C_f}{gh}v\sqrt{u^2+v^2} \end{pmatrix}$$

where C_f = coefficient of friction; n = Manning's roughness coefficient; and C_D = 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:

$$Q(x,y,t) \approx \sum_j \Phi_j(x,y)Q_j(t)$$

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

$$M\frac{\partial_q}{\partial_t}+A\frac{\partial_q}{\partial x}+B\frac{\partial_q}{\partial_y}+h=0$$

$$q=\begin{pmatrix} h \\ u \\ v \end{pmatrix}$$

$$M=\begin{pmatrix} 1 & 0 & 0 \\ 0 & h & 0 \\ 0 & 0 & h \end{pmatrix}$$

$$A=\begin{pmatrix} u & h & 0 \\ c^2 & uh & 0 \\ 0 & 0 & uh \end{pmatrix}$$

$$B=\begin{pmatrix} v & 0 & h \\ 0 & vh & 0 \\ c^2 & 0 & vh \end{pmatrix}$$

$$and$$

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

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

$$\psi_i^T=\phi_i^T+\alpha(\Delta x\frac{\partial\phi_i^T}{\partial x}MP^{-1}\Lambda_x PM^{-1}+\Delta y\frac{\partial\phi_i^T}{\partial y}MR^{-1}\Lambda_y RM^{-1}$$

$$or$$

$$\psi_i^T=\phi_i^T+\varphi_i^T$$

where, α is a coefficient between 0 and 0.5,

$$\Delta x = 2\left[(\frac{\partial x}{\partial \xi})^2 + \frac{\partial x}{\partial \eta})^2 \right]^{\frac{1}{2}}$$

$$\Delta y = 2\left[(\frac{\partial y}{\partial \xi})^2 + \frac{\partial y}{\partial \eta})^2 \right]^{\frac{1}{2}}$$

(ξ, η are the local variables that have values between 0 and 1)

$$P = \begin{pmatrix} 0 & 0 & 1 \\ 1 & -\frac{c}{g} & 0 \\ 1 & \frac{c}{g} & 0 \end{pmatrix}$$

$$R = \begin{pmatrix} 0 & 0 & 1 \\ 1 & -\frac{c}{g} & 0 \\ 1 & \frac{c}{g} & 0 \end{pmatrix}$$

$$\Delta x = 2\left[(\frac{\partial_x}{\partial_\xi})^2 + (\frac{\partial_x}{\partial_\eta})^2 \right]^{\frac{1}{2}}$$

$$\Delta y = 2\left[(\frac{\partial_y}{\partial_\xi})^2 + (\frac{\partial y}{\partial_\eta})^2 \right]^{\frac{1}{2}}$$

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

The weak form finite element approximation is then

$$\sum_e \left[\int_{\Omega_e} (\psi_i^T \frac{\partial Q}{\partial t} - \frac{\partial \phi_i^T}{\partial x} F_x - \frac{\partial \phi_i^T}{\partial y} F_y + \psi_i^T (\frac{\partial F_x}{\partial x} + \frac{\partial F_y}{\partial y}) + \psi_i^T H) d\Omega_e + \oint_{\Gamma_e} \phi_i^T (F_x n_x + F_y n_y) d\Gamma_e \right] = 0$$

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

PRECISION OF CONSERVATION IN AdH

ADH solves conservation statements concerning water volume, momentum, and constituent mass. ADH is written in conservative form and thus can be regarded as both a finite element method and also a finite volume method. As such ADH can be shown to represent a sum of fluxes around the edge of an element to be in balance with the mass or volume change within the element. In the case of the momentum equation it will be a sum of fluxes and forces balancing the momentum change within an element. We shall focus on water and constituent conservation, and will refer to these conservation properties of balance over an element as elemental conservation of mass.

Another factor that must be considered in relation to elemental conservation is that of the nonlinearity of the differential equations. This means that items that we don't know are multiplied times each other, times themselves, raised to a power, etc. For a linear problem the elemental conservation of mass in ADH is exact. That is, it is as good as the computational accuracy of the machine that solves it. This is so for the convection-diffusion equation, typically. However, for the conservation of water mass equation we don't have a linear equation. In the 2D shallow water equation we have terms involving the velocity times the depth. There are at least two common ways to linearize the problem so that it can be solved. One is to use lagging of terms. That is we could use old velocity (that we know) times new depths. If we lag all the non-temporal terms we would have an explicit method. An explicit method has a stringent time step restriction. Even lagged methods have pretty severe restrictions due to accuracy. The other method, and, in fact, the one used in ADH is to use the Newton method.

In the Newton method we linearize the discrete equations and solve for the change in the unknowns, rather than for the unknowns themselves. This method is repeated (iterated) until the original differential equations (in discrete form) are satisfied below a tolerance. With this approach the time step size can be selected with much less concern for stability, but it still must be small enough for accuracy. The tolerance of the convergence then influences how tightly conservation is achieved. For problems in which we are not solving problems that wet and dry, it turns out that the conservation of mass equations for water are less nonlinear than the momentum equation. In fact, if the conservation of mass equation were linear and the momentum equation nonlinear, the conservation of mass would be satisfied to machine precision at each iteration regardless of the user defined tolerance. The solver would continue iterating until the momentum equations were satisfied within the tolerance, but the conservation of mass would be exact. However since the conservation of mass is somewhat nonlinear, but less so than the momentum equations, it will converge much more tightly than the tolerance. So for a problem in which all the domain remains wet the elemental conservation of mass for water (and also constituent) should be much better than the tolerance.

When ADH is solving a problem in which regions of the model wet and dry, it is solving not only for the depth and velocity, but also for the boundary of the domain. ADH solves for the water edge by allowing both positive and negative depths. The equations are only solved in the wet (or positive depth regions). But now the Newton method is responsible to find the water surface elevation that not only satisfies the differential equations but also what is the wet region of the domain. This is a much more nonlinear problem. The conservation of mass equation is much more nonlinear. The Newton method will have more trouble converging, and it isn't assured that the conservation of mass equation will converge more quickly than the momentum equation. Therefore, for problems with significant wetting and drying regions the elemental conservation of mass is more directly related to the tolerance used to determine that the equations are satisfied. These tolerances are specified by either the IP NTL or the IP ITL or both.

Technically the drying tolerance (specified via the MP DTL) doesn't directly affect elemental mass conservation. What the DTL specifies is a value that can be thought of as the thickness of a groundwater layer that allows flow within a partially wet element to be determined solely by the pressure differential created by water surface slope and friction. It actually isn't really groundwater but it is a useful analogy. The fluxes within this drying-wetting thickness are included in the mass conservation, so that elemental mass conservation is preserved. We would want this thickness (DTL) to be large enough to keep the model stable during wetting and drying but small enough not to severely modify the equations we wish to solve. So the selection of DTL can impact how well ADH converges. But once it does converge the results should satisfy the elemental mass conservation within a limit associated with the user defined tolerance set via NTL and ITL.

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.

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

- The percent area change between adjacent elements should not exceed 50%.
- Typically the minimum for interior angles is 20 degrees and the maximum is 130 degrees.
- No more than eight elements should join at one node.
- Try to avoid large gradients due to slope in bed elevation.

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

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

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

Run Pre_ADH first, and 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 a depth file with a .dat extension. These files can be opened and viewed in SMS or GMS without any further post-processing.

Sediment transport simulation requires the use of SI units. If converting input files from English to SI units, be certain to convert 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.

Refinement Levels allows an element to be divided into a maximum of $2^{lev\#}$ of elements, you may not use all of them depending on the flow conditions and the SRT.

Refinement Tolerances for Adaption are determined by running a short simulation with ML = 0 and SRT = 1, view the *err.dat output file to determine the magnitude of the residual error values, use the SMS contour options to determine which elements may be refined for a given tolerance value. The values in the *err.dat file are ratios of residual error to tolerance value. When this ratio is greater than 1, an element can be refined. The values in the *err_hydro.dat and *err_con#.dat files are non-ratioed residual error values for the hydrodynamic equations and transport equations, respectively. Once an appropriate SRT value is determined for each material from the initial run, set the SRT values and run again. This time the *err.dat file should contain much lower values. Locations where the values remain greater than 1 usually require more refinement levels. A standard rule of thumb is to not refine more than about 20% of your mesh at a given time, so select the appropriate SRT with this in mind. Watch the run and check the amount of resolution that is being added to make sure that you are getting the expected results.

Wetting/Drying Tolerance defaults to 0 if the card is not included and it is not needed when the domain remains wet. However, once drying begins to occur, the model can become unstable unless this value is set greater than 0. This value has a unit of length. Once the depth becomes equal to or less than the tolerance depth a stability factor is applied to prevent large velocity results and instabilities (essentially placing mass balance as the priority and not worrying with the physics). It is best to keep this value as low as possible…starting around 0.001*max depth in the model and then adjusting to improve model performance is a good method.

The wet/drying fraction can be used to remove elements from the computations, thereby speeding up your simulation. The value on this card is a fraction of an element that is wet at which point it is removed from the equations. This fraction should never be greater than 0.65.

Steady State Initial Timestep (STD #) option doesn't care about time but AdH still computes through a timestepping method. When using this option, an initial timestep is requested. This value should be fairly small, on the order of 5 or 10 in most cases. The code will increase or decrease this value as is runs in an effort to reach convergence to the NTL value. Since the product of the initial residual and the initial timestep size is kept constant to determine how to vary the timesteps as the residual changes throughout the run, the initial timestep size can have a large effect on the solution. If the run fails, drop the initial timestep by about half and try again.

Eddy Viscosity (EEV and EVS) is a parameter that helps numerical solutions match reality. Higher values indicate more viscous flow and vice versa. It is often difficult to determine how to set the various EVS values so the Estimated Eddy Viscosity (EEV) can help. This parameter computes the appropriate viscosity term depending on the depth and velocity magnitude at each location and time. This option requires the user to input a coefficient greater than 0. There are three methods, one in which the eddy viscosity is applied

isotropically one in which the eddy viscosity is directed in the direction of flow, and a Smagorinsky method. Typically 0.5 is a good place to start and sensitivity checks can be performed to determine if this coefficient should be adjusted. If using EVS, an initial estimate can be calculated as 1/40*h*u, where u is a representative depth in the model and u is a representative velocity magnitude.

Setting the convergence Values - the NTL value converges on the maximum residual norm. The ITL value converges on the maximum increment norm or the change in the solution. AdH will converge on all tolerances included in the boundary conditions file, so if you have both an ITL and NTL, you will converge to both. An estimate for an appropriate NTL value for a problem is (total_area*10^{-6})/#_elements. The NTL is the maximum allowance, so you will likely want to set it higher than the estimate so that your simulation will progress.

Common Error Messages

- *2D elements do not exist to match 1D elements*

  Check node numbers in edge strings, there is likely a wrong node number or a gap

- *Overland flow does not match available types*

  Either the hotstart file does not contain correct initial depths (the mesh is dry) or there is a string that does not have an associated boundary condition

AdH-DSS INSTRUCTIONS

DSS Input:

1. Export your HEC-RAS data to a *.dss file.

2. If a *.dsc file is not generated, open the *.dss file in DSSVue. That will automatically generate one for you.

3. Once you have selected which data set(s) within the *.dss file you would like to use, open the *.dsc file and take note of the data set(s) you would like to use.

4. With that open, also open the *.bc file and add the DSS card, which is input thusly:
```
DSS [time series ID] [DSS file name] [pathname]
```

5. Continue adding DSS time series, pasting the data set pathname(s) from the *.dsc file, until you are finished. Note: The time parameter of the pathname is ignored.

6. Pick a time window to use for all of the data that was just included in the *.bc file.

7. Include the window start time in the *.bc file:
```
TC DS0 [MM]-[DD]-[YYYY] [TIME (24 hour)]
```

8. Include the window end time in the *.bc file:
```
TC DSF [MM]-[DD]-[YYYY] [TIME (24 hour)]
```

9. Run AdH.

DSS Output:

1. Locate the nodes in the mesh you would like to use as output stations.

2. Create a node string using those nodes in the *.bc file.

3. Add the DSS output card to the *.bc file:
```
PC DSS [node string ID]
```

4. Run AdH.

References

Bernard, B., (1992) "Depth-Average Numerical Modeling for Curved Channels" Technical Report HL-92-9, U.S. Army Corps of Engineers, Engineering Research and Development Center, Vicksburg, MS.

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.

Hammack, E. A., Smith, D. S., and Stockstill, R. L. "Modeling vessel-generated currents and bed shear stresses," Technical Report ERDC/CHL TR-08-7, U.S. Army Engineer Research and Development Center, Vicksburg, MS, June 2008.

Hammack, E. A. and J. N. Tate. 2008. Considerations for modeling vessel-generated currents and bed shear stresses. Coastal and Hydraulics Engineering Technical Note, http://chl.erdc.usace.army.mil/chetn, ERDC/CHL CHETN-IX-17. Vicksburg, MS: U.S. Army Engineer Research and Development Center.

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).

Lilly, D.K (1967) "The Representation of Small-Scale Turbulence in Numerical Experiments", Proc. IBM Scientific Computing Symposium on Environmental Sciences, pp 195-210.

Press, William H, Teukolsky, Vetterling, William T, Flannery Brian P "Numerical Recipes in C, The Art of Scientific Computing" Cambridge University Press, Second Edition, 1992.

Rodi, W. "Turbulence Models and their Applications in Hydraulics – A State of the Art Review", International Association for Hydraulics Research, 1984.

Savant, Gaurav and Berger, Charlie (2009) "Considerations for Modeling Flow Control Structures in Adaptive Hydraulics (ADH)" System Wide Water Resources Program Technical Note, ERDC-TN-SWWRP-09-3, 2009.

Savant, Gaurav and Berger, Charlie (2010) "Intelligent Adaptive Time Step Control for Modeling Rapidly-Evolving Hydrodynamic Flows in Adaptive Hydraulics (ADH)", System Wide Water Resources Program Technical Note, 2010.

Teeter, Allen Michael (2002) "Sediment Transport in Wind-Exposed Shallow, Vegetated Aquatic Systems" A Dissertation. Louisiana State University.

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.

Webel, G. and Schatzmann, M. (1984). “Transverse Mixing in Open Channel Flow” *Journal of Hydraulic Engineering*, Vol 110, No. 4, April 1984, pp 423-435.

Wu, Jin (1982). “Wind-Stress Coefficients Over Sea Surface From Breeze to Hurricane” *Journal of Geophysical Research,* Vol 87, No. C12, November 1982, pp 9704-9706.

Zienkiewicz, O.C and Zhu, J.Z “The Superconvergent Patch Recovery and *a Posteriori* Error Estimates. Part 1: The Recovery Technique”, International Journal for Numerical Methods in Engineering, 33(7), 1331-1364, 2005.

Footnotes

1

USAE Research and Development Center