

SMS User Manual 11.2, Vol. 3 Models

Contents

4. General Numeric Models	1
SMS Models	1
4.1. Generic Mesh Model	3
Generic Mesh Model	3
Generic Model Files	4
Generic Model Graphical Interface	5
4.2. Finite Volume Coastal Ocean Model (FVCOM)	11
FVCOM	11
4.3. Particle Tracking Model (PTM)	14
PTM	14
PTM Model Control	17
PTM Graphical Interface	21
PTM Particle Filters	22
PTM Traps	23
4.3.a. PTM Coverages	25
PTM Coverages	25
PTM Gage Coverage	25
4.3.a.1. PTM Sources	26
PTM Sources	26
PTM Arc Attributes Dialog	28
PTM Feature Point Attributes Dialog	29
4.3.b. PTM Files	31
PTM Files	31
PTM Control File	32
PTM Sediment File	41
PTM Source File	43
PTM Wave File	48
PTM Trap File	49
PTM Trap Output	52
PTM Boundary Condition File	53

4.4. Generic Grid Model	54
Generic Grid Model	54
Creating a Generic Grid	55
5. Coastal Numeric Models	56
5.1. ADCIRC (Advanced Circulation) Model	57
ADCIRC	57
ADCIRC Database	60
ADCIRC Files	61
ADCIRC Graphical Interface	63
ADCIRC Menu	64
ADCIRC Mesh	65
ADCIRC Model Control	74
ADCIRC Spatial Attributes	79
LTEA	83
5.1.a. ADCIRC Boundary Conditions	89
ADCIRC Boundary Conditions	89
ADCIRC Weirs and Island Barriers	93
5.2. BOUSS-2D - A Boussinesq Wave Model for Coastal Regions and Harbors	95
BOUSS-2D	95
BOUSS-2D Using the Model	99
BOUSS-2D Simulations	100
BOUSS-2D Calculators	102
BOUSS-2D Files	104
BOUSS-2D Graphical Interface	105
BOUSS-2D Model Control	108
BOUSS-2D Parameter File	110
BOUSS-2D Probes	113
BOUSS-2D Test Cases	114
5.3. BOUSS-2D Runup/Overtopping	115
BOUSS2D Runup / Overtopping	115
BOUSS2D Runup / Overtopping Input Files	117
BOUSS2D Runup / Overtopping Viewing Data	118

5.4. CGWAVE	121
CGWAVE	121
CGWAVE Overview	128
CGWAVE Math Details: Boundary Conditions	129
CGWAVE Boundary Conditions Dialog	133
CGWAVE Files	134
CGWAVE Math Details: Governing Equations	138
CGWAVE Graphical Interface	141
CGWAVE Incident Wave Conditions	144
CGWAVE Model Control	145
CGWAVE Math Details: Numerical Solution	147
CGWAVE Practical Notes	148
CGWAVE Test Cases	150
Long Wave Input Toolbox	153
5.5. Coastal Modeling System	156
CMS	156
CMS-Flow/CMS-Wave Steering	157
CMS-Flow/Save Points	158
5.5.a. CMS-Flow	161
CMS-Flow	161
CMS-Flow Cell Attributes	163
CMS-Flow Graphical Interface	164
CMS-Flow Grid Modification Flag	167
CMS-Flow Hard Bottom	168
CMS-Flow Hot Start	170
CMS-Flow Menu	171
CMS-Flow Model Control	173
CMS-Flow Observation Cells	183
CMS-Flow Output Control	184
CMS-Flow Spatial Datasets	185
CMS-Flow Time Control	187
CMS-Flow Transport Control	188
CMS-Flow User-Editable Datasets	190
Lund Cirp and Watanabe Formula	191
CMS-Flow Advanced Cards	192

M2D	192
Telescoping Grids	193
5.5.b. CMS-Wave	194
CMS-Wave	194
CMS-Wave Cell Attributes Dialog	196
CMS-Wave Graphical Interface	198
CMS-Wave Menu	199
CMS-Wave Model Control	200
CMS-Wave Nesting Options Dialog	202
5.5.b.1. CMS-WAVE Files	203
CMS-Wave Files	203
CMS-Wave Control File	205
CMS-Wave Depth File	208
CMS-Wave Simulation File	209
CMS-Wave Spectral Energy File	210
CMS-Wave Spectral Table File	211
CMS-Wave STD	212
CMS-Wave Structure File	213
5.6. CSHORE	215
CSHORE	215
5.7 CSTORM	219
CSTORM-MS	219
5.8. GenCade	224
GenCade	224
GenCade Files	226
GenCade Modeling Process	227
5.8.a. GenCade Graphical interface	229
GenCade Graphical Interface	229
GenCade Arc Attributes	231
GenCade Events	232
GenCade Menu	233
GenCade Model Control Dialog	234
GenCade Result Visualization	236

GenCade Structures	237
Wave Gages	242
5.9. STWAVE - Steady State Spectral Wave	244
STWAVE	244
Grid Nesting	246
STWAVE Graphical Interface	247
STWAVE Iterations Control Dialog	248
STWAVE Menu	249
STWAVE Model Control	250
STWAVE Output Control Dialog	252
5.10. WAM - Wave Prediction Model	253
WAM	253
WAM Map to Raster Utility	254
5.10.a. WAM Graphical Interface	256
WAM Graphical Interface	256
WAM Grid Options	258
WAM Simulation Model Control	262
WAM Spectra from STWAVE Grids	264
6. Riverine and Estuarine Models	266
6.1. ADH - Adaptive Hydraulics Modeling	267
ADH	267
ADH Bed Layers Assignment	269
ADH Boat Definition File Cards	270
ADH Extract WSE	271
ADH Hot Start File	271
ADH Hot Start Initial Conditions	272
ADH Material Properties	273
ADH Run Model	275
Sediment Library Control	276
ADH Sediment Transport and Bed Layers	277
ADH Time Series	279
ADH Time Series Attributes	281
ADH Velocity Series Editor	281
ADH Vessel Coverage	282

ADH Wind Stations	284
6.1.a. ADH Boundary Condition	286
ADH Boundary Condition	286
ADH Boundary Condition File Cards	289
6.1.b. ADH Model Control	293
ADH Model Control	293
ADH Model Control Model Parameters	295
ADH Model Control Iterations	297
ADH Model Control Time	299
ADH Model Control Output	301
ADH Model Control Global Material Properties	302
ADH Model Control Transport Constituents	304
ADH Model Control Consolidation	306
ADH Model Control Advanced	307
ADH Model Control Solver	308
6.2. Finite Element Surface Water Modeling System (FESWMS)	309
FESWMS	309
FESWMS Arc Attributes Dialog	312
FESWMS BC Nodestrings	312
FESWMS Executable Known Issues	314
FESWMS Files	315
FESWMS Graphical Interface	316
FESWMS Hydraulic Structures	317
FESWMS Material Properties	324
FESWMS Menu	326
FESWMS Model Control Dialog	328
FESWMS Point Attributes Dialog	333
FESWMS Sediment Control	334
FESWMS Spindown	336
6.3. HYDRO AS-2D	338
HYDRO AS-2D	338
6.4. RIVERFLO-2D	340
RIVERFLO-2D	340

6.5. SRH-2D	343
SRH-2D	343
6.6. Steering	348
Steering	348
6.7. TABS-MD (Multi-Dimensional) Numerical Modeling System - RMA2/RMA4	349
TABS	349
TABS Attribute Dialog	350
Total Flow Nodestring	352
6.7.a. GFGEN	353
GFGEN	353
GFGEN Executable Known Issues	354
6.7.b. RMA2 - Resource Management Associates	355
RMA2	355
Nodal Transition (Marsh Porosity) Dialog	357
Rainfall Values Dialog	358
RMA2 1D Control Structure	359
RMA2 Boundary Conditions	360
RMA2 Files	362
RMA2 Graphical Interface	363
RMA2 Material Properties	364
RMA2 Menu	367
RMA2 Model Control Dialog	368
RMA2 Spindown	374
Roughness Options Dialog	376
RMA2 GCL Card	377
6.7.c. RMA4 - Resource Management Associates	378
RMA4	378
RMA4 Boundary Conditions	380
RMA4 Files	383
RMA4 Graphical Interface	383
RMA4 Material Properties	384
RMA4 Menu	385

RMA4 Model Control	385

6.8. TUFLOW 388

TUFLOW	388
TUFLOW Overview	389
Batch Runs	392
Building TUFLOW 2D Domains	393
Command Objects	393
Define TUFLOW domain	395
Reading a TUFLOW Simulation	397
TUFLOW 2D Geometry Components	404
TUFLOW AD	405
TUFLOW Boundary Conditions	406
TUFLOW Check Files	410
TUFLOW Combining 1D and 2D Domains	411
TUFLOW Grid Options	413
TUFLOW Inlet Database	414
TUFLOW Irregular Culverts	415
TUFLOW Linking 2D Domains	415
TUFLOW Links	416
TUFLOW Manholes	417
TUFLOW Material properties	418
TUFLOW Menu	419
TUFLOW Model Parameters	420
TUFLOW Numeric Engine	429
TUFLOW Simulation	430
TUFLOW ZShape	430

References

Article Sources and Contributors	434
Image Sources, Licenses and Contributors	439

4. General Numeric Models

SMS Models

SMS provides pre- and post- processing for several numeric models. These models are developed and maintained by government or commercial entities rather than the developers of SMS.

Hydraulic Models

- ADCIRC (ADvanced CIRCulation Model) – Widely applied coastal circulation and coastal flooding model. Developed commercially.
- Coastal Modeling System CMS-FLOW – Suite of models that simulates a wide variety of coastal processes. Developed and maintained by the USACE.
- FESWMS – Developed in cooperation with the Federal Highway Administration (FHWA).
- Accessed through the generic model interface in cooperation with Aquaveo
 - SRH-2D – Developed at the United States Bureau of Reclamation.
 - RIVERFLO-2D – Commercially developed finite element model to route floods. Provides high resolution flood hydraulics.
 - HYDRO AS-2D – Developed commercially in Germany.
 - TUFLOW-FV – Finite volume model developed by the makers of TUFLOW (WBM).
- TABS – Suite of models for solving hydrodynamics and transport problems maintained by the USACE.
- TUFLOW – Finite difference model featuring combined 1D/2D models, very stable wetting drying, and advanced simulation management for Coastal, Riverine or Urban applications.

Wave Models

Harbor Design

- BOUSS-2D – Wave climate, circulation from waves, and sea state using the Boussinesque equations
- CGWAVE – Phase resolving wave reflection and refraction analysis

Wave Generation and Transformation

- STWAVE – Finite difference spectral wave energy model
- CMS-Wave (WABED)

Other Models

- Generic – Rather than a specific model interface, the "Generic" model interface in the mesh module is a collection of interface objects that can be configured by the user to generate specific types of data. The developers at Aquaveo work with some engine developers to utilize this tool. These engines are included in the list above. Other model developers are encouraged to contact Aquaveo for assistance in using these tools. The tools of the generic model interface can be utilized without coordination with Aquaveo. Se are aware of partial interfaces with the following:
 - FVCOM
 - HYDRO AS-2D
-

- SRH-W
- PTM – Lagrangian Particle Tracking Model which tracks sediment particles based upon input hydrodynamics and wave effects.

Hydraulic Models (Feature Comparison)

Model	Riverine	Tidal Forcing	Wave Forcing	1D	2D	Hydraulic Structures	Sediment Transport	Advection / Dispersion
ADCIRC	no	yes	yes	no	yes	yes	no	no
CMS-Flow	no	yes	yes	no	yes	no	yes	no
FESWMS	yes	no	no	no	yes	yes	yes	no
TABS	yes	yes	no	yes	yes	yes	no	yes
TUFLOW	yes	yes	no	yes	yes	yes	no	no

Model Linkages / Steering

- General Steering
- RMA2 Spindown
- FESWMS Spindown
- CMS-Flow / CMS-Wave Steering

Related Topics

- SMS Main page

4.1. Generic Mesh Model

Generic Mesh Model

While SMS contains several custom interfaces for specific numerical models, it is not possible to build interfaces in SMS for every model in existence. SMS has a generic interface whereby any two-dimensional finite element or finite difference model can be run using SMS as a pre- and post-processor.

Functionality

The following can be defined for a generic model:

- Global Model Parameters
- Boundary Conditions
- Material Properties

With the model attributes defined, the user can generate a mesh or grid using the geometric tools in the Map, and Mesh or Cartesian Grid Modules. Model parameters can then be set, boundary conditions specified, and material properties applied to the geometric data and saved out to a generic file format.

Using Multiple Meshes

Starting in SMS 11.2, multiple generic mesh model definitions can exist at the same time. When creating a new generic 2D mesh coverage, if multiple generic model definitions exist, a dialog will appear to ask the user which model definition they wish to use. When performing a map to 2D mesh operation, the definition and values will be copied from the coverage to the new mesh.

The define model menu item will define the definition for the mesh, if the menu item is in the mesh module menu, or for the coverage, if in the feature objects menu. The generic model definition, or template, is stored in a *.2dm file with the SMS project. The *.2dm file will not contain a mesh or parameter values. This template will be used whenever a new mesh or coverage of that type is created.

Graphical Interface

SMS provides a graphical interface that is designed to allow users to visualize the projects they are creating, easily modify project parameters, and view the solutions produced by the Generic Model. See Generic Model Graphical Interface for more information.

The Generic Model Graphical Interface contains tools to create and edit a Generic Model simulation. The simulation consists of a geometric definition of the model domain (the mesh) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the 2D Mesh Module and setting the current model to Generic. If a mesh has already been created for a Generic Model simulation or an existing simulation read, the mesh object will exist in the Project Explorer and selecting that object will make the 2D Mesh module active and set the model to Generic. See the Mesh Module documentation for guidance on building and editing meshes as well as visualizing mesh results.

The interface consists of the 2D Mesh Module Menus and tools augmented by the Generic Model Menu. See Generic Model Graphical Interface for more information.

Case Studies / Sample Problems

The following tutorials may be helpful for learning to use the Generic Model Interface in SMS:

- General Section
 - Mesh Editing
 - Observation
- Models Section
 - Generic Mesh Model

Related Topics

- BASEMENT
- HYDRO AS-2D
- SRH-2D
- ELCIRC
- FVCOM

Generic Model Files

SMS stores all of the Generic Model input parameters in a 2D Mesh File (*.2dm). This 2D Mesh file includes:

- The mesh
 - Node locations
 - Element type
 - Triangle or quadrilateral
 - Linear or quadratic
 - Element connectivity
- Global parameters
- Boundary conditions
 - Nodal boundary conditions
 - Nodestring boundary conditions
 - Element boundary conditions
- Material properties

For more information, see the 2D Mesh File (*.2dm) article.

Related Topics

- HYDRO AS-2D
 - SRH-W
-

Generic Model Graphical Interface

The Generic Model Graphical Interface includes tools to assist with creating, editing and debugging a Generic Model. The Generic Model interface exists in the Mesh Module.

Define Model

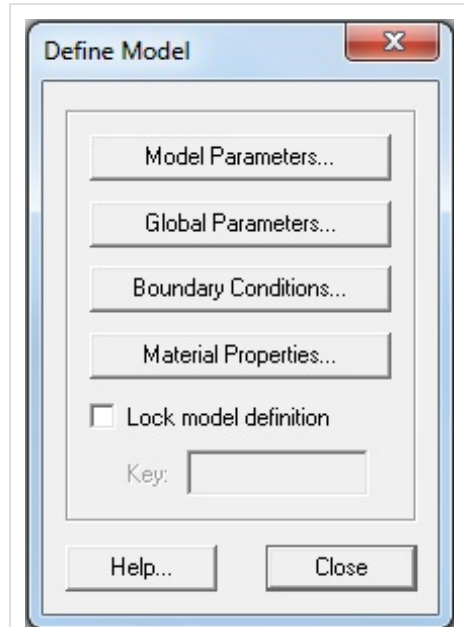
The *Define Model* dialog is used to customize and define the model interface parameters that define various states and characteristics of a model. These model parameters may include items such as those needed to describe flow, channel roughness, and control structures. The parameters, names, and ranges can be created and customized by the user.

Parameters are organized into groups and are given suitable value ranges depending on the purposes of the model. Proper organization of parameters will increase the abilities of SMS as an interface.

Using the *Define Model* dialog the model interface can be renamed by the user. The *Define Model* dialog can be accessed when the Mesh Module is the active module.

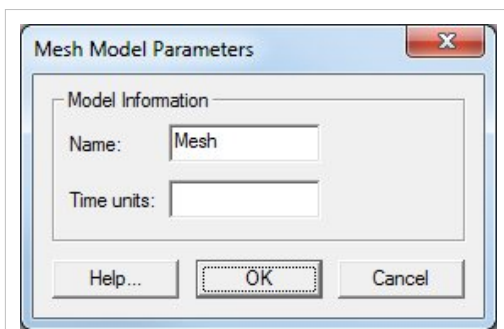
The *Define Model* dialog is used to setup the options that apply to the simulation as a whole.

- Model Parameters
- Global Parameters
- Boundary Conditions
- Material Properties
- Lock Model Definition with Key



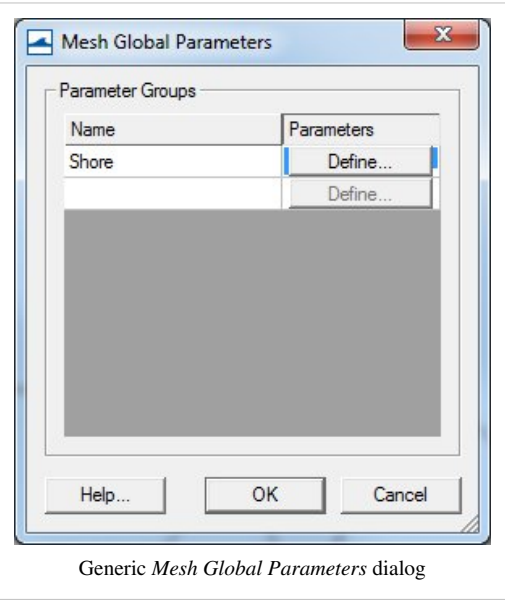
Define Model dialog for the Generic Model

Model Parameters



Generic *Mesh Model Parameters* dialog

Global Parameters

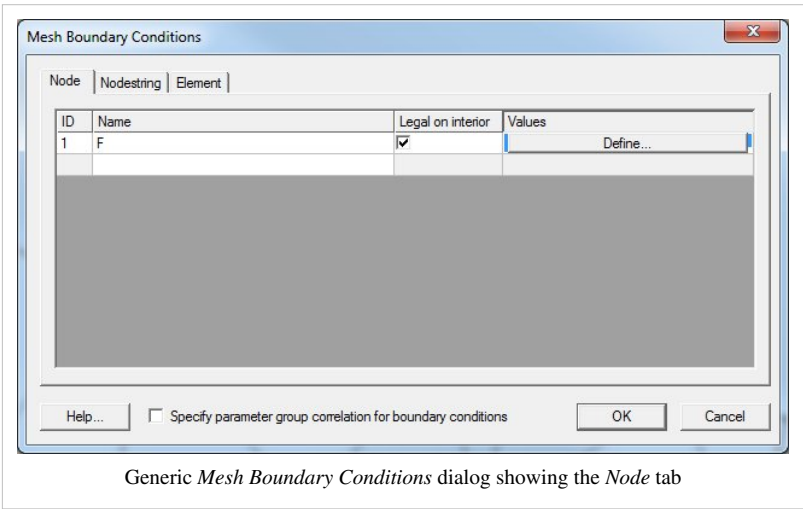


Boundary Conditions

All numeric models require boundary condition (BC) data. Generic Mesh Model boundary conditions can be defined on nodestrings, nodes, and elements. An entity (nodestring, node, or element) may have multiple boundary conditions set. To add or remove a boundary condition, select a node, nodestring, or element. Then right-click and select **Assign BC**. A dialog with a tree item appears listing all possible boundary conditions. Checked tree items are active boundary conditions. Toggle the tree items as desired to activate/deactivate boundary conditions. The settings in *Display | Display Options* will determine how the boundary conditions are displayed.

The model developer can define boundary conditions as constant or as dynamic. This is stored in the BD card as the next to last field. The user can also assign multiple boundary conditions to

nodes, nodestrings, and elements. To add or remove a boundary condition, select a node, nodestring, or element. Then right-click and select **Assign BC**. A dialog with a tree item appears listing all possible boundary conditions. Checked tree items are assigned. Toggle the tree items as desired to assign/unassign boundary conditions. The settings in *Display | Display Options* will determine how the boundary conditions are displayed.



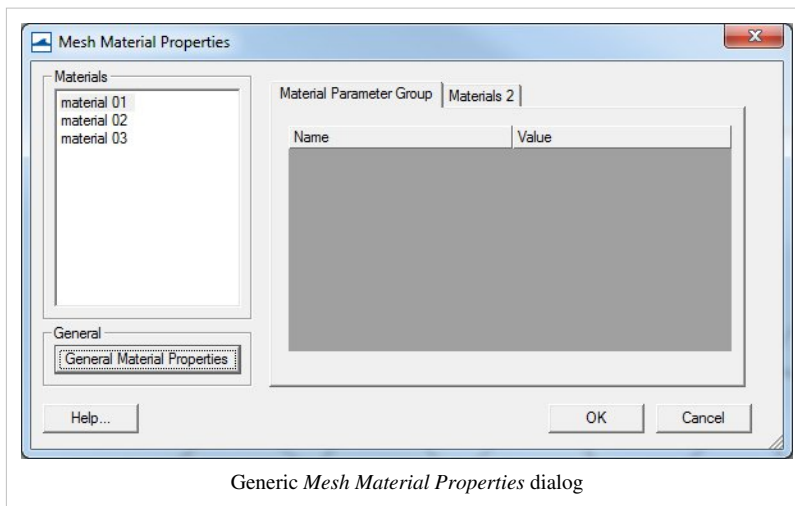
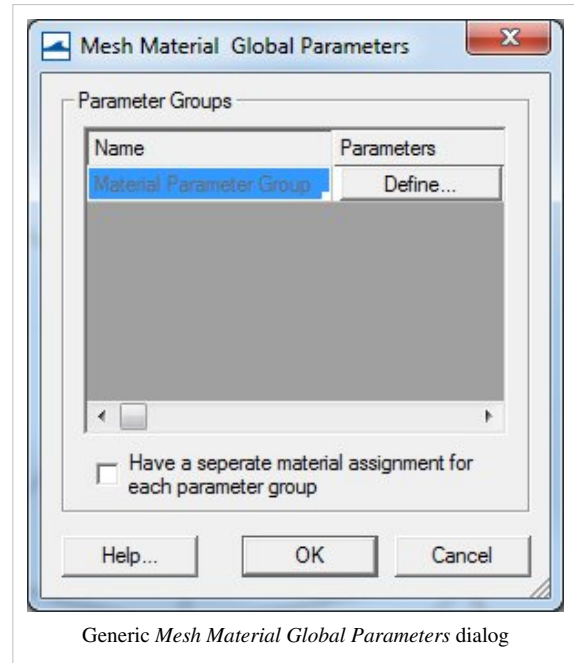
Material Properties

Each element is assigned a material type. Material properties describe the hydraulic characteristics of each material type.

Material Parameter Groups are the same as the Global Parameter Groups. To add a new Material Parameter Group, add it in the Global Parameter Group by *Gen2DM | Define Model | Global Parameters*.

Designer /User can choose whether to have a single material group or multiple (1 for each group) by *Gen2DM | Define Model | Material Properties* then select or unselect *Have a seperate material assignment for each parameter group*.

- Multiple
 - User can change the active group by *Gen2DM | Set Active Material Group*.
 - Changing the "active material group" effects the display and assignment of materials.
 - In the *Gen2DM | Material Properties* dialog, only the active/assigned materials are displayed.
- Single
 - *Set Active Material Group* menu is hidden under the *Gen2DM* menu
 - In the *Gen2DM | Material Properties* dialog, all materials are displayed. The tabs across the top correspond to the Groups.

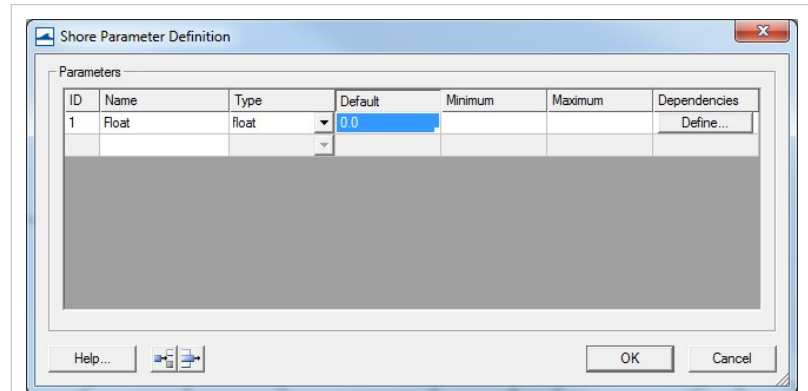


Dependencies

The generic model interface lets a generic model designer create a custom interface by setting up input parameters for their model.

The parameters may be global parameters, bc parameters (applied to node, element, or nodestring), or material parameters.

It is often useful to have certain parameters displayed only in some situations.



Generic Mesh Global *Parameters Definition* dialog

For example, the numeric engine may support manning values by depth or a single manning value for all depths. If the user chooses to use manning values by depth he or she would need to provide a curve for manning values based upon depth. If the user chooses to do a single manning value he or she would need to provide a single manning value.

This manning example could be expanded another level. Suppose the manning value mentioned above was part of the material properties for each material. Also, suppose that the engine supported chezy as well as manning to represent roughness but this had to be applied at a global level. If the global parameter is chezy, the *material properties* dialog should only show the option to enter a chezy value. If the global parameter is manning, the *material properties* dialog would let the user choose whether to provide this using a single value or a curve by depth. The controls would work as described in the preceding paragraph.

The generic model designer can use dependencies to accomplish both situations above. Dependencies show/hide parameters based upon the setting of a parent parameter. The parent parameter must be before the child parameter. If the child parameter is a material or boundary condition parameter, the parent can also be a global parameter.

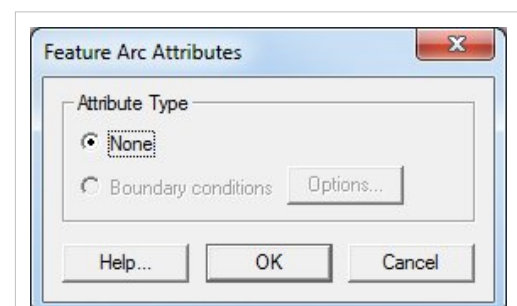
Dependencies are controlled on the child level when defining the model. To assign a dependency the parent and child parameters must exist and the parent must have its options defined. Dependencies are specified by clicking on the **None** button in the dependency column for the child parameter. You specify the parent you want to use for this parameter whether a global parameter or local (whatever level you currently are defining). Then check the boxes next to the parent parameters that will allow the child parameter to be visible. A child option may be visible for several parent options.

Whenever a parent object is invisible all children that are dependent upon the option are also invisible.

Generic 2D Mesh Arc Attributes Dialog

The Generic 2D Mesh *Feature Arc Attributes* dialog is used to set the attributes for feature arcs. Attributes that can be specified for each feature arc include:

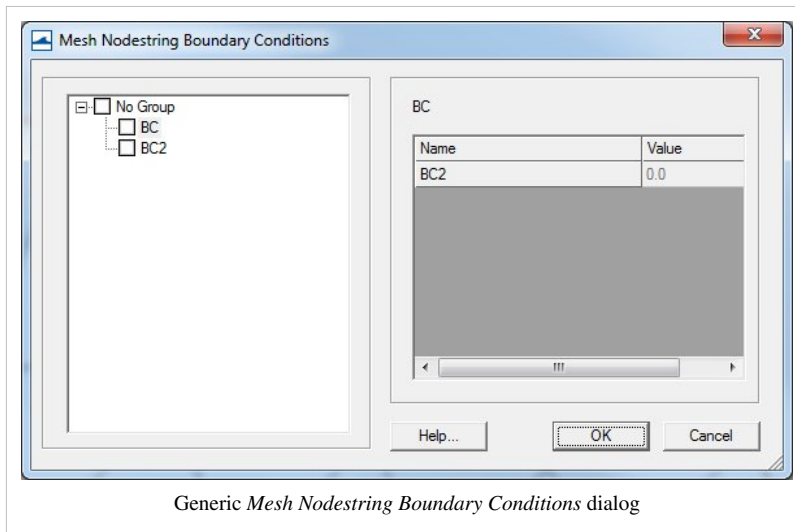
- Arc Type
 - None
 - Boundary Conditions – Options button opens the *Mesh Nodestring Boundary Conditions* dialog.



Generic *Feature Arcs Attributes* dialog

Mesh Nodestring Boundary Conditions Dialog

Generic 2D Mesh Boundary conditions are generally defined on feature arcs in the conceptual model or nodestrings on the 2D mesh. Boundary conditions constrain the water surface elevation and/or flow at the model boundary. The options available will be based on the current generic model in use and are defined in the *Define Model* dialog.



Generic 2D Nodal BC, Nodestring and Element Display Options

The properties of all Gen2DM (Generic 2D Mesh) node boundary conditions, nodestrings, and elements that SMS displays on the screen can be controlled here. This window is accessible only if the current 2D Mesh model is Generic (Mesh module's *Data | Switch Current Model...* menu command). Open *Display Options*, turn on either *Nodal BC*, *Nodestrings*, or *Elements* and then select the corresponding **Options** button.

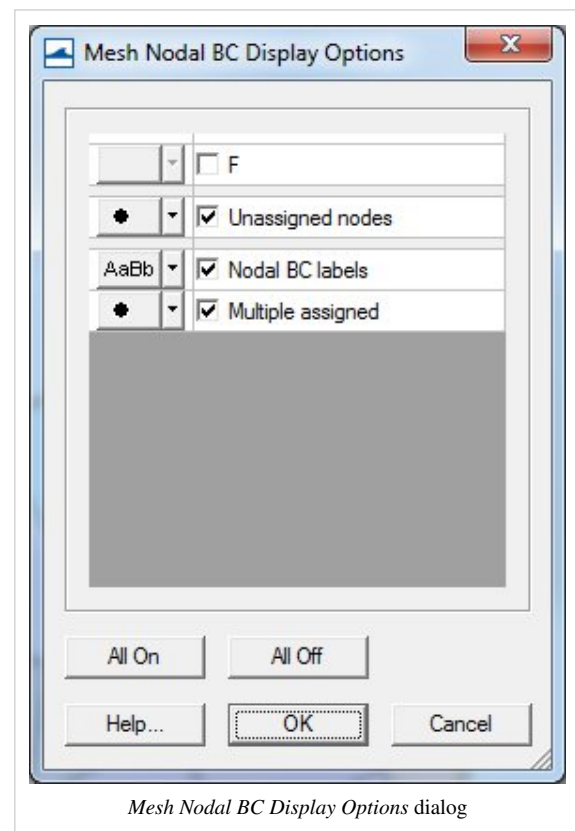
Display options are available for all boundary condition entities defined in the Gen2DM model definition. If no BC entities have been defined then a message will state that fact and display options will not be shown. Each defined BC entity has a checkbox to toggle the display of the item and an **Attribute Options** button to adjust style and color.

Below the list of BC entities is the display options for Inactive entities. Inactive entities are BC entities that are associated with an inactive parameter group (activation is controlled via the *Gen2DM | Global Parameters* menu item).

The BC entities can be displayed with labels beside them. Labels allows the user to adjust the label font, style, size, color and whether they are displayed. When *Labels* is checked for display, the auxiliary *Show BC Values* in labels is available. If checked, BC entities' values will be displayed following the entities' description.

All on checks all BC entity display options.

All off unchecks all BC entity display options.



Overview

The designer/user can define whether the curves in their model are interpolated to the time step duration or not.

Saving the Model

The Generic Model Files are written automatically with the SMS project file or can be saved separately using the *File* | **Save Mesh** or *File* | **Save As** menu commands. See Generic Model Files for more information on the files used for the Generic Model run.

Generic Model Menu

See Generic Model Menu for more information.

Related Topics

- Mesh Module
- Generic Model Graphical Interface

4.2. Finite Volume Coastal Ocean Model (FVCOM)

FVCOM

FVCOM	
Model Info	
Model type	Prognostic, unstructured-grid, finite-volume, free-surface, 3-D primitive equation coastal ocean circulation model
Developer	Changsheng Chen University of Massachusetts-Dartmouth
Web site	FVCOM web site ^[1]
Tutorials	General Section • Data Visualization • Mesh Editing • Observation

As stated on the FVCOM website (<http://fvcom.smast.umassd.edu/FVCOM/index.html>):

The Unstructured Grid Finite Volume Coastal Ocean Model (FVCOM) is a prognostic, unstructured-grid, finite-volume, free-surface, 3-D primitive equation coastal ocean circulation model developed by UMASSD-WHOI joint efforts. The model consists of momentum, continuity, temperature, salinity and density equations and is closed physically and mathematically using turbulence closure submodels. The horizontal grid is comprised of unstructured triangular cells and the irregular bottom is presented using generalized terrain-following coordinates. The General Ocean Turbulent Model (GOTM) developed by Burchard's research group in Germany (Burchard, 2002) has been added to FVCOM to provide optional vertical turbulent closure schemes. FVCOM is solved numerically by a second-order accurate discrete flux calculation in the integral form of the governing equations over an unstructured triangular grid. This approach combines the best features of finite-element methods (grid flexibility) and finite-difference methods (numerical efficiency and code simplicity) and provides a much better numerical representation of both local and global momentum, mass, salt, heat, and tracer conservation. The ability of FVCOM to accurately solve scalar conservation equations in addition to the topological flexibility provided by unstructured meshes and the simplicity of the coding structure has made FVCOM ideally suited for many coastal and interdisciplinary scientific applications.

FVCOM was originally developed for the estuarine flooding/drying process in estuaries and the tidal-, buoyancy- and wind-driven circulation in the coastal region featured with complex irregular geometry and steep bottom topography. This model has been upgraded to the spherical coordinate system for basin and global applications. A non-hydrostatic version of FVCOM has been coded and is being tested.

Features

The present version of FVCOM includes:

- Choice of Cartesian or spherical coordinate system
- A mass-conservative wet/dry point treatment for the flooding/drying process simulation
- The General Ocean Turbulent Model (GOTM) modules (Burchard et al., 1999; Burchard, 2002) for optional vertical turbulent mixing schemes
- A water quality module to simulate dissolved oxygen and other environmental indicators
- 4-D nudging and Reduced/Ensemble Kalman Filters (implemented in collaboration with P. Rizzoli at MIT) for data assimilation
- Fully-nonlinear ice models (implemented by F. Dupont)
- A 3-D sediment transport module (based on the U.S.G.S. national sediment transport model) for estuarine and near-shore applications
- A flexible biological module (FBM) for food web dynamics study. With various pre-built functions and parameters for these groups, FBM allows users to either select a pre-built biological model (such as NPZ, NPZD, etc.) or to build their own biological model using the pre-defined pool of biological variables and parameterization functions. FBM includes seven groups:
 - Nutrients
 - Autotrophy
 - Heterotrophy
 - Detritus
 - Dissolved organic matter
 - Bacteria
 - Other

FVCOM was originally coded for sigma-coordinates in the vertical and now has been upgraded to a generalized terrain-following coordinate system with choices of various topographic-following coordinates. FVCOM is written with Fortran 90 with MPI parallelization, and runs efficiently on single and multi-processor machines.

FVCOM is an open source code ocean community model that always welcomes new users. **This program is only permitted for use in non-commercial academic research and education.** Users are required to register ^[2] to receive the source codes, demo examples, and user manuals as well as some recommended postprocessing tools.

Graphical Interface

FVCOM uses the Generic Model Graphical Interface.

External Links

- FVCOM Model Developer Website ^[1]
- FVCOM Publications ^[3]
- FVCOM Forum ^[4]
- FVCOM Help ^[5]

Related Topics

- [Generic Model Interface](#)

4.3. Particle Tracking Model (PTM)

PTM

PTM	
Model Info	
Model type	Lagrangian particle tracker designed to allow the user to simulate particle transport processes.
Developer	Neil J. MacDonald, Ph.D. Michael H. Davies, M.Sc., Ph.D., P.Eng. Coldwater Consulting Ltd ^[1]
Web site	PTM web site ^[2]
Tutorials	Models Section PTM (Pending)

The Particle Tracking Model (PTM) is a Lagrangian particle tracker designed to allow the user to simulate particle transport processes. PTM is funded through two US Army Corps of Engineers Engineering Research and Development Center (ERDC) research programs, the Coastal Inlets Research Program (CIRP) and the Dredging Operations and Environmental Research (DOER) Program.

Functionality

PTM has been developed for application to dredging and coastal projects including dredged material dispersion and fate, sediment pathway and fate, and constituent transport. The model contains algorithms that appropriately represent transport, settling, deposition, mixing, and resuspension processes in nearshore wave/current conditions. It uses waves and currents developed through other models and input directly to PTM as forcing functions.

Using the Model / Practical Notes

- The horizontal and vertical coordinates used for all PTM input files must be in meters. Geographic Coordinates cannot be used since it is a latitude/longitude system defined in decimal degrees.
 - If last step trap is checked, the traps will not become active until the last time step.
-

How do I use PTM?

The SMS tutorials are a good place to start learning to use SMS and associated models. A tutorial is available for PTM.

Graphical Interface

SMS provides a graphical interface that is designed to allow users to visualize the projects they are creating, easily modify project parameters, and view the solutions produced by the PTM model. See PTM Graphical Interface for more information.

The PTM Graphical Interface contains tools to create and edit an PTM simulation. The simulation consists of a geometric definition of the sources, traps, and a set of numerical parameters. The parameters define the hydrodynamic input and options pertinent to the model.

The interface is accessed by selecting the Particle Module and setting the current model to PTM. If a simulation has already been created or an existing simulation read, the particle object will exist in the Project Explorer and selecting that object will make the Particle module active and set the model to PTM. See the Particle Module documentation for guidance on visualizing results.

The interface consists of the Particle Module Menus and tools augmented by the PTM Menu. See PTM Graphical Interface for more information.

PTM Menu

The *PTM* menu becomes active when a PTM model coverage has been created. For more information see PTM Graphical Interface.

Tools

PTM sources and traps are created in the Map Module as feature objects using the Map Module Tools.

Theoretical Basis / Mathematical Details

Please refer to the model developer provided documentation listed in the external links section.

Numeric Engine Background

Please refer to the model developer provided documentation listed in the external links section.

General Steps to Build a PTM Model

1. Open SMS
 2. Although not required, it is generally easier to visualize your model if you read in the hydrodynamic solution
 3. Convert the horizontal and vertical coordinates to meters. PTM cannot be run in Geographic Coordinates since it is a latitude/longitude system defined in decimal degrees.
 4. Switch to the Map Module
 5. Create a PTM type coverage
 6. Create sources and traps using feature points, arcs, and polygons
 7. Switch to the Particle Module
 8. Use the *PTM* menu to create a new PTM simulation
 9. Use the *PTM* menu to open the *PTM Model Control* dialog
 1. Specify input and output parameters in the model control
-

2. Click on the **Create input file(s) from data** button to create a native sediment file, and hydrodynamic input files if needed
10. Save the simulation
11. Use the *PTM* menu to perform a model check
12. Use the *PTM* menu to run PTM

General Post-Processing Steps

1. Use the *display options* to visualize the results
2. Use the *Data Calculator* or *Create Datasets* dialog to create particle datasets
3. Create a Cartesian Grid and use the *Create Grid Datasets* dialog to create Cartesian grid datasets
4. Review the Trap Output files in an ASCII file viewer

External Links

- U.S. Army Corps of Engineers DOER PTM website [2]
- Sep 2006 ERDC/CHL TR-06-20 PTM: Particle Tracking Model Report 1: Model Theory, Implementation, and Example Applications [3]
- Jul 2005 ERDC TN-DOER-D4 Particle Tracking Model (PTM) in the SMS: I. Graphical Interface [4]
- Jul 2005 ERDC TN-DOER-D5 Particle Tracking Model (PTM): II. Overview of Features and Capabilities [5]
- Jul 2005 ERDC TN-DOER-D6 Particle Tracking Model (PTM) in the SMS: III. Tutorial with Examples [6]
- Jul 2008 ERDC/CHL CHETN-IV-71 Particle Tracking Model (PTM) in the SMS 10: IV. Link to Coastal Modeling System [7]
- 2007 The Particle Tracking Model: Description and Processes [8]
- Application of the Particle Tracking Model to Predict the Farfield Fate of:
 - 2007 Sediment Suspended by Nearshore Dredging and Placement, Brunswick, GA [9]
 - 2008 Dredged Suspended Sediment at the Willamette River [10]
- 2009 Assessment of Dredging-Induced Sedimentation on Winter Flounder Spawning habitat [11]

Related Topics

- Particle Module
 - Mesh Module
-

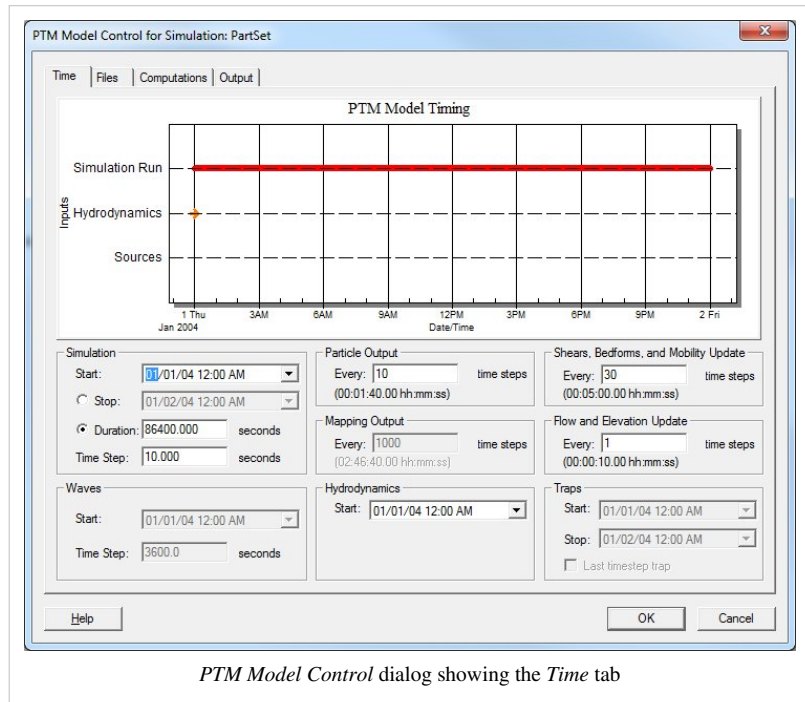
PTM Model Control

The PTM model control is used to create the program control file, which contains general simulation options. Refer to the PTM manual for a more detailed description of how these parameters affect the model results.

Time

The *Time* tab is used to specify time information to be written to the program control file. If a control on the dialog is not enabled, the corresponding keyword will not be written to the program control file. Controls on the dialog are disabled based on the options you have chosen on the other model control tabs. For example, if no trap file is specified on the *Files* tab, the time controls related to traps will not be available. There is a plot on the dialog of the specified times. This tab has the following input fields:

- Simulation
 - Start
 - Stop
 - Duration
 - Time Step
- Particle Output
- Shear, Bedforms, and Mobility Updates
- Mapping Output
- Flow and Elevation Update
- Waves
 - Start
 - Time Step
- Hydrodynamics
- Traps
 - Start
 - Stop
 - Last time step trap



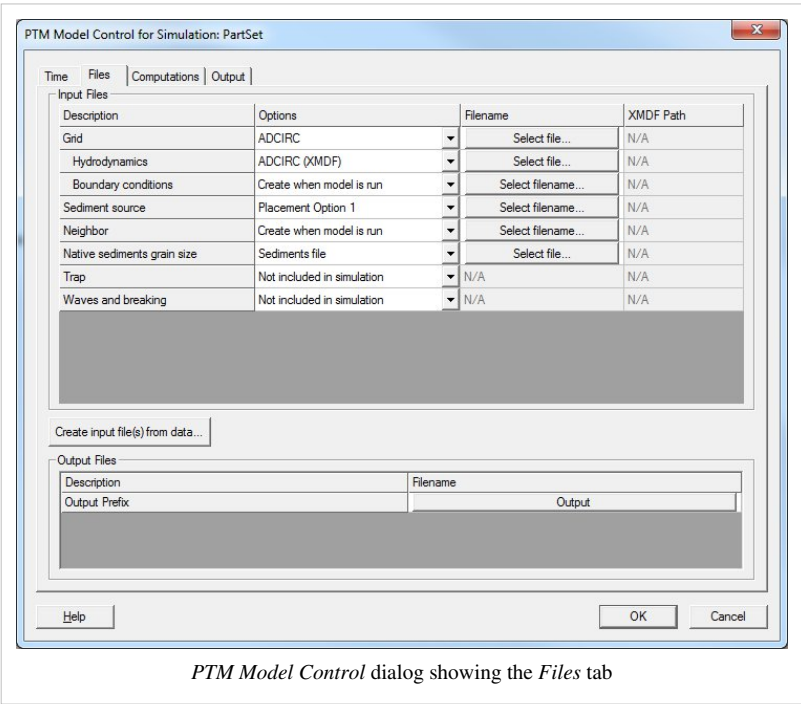
Files

The *Files* tab is used to specify input and output file information to be written to the program control file.

Pressing the **Create input file(s) from data...** button on the *Files* tab will open the *Create PTM External Input Files* dialog.

Native Sediment Grain Size

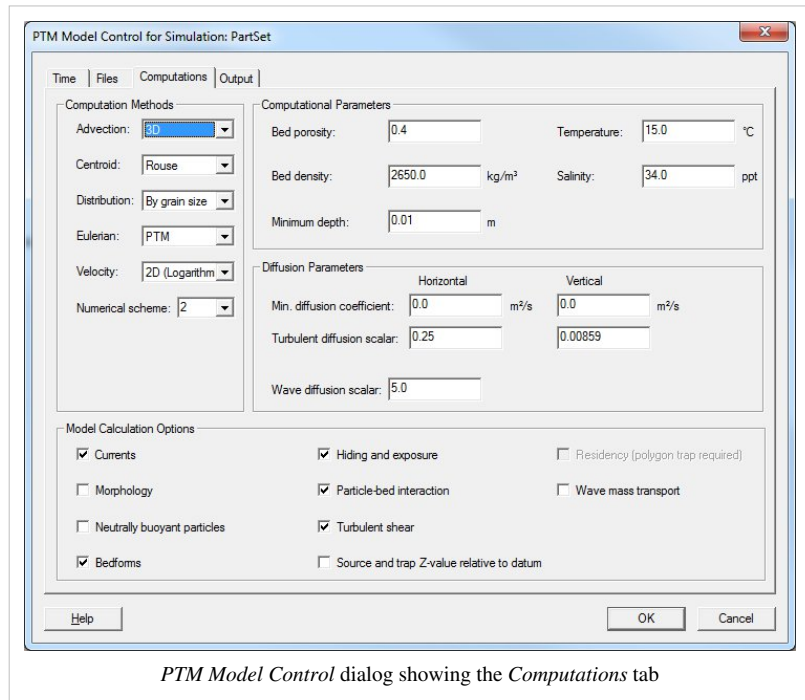
1. If "Uniform bed" is selected from the "Options" column an **Options** button is displayed under "Filename". Pressing this button allows the user to create uniform grain sizes for D35, D50 and D90. If the project is saved, the :SEDIMENT_FORMAT card is created, with the keyword UNIFORM and the 3 grain sizes being written out.
2. If Sediment file is selected, the user can manually select the filename by pressing the button under "Filename".
3. Pressing the **Create input file(s) from data** and selecting to create a Native sediment grain size file will automatically toggle the native sediments grain size to "sediments file" and place the new file under the "Filename".



Computations

The *Computations* tab is used to specify model computation options to be written to the program control file. This tab has the following input fields:

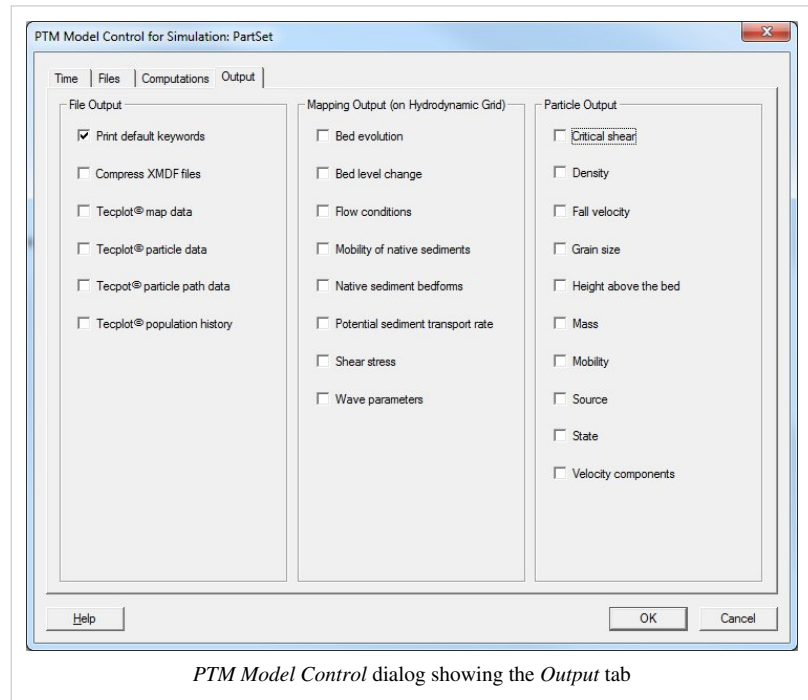
- Computation Methods
 - Advection
 - Centroid
 - Distribution
 - Eulenian
 - Velocity
 - Numerical scheme
- Computational Parameters
 - Bed porosity
 - Temperature
 - Bed density
 - Salinity
 - Minimum depth
- Diffusion Parameters
 - Min. diffusion coefficient
 - Turbulent diffusion scalar
 - Wave diffusion scalar
- Model Calculation Options
 - Currents
 - Morphology
 - Neutrally buoyant particles
 - Bedforms
 - Hiding and exposure
 - Particle-bed interaction
 - Turbulent shear
 - Source and trap Z-value relative to datum
 - Residency (polygon trap required)
 - Wave mass transport



Output

The *Output* tab is used to specify model output options to be written to the program control file. This tab has the following input fields:

- File Output
 - ☒ Print default keywords
 - ☐ Compress XMDf files
 - ☐ Tecplot® map data
 - ☐ Tecplot® particle data
 - ☐ Tecplot® particle path data
 - ☐ Tecplot® population history
- Mapping Output (on Hydrodynamic Grid)
 - ☐ Bed evolution
 - ☐ Bed level change
 - ☐ Flow conditions
 - ☐ Mobility of native sediments
 - ☐ Native sediment bedforms
 - ☐ Potential sediment transport rate
 - ☐ Shear stress
 - ☐ Wave parameters
- Particle Output
 - ☐ Critical shear
 - ☐ Density
 - ☐ Fall velocity
 - ☐ Grain size
 - ☐ Height above the bed
 - ☐ Mass
 - ☐ Mobility
 - ☐ Source
 - ☐ State
 - ☐ Velocity components



PTM Model Control dialog showing the Output tab

Related Topics

- PTM

PTM Graphical Interface

The SMS interface to PTM includes tools for creating input files as well as post-processing capabilities. The PTM interface includes the following components:

PTM Menu

The following menu commands are available in the PTM Menu:

- **New Simulation**
Creates a new PTM simulation and adds it to the Project Explorer.
- **Model Check**
Checks the active PTM simulation for common input errors.
- **Model Control**
Opens the *Model Control* dialog (used to organize input files, specify model parameters, choose output options, etc.).
- **Run Model**
Launches the PTM model using the active PTM simulation input files.

Model Control

The *PTM Model Control* dialog is used to setup the options that apply to the simulation as a whole. These options include time controls, computation parameters, output options, and other global settings.

Running the Model

The PTM Files are written automatically with the SMS project file or can be saved separately using the *File | Save PTM* or *File | Save As* menu commands. See PTM Files for more information on the files used for the PTM run.

PTM can be launched from SMS using the *PTM | Run Model* menu command. A check of some of the common problems called the *Model Checker* is done each time the model is launched, or by selecting the *PTM | Model Check* menu command.

PTM Model Check

The model check will warn you if it suspects a potential error has been made in creating the input files for PTM and suggest how to resolve the issue. The issues shown in the model checker may or may not cause problems, it is left to the modeler to decide if an issue will affect the results of their simulation.

Map Module

PTM sources and traps are created in the Map Module as feature objects in a PTM type coverage.

Feature Points

- Instant Mass Source – Release at a single point in time (Ex. accidental vessel spill)
 - Point Mass Rate Source – Release over a period of time (Ex. leaking pipeline)
 - Vertical Line Source – Release with a uniform distribution along the line and a two-dimensional Gaussian distribution in the plane perpendicular to the line (Ex. bucket dredge)
-

Feature Arcs

- Horizontal Line Source – Release with a uniform distribution along the line and a two-dimensional Gaussian distribution in the plane perpendicular to the line
- Horizontal Line Trap – Count particles crossing a line
- Vertical Area Source – Under development

Feature Polygons

- Horizontal Area Source – Release with a uniform distribution over the area and a Gaussian distribution perpendicular to the source
- Horizontal Area Trap – Count particles entering an area

For more information on sources, see PTM Sources. For more information on traps, see PTM Traps.

Polygon Attributes Dialog

Feature polygons created in a PTM type coverage can be used to create either sources or traps.

Related Topics

- PTM

PTM Particle Filters

Particle Filters

The particle filters allow the user to evaluate specific particles based upon different criteria. The filters act exclusively upon particle datasets. Multiple filters can be used together to filter multiple datasets.

The filters affect:

- Displayed particles
- Selectable particles (hence selected particle info)
- Compute Grid datasets functionality

Dialog Description

The following controls are used to define the particle filters.

Filters

Use the New and Delete buttons to create / remove filters. Each filter may be enable / disabled by clicking its toggle box. The filters are defined by selecting a Dataset whose values will be compared against the operands / operators pairs defined in the Options control.

Datasets

Each filter operates on a single Particle Module dataset. Examples of datasets in the Particle Module include solutions created by the PTM model or using the data calculator.

Options

- Operators – Comparison operators

<	Less than
<=	Less than or equal to
equal	Equal to
not equal	Not equal to
>	Greater than
>=	Greater than or equal to

- Operands – Values compared against the selected dataset using the corresponding operators
- Conjunction – This toggle box disables the second operator / operand pair or performs the and function on them against the first operator / operand pair.

PTM Traps

Overview

A trap is defined as an area into which particles enter and are counted.

Global Trap Options

Global options affect all traps.

- Time limits can be specified on the trap's operation in the *Time* tab of the *PTM Model Control* dialog.
- Traps can be set to only be active on the last time step of the PTM simulation by selecting "Last timestep trap" in the *Time* tab of the *PTM Model Control* dialog.

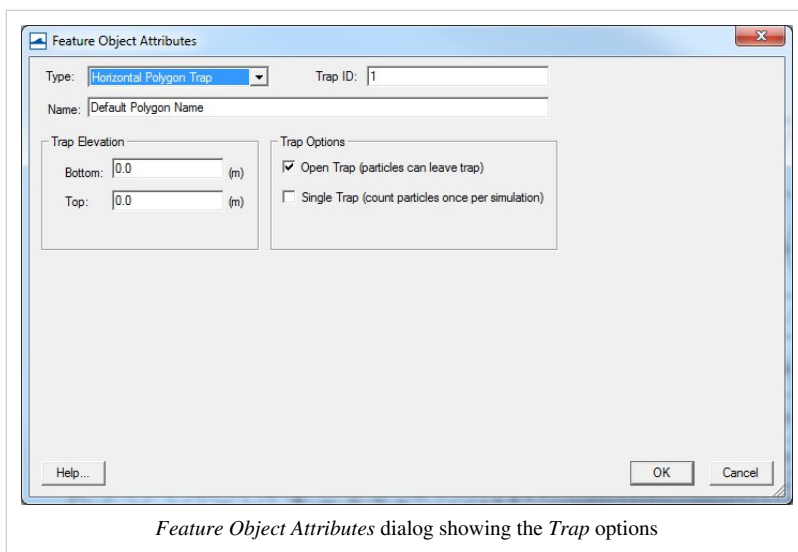
Individual Trap Options

Individual options are specified on a trap by trap basis.

- The trap may be open (particles are free to leave) or closed.
 - Particles can be counted once per simulation (single trap) or every time they re-enter an open trap.
 - The trap can have a bottom and top elevation defining the portion of the water column the trap exists within.
 - ID – Integer value to identify the trap
 - Name – Text to identify the trap
-

PTM Polygon Attribute Dialog

The options for an individual trap can be set in the *Feature Object Attributes* dialog. The dialog is accessed by right-clicking on the trap polygon and selecting the **Attributes** menu command.



Feature Object Attributes dialog showing the Trap options

passing through the trap regardless of direction.

- Increasing x-coordinate – only counts particles passing through the trap moving in the positive x-direction.

Horizontal Polygon Traps

Horizontal polygon traps lie on a horizontal plane and are extruded in the vertical direction based on the trap's top and bottom elevation.

Related Topics

- PTM Trap File
- PTM Sources

Horizontal Line Traps

Horizontal line traps lie in a vertical plane, defined by the trap end points and a top and bottom elevation.

Horizontal line traps have a direction associated with them. The direction of a trap determines which particles the trap will count.

- Decreasing x-coordinate – only counts particles passing through the trap moving in the negative x-direction.
- Either direction – counts particles

4.3.a. PTM Coverages

PTM Coverages

In the SMS interface, two types of coverages can be applied to a PTM simulation. These include:

- The PTM Source/Trap Coverage: used to define input for a PTM simulation.
- The PTM Gage Coverage: used to view time series of output data computed by the PTM engine.

PTM Gage Coverage

A "PTM Gage" coverage has been added to allow the creation of virtual gages. A virtual gage can be associated with either a point or a polygon. When a virtual gage is associated with a point, the user also specifies a radius that defines the area being monitored by the gage (creating a circle).

The gages can be thought of as open traps which monitor the number parcels and material that enter the specified region during the simulation.

When the user invokes the **Particle Module Compute Grid Datasets** command, the user can choose to compute datasets associated with a grid, or with a PTM Gage coverage. When the gage coverage option is selected, SMS computes output data associated with each point and/or polygon. This data consists of sets of xy series curves organized by solution and dataset name. Each curve represents a quantity related to the specific gage. These quantities include count, accumulation, concentrations, rate of exposure and all of the options supported by the compute dataset command including the options to bin concentration, exposure and dosage in the "Z" direction (creating 3D data). The output data is visualized using a PTM Gage plot type in the plot wizard for 2D Plots.

The polygons and points in the virtual gage coverage are created with discrete color attributes. Each point has a radius attribute that represents the area over which the computations will apply.

PTM Trap Output as Gages

In addition to creating virtual point and polygon gages, the PTM Gage Coverage can also be used to view data output by PTM in its Trap Output file. This functionality does not include the concentration, dosage and exposure options, but it does allow the user to visualize data related to closed traps.

4.3.a.1. PTM Sources

PTM Sources

Overview

The material which is to be modeled in PTM is released from sources. The amount of material released from each source is specified as a mass, either as an instantaneously released total mass, or as a mass release rate over a given time period. PTM represents this mass by a finite number of particles.

Particles can be introduced via three different types of sources:

- Point Sources
- Line Sources
- Area Sources

There can be any number of any source type used in a simulation, and different source types can be specified in the same simulation.

Point Sources

Instantaneous

If the material to be modeled is to be released at a single point in time, then an instantaneous source should be specified. An example of this type of release is an accidental spill from a vessel. This type of release occurs at a fixed location, and the full release of material occurs at the time given and with the properties specified.

Varying-Release

If the release of material occurs over a period of time, then a varying-release point source should be specified. An example of this type of release is a leak from a pipeline. The characteristics of release point sources can vary with time (e.g., release rate, three dimensional positions, etc.). Varying-release point sources can be started, stopped, re-started, moved, etc., as directed in a source release schedule, which is developed through the SMS interface.

The horizontal and vertical radii of both types of point sources can be specified in the source release schedule. If either radius is greater than zero, then the initial locations of the individual particles are varied so as to produce a two- or three-dimensional Gaussian-distributed cloud.

Line Sources

Line sources must either be vertical or horizontal and are varying-release. Particles released from a line source will have a uniform distribution along the line and a two-dimensional Gaussian distribution in the plane perpendicular to the line. Line sources are specified by their end points. Line sources may move or change length, position, or discharge properties with time. Linear interpolation in time is used for most properties in a line source, but the characteristics of the release do not vary along the line source (e.g., the release rate of particles can vary with time for a line, but the rate will be the same over the length of that line). To model a line source with varying characteristics along the line, one could use a series of lines positioned end to end, each with different characteristics.

Vertical Line Source Datums

The following vertical line source datums can be used to specify the vertical line source top and bottom elevation:

- Bed (default) – Elevations are specified relative to the bed. The rate is given in kg/m/s by the user.
- Surface – Elevations are specified relative to the water surface. The rate is given in kg/m/s by the user.
- Depth distributed – Elevations are specified as percentages of the water depth. The rate is given in kg/s by the user in this case. An example is if you want to distribute the source over a fraction of the water depth, you could set $z1=0.33$ and $z2=0.67$ to introduce the source over the middle third of the water column. The depth-distributed source is taken as a percentage from the bed. So if $z1 = 0.0$ and $z2$ is 0.5, this suggests a vertical line source that is the lower one half of the water depth.

Area Sources

Area sources lie in a vertical or horizontal plane and are varying-release. Area sources are polygons and are specified by the locations of their vertices. The vertices must be ordered with a counter-clockwise convention. Particles are released from an area source such that there is a uniform distribution over the area and a Gaussian distribution perpendicular to the source. Source properties within an area source are uniform across the polygon.

Estimating generated Parcels

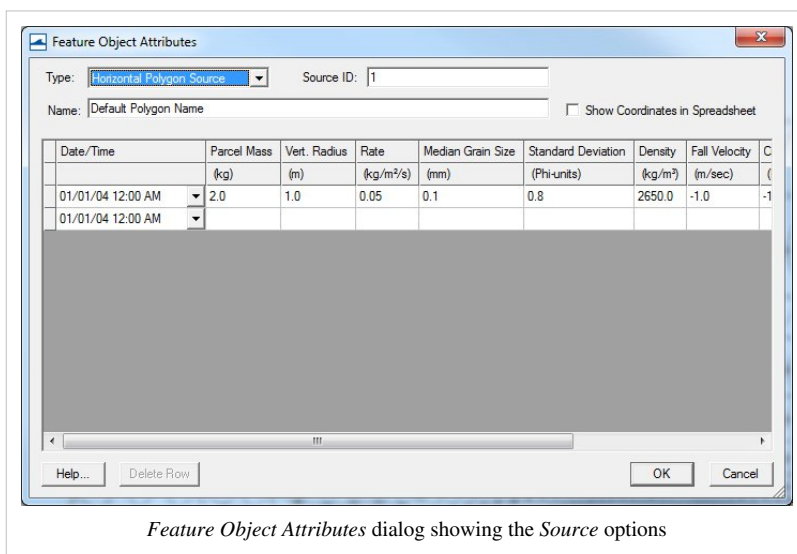
The *Estimate Generate Parcels* dialog allows the user to edit the parcel mass of PTM sources (point, line, or polygon) to estimate the number of parcels created. This can be accessed by right-clicking on the PTM coverage and selecting **Estimate generated Parcels**.

Calculated vs. Specified Values

PTM can calculate the Fall Velocity, Critical Shear Initiation, and Critical Shear Deposition. Specifying a value of -1.0 for any of these parameters will result in the model calculating the value. See the model user manual for information on the equations used to calculate these values.

PTM Polygon Attributes Dialog

The options for an individual source can be set in the *Feature Object Attributes* dialog. The dialog is accessed by right-clicking on the source object and selecting the **Attributes** menu command.



Related Topics

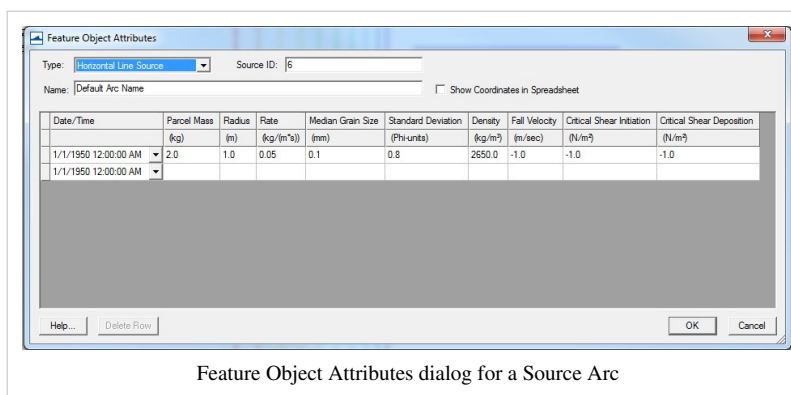
- PTM Source File
- PTM Traps

PTM Arc Attributes Dialog

Feature arcs created in a PTM type coverage can be used to create either sources or traps.

Feature Object Attributes dialog for a Source Arc

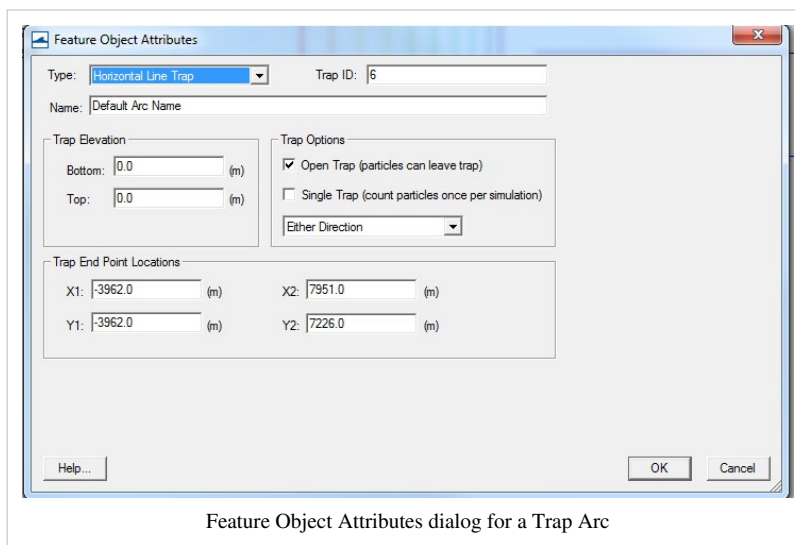
- Type
- Source ID
- Name
- Show Coordinates in Spreadsheet
- Date/Time
- Parcel Mass
- Radius
- Rate
- Median Grain Size
- Standard Deviation



- Density
- Fall Velocity
- Critical Shear Initiation
- Critical Shear Deposition

Feature Object Attributes dialog for a Trap Arc

- Type
- Trap ID
- Name
- Trap Elevation
 - Bottom
 - Top
- Trap Options
 - Open Trap (particles can leave trap)
 - Single Trap (Count particles once per simulation)
- Trap End Point Locations
 - X1
 - X2
 - Y1
 - Y2



Related Topics

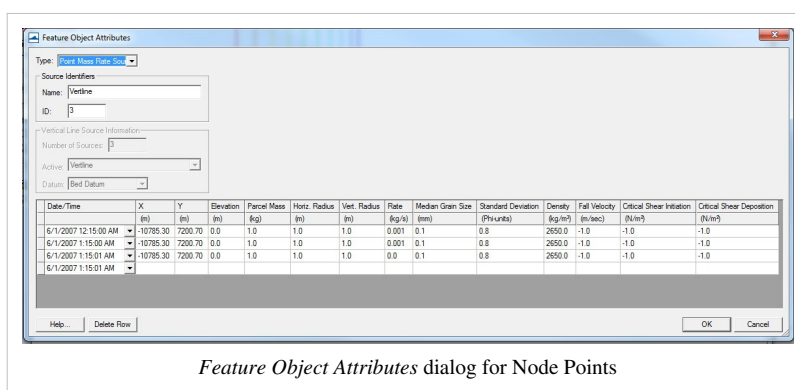
- PTM Sources
- PTM Traps

PTM Feature Point Attributes Dialog

The PTM *feature point attributes* dialog is used to specify point source attributes.

Feature Object Attributes dialog for Node Points

- Type
- Source Identifiers
 - Name
 - ID
- Date/Time
- X
- Y
- Elevation
- Parcel Mass



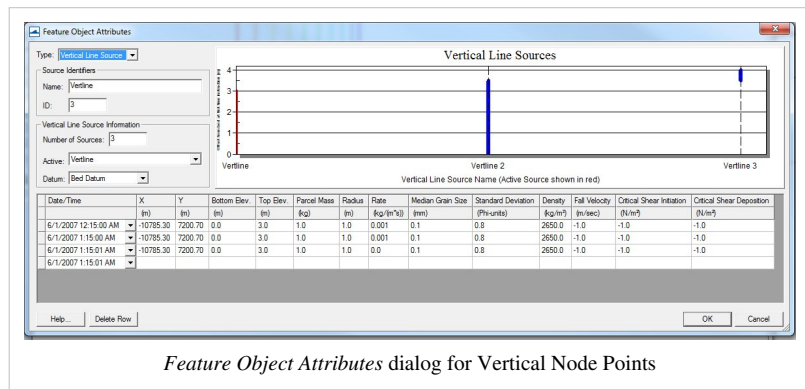
- Horiz. Radius
- Vert. Radius
- Rate
- Median Grain Size
- Standard Deviation

- Density
- Fall Velocity
- Critical Shear Initiation
- Critical Shear Deposition
- Delete Row

Feature Object Attributes dialog for Vertical Node Points

This dialog is the same as the *Feature Object Attributes* dialog for Node Points with the following additions:

- Vertical Line Sources window
- Vertical Line Source Information
 - Numbers of sources
 - Active
 - Datum



Feature Object Attributes dialog for Vertical Node Points

4.3.b. PTM Files

PTM Files

Input and Output files for PTM.

Input Files

- Required files
 - Program Control File (*.pcf) – run time instruction for the model.
 - Mesh file – bathymetry and boundary information for the model.
 - PTM Version 1.0 supports ADCIRC compatible three-noded triangular finite element mesh files (fort.14 or .grd). SMS contains tools to convert mesh files to this format.
 - Hydrodynamic input file – time varying water surface elevation and depth-averaged velocities.
 - The SMS 9.2 interface to PTM Version 1.0 supports XMDf (*.h5) format hydrodynamic files. SMS contains tools to convert mesh output files to this format.
 - Native sediment file – native sediments over the domain defined by the mesh file.
- Optional files
 - Sediment source file – the sediment sources in the simulation.
 - Trap file – locations where information about the simulation should be extracted.
 - Wave file(s) – time varying information about the wave field.
 - Wave breaking file(s) – time varying information about wave breaking.
 - PTM Boundary Condition file – describes the wet/dry interface.

Output Files

The output files use the name given after the keyword :OUTPUT_PREFIX in the program control file and are appended with the following endings:

- **Mapping Output File** (*_maps.h5) – behavior of the flow and native sediments over the whole domain incremented by time.
 - **Parcel Data File** (*_particles.h5) – parcel information incremented by time.
 - **Echo file** (*_input.out) – ASCII file echoing the input parameters of the simulation.
 - **Neighbor file** (*.neighbors) – contains mesh connectivity information. Once generated, can be specified as an input file to avoid regenerating for subsequent simulations with a given mesh. If the mesh changes, the neighbor file should be regenerated.
 - **Trap Count File** (*_count.out) – contains the time and ID of particles entering a trap
 - **Trap Residency File** (*_residency.out) – contains the time particles enter and exit a trap and the total time spent in the trap.
-

PTM Control File

The PTM program control file (*.pcf) contains run time instruction for the model.

File Overview

The program control file (PCF) contains all the information necessary for the PTM to perform the simulation requested. The PCF file is a keyword oriented file. Lines in the file should contain either comments, with the line beginning with a "#" symbol, or a keyword, a single word beginning with a colon, ":", and its associated values. Some keywords have no associated values, e.g.:

```
:MOBILITY_MAPPING
```

These are generally on/off instructions to the model. Keywords pointing to filenames or setting constants have one or two associated values, e.g.:

```
:MESH_FORMAT ADCIRC
:MESH_FILE estuary.grd
```

Keywords and values are not case sensitive with the single exception of the simulation output file names (i.e. file names like "Run_C1_Tp_12s_paths.out" are possible).

The PCF file is read until the instruction :

```
:END_DATA
```

is encountered.

The following list summarizes the rules for construction of a PCF:

- Keywords should be preceded with ":".
- Comments should be preceded with "#".
- Keywords are not case sensitive.
- Keywords and value(s) are white space delimited (tab or space) and, so, should not contain blanks.
- Keywords may be placed in any order in the file.
- If a keyword appears more than once, then the final incidence is used.
- All value units are S.I.
- All times are relative to the clock defined by the currents file.

Example PCF File

```
#####
# PTM Simulation file written by SMS 9.2.1
# SMS Build Date: Mar  9 2007
# Date: 03/09/07
# Time: 16:36:04
#####
:CURRENTS
#
:ID_OUTPUT
:ELEVATION_OUTPUT
:GRAIN_SIZE_OUTPUT
:MOBILITY_OUTPUT
```

```
:STATE_OUTPUT
:PARCEL_MASS_OUTPUT
:OUTPUT_INC 10
:MAPPING_INC 1000
#
:BY_WEIGHT
:GRID_UPDATE 300
#
:MESH_FORMAT ADCIRC
:MESH_FILE estuary.grd
:FLOW_FORMAT XMDF
:FLOW_FILE_XMDF estuary_tide.h5
:XMDF_VEL_PATH Datasets/tide only velocity
:XMDF_WSE_PATH Datasets/tide only wse
:NEIGHBOR_FILE estuary.neighbors
:SOURCE_FILE estuary_Tides.source
:SEDIMENT_FILE estuary.sediments
:OUTPUT_PREFIX estuary_Tides
#
:START_RUN 2004 2 11 0 0 0
:STOP_RUN 2004 2 12 7 45 0
:START_FLOW 2004 2 9 0 0 0
#
:TIME_STEP 3.000000
#
:KET 0.25
:KEV 0.00859
:TEMPERATURE 15
:SALINITY 34
:RHOS 2650
:MIN_DEPTH 0.01
#
:ETMIN 0.02
:EVMIN 0
#
:EMRL_GUID_COV 7d0bcd2f2-f218-4046-ba24-218294d74794
:EMRL_PARTICLE_SET_GUID b7183313-bcba-45d9-8cf7-f6ce96cf1d45
:END_DATA
```

Keywords

Keyword	Value(s)	Action
:waves		Instructs the model that waves are to be runs in the model. Turns on the various wave flags. Default is inactive.
:currents		Instructs the model that currents are to be runs in the model. Turns on the various wave flags. Default is inactive.
:no_parcel		Instructs the model to only perform Eulerian calculations (i.e. no particles are released). Source file is not required if this option is used. Default is inactive.
:ensim		Tells the model that the input bathymetry file uses the CHC EnSim convention of positive upwards. Default is inactive. Default is inactive.
:paths		Instructs the model to output the parcel path files. Default is inactive.
:morphology		Instructs the model to output the predicted bed evolution on the Eulerian mesh node locations. Frequency of output is given by 'mapping_inc'. Same result as ":bed_level_mapping ". Default is inactive.
:flow_mapping		Instructs the model to output the flow conditions on the Eulerian mesh node locations. Frequency of output is given by 'mapping_inc'. Default is inactive.
:bedform_mapping		Instructs the model to output the predicted native sediment bedforms on the Eulerian mesh node locations. Frequency of output is given by 'mapping_inc'. Default is inactive.
:wave_mapping		Instructs the model to output the three predicted wave parameters (H, T, θ) on the Eulerian mesh node locations. Frequency of output is given by 'mapping_inc'. Default is inactive.
:mobility_mapping		Instructs the model to output the predicted mobility of the native sediments on the Eulerian mesh node locations. Frequency of output is given by 'mapping_inc'. Same result as ":morphology". Default is inactive.
:bed_level_mapping		Instructs the model to output the predicted bed evolution on the Eulerian mesh node locations. Frequency of output is given by 'mapping_inc'. Same result as ":morphology". Default is inactive.
:bed_level_change_mapping		Instructs the model to output the predicted bed level change on the Eulerian mesh node locations. Frequency of output is given by 'mapping_inc'. Default is inactive.
:transport_mapping		Instructs the model to output the predicted potential sediment transport rate of the native sediments on the Eulerian mesh node locations. Frequency of output is given by 'mapping_inc'. Default is inactive.
:grain_size_output		Instructs the model to output the grain size of the parcels. Frequency of output is given by 'output_inc'. Default is inactive.
:mobility_output		Instructs the model to output the mobility of each parcel. Frequency of output is given by 'output_inc'. Default is inactive.
:elevation_output		Instructs the model to output the elevation of each parcel. Frequency of output is given by 'output_inc'. Default is inactive.
:height_output		Instructs the model to output the height of each parcel above the bed. Frequency of output is given by 'output_inc'. Default is inactive.
:state_output		Instructs the model to output the state of each parcel (active=1, dormant=0). Frequency of output is given by 'output_inc'. Default is inactive.
:id_output		Instructs the model to output the identification number of each parcel. Frequency of output is given by 'output_inc'. Default is inactive.
:source_output		Instructs the model to output the original source of the each parcel. Frequency of output is given by 'output_inc'. Default is inactive.
:flow_output		Instructs the model to output the velocity components of each parcel. Frequency of output is given by 'output_inc'. Default is inactive.

:density_output		Instructs the model to output the density of each parcel. Frequency of output is given by 'output_inc'. Default is inactive.
:parcel_mass_output		Instructs the model to output the mass of each parcel. Frequency of output is given by 'output_inc'. Default is inactive.
:time_step	<real>	The time step of the simulation in seconds, e.g.: - time_step 2. - Input value required.
:by_weight		Instructs the model to build up the distributions of parcel grain sizes at each source such that they are Gaussian in terms of weight (i.e. "percentage finer by weight"), rather than by grain size. Default is inactive.
:no_bedforms		Instructs the model to skip bedform calculations on the Eulerian grid. Default is inactive.
:bed_porosity		Input statement for bed porosity. Default value is 0.4.
:eulerian_method	ptm or van_rijn	Selects which method to use to determine the native sediment mobility and shears on the Eulerian grid. The two choices are ptm and van_rijn, e.g.: - eulerian_method van_rijn - The ptm method is much faster. Default is ptm.
:centroid_method	rouse or van_rijn	Selects which method to use to determine the parcel mobility and shears. The two choices are rouse and van_rijn, e.g.: - lagrangian_method rouse - The rouse method uses empirical approximations of integrated Rouse curves. The van_rijn method uses a numerical integration of the total sediment transport at the location of the parcel to compute the centroid elevation. The rouse method is much faster. Default is rouse.
:advection_method	1d (formerly 2d – not available in SMS interface) or 2d (formerly q3d - default) or 3d	Selects which method to use for the advection of the parcels. The three choices are 1d, 2d and 3d, e.g.: - advection_method 2d - The 1d method places the parcel at the elevation of the centroid of the local total load distribution. The 2d method allows the parcel to be above or below the total load centroid elevation with the parcel moving vertically towards the centroid. The 3d option allows the parcel to move freely in the vertical in response to a vertical force balance. Default is 2d.
:velocity_method	2d (log) or 2d (uniform) or 2d (two-point) or 3d or 3ds or 3dz	Selects which method to use to compute the velocity profile in the vertical, e.g.: - velocity_method 3dz - The 2d (uniform) option causes the model to use the given velocity throughout the water column. This method is designed primarily for testing. The 2d (log) option causes the model to use a logarithmic velocity profile. The 3d variants force the model to use the given three-dimensional velocity components. These should only be used with three-dimensional input. Default is 2d (log)
:mesh_format	ADCIRC	Gives the format of the mesh file, e.g.: - mesh_format ADCIRC - Options are CMS-2D, CMS-3D, ADCIRC, M2D and CH3D Default value is ADCIRC.

:mesh_file	<character>	Gives the name of the mesh file, e.g.: • mesh_file test.grd • Input value required.
:flow_format	ADCIRC or CMS-2D or CMS-3D or CH3D	Gives the format of the hydrodynamic files: • flow_format ADCIRC • Options are ADCIRC, CMS-2D (for M2D), CMS-3D (for M3D) and CH3D. Default value is ADCIRC.
:flow_files	<character> <character>	Gives names of the hydrodynamic files, e.g.: • flow_files A1.63 A1.64 • Note that the extensions are optional; if a '.' is not found in the filename, the *.63 and *.64 will be appended. Input values required.
:flow_file_uv	<character>	Gives names of the hydrodynamic flow file, e.g.: • flow_file_uv A1.64 • Note that the extensions are optional; if a '.' is not found in the filename, the *.64 will be appended. Input values required. This command allows names with spaces, including directories.
:flow_file_z	<character>	Gives names of the hydrodynamic elevation file, e.g.: • flow_file_z A1.63 • Note that the extensions are optional; if a '.' is not found in the filename, the *.63 will be appended. Input values required. This command allows names with spaces, including directories.
:flow_file_xmdf	<character>	Gives the name of the XMDF format flow file. This is a single file in h5 format.
:flow_file_dst	<character>	Gives name of the density/salinity/temperature file, e.g.: • flow_file_dst A1.45 • This is only used for some ADCIRC-3D model input. If IDEN=0, then no file is required.
:bc_file	<character>	Gives the name of the boundary condition file. Format depends on model type.
:xmdf_wse_path	<character>	Gives the internal data structure path in the XMDF format flow file. The default is 'Datasets/Water Surface Elevation (63)'
:xmdf_vel_path	<character>	Gives the internal data structure path in the XMDF format flow file. The default is 'Datasets/Velocity (64)'
:xmdf_u_path	<character>	Gives the internal data structure path for the U velocity component in the XMDF format flow file (M3D uses three scalars for the three components of velocity). The default is 'Datasets/U'
:xmdf_v_path	<character>	Gives the internal data structure path for the V velocity component in the XMDF format flow file (M3D uses three scalars for the three components of velocity). The default is 'Datasets/U'
:xmdf_w_path	<character>	Gives the internal data structure path for the W velocity component in the XMDF format flow file (M3D uses three scalars for the three components of velocity). The default is 'Datasets/U'
:m2d_control_file	<character>	Gives the name of the M2D control file {*.m2c}. This is only used for M2D version 3.02 and requires the mesh format to be specified as 'm2d', rather than 'cms-2d'
:xmdf_dep_path	<character>	Gives the internal data structure path for the depth in the XMDF format file (M3D only). The default is 'Datasets/Depth'

:xmdf_sal_path	<character>	Gives the internal data structure path for the salinity in the XMDF format file (M3D and CH3D). The default is 'Datasets/Salinity'
:xmdf_temp_path	<character>	Gives the internal data structure path for the temperature in the XMDF format file (M3D and CH3D). The default is 'Datasets/Temperature'
:xmdf_M3D_path	<character>	Gives the internal data structure path for the 3D data in the XMDF format file (M3D only). The default is blank.
:xmdf_d35_path	<character>	Gives the internal data structure path for the D35 values in the XMDF format sediment file. The default is blank.
:xmdf_d50_path	<character>	Gives the internal data structure path for the D50 values in the XMDF format sediment file. The default is blank.
:xmdf_d90_path	<character>	Gives the internal data structure path for the D90 values in the XMDF format sediment file. The default is blank.
:start_flow	yyyy mm dd hh mm ss	Time origin of the ADCIRC flow files, e.g.: - start_flow 1999 12 3 22 30 0 - Each output step of the ADCIRC .63 and .64 files has a timestamp in seconds that is relative to a UTC time selected when the ADCIRC model control file is established. Input values required.
:wave_format	STWAVE or WABED or XMDF	Gives the format of the wave file, e.g.: - wave_format STWAVE - Options are STWAVE, WABED or XMDF. Default value is STWAVE.
:wave_files	<integer>	Gives the number of STWAVE .wav files that are to be used in the present simulation, e.g.: - wave_files 100 - The list of file names should start on the line immediately after the :end_data statement. Input value required if waves are used.
:wave_file_xmdf	<character>	Gives the name of the XMDF input file with wave data, e.g.: - wave_file_xmdf south_waves_example.h5 - This is the main (parent) grid. Input value required.
:wave_file_2_xmdf	<character>	Gives the name of the XMDF input file with wave data, e.g.: - wave_file_xmdf south_waves_example.h5 - This is the nested (child) grid. Input value required.
:wave_frames	<integer>	Gives the number of output frames (steps) that appear in each STWAVE .wav file, e.g.: - wave_frames 1 - In general, STWAVE files will contain a single frame if produced using the SMS steering module and more than one frame if produced using a stand-alone version of STWAVE (e.g. steering module run -> 100 files with one frame each; stand-alone -> 1 file with 100 frames. Input value required if waves are used.
:wave_step	<real>	Gives the time (in seconds) between each output frame (step) in the STWAVE .wav file(s), e.g.: - wave_step 7200. - Input value required if waves are used.
:m2d_step	<real>	Gives the time (in seconds) between each output frame (step) in the XMDF-format M2D file, e.g.: - m2d_step 3600. - New XMDF files have the time units specified, so this may now longer be required.

:wave_grid_angle	<real>	Gives the orientation angle (in degrees, clockwise from y-axis to N) of the STWAVE grid, e.g.: • wave_grid_angle 20. • Input value required if waves are used.
:wave_x_origin	<real>	x-ordinate of the STWAVE grid origin, e.g.: • wave_x_origin 1226722.50 • Input value required if waves are used.
:wave_y_origin	<real>	y-ordinate of the STWAVE grid origin, e.g.: • wave_y_origin 39990.15 • Input value required if waves are used.
:nested		Instructs the model that nested STWAVE grids are to be used. The list of file names should start on the line immediately after the list of parent STWAVE file names. The nested and parent STWAVE grids must have the same number of frames and the same times associated with each frame.
:wave_grid_angle_2	<real>	Gives the orientation angle (in degrees, clockwise from y-axis to N) of the nested STWAVE grid, e.g.: • wave_grid_angle 20. • Input value required if waves are used.
:wave_x_origin_2	<real>	x-ordinate of the nested STWAVE grid origin, e.g.: • wave_x_origin 1226722.50 • Input value required if waves are used.
:wave_y_origin_2	<real>	y-ordinate of the nested STWAVE grid origin, e.g.: • wave_y_origin 39990.15 • Input value required if waves are used.
:start_waves	yyyy mm dd hh mm ss	Time of the first STWAVE frame, e.g.: • start_waves 2004 12 3 0 30 0 • Input values required if waves are used.
:exceed_waves		Instructs model to hold waves steady, if simulation exceeds duration of wave input file
:source_file	<character>	Gives the name of the file containing the sediment source information, e.g.: • source_file test_2.source • Input value required.
:neighbor_file	<character>	Gives the name of the file containing the neighbor source information, e.g.: • neighbor_file test.neighbor • The model will generate a file if one is not supplied. Input value required.
:output_prefix	<character>	Gives the prefix of the files used for output, e.g.: • output_prefix west_test_1 • The model will append an extension depending on the nature of the output file. Input value required.
:undertow		The model computes wave undertow. Default is inactive. Needs to be made more general.
:trap_file	<character>	Gives the name of the trap file, e.g.: • trap_file trap_A.dat • See trap section for details. If this keyword command is not issued, then no trap is present. Default is no trap.
:xmdf_grid_path	<character>	Path to the main folder in an M2D h5 file. This is the folder with such data as the Origin, Bearing etc. The datasets folder is usually below this one.
:last_step_trap		Surveys all particles on the last time step and places them in a trap, whether they are moving or not.

:start_trap	yyyy mm dd hh mm ss	Time at which the trap, if specified, is operational, e.g.: - start_trap 2004 12 4 16 0 50 - Input values required if trap is used. Default start time is the start of the simulation.
:stop_trap	yyyy mm dd hh mm ss	Time at which the trap, if specified, ceases operation, e.g.: - stop_trap 2004 12 4 19 20 0 - Input values required if trap is used. Default stop time is the end of the simulation.
:sediment_format	ADCIRC or X MDF_PROPERTY or X MDF_DATASET or m2d	Gives the format of the native sediment file, e.g.: - sediment_format ADCIRC - Default value is ADCIRC.
:sediment_file	<character>	Gives the name of the file containing the grain sizes of the native sediments, e.g.: sediment_file sed-A2.grd
:start_run	yyyy mm dd hh mm ss	Start time of the simulation, e.g.: - start_run 2004 11 4 16 0 0 - Input value required.
:stop_run	yyyy mm dd hh mm ss	Stop time of the simulation, e.g.: - stop_run 2004 11 14 13 30 0 - Must specify either :duration or :stop_run.
:duration	<real>	The duration of the simulation in seconds. Must specify either :duration or :stop_time.
:grid_update	<integer>	The increment in steps between updating of the mobility, shears and bedforms of the Eulerian grid
:flow_update	<Integer>	The increment in steps between updating of the flows and elevations of the Eulerian grid
:ket	<real>	Diffusion scalar. Default value is 0.25.
:temperature	<real>	The water temperature in °C. The default value is 15°C.
:salinity	<real>	The water salinity in ‰. The default value is 34 ‰.
:output_inc	<integer>	The increment in number of steps between output of the Lagrangian parcel data. The default value is 10.
:mapping_inc	<integer>	The increment in number of steps between output of the Eulerian grid data. The default value is 1000.
:min_depth	<real>	The minimum computation depth in m. The default value is 0.01 m.
:debug		Tells the model to turn off certain random generators routines and output stage times
:ensim_parcel		The model outputs a file with particle data in EnSim format (.pcl)
:ensim_maps		The model outputs a file with map data in EnSim format (.t3s)
:tecplot_parcel		The model outputs a file with particle data in Tecplot format (.plt)
:tecplot_maps		The model outputs a file with map data in Tecplot format (.plt)
:no_xmdf_parcel		The model will not output a parcel data file in X MDF format (.h5)
:no_xmdf_maps		The model will not output a map data file in X MDF format (.h5)
:population_record		A file with population history (e.g. particles born, trapped, deposited, etc.) is output as in Tecplot format as *_population.plt
:tau_cr_output		Instructs the model to output the critical shear for the initiation of motion of each parcel. Frequency of output is given by 'output_inc'. Default is inactive.

:fall_velocity_output		Instructs the model to output the fall velocity of each parcel. Frequency of output is given by 'output_inc'. Default is inactive.
:xmdf_compressed		Instructs the model to use compressed XMDF files. Default is inactive.
:no_hiding_exposure		Turns off hiding and exposure routines. Default is active.
:no_turbulent_shear		Turns off probabilistic shear calculations. Default is active.
:no_bed_interaction		Turns off bed interaction routines. Default is active.
:no_wave_mass_transport		Turns off wave mass transport calculations. Default is active.
:wave_mass_transport		Turns on wave mass transport calculations. Default is inactive.
:residence_calc		Model performs residency calculations (requires a polygon trap to be active)
:neutrally_buoyant		Model assumes all particles are neutrally buoyant (i.e. particle density set to that of the fluid and fall velocity is set to 0.). Default is inactive.
:bottom_flow_format	ADCIRC or XMDF	Gives the format of the bottom currents file. Options are 'Default is 'adcirc'.
:xmdf_bot_path	<character>	Gives the internal data structure path in the XMDF format flow file. The default is 'Datasets/Velocity (64)'
:bottom_flow_file	<character>	Gives the name of the bottom currents file.
:bottom_flow_height	<real>	Sets the height of the input bottom currents.
:bottom_mask_file	<character>	Gives the locations of the nodes where bottom currents are provided (1) and not provided (0)
:bottom_start_flow	yyyy mm dd hh mm ss	Time origin of the bottom flow files, e.g.: - bottom_start_flow 1997 10 5 12 45 0 - Each output step of the bottom flow file has a timestamp in seconds that is relative to a UTC time selected when the ADCIRC model control file is established. Input values required.
:wave_breaking		Instructs model to open a *.brk file for each *.wav file opened and to use breaking indices (Bi) estimate increased surf zone mixing.
:kew	<real>	Sets diffusion coefficient for waves. Default is 5. The horizontal eddy viscosity is adjusted by the factor:
:kev	<real>	Sets vertical diffusion coefficient. Default is 0.00859.
:etmin	<real>	Sets minimum vertical eddy viscosity. Default is 0.02.
:evmin	<real>	Sets minimum vertical eddy viscosity. Default is 0.0.
:source_to_datum		Sets the z-value of parcels released by sources to a z-value relative to datum rather than the bed
:numerical_scheme	<Integer>	Controls the order of the numerical scheme. Only valid options are 2 or 4.
:rhos	<real>	Sets the density of the bed sediments. Default is 2650.
:emrl_particle_set_guid	<character>	Used by SMS. Model will pass string to output files.
:emrl*		Model will ignore lines with these first four characters, except :emrl_particle_set_guid.
:end_data		Indicates the end of the keyword part of the file. The only contents of the file beyond this point should be the names of the wave files (if required).

Related Topics

- Particle Module
- Particle Tracking Model (PTM)
- Sediment File
- Source File
- Trap File

PTM Sediment File

The PTM native sediment file contains the grain size information for the native sediments in terms of its D35, D50, and D90.

File Overview

The native sediments file contains the spatially-varying grain size information for the native sediments in terms of D35, D50, and D90. Frictional characteristics of the bed are computed with D90. The D50 value is used in the prediction of bed forms, in the determination of sediment mixing routines that influence reentrainment of deposited particles, and in the hiding and exposure routines that influence the critical shear stress of deposited particles. The D35 value is used in the determination of the suspended sediment transport if the van Rijn approach is selected for the centroid method. Non-erodible areas (e.g., rock outcroppings) can be identified by a negative input grain size. This absolute value of the grain size is treated as an effective roughness height, ks' and ks'' .

The native sediments filename is specified on the Files page of the PTM model control. A native sediments file can be generated by the SMS interface, if one is required, by pressing the "Create input file(s) from data..." button on the Files tab of the PTM model control. This will open the Create PTM External Input Files dialog.

The native sediment file can be in ASCII or XMDF format.

ASCII File Overview

An ASCII format native sediment file is used when using an ADCIRC (fort.14 or *.grd), or CMS-Flow format mesh file (see PCF File keywords :MESH_FORMAT and :SEDIMENT_FORMAT).

1. The first line is a comment line that is not read by PTM.
2. The second line contains the number of elements and the number of nodes of the mesh.
3. The next section contains a single line for each node, which gives the node number, easting (m), northing (m), D35 (mm), D50 (mm), and D90 (mm).
4. The file extension should be specified as .sediments (for SMS)

Example ASCII Native Sediment Data File

Node	ID	X (m)	Y (m)	D35 (mm)	D50 (mm)	D90 (mm)
2704	1401					
1		54010.00	60160.00	0.13	0.19	0.3
2		52008.90	60146.50	0.13	0.19	0.3
3		50007.80	60133.00	0.13	0.19	0.3
4		48006.70	60119.40	0.13	0.19	0.3
5		46005.60	60105.90	0.13	0.19	0.3

[Continued...]

1397	-6825.58	-408.412	0.13	0.19	0.3
1398	-6876.47	-481.815	0.13	0.19	0.3
1399	-6950.30	-414.314	0.13	0.19	0.3
1400	-6693.29	-533.139	0.13	0.19	0.3
1401	-6793.13	-526.903	0.13	0.19	0.3

In this example, there are two information lines (one comment line and one line with the number of elements and number of nodes) followed by 1,401 lines of data. Each node has sediment with a D35 of 0.13 mm, D50 of 0.19 mm, and D90 of 0.3 mm.

XMDF File Overview

An XMDF format native sediment file is used when using an XMDF format mesh file (see keywords :MESH_FORMAT and :SEDIMENT_FORMAT).

1. D35, D50, and D90 will be the same for every node when given as a property.
2. D35, D50, and D90 can be spatially varied when given as a dataset.

Related Topics

- Particle Module
- Particle Tracking Model (PTM)
- Program Control File (PCF File)
- Source File
- Trap File

PTM Source File

The PTM sediment source file contains the time varying location and parameters assigned to a source.

File Overview

The first line contains the number of Instant Mass Sources, NIMS. This is a single integer value. Any comments can follow on the line, e.g.:

```
1 Instant Mass Source(s)
```

This is followed by NIMS blocks of data. Each block of data contains:

- An identification line giving the source id number, the number of instructions, NIS, and a label. In this example, there is one instant mass source, known as “My Instant Mass Source”, that has an id of zero and two instructions:

```
0 2 My Instant Mass Source
```

- NIS lines of instructions giving the time to issue the instruction (year, month, day, hour, minute, second), the x, y, and z location of the point at that time (m), the mass of each parcel (kg), the horizontal radius of the source (m), the vertical radius of the source, the mass of the source (kg), the grain size (m), the standard deviation of the sediment distribution (Phi-units), the density (kg/m³), the fall velocity (-1 to have PTM compute) (m/sec), Tau critical initiation (-1 to have PTM compute) (N/m²), and Tau critical deposition (-1 to have PTM compute) (N/m²) e.g.;

```
2004 10 6 12 0 0 4.764000000000000e+003 2.638000000000000e+003 8 2 1 1 180 2e-007 0.8 2650 -1 -1 -1
2004 10 7 20 0 0 4.764000000000000e+003 2.638000000000000e+003 8 2 1 1 180 2e-007 0.8 2650 -1 -1 -1
```

The next set of data contains information for Point Mass Rate Sources. This begins with the number of Point Mass Rate sources, NPMRS. This is a single integer value. Any comments can follow on the line, e.g.:

```
2 Point Mass Rate Source(s)
```

This is followed by NPMRS blocks of data. Each block of data contains:

- An identification line giving the source id number, the number of instructions, NIS, and a label. e.g.:

```
0 4 My Mass Rate Source
```

- NIS lines of instructions giving the time to issue the instruction (year, month, day, hour, minute, second), the x, y, and z location of the point at that time (m), the horizontal radius of the source (m), the vertical radius of the source, the mass rate of the source (kg/s), the grain size (m), the standard deviation of the sediment distribution (Phi-units), the density (kg/m³), the fall velocity (-1 to have PTM compute) (m/sec), Tau critical initiation (-1 to have PTM compute) (N/m²), and Tau critical deposition (-1 to have PTM compute) (N/m²) e.g.;

```
2004 10 6 10 0 0 4.745000000000000e+003 3.762000000000000e+003 10 2 2 2 0.05 2e-006 0.8 2650 -1 -1 -1
2004 10 6 11 0 0 4.745000000000000e+003 3.762000000000000e+003 10 2 2 2 0.05 2e-006 0.8 2650 -1 -1 -1
2004 10 6 11 0 1 4.745000000000000e+003 3.762000000000000e+003 10 2 2 2 0 2e-006 0.8 2650 -1 -1 -1
2004 10 7 20 0 0 4.745000000000000e+003 3.762000000000000e+003 10 2 2 2 0 2e-006 0.8 2650 -1 -1 -1
```

The next set of data contains information for Line Sources. This begins with the number of Line Sources, NLS. This is a single integer value. Any comments can follow on the line, e.g.:

```
1 Line Source(s)
```


This is followed by NLS blocks of data. Each block of data contains:

- An identification line giving the line source id number, the number of instructions, NIS, and a label, e.g.:

```
1 4 My Horizontal Line Source
```

- NIS lines of instructions giving the time to issue the instruction (year, month, day, hour, minute, second), the x, y, and z location of the first end point of the line at that time (m), the x, y, and z location of the second end point of the line at that time (m), the horizontal radius of the source (m), the vertical radius of the source, the mass rate of the source (kg/s/m), the grain size (m), the standard deviation of the sediment distribution (Phi-units), the density (kg/m³), the fall velocity (-1 to have PTM compute) (m/sec), Tau critical initiation (-1 to have PTM compute) (N/m²), and Tau critical deposition (-1 to have PTM compute) (N/m²) e.g.;

```
2004 10 6 18 0 0 3.575000000000000e+003 3.845000000000000e+003 10
3.575000000000000e+003 1.570000000000000e+003 10 2 1 1 2.2e-005 2.5e-008
0.8 2650 -1 -1 -1
2004 10 6 19 0 0 3.575000000000000e+003 3.845000000000000e+003 10
3.575000000000000e+003 1.570000000000000e+003 10 2 1 1 2.2e-005 2.5e-008
0.8 2650 -1 -1 -1
2004 10 6 19 0 1 3.575000000000000e+003 3.845000000000000e+003 10
3.575000000000000e+003 1.570000000000000e+003 10 2 1 1 0 2.5e-008
0.8 2650 -1 -1 -1
2004 10 7 20 0 0 3.575000000000000e+003 3.845000000000000e+003 10
3.575000000000000e+003 1.570000000000000e+003 10 2 1 1 0 2.5e-008
0.8 2650 -1 -1 -1
```

The next set of data contains information for Polygon Sources. The points must be ordered using a standard counter-clockwise convention. This begins with the number of Polygon Sources, NPS. This is a single integer value. Comments can follow on the line, e.g.:

```
1 Polygon Source(s)
```

This is followed by NPS blocks of data. Each block of data contains:

- An identification line giving the source id number, the number of instructions, NIS, and a label, e.g.:

```
1 2 My Polygon Source
```

- NIS blocks of instructions. Each block of instructions contains:
 - A line with the time to issue the instruction (year, month, day, hour, minute, second). For each point in the polygon (1 point per line), the x, y, and z location of the point (m). The final line of the block of instructions contains the horizontal radius of the source (m), the vertical radius of the source (m), the mass rate of the source (kg/s/m²), the grain size (m), the standard deviation of the sediment distribution (Phi-units), the density (kg/m³), the fall velocity (-1 to have PTM compute) (m/sec), Tau critical initiation (-1 to have PTM compute) (N/m²), and Tau critical deposition (-1 to have PTM compute) (N/m²) e.g.;

```
2000 1 1 8 0 0
2451 4195 2
2453 4560 2
2183 4544 2
2000 4342 2
2261 4129 2
1 0 0 .00001 0.0003 0.4 2650 -1 -1 -1
```

The following list summarizes the rules for construction of a sediment source file:

- Data files must contain an entry stating the number of each type of sediment source, even if one or more sediment source types are not used.
- All sediment sources are time varying and are accompanied by a list of instructions.
- Linear interpolation between instructions is used to obtain source characteristics at a given time. All characteristics are interpolated. A constant source can be defined by two instructions with all data, except the start and stop times constant.
- The final instruction must have a time later than the stop time of the model.

Example Source File

```
1 Instant Mass Source(s)
3 2 My Instant Mass Source
2004 10 6 12 0 0 4.764000000000000e+003 2.638000000000000e+003 8 2 1 1
180 2e-007 0.8 2650 -1 -1 -1
2005 10 6 20 0 0 4.764000000000000e+003 2.638000000000000e+003 8 2 1 1
180 2e-007 0.8 2650 -1 -1 -1
2 Point Mass Rate Source(s)
2 4 My Moving Point Mass Rate Source
2004 10 6 16 0 0 4.091000000000000e+003 3.375000000000000e+003 8 2 3 3
0.05 5e-008 0.8 2650 -1 -1 -1
2004 10 6 17 0 0 4.186000000000000e+003 1.907000000000000e+003 8 2 3 3
0.05 5e-008 0.8 2650 -1 -1 -1
2004 10 6 17 1 0 4.091000000000000e+003 3.375000000000000e+003 8 2 3 3
0 5e-008 0.8 2650 -1 -1 -1
2005 10 6 20 0 0 4.091000000000000e+003 3.375000000000000e+003 8 2 3 3
0 5e-008 0.8 2650 -1 -1 -1
1 4 My Point Mass Rate Source
2004 10 6 10 0 0 4.745000000000000e+003 3.762000000000000e+003 10 2 2 2
0.05 2e-006 0.8 2650 -1 -1 -1
2004 10 6 11 0 0 4.745000000000000e+003 3.762000000000000e+003 10 2 2 2
0.05 2e-006 0.8 2650 -1 -1 -1
2004 10 6 11 1 0 4.745000000000000e+003 3.762000000000000e+003 10 2 2 2
0 2e-006 0.8 2650 -1 -1 -1
2005 10 6 20 0 0 4.745000000000000e+003 3.762000000000000e+003 10 2 2 2
0 2e-006 0.8 2650 -1 -1 -1
1 Line Source(s)
4 4 My Horizontal Line Source
2004 10 6 18 0 0 3.575000000000000e+003 3.845000000000000e+003 10
3.575000000000000e+003 1.570000000000000e+003 10 2 1 1 2.2e-005 2.5e-008
0.8 2650 -1 -1 -1
2004 10 6 19 0 0 3.575000000000000e+003 3.845000000000000e+003 10
3.575000000000000e+003 1.570000000000000e+003 10 2 1 1 2.2e-005 2.5e-008
0.8 2650 -1 -1 -1
2004 10 6 19 1 0 3.575000000000000e+003 3.845000000000000e+003 10
3.575000000000000e+003 1.570000000000000e+003 10 2 1 1 0 2.5e-008 0.8
2650 -1 -1 -1
```

```

2005 10 6 20 0 0 3.575000000000000e+003 3.845000000000000e+003 10
3.575000000000000e+003 1.570000000000000e+003 10 2 1 1 0 2.5e-008 0.8
2650 -1 -1 -1

```

```
1 Polygon Source(s)
```

```
5 12 2 My Polygon Source
```

```
2004 10 6 12 0 0
```

```

4.258000000000000e+003 1.884000000000000e+003 0
4.139000000000000e+003 2.040000000000000e+003 0
4.144000000000000e+003 2.191000000000000e+003 0
4.196000000000000e+003 2.289000000000000e+003 0
4.377000000000000e+003 2.341000000000000e+003 0
4.533000000000000e+003 2.310000000000000e+003 0
4.595000000000000e+003 2.279000000000000e+003 0
4.647000000000000e+003 2.175000000000000e+003 0
4.647000000000000e+003 2.082000000000000e+003 0
4.585000000000000e+003 1.968000000000000e+003 0
4.507000000000000e+003 1.916000000000000e+003 0
4.377000000000000e+003 1.900000000000000e+003 0

```

```
2 1 1 0.05 0.0001 0.8 2650 -1 -1 -1
```

```
2005 10 6 20 0 0
```

```

4.258000000000000e+003 1.884000000000000e+003 0
4.139000000000000e+003 2.040000000000000e+003 0
4.144000000000000e+003 2.191000000000000e+003 0
4.196000000000000e+003 2.289000000000000e+003 0
4.377000000000000e+003 2.341000000000000e+003 0
4.533000000000000e+003 2.310000000000000e+003 0
4.595000000000000e+003 2.279000000000000e+003 0
4.647000000000000e+003 2.175000000000000e+003 0
4.647000000000000e+003 2.082000000000000e+003 0
4.585000000000000e+003 1.968000000000000e+003 0
4.507000000000000e+003 1.916000000000000e+003 0
4.377000000000000e+003 1.900000000000000e+003 0

```

```
2 1 1 0.05 0.0001 0.8 2650 -1 -1 -1
```

```
*****
```

```
* Guide:
```

```
*****
```

```
Number of Instant Mass Source(s)
```

```

time(6), x_location, y_location, z_location, Pmass(kg), Hradius(m),
Vradius(m), mass(kg), grain_size(m), standard_dev(Phi-units),
density(kg/m^3), fall_velocity(m/s) (-1 = compute),
TAU_critical_initiation(N/m^2) (-1 = compute),
TAU_critical_deposition(N/m^2 (-1 = compute)

```

```
Number of Point Mass Rate Source(s)
```

```

time(6), x_location, y_location, z_location, Pmass(kg), Hradius(m),
Vradius(m), rate(kg/s), grain_size(m), standard_dev(Phi-units),
density(kg/m^3), fall_velocity(m/s) (-1 = compute),
TAU_critical_initiation(N/m^2) (-1 = compute),

```

```

TAU_critical_deposition(N/m^2 (-1 = compute)
Number of Line Source(s)
    time(6), x1_loc, y1_loc, z1_loc, x2_loc, y2_loc, z2_loc, Pmass(kg),
Hradius(m), Vradius(m), rate(kg/s/m), grain_size(m), phi_sd(Phi-units),
    density(kg/m^3), fall_velocity(m/s) (-1 = compute),
TAU_critical_initiation(N/m^2) (-1 = compute),
TAU_critical_deposition(N/m^2 (-1 = compute)
Number of Polygon Source(s)
    time(6)
    For each point (1 point per line): x_location, y_location,
z_location
    Pmass(kg), Hradius(m), Vradius(m), rate(kg/s/m^2), grain_size(m),
standard_dev(Phi-units), density(kg/m^3), fall_velocity(m/s) (-1 =
compute), TAU_critical_initiation(N/m^2) (-1 = compute),
TAU_critical_deposition(N/m^2 (-1 = compute)
*****
* Source file written by SMS 10.0.0 Development
* SMS Build Date: Jun 20 2007
* Date saved: 06/20/07
* Time saved: 09:43:14
*****

```

Vertical Line Source Datums

The source file format has been changed to allow the specification of vertical line source datums:

Old Format

```

# Line Source(s)
LineSourceID  #Instructions  SourceName

```

New Format

```

# Line Source(s)
LineSourceID  #Instructions  SourceName  SourceDatum

```

The values available for the SourceDatum field are:

- beddatum (default)
- surfacedatum
- depthdistributed

The Sourcedatum has a default value of beddatum. If no datum is given in the file, then it is assumed to use the bed as the datum. Therefore, PTM will still process source files in the old format correctly.

Relative to the bed

This is the default and is what was already in PTM. Given the following input from the source file:

Time	X1	Y1	Z1	X2	Y2	Z2	PMass	Hrad	Vrad	Rate	Additional Info

Relative to the water surface

If the SourceDatum is surfacedatum then the code takes Z1 and Z2 as relative to the water surface. The rate is given in kg/m/s by the user.

Depth-distributed

If the SourceDatum is depthdistributed then the code takes the Z1 and Z2 as percentages of the water depth. The rate is given in kg/s by the user in this case. An example is if you want to distribute the source over a fraction of the water depth, you could set z1=0.33 and z2=0.67 to introduce the source over the middle third of the water column. The depth-distributed source is taken as a percentage from the bed. So if z1 = 0 and z2 is 0.5, this suggests a segment that is the lower one half of the water depth.

Related Topics

- Particle Module
- Particle Tracking Model (PTM)
- Program Control File (PCF File)
- Sediment File
- Trap File

PTM Wave File

The PTM wave input files contain the time-varying phase-averaged wave information for the simulation in terms of significant wave height, Hs, peak period, Tp, and direction.

File Overview

The wave input files must contain information for each node in the mesh input file. Wave input files are usually time-varying. The wave input file can be in ASCII or XMDF format.

ASCII File Overview

ASCII format wave input files are used when using STWAVE or CMS-Wave (WABED) results.

Example ASCII Wave File

Examples of ASCII Wave files can be viewed in the CMS-Wave Wave Breaking File and CMS-Wave Wave Output File articles.

XMDF File Overview

Example XMDF Wave File

TODO: Link to CMS-Wave file

Related Topics

- PTM Files

PTM Trap File

A PTM trap file defines areas into which parcels enter and are counted. The traps can be open or closed traps. When a parcel enters an open trap, it is free to leave the trap. When a parcel enters a closed trap, it is not allowed to leave the trap. If the trap is a single trap, each parcel is only counted once per simulation.

File Overview

The first line contains the number of line traps, NLT. This is a single integer value. Any comments can follow on the line, e.g.:

```
1 Line Trap(s)
```

This is followed by NLT blocks of data. Each block of data contains:

- An identification line giving the trap id number, and a label. In this example, there is one line trap, known as “My Line Trap”, that has an id of two:

```
2 My Line Trap
```

- A single line giving the trap direction, trap bottom elevation, trap top elevation, open trap flag, and single count trap flag:

```
0 3 15 0 1
```

- Two lines giving the x, y location of the line trap endpoints:

```
3.990000000000000e+001 1.076000000000000e+002
4.920000000000000e+001 1.782000000000000e+002
```

The next set of data contains information for Polygon Traps. The points must be ordered using a standard counter-clockwise convention. This begins with the number of Polygon Traps, NPT. This is a single integer value. Comments can follow on the line, e.g.:

```
1 Polygon Trap(s)
```

This is followed by NPS blocks of data. Each block of data contains:

- An identification line giving the trap id number, and a label, e.g.:

```
1 My Polygon Trap
```

- A single line giving the number of points in the trap, NP, trap bottom elevation, trap top elevation, open trap flag, and single count trap flag:

```
8 2 5 1 0
```

- For each point in the polygon (1 point per line), the point id, x, y location of the point:

```
1 1.7860000000000000e+001 2.3930000000000000e+001
2 3.5420000000000000e+001 1.4670000000000000e+001
3 7.2940000000000000e+001 2.0420000000000000e+001
4 7.9160000000000000e+001 5.4260000000000000e+001
5 6.1120000000000000e+001 7.8840000000000000e+001
6 2.9360000000000000e+001 8.1400000000000000e+001
7 9.0800000000000000e+000 6.5590000000000000e+001
8 3.9700000000000000e+000 5.1860000000000000e+001
```

Trap File Flag Values

Flag	Value(s)
Trap direction	-1 (decreasing x-coordinate), 0 (any direction), or 1 (increasing x-coordinate)
Open trap	0 (closed trap) or 1 (open trap)
Single count trap	0 (Count parcels every time they enter a trap) or 1 (Count parcels once per simulation)

If the keyword :LAST_STEP_TRAP is included in the Program Control File (PCF File), the traps will only be active on the last time step of the PTM simulation.

Example Trap File

```
1 Line Trap(s)
0 My Line Trap
0 2 12 1 0
-5.2800000000000000e+000 7.0860000000000000e+001
4.6760000000000000e+001 8.5710000000000000e+001
1 Polygon Trap(s)
1 My Polygon Trap
6 5 20 0 1
1 3.5580000000000000e+001 1.5630000000000000e+001
2 6.5430000000000000e+001 1.4510000000000000e+001
3 7.6450000000000000e+001 2.9040000000000000e+001
4 5.9210000000000000e+001 5.4260000000000000e+001
5 3.3670000000000000e+001 5.1860000000000000e+001
6 2.1850000000000000e+001 3.0630000000000000e+001
*****
* Guide:
*****
Number of Line Traps(s)
For each Line Trap:
Trap ID, Trap Name
Trap Direction, Trap Bottom, Trap Top, Open Trap, Single Trap
```

```

    First node x1_location, y1_location
    Second node x2_location, y2_location
Number of Polygon Trap(s)
For each Polygon Trap:
    Trap ID, Trap Name
    Number of points in trap, Trap Bottom, Trap Top, Open Trap, Single Count Trap
    For each point: (1 point per line): x_location, y_location
*****
* Trap Keyword Values:
*****
Line Trap Direction:
    -1 (decreasing x-coordinate), 0 (any direction), or 1 (increasing x-coordinate)
    Default value is 0 (any direction)
Open Trap:
    0 (closed trap) or 1 (open trap)
    Default value is 1 (open trap)
Single Count Trap:
    0 (Count parcels every time they enter a trap) or 1 (Count parcels once per simulation)
    Default value is 0 (Count parcels every time they enter a trap)
*****
* Trap file written by SMS 10.1.0 Development Release
* SMS Build Date: Sep  9 2008
* Date saved: 09/09/08
* Time saved: 14:05:03
*****

```

Related Topics

- Particle Module
- Particle Tracking Model (PTM)
- Program Control File (PCF File)
- Sediment File
- Source File

PTM Trap Output

Trap output files are generated by PTM when one or more traps have been specified (this can be done by linking to an existing "*.trap" file or SMS will create this file from a PTM coverage containing trap objects).

A PTM trap file is a tabular ASCII file which contains information about when parcels entered specific traps in a PTM simulation. The first row of the file must be a header row defining the contents of the file. Each row after the header defines a single entry event of a parcel into a trap

File Overview

The first line of a standard trap output files contains the header defining the standard five columns of output, e.g.:

STEP	DATE	TIME	PARTICLE	TRAP

The trap output file may also include additional columns which give extra information about the parcel, e.g.:

STEP	DATE	TIME	PARTICLE	TRAP	MASS	MEDIAN_SIZE	SOURCE

The header line is followed by entry lines. Each line defines an entry event of a parcel into a trap:

STEP	DATE	TIME	PARTICLE	TRAP	MASS	MEDIAN_SIZE	SOURCE
1178	2001/01/01	02:38:10.000	53	1	2	0.1	2
1215	2001/01/01	02:41:15.000	47	3	3	0.05	1
1233	2001/01/01	02:42:45.000	99	1	3	0.2	2
.							
.							

When SMS is asked to read a trap output file, the file is identified by the extension "*_count.out". If the extension has been modified, SMS can be instructed to read this file using the file import wizard.

When SMS reads the standard trap output file, a time series curve for each trap represented in the file is created counting the number of parcels which entered the trap and logging the time (in hours from the first parcel) that the parcels entered. This is a "Count" time series curve. These time series curves are monotonically increasing.

If one or more "Value" column is specified when reading the file, SMS also creates one curve for each trap showing the accumulated value from this column. In the example above, the "MASS" column could be mapped as a value and the cumulative mass of each parcel would be summed as it enters the trap.

If one or more "Filter" column is specified, SMS asks for filter ranges associated with this quantity. For example, the user may wish to only count the parcels with a mean grain size below 0.15 mm. In this case, the "MEDIAN_SIZE" column above would be mapped as a "Filter" and a range of (0.00-0.15) specified in the filter range dialog that appears after the import wizard specification is complete. A parcel will be counted as in the range if its value is greater than the minimum and less than or equal to the maximum). Ranges that are specified with the same minimum and maximum values are ignored. The values are always interpreted as doubles even when they represent an integer quantity such as "SOURCE". In this case, the ranges could be specified as (0.5-1.5) and (1.5-2.0) to filter out parcels from a specific source. SMS will create additional curves (both for count and cumulative value) for each trap for the parcels that fall into each filter range.

Therefore, the total number of curves created when reading a trap output file would be:

$$\text{Num\_Curves} = (1 + \text{Num\_Value\_Columns}) * (1 + \text{Number\_Filter\_Ranges}) * \text{Number\_Traps}$$

These curves can be viewed using the *Plot Wizard* and selecting *PTM Gage plots*.

PTM Boundary Condition File

This file follows the ADCIRC fort.14 format. The only difference is that the header line will identify the file as a PTM_BC file with a version number. Here is an example:

```
PTM_BC CMS-PTM v2.1.025
```

When SMS reads the keyword (PTM_BC) in the header, it will then convert the file into a MIF/MID file, and read it into the GIS module. Blue is used to indicate open boundaries, and brown for closed boundaries. The reason for reading the file into the PTM interface and not the ADCIRC is that some users may not have the ADCIRC interface enabled, or may have an existing mesh that they don't want to replace.

4.4. Generic Grid Model

Generic Grid Model

The Generic Cartesian Grid Model provides a limited number of tools and options in the Cartesian Grid Module.

Generic Cartesian Grid Tools

The generic grid model has following tools available:

- Select Grid Cell
- Create Cartesian Grid
- Add Contour Label
- Select Grid Cell Points

To learn more, see Cartesian Grid Tools.

Generic Cartesian Grid Menus

The generic grid model the following standard menus:

- File
- Edit
- Display
- Cartesian Grid Data Menu
- Window
- Help

Related Topic

- Generic Cartesian Grid Coverage
 - Creating a Generic Grid
 - Grid Frame Dialog
 - Grid Frame Properties
-

Creating a Generic Grid

SMS provides the option to work in a Generic Cartesian Grid coverage. Selecting this coverage allows the user to use the **Create 2D Grid Frame** tool.

- Select the **Create 2D Grid Frame** tool
- Click a grid frame in the desired coverage area in the Graphics Window.

After creating a grid frame in the map module, a user can use the **Map → 2D Grid** command.

Related Topics

- Generic Cartesian Grid Coverage
- Generic Grid Model
- Grid Frame Dialog
- Grid Frame Properties

5. Coastal Numeric Models

5.1. ADCIRC (Advanced Circulation) Model

ADCIRC

ADCIRC	
Model Info	
Model type	Finite element hydrodynamic model for coastal oceans, inlets, rivers and floodplains.
Developer	Rick Luettich Joannes Westerink Randall Kolar Cline Dawson
Web site	http://www.adcirc.org
Tutorials	General Section • Data Visualization • Mesh Editing • Observation Models Section • ADCIRC Several Sample problems can be found on the ADCIRC model developer's webpage ^[1]

The ADCIRC (Advanced Circulation) model is a finite element hydrodynamic model for coastal oceans, inlets, rivers and floodplains. The initial developers of the code were Rick Luettich (University of North Carolina at Chapel Hill) and Joannes Westerink (University of Notre Dame). Other principal developers include Randall Kolar (University of Oklahoma at Norman) and Cline Dawson (University of Texas at Austin). Various other groups are involved in development and support around the country.

Graphical Interface

SMS provides a graphical interface that is designed to allow users to visualize the projects they are creating, easily modify project parameters, and view the solutions produced by the ADCIRC model. See ADCIRC Graphical Interface for more information.

The ADCIRC Graphical Interface contains tools to create and edit an ADCIRC simulation. The simulation consists of a geometric definition of the model domain (the mesh) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the 2D Mesh Module and setting the current model to ADCIRC. If a mesh has already been created for a ADCIRC simulation or an existing simulation read, the mesh object will exist in the Project Explorer and selecting that object will make the 2D Mesh module active and set the model to ADCIRC. See the Mesh Module documentation for guidance on building and editing meshes as well as visualizing mesh results.

The interface consists of the 2D Mesh Module Menus and tools augmented by the *ADCIRC* Menu. See ADCIRC Graphical Interface for more information.

ADCIRC Files

The list of files (both input and output) that may be associated with an ADCIRC simulation is quite long. The ADCIRC.org ^[2] web site lists all of these with details describing format and purpose. A brief summary of the most common file types is included here:

- Input files
 - Required
 - fort.14: Grid file – saved as *proj_name.grd* by SMS and copied to fort.14 for use in an ADCIRC run.
 - fort.15: Control file – saved as *proj_name.ctl* by SMS and copied to fort.15 for use in an ADCIRC run.
 - Optional
 - fort.13: Nodal or Spatial attributes
 - fort.19: Specified water surfaces (non-periodic elevation)
 - fort.20: Specified flow rates (non-periodic flow/flux such as a river)
 - fort.22, fort.2**: Meteorologic conditions (winds and atmospheric pressure) – several formats depending on the option being used
 - fort.23: Wave radiation stress forcing – fort.23
- Output files
 - Diagnostic
 - fort.6: Screen output
 - fort.16: General information
 - Global
 - fort.63 or fort.63.nc: Water surface at each node
 - fort.64 or fort.64.nc: Velocity components at each node
 - fort.53: Elevation Harmonic constituents at each node
 - fort.54: Velocity Harmonic constituents at each node
 - fort.73: Atmospheric pressure at each node
 - fort.74: Wind stress or velocity at each node
 - At observation stations
 - fort.61: Water surface at stations
 - fort.62: Velocity components at stations
 - fort.51: Elevation Harmonic constituents at stations
 - fort.52: Velocity Harmonic constituents at stations
 - fort.71: Atmospheric pressure at stations
 - fort.72: Wind stress or velocity at stations

Global Output Format

Generally, ADCIRC has the ability to output global data in three formats. These include:

- Standard ASCII – This format loops through the time steps including a value for each node (both node ID and solution value). These files are commonly very large (multiple GB) and can take a significant amount of time to load (sometimes as long as half an hour) because SMS has to process each time step of each dataset and build information about the solution for faster access. When a user instructs SMS to read a file of this type, SMS recognizes that the ASCII format is not efficient and converts the data to XMDf format in an "h5" file. The user can specify the name of the XMDf file that will be created. Multiple standard ascii files can be combined into a single "h5" file. The new "h5" file has the following advantages:
 - The "h5" file is binary and compressed so it is much smaller than the standard ASCII file.

- SMS can read the "h5" file almost instantaneously because all of the time step information is already compiled and a single time step is retrieved rather than processing the entire dataset.
- Sparse ASCII – This format loops through the time steps includes a default value for the time step and a number of nodes that don't have this default. Most commonly, the default would be -9999 indicating dry nodes. The file then includes the *exceptions* consisting of node ID and solution values for nodes that are not the default value. These lines are identical to the value lines in the standard ASCII format. (This format is supported in SMS starting at version 11.2) SMS converts sparse ASCII files to X MDF files just as it does the standard ASCII files.
- NetCDF – This format is a binary library format using the NetCDF library. The data can be viewed using an HDF viewer. (This format is supported in SMS starting at version 11.2)

(Note: ADCIRC documentation references a global binary format as an option. These options correspond to NOUT** values of 2 and -2 generally. There is no evidence that this option is functional in the current version of ADCIRC. It has been removed from the SMS interface.)

Functionality

ADCIRC is a system of computer programs for solving time dependent, free surface circulation and transport problems in two and three dimensions. These programs utilize the finite element method in space allowing the use of highly flexible, unstructured grids. Typical ADCIRC applications have included: (i) modeling tides and wind driven circulation, (ii) analysis of hurricane storm surge and flooding, (iii) dredging feasibility and material disposal studies, (iv) larval transport studies, (v) near shore marine operations.

For more information about the ADCIRC model visit www.adcirc.org ^[3].

Saving ADCIRC

When you do a *File | Save As...* the following files get saved in the SMS file.

- *.mat referenced to new save location
- *.map referenced to new save location
- *.grd referenced to new save location
- *.ctl referenced to new save location
- *.h5 referenced to new save location
- *.dat referenced to new save location

Using the Model / Practical Notes

- There is an ADCIRC listserv that may be useful to keep up-to-date about the latest releases of ADCIRC and to post any questions about ADCIRC. It is adcirc@listserv.unc.edu ^[4]. If you would like to join please email Crystal Fulcher ^[5].

Related Topics

- SMS Models
- LTEA – Linear Truncation Error Analysis
- ADCIRC Database

External Links

- ADCIRC Home page ^[6]
- Mar 2002 ERDC/CHL CHETN-IV-40 Guidelines for Using Eastcoast 2001 Database of Tidal Constituents within Western North Atlantic Ocean, Gulf of Mexico and Caribbean Sea [7]
- Jun 2001 ERDC/CHL CHETN-IV-32 Leaky Internal-Barrier Normal-Flow Boundaries in the ADCIRC Coastal Hydrodynamics Code [8]
- Mar 2001 Technical Report CHL-98-32 Shinnecock Inlet, New York, site Investigation Report 4, Evaluation of Flood and Ebb shoal Sediment Source Alternatives for the West of Shinnecock Interim Project, New York [9]
- Dec 1999 Coastal Engineering Technical Note IV-21 Surface-Water Modeling System Tidal Constituents Toolbox for ADCIRC [10] [11]
- ADCIRC wiki hosted by Seahorse Coastal Consulting ^[12]
- Glacier Bay Test Case by Dave F. Hill ^[13]
- Assessment of ADCIRC's Wetting and Drying Algorithm ^[14]

ADCIRC Database

In order to work with the ADCIRC Database, you will need to download the files separately. The needed files are:

1. Atlantic Database and Grid Files ^[1] – 74.9 MB
 - `adcircnwattides.exe` (utility)
 - `ec2001.grd` (adcirc grid file)
 - `ec2001.tdb` (adcirc harmonic output file)
 - `tides.in` (sample file)
 - `tides_ec2001.f` (source code)
3. Pacific Database and Grid Files ^[2] – 86.2 MB
 - `adcircnepactides.exe` (utility)
 - `enpac2003.grd` (adcirc grid file)
 - `enpac2003.tdb` (adcirc harmonic output file)
 - `tides.in` (sample file)
 - `tides_enpac2003.f` (source code)

These files should be unzipped and placed in `C:\Program Files\SMS X.X\models` (or a location of your choosing). You will then need to tell SMS where these files are located by going to *Edit | Preferences | File Locations*.

Related Topics

- ADCIRC
-

ADCIRC Files

Here are tables of all the available Input and Output files with the ADCIRC model (All formats might not be supported by SMS)

- For more information on the formatting of these files click the link on the file name and you will be redirected to the ADCIRC online documentation (ADCIRC.org^[6])

Input Files

Name	Description	Required/Conditional
fort.14 ^[1]	Grid and Boundary Information File	Required
fort.15 ^[2]	Model Parameter and Periodic Boundary Condition File	Required
fort.10 ^[3]	Passive Scalar Transport Input File	Conditional
fort.11 ^[4]	Density Initial Condition Input File	Conditional
fort.13 ^[5]	Nodal Attributes File	Conditional
fort.19 ^[6]	Non-periodic Elevation Boundary Condition File	Conditional
fort.20 ^[7]	Non-periodic, Normal Flux Boundary Condition File	Conditional
fort.22 ^[8]	Single File Meteorological Forcing Input	Conditional
fort.200,... ^[9]	Multiple File Meteorological Forcing Input	Conditional
fort.23 ^[10]	Wave Radiation Stress Forcing File	Conditional
fort.24 ^[11]	Self Attraction/Earth Load Tide Forcing File	Conditional
fort.67 or fort.68 ^[12]	2DDI Hot Start Files	Conditional

Output Files

Name	Description
fort.6 ^[13]	Screen Output
fort.16 ^[14]	General Diagnostic Output
fort.33 ^[15]	Iterative Solver ITPACKV 2D Diagnostic Output
fort.41 ^[16]	3D Density, Temperature and/or Salinity at Specified Recording Stations
fort.42 ^[17]	3D Velocity at Specified Recording Stations
fort.43 ^[18]	3D Turbulence at Specified Recording Stations
fort.44 ^[19]	3D Density, Temperature and/or Salinity at All Nodes in the Model Grid
fort.45 ^[20]	3D Velocity at All Nodes in the Model Grid
fort.46 ^[21]	3D Turbulence at All Nodes in the Model Grid
fort.51 ^[22]	Elevation Harmonic Constituents at Specified Elevation Recording Stations

fort.52 ^[23]	Depth-averaged Velocity Harmonic Constituents at Specified Velocity Recording Stations
fort.53 ^[24]	Elevation Harmonic Constituents at All Nodes in the Model Grid
fort.54 ^[25]	Depth-averaged Velocity Harmonic Constituents at All Nodes in the Model Grid
fort.55 ^[26]	Harmonic Constituent Diagnostic Output
fort.61 ^[27]	Elevation Time Series at Specified Elevation Recording Stations
fort.62 ^[28]	Depth-averaged Velocity Time Series at Specified Velocity Recording Stations
fort.63 ^[29]	Elevation Time Series at All Nodes in the Model Grid
fort.64 ^[30]	Depth-averaged Velocity Time Series at All Nodes in the Model Grid
fort.67, fort.68 ^[12]	Hot Start Output
fort.71 ^[31]	Atmospheric Pressure Time Series at Specified Meteorological Recording Stations
fort.72 ^[32]	Wind Velocity Time Series at Specified Meteorological Recording Stations
fort.73 ^[33]	Atmospheric Pressure Time Series at All Nodes in the Model Grid
fort.74 ^[34]	Wind Stress or Velocity Time Series at All Nodes in the Model Grid
fort.81 ^[35]	Depth-averaged Scalar Concentration Time Series at Specified Concentration Recording Stations
fort.83 ^[36]	Depth-averaged Scalar Concentration Time Series at All Nodes in the Model Grid
fort.91 ^[37]	Depth-averaged Density Fields at Specified Recording Stations
fort.93 ^[38]	Depth-averaged Density Fields at All Nodes in the Model Grid

Related Topics

- Run ADCIRC

ADCIRC Graphical Interface

The ADCIRC Graphical Interface includes tools to assist with creating, editing and debugging an ADCIRC model. The ADCIRC interface exists in the 2D Mesh Module.

Model Control

The *ADCIRC Model Control* dialog is used to setup the options that apply to the simulation as a whole. These options include time controls, run types, output options, global parameters, print options and other global settings.

Boundary Conditions

All numeric models require boundary condition data. In ADCIRC boundary conditions are defined on nodestrings. The default boundary condition is a closed boundary (no flow). See ADCIRC BC Nodestrings for more information.

Running the Model

The ADCIRC files are written automatically with the SMS project file or can be saved separately using the *File | Save ADCIRC* or *File | Save As* menu commands. See ADCIRC Files for more information on the files used for the ADCIRC run.

ADCIRC can be launched from SMS using the *ADCIRC | Run ADCIRC* menu command. A check of some of the common problems called the Model Checker is done each time the model is launched, or by selecting the *ADCIRC | Model Check* menu command.

ADCIRC Menu

See ADCIRC Menu for more information.

Related Topics

- Boundary Types
 - Boundary Conditions – tidal forcing
 - Coverage
 - Linear Truncation Error Analysis (LTEA)
 - Meshes
 - Model Control
 - Spatial Attributes
 - Steering
-

ADCIRC Menu

The following menu commands are available in the ADCIRC Menu:

Boundary Conditions

- Assign BC (Boundary Condition)
- Define Nodal Attributes
- Assign Nodal Attributes
- Create Observation Station

Model Parameters

- Spatial Attributes – Opens the *Spacial Attributes* dialog.
- Model Check – starts the model checker.
- Model Control – Opens the *ADCIRC Model Control* dialog.
- Run ADCIRC – launches the ADCIRC engine and generates output files.

Run ADCIRC

Presently, ADCIRC uses a specific naming convention for its input and output files. Therefore, before ADCIRC can start, the basic input files must be present in the working directory, which SMS takes care of automatically.

To run ADCIRC:

1. Select *ADCIRC | Model Control*.
2. Select the *General* Tab.
3. Name your project in the *Project title:* field and enter a name in the *Run ID:* field as well.
4. Press **OK**.
5. Select *ADCIRC | Run ADCIRC*.
6. If the name of the ADCIRC executable does not appear, click the folder icon, locate the ADCIRC executable, and click **OK**.

The ADCIRC model wrapper appears and gives status for 64800 time steps while the model runs. On a typical desktop machine, this will take around 15 minutes. Once the ADCIRC run has completed, there will be several new files created. See ADCIRC Files for more information about the files generated.

Mesh Generation

- Mesh Generation Toolbox
 - Localized Truncation Error Analysis (LTEA)

Related Topics

- ADCIRC Coverage
 - ADCIRC Mesh
 - Steering
-

ADCIRC Mesh

ADCIRC Meshes

ADCIRC computations are performed on a finite element mesh. ADCIRC meshes can only contain linear triangular elements. For more information about the mesh structure and using meshes see Mesh Module.

ADCIRC meshes are often generated using scalar density paving which helps define meshes that transition well between different sizes of elements (see Mesh Generation). Size functions used for scalar density paving can be defined using Linear Truncation Error Analysis (LTEA).

Example ADCIRC Mesh File (fort.14 or *.grd)

My Mesh File

```
200 125
  1  0.0000000000 5000.0000000000 10.0000000000
  2  0.0000000000 4500.0000000000 10.0000000000
  3  0.0000000000 4000.0000000000 10.0000000000
  4  0.0000000000 3500.0000000000 10.0000000000
  5  0.0000000000 3000.0000000000 10.0000000000
  6  0.0000000000 2500.0000000000 10.0000000000
  7  0.0000000000 2000.0000000000 10.0000000000
  8  0.0000000000 1500.0000000000 10.0000000000
  9  0.0000000000 1000.0000000000 10.0000000000
 10  0.0000000000  500.0000000000 10.0000000000
 11  0.0000000000   0.0000000000 10.0000000000
 12 387.2875168500 4679.5600328850 10.0000000000
 13 500.0000000000 5000.0000000000 10.0000000000
 14 448.0608478950 4243.2347889200 10.0000000000
 15 439.9566817750 3740.3946708250 10.0000000000
 16 438.0183235050 3237.7140888100 10.0000000000
 17 434.2179653200 2728.5012612700 10.0000000000
 18 434.4501409550 2223.5360522450 10.0000000000
 19 445.6728191600 1709.4083456900 10.0000000000
 20 441.7355135250 1217.0963461800 10.0000000000
 21 446.7690604800  665.5453978200 10.0000000000
 22 500.0000000000   0.0000000000 10.0000000000
 23 879.4691569000 4527.8977321650 10.0000000000
 24 1000.0000000000 5000.0000000000 10.0000000000
 25 915.9217524550 3970.1981044250 10.0000000000
 26 908.0623079800 3485.8982158350 10.0000000000
 27 865.4508483900 2975.6001854250 10.0000000000
 28 870.3584492700 2470.6287334900 10.0000000000
 29 865.3646748800 1963.5880264200 10.0000000000
 30 958.2489022100 1408.2747297650 10.0000000000
 31 846.2862793400  940.9107967100 10.0000000000
 32 894.5851194050  468.8869156450 10.0000000000
 33 1000.0000000000   0.0000000000 10.0000000000
```

34	1407.4389323650	4568.9272855350	10.0000000000
35	1319.0864158900	4152.9138196250	10.0000000000
36	1500.0000000000	5000.0000000000	10.0000000000
37	1519.2308151400	3630.3323211050	10.0000000000
38	1289.5949017700	3195.6810638750	10.0000000000
39	1309.3380986300	2727.7292707550	10.0000000000
40	1305.6561222300	2246.1068999650	10.0000000000
41	1287.1975618200	1772.0435687850	10.0000000000
42	1581.6880220050	1382.8502340350	10.0000000000
43	1352.9489540350	880.7930319200	10.0000000000
44	1338.3794076000	386.3892121500	10.0000000000
45	1500.0000000000	0.0000000000	10.0000000000
46	1912.7374514350	4563.7881224100	10.0000000000
47	1799.9414465350	4111.9865086650	10.0000000000
48	2000.0000000000	5000.0000000000	10.0000000000
49	2095.6095101550	3684.4249560900	10.0000000000
50	2029.7448911600	3256.1867318100	10.0000000000
51	1724.4261905750	3002.3231849500	10.0000000000
52	1757.6153478550	2485.5439440750	10.0000000000
53	1700.5538408750	1971.3218032750	10.0000000000
54	2022.3261809400	1718.2985960850	10.0000000000
55	2115.1877830250	1308.3558784300	10.0000000000
56	1908.2173673400	884.8414132300	10.0000000000
57	1796.4311279300	444.4909875000	10.0000000000
58	2000.0000000000	0.0000000000	10.0000000000
59	2421.8299088200	4565.2347923850	10.0000000000
60	2296.2815207500	4136.2010145200	10.0000000000
61	2500.0000000000	5000.0000000000	10.0000000000
62	2548.1854914450	3736.1664761600	10.0000000000
63	2245.7592831150	2715.5365617900	10.0000000000
64	2156.5989885850	2163.7822188800	10.0000000000
65	2576.1588612050	1255.8568827850	10.0000000000
66	2424.8132302500	862.3361492400	10.0000000000
67	2296.5804477550	435.0891291250	10.0000000000
68	2500.0000000000	0.0000000000	10.0000000000
69	2928.9231473700	4562.5355493550	10.0000000000
70	2813.0832006250	4131.9646648200	10.0000000000
71	3000.0000000000	5000.0000000000	10.0000000000
72	3005.2616900500	3650.6086203300	10.0000000000
73	3079.6575104700	3155.0929235250	10.0000000000
74	3026.1311773200	1347.8384132750	10.0000000000
75	3074.0212166200	1837.0105551000	10.0000000000
76	2941.1113740800	868.2621980900	10.0000000000
77	2803.0270930300	435.0218954200	10.0000000000
78	3000.0000000000	0.0000000000	10.0000000000
79	3428.1706765000	4553.5572992900	10.0000000000
80	3334.2421005900	4097.5833709050	10.0000000000

81	3500.0000000000	5000.0000000000	10.0000000000	
82	3585.5001362100	3580.4710896600	10.0000000000	
83	3644.1599668700	3020.1398601150	10.0000000000	
84	3268.7966206450	2718.6831817150	10.0000000000	
85	3591.9118685300	1432.2295580150	10.0000000000	
86	3444.7797743450	901.6282802800	10.0000000000	
87	3639.1938445900	1996.7994288000	10.0000000000	
88	3191.2693888900	2293.1529447050	10.0000000000	
89	3312.8201976500	437.8216047050	10.0000000000	
90	3500.0000000000	0.0000000000	10.0000000000	
91	3875.4018613600	4597.6915792100	10.0000000000	
92	3897.1163185450	4126.3881442500	10.0000000000	
93	4000.0000000000	5000.0000000000	10.0000000000	
94	4132.8748876900	3728.5069540050	10.0000000000	
95	4108.4300319250	3272.5630823750	10.0000000000	
96	4124.3068688700	2760.4946533350	10.0000000000	
97	3697.0596781500	2499.0463538250	10.0000000000	
98	4103.2022573000	1742.3164195700	10.0000000000	
99	4136.5962459150	1283.2373595850	10.0000000000	
100	3961.1675874700	903.0618231000	10.0000000000	
101	3813.1576567050	450.2947123950	10.0000000000	
102	4125.8315088050	2248.5711365450	10.0000000000	
103	4000.0000000000	0.0000000000	10.0000000000	
104	4367.7984401650	4498.4664948950	10.0000000000	
105	4521.7565602000	4046.3826573000	10.0000000000	
106	4500.0000000000	5000.0000000000	10.0000000000	
107	4598.8223028500	3503.3617712150	10.0000000000	
108	4561.3391885400	2955.6940550700	10.0000000000	
109	4563.9298111000	2441.5637730600	10.0000000000	
110	4561.8541878800	1933.3440965250	10.0000000000	
111	4569.5749317300	1429.5434022850	10.0000000000	
112	4527.5800133650	930.2255863050	10.0000000000	
113	4350.9621391150	487.6105782500	10.0000000000	
114	4500.0000000000	0.0000000000	10.0000000000	
115	5000.0000000000	4500.0000000000	10.0000000000	
116	5000.0000000000	4000.0000000000	10.0000000000	
117	5000.0000000000	5000.0000000000	10.0000000000	
118	5000.0000000000	3500.0000000000	10.0000000000	
119	5000.0000000000	3000.0000000000	10.0000000000	
120	5000.0000000000	2500.0000000000	10.0000000000	
121	5000.0000000000	2000.0000000000	10.0000000000	
122	5000.0000000000	1500.0000000000	10.0000000000	
123	5000.0000000000	1000.0000000000	10.0000000000	
124	5000.0000000000	500.0000000000	10.0000000000	
125	5000.0000000000	0.0000000000	10.0000000000	
1	3	1	12	13
2	3	12	1	2

3	3	12	2	14
4	3	2	3	14
5	3	15	14	3
6	3	3	4	15
7	3	4	16	15
8	3	4	5	16
9	3	17	16	5
10	3	5	6	17
11	3	18	17	6
12	3	6	7	18
13	3	19	18	7
14	3	19	7	8
15	3	8	20	19
16	3	20	8	9
17	3	9	21	20
18	3	9	10	21
19	3	22	21	10
20	3	10	11	22
21	3	13	12	23
22	3	23	12	14
23	3	13	23	24
24	3	14	25	23
25	3	14	15	25
26	3	25	15	26
27	3	15	16	26
28	3	26	16	27
29	3	17	27	16
30	3	17	28	27
31	3	17	18	28
32	3	28	18	29
33	3	29	18	19
34	3	19	30	29
35	3	30	19	20
36	3	20	31	30
37	3	20	21	31
38	3	31	21	32
39	3	22	32	21
40	3	32	22	33
41	3	24	23	34
42	3	23	35	34
43	3	23	25	35
44	3	36	24	34
45	3	35	25	37
46	3	25	26	37
47	3	37	26	38
48	3	27	38	26
49	3	38	27	39

50	3	39	27	28
51	3	28	40	39
52	3	40	28	29
53	3	40	29	41
54	3	29	30	41
55	3	41	30	42
56	3	42	30	43
57	3	31	43	30
58	3	31	32	43
59	3	43	32	44
60	3	33	44	32
61	3	33	45	44
62	3	36	34	46
63	3	46	34	47
64	3	34	35	47
65	3	35	37	47
66	3	48	36	46
67	3	47	37	49
68	3	49	37	50
69	3	50	37	51
70	3	37	38	51
71	3	38	39	51
72	3	51	39	52
73	3	39	40	52
74	3	40	53	52
75	3	53	40	41
76	3	41	42	53
77	3	42	54	53
78	3	54	42	55
79	3	42	56	55
80	3	43	56	42
81	3	57	56	43
82	3	57	43	44
83	3	44	45	57
84	3	58	57	45
85	3	59	48	46
86	3	46	60	59
87	3	47	60	46
88	3	60	47	49
89	3	59	61	48
90	3	49	62	60
91	3	62	49	50
92	3	51	63	50
93	3	63	51	52
94	3	52	64	63
95	3	52	53	64
96	3	64	53	54

97	3	65	54	55
98	3	55	66	65
99	3	56	66	55
100	3	56	67	66
101	3	56	57	67
102	3	58	67	57
103	3	67	58	68
104	3	61	59	69
105	3	70	69	59
106	3	59	60	70
107	3	60	62	70
108	3	69	71	61
109	3	70	62	72
110	3	72	62	73
111	3	74	75	65
112	3	65	76	74
113	3	76	65	66
114	3	76	66	77
115	3	67	77	66
116	3	77	67	68
117	3	77	68	78
118	3	79	71	69
119	3	69	80	79
120	3	70	80	69
121	3	70	72	80
122	3	81	71	79
123	3	80	72	82
124	3	72	73	82
125	3	82	73	83
126	3	73	84	83
127	3	75	74	85
128	3	85	74	86
129	3	76	86	74
130	3	88	75	87
131	3	87	75	85
132	3	76	89	86
133	3	76	77	89
134	3	78	89	77
135	3	78	90	89
136	3	79	91	81
137	3	79	92	91
138	3	92	79	80
139	3	80	82	92
140	3	91	93	81
141	3	94	92	82
142	3	94	82	95
143	3	95	82	83

144	3	95	83	96
145	3	83	97	96
146	3	83	84	97
147	3	84	88	97
148	3	87	85	98
149	3	98	85	99
150	3	85	100	99
151	3	100	85	86
152	3	100	86	101
153	3	86	89	101
154	3	97	88	87
155	3	97	87	102
156	3	102	87	98
157	3	101	89	90
158	3	90	103	101
159	3	104	93	91
160	3	104	91	92
161	3	104	92	105
162	3	92	94	105
163	3	106	93	104
164	3	94	107	105
165	3	107	94	95
166	3	95	108	107
167	3	96	108	95
168	3	96	109	108
169	3	96	102	109
170	3	97	102	96
171	3	110	102	98
172	3	110	98	111
173	3	98	99	111
174	3	111	99	112
175	3	100	112	99
176	3	100	113	112
177	3	100	101	113
178	3	113	101	103
179	3	102	110	109
180	3	103	114	113
181	3	104	115	106
182	3	115	104	105
183	3	105	116	115
184	3	116	105	107
185	3	117	106	115
186	3	116	107	118
187	3	118	107	119
188	3	119	107	108
189	3	119	108	120
190	3	120	108	109

191	3	109	121	120
192	3	109	110	121
193	3	122	121	110
194	3	110	111	122
195	3	123	122	111
196	3	111	112	123
197	3	123	112	124
198	3	124	112	113
199	3	114	124	113
200	3	124	114	125

1 = Number of open boundaries

11 = Total number of open boundary nodes

11 = Number of nodes for open boundary 1

1

2

3

4

5

6

7

8

9

10

11

4 = Number of land boundaries

44 = Total number of land boundary nodes

11 1 = Number of nodes for land boundary 1

50

62

73

84

88

75

65

54

64

63

50

11 22 = Number of nodes for land boundary 2

125

124

123

122

121

120

119

118

```
116
115
117
11 0 = Number of nodes for land boundary 3
11
22
33
45
58
68
78
90
103
114
125
11 0 = Number of nodes for land boundary 4
117
106
93
81
71
61
48
36
24
13
1
```

Related Links

- [ADCIRC](#)
- [Boundary Conditions](#)
- [Coverage](#)
- [Linear Truncation Error Analysis \(LTEA\)](#)
- [Model Control](#)
- [Spatial Attributes](#)
- [Steering](#)

ADCIRC Model Control

ADCIRC Model Control

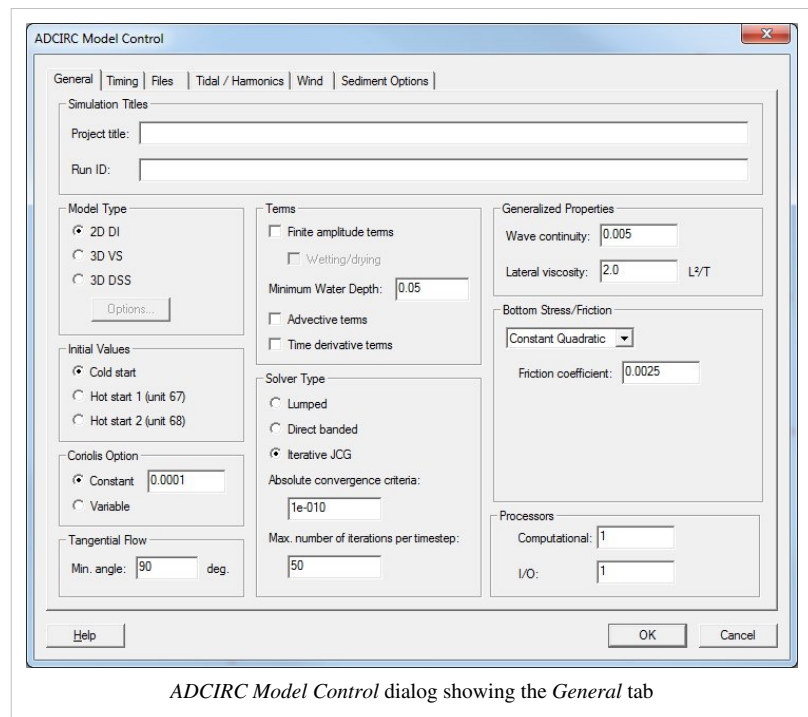
The *ADCIRC Model Control* dialog is where important project parameters are chosen and defined. There are 6 different tabs the user should be aware of. They are listed below.

- **General** – contains such options as model type, cold/hot start, terms, maximum number of iterations, bottom stress/friction, etc.
- **Timing** – define simulation run time, start time, and time step options
- **Files** – specify which input and output files are desired for the project
- **Tidal/Harmonics** – define tidal constituents and harmonic analysis options
- **Wind** - enable specific wind types and their options
- **Sediment Options** – enable sediment transport options

General Tab

Here is a list of terms found on the *General* tab:

- **Model Type** – 2D DI, 3D VS, 3D DSS. The two dimensional depth integrated is the only option that is recommended for commercial applications at this time.
- **Initial Values** – Cold Start, Hot Start 1, Hot Start2. This option allows you to choose a file to initialize the values of water level and velocity for a simulation. The files must have been saved from a previous run using the same finite element grid.



ADCIRC Model Control dialog showing the *General* tab

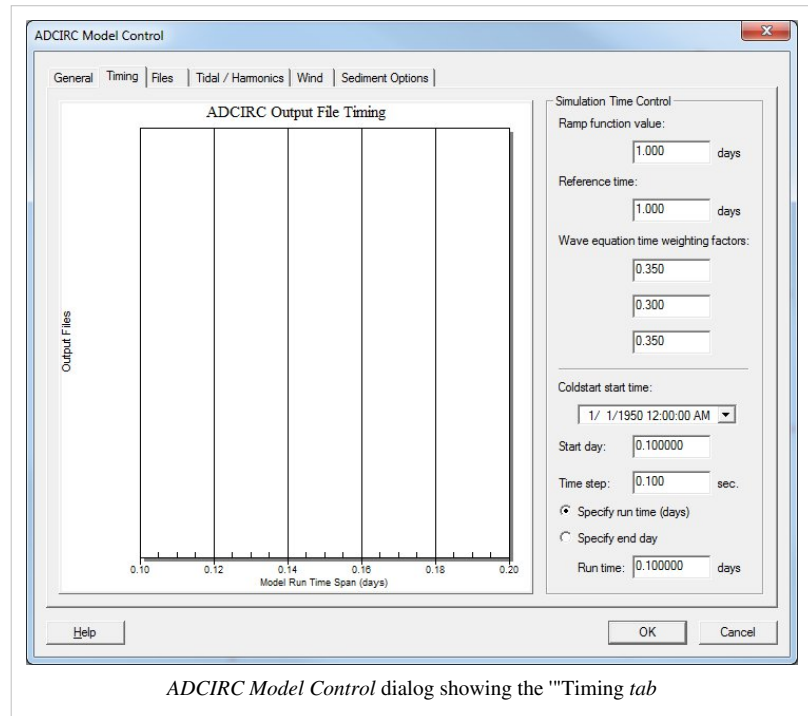
- **Coriolis Option** – Constant, Variable. The type of Coriolis forcing used does not impact run time for ADCIRC at all. Variable should be used when the coordinate system is geographic. In this case, ADCIRC will load a Coriolis coefficient for each node based on the latitude of the node. If you wish to run ADCIRC in rectilinear coordinates you should specify a constant Coriolis option because the model does not support reprojection from rectilinear to geographic to determine individual latitudes for each node.
- **Minimum Angle For Tangential Flow**
- **Terms** – Finite Amplitude Terms, Advective Terms, Time Derivative Terms
- **Solver Type** – Lumped, Direct Banded, Iterative JCG – The Iterative JCG is the default and recommended solver. The other options are in the interface for backward compatibility and analytic tests.
- **Absolute Convergence Criteria**
- **Maximum Number of Iterations per Time Step**
- **Generalized Properties** – Wave Continuity, Lateral Viscosity

- **Bottom Stress/Friction** – Constant Linear, Constant Quadratic, Constant Hybrid, Varying Linear, Varying Quadratic, Friction Coefficient – The "Constant Linear" option should only be used for analytical cases when you are verifying the code. Linear friction causes overdamping in deep water. For "Constant Quadratic" the value of TAU0 should be set based on the principle depth. A value of TAU0 = 0.005 is suggested for deeper water (greater than 10 m depth). For shallow water, TAU0 = 0.02 is recommended. If your domain includes both shallow and deep water, you may want to consider the varying quadratic option.

Timing Tab

Important parameters to be aware of:

- **Ramp Function Value** – this value is a period of time, starting at the beginning of a simulation, that allows the full forcing of the tidal constituents to be gradually applied to the model. This prevents any issues or negative effects from occurring that could harm the results of a simulation.^[1]
- **Reference time** – You can use this to specify that the extracted tidal constituents are earlier or later than the start time of the simulation (see ADCIRC documentation Reftime ^[2]).
- **Start Day** – the start time of a project
- **Time Step** – time interval between successive measurements
- **Run Time** – total length of time a simulation is run
- **End Time** – day when simulation should finish

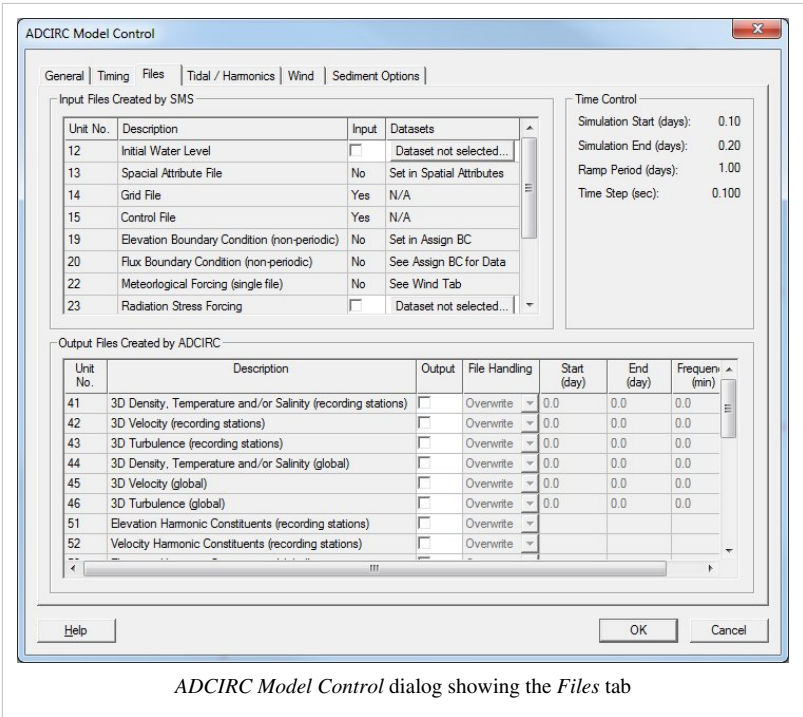


Files Tab

This tab contains two spreadsheets. The first of which is concerned with the input information ADCIRC can use before a simulation is run. The user may toggle on or off any of the enabled input files by checking the checkbox in the *Input* column. Some of the more common input files that are used are:

- **Initial Water Level** – provides initial water elevation levels
- **Hotstart 1 (fort.67)** – see ADCIRC Hot Start Files (fort.67 & fort.68) ^[3]
- **Hotstart 2 (fort.68)** – see ADCIRC Hot Start Files (fort.67 & fort.68) ^[3]

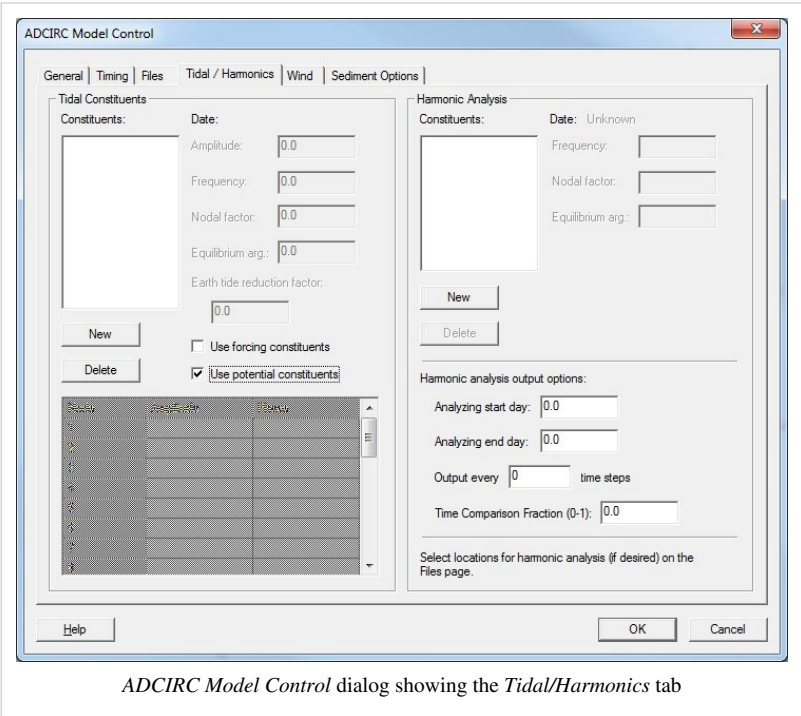
The second spreadsheet on this tab lists all of the possible output/solution files that ADCIRC can produce. This spreadsheet works similarly to the *Input Files* spreadsheet. A user may toggle on or off any of the output files listed by clicking in a checkbox in the *Output* column. Some of the files also require a start time, end time, or frequency to be specified for results to be produced. The start time and end time do not have to be the same as the start and end time of the entire simulation, but they do need to be somewhere within that time frame.



ADCIRC Model Control dialog showing the Files tab

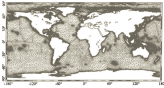
Tidal/Harmonics Tab

Tidal constituents are variations in tides that are created by different frequencies of astronomical forcing. They arise due to the gravitational influences of the Moon and Sun on the Earth, the tilt of the rotational axis of the Earth, the elliptical shape of the Moon's orbit around the Earth, the shape of the Earth's orbit around the Sun, and other such factors. Many constituents have been defined and are classified based on their cycle lengths. Most of the tidal constituents used in the ADCIRC interface of SMS are either diurnal (one cycle per day) or semidiurnal (two cycles per day) in nature. Harmonic constituents are those variations that have periods of less than half a day. For a table of commonly used constituents see Principal Tidal Constituents^[11].



ADCIRC Model Control dialog showing the Tidal/Harmonics tab

SMS uses the following databases to provide the tidal constituent data to an ADCIRC simulation:

- LeProvost – The LeProvost database is a set of *.legi files which provide amplitude and phase information. The LeProvost database can be downloaded at: sms.aquaveo.com/leprovost.zip^[4].
- An image of the LeProvost tidal database domain coverage can be found here – ^[2]

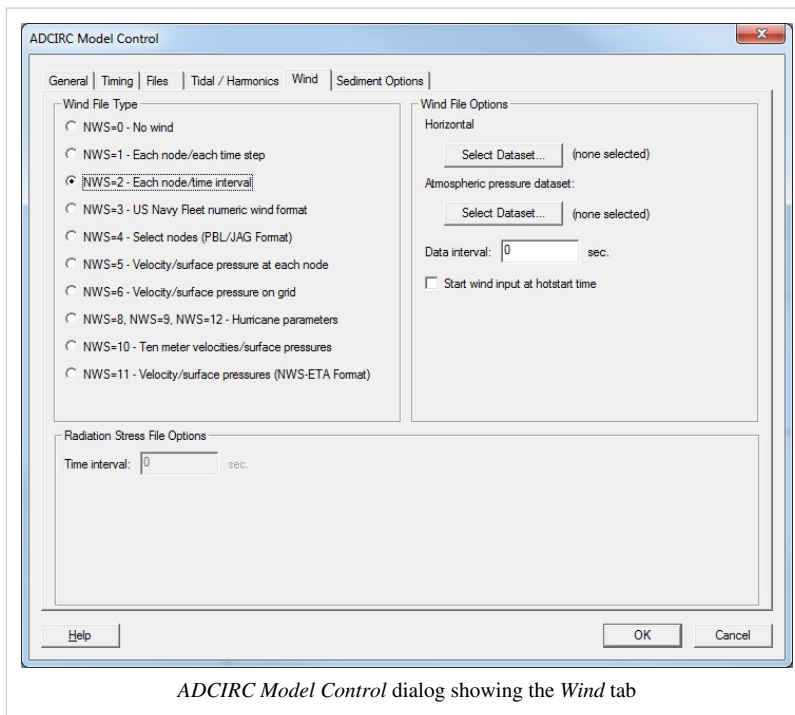
- ADCIRC – The ADCIRC database includes a grid file and a harmonics file that uses an *.exe extractor to put information into SMS. The ADCIRC database can be downloaded at: sms.aquaveo.com/adcirc_tides.zip ^[6]. More information about the ADCIRC database can be obtained at the ADCIRC web page: ADCIRC Tides Databases ^[7]. The ADCIRC database is only valid for projections in North America.

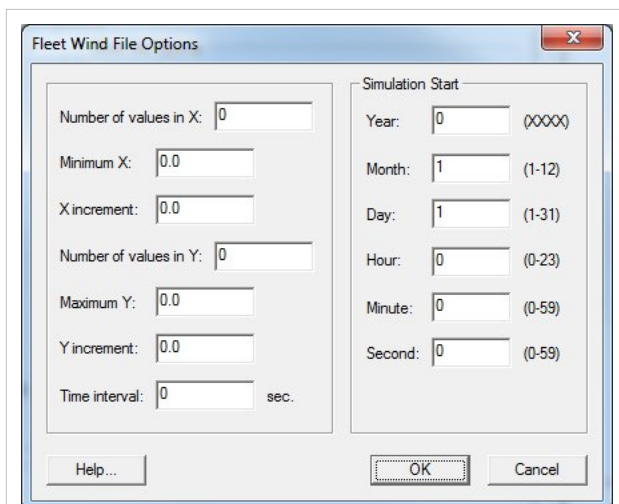
The path to each database on your computer must be specified in the *Preferences* dialog by going to *Edit/Preferences/File Locations*. The two tidal constituent databases (LeProvost and ADCIRC) are only accessible if the SMS project has a global (not local) display projection. In the *New Constituent* dialog, the radio buttons that allow you to select either of these databases are either enabled or grayed out depending on the display projection setting.

Wind Tab

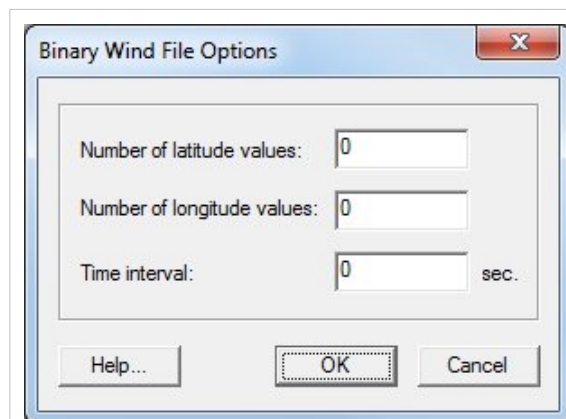
ADCIRC can model and compute wind velocities and stresses. The ADCIRC model is able to use wind data from a variety of different file types. The SMS interface is not able to read and create all of the available wind data file types. There are three types that SMS is able to read and create. Even though SMS does not read and create these other file types, it does tell ADCIRC that there is data there, where it is, and how to access it for a simulation to run correctly.

One of the parameters found in the fort.22 file (Single File Meteorological Forcing Input File ^[8]) will tell a user which of the wind data file types SMS can read and create. The first parameter specified in the fort.22 file is called the NWS value. This value represents what type of wind data is in the file. It can range from 0 to 100+. SMS can read and create fort.22 files that are of the NWS value of 1, 2, or 5. All other values of the NWS value are only shown and directed to the ADCIRC model by SMS so that a simulation will run correctly.





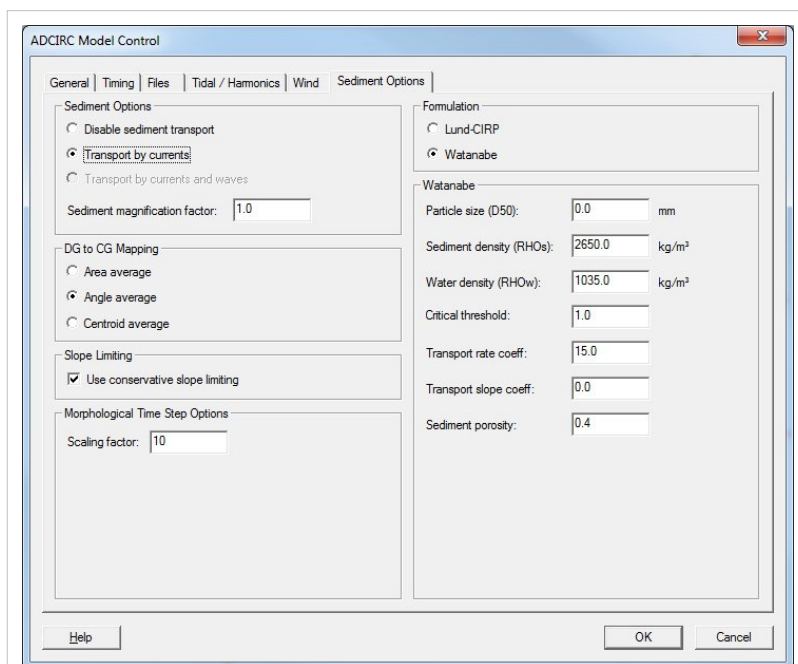
*Fleet Wind File Options dialog accessed from the **Options** button in the **Wind Tab***



*Binary Wind File Options dialog accessed from the **Options** button in the **Wind Tab***

Sediment Options Tab

This section of the SMS interface is still under development and is not fully functional. It will be updated sometime in the future.



*ADCIRC Model Control dialog showing the **Sediment Options** tab*

References

- [1] Militello, A., and Zundel, A. K. (1999). "Surface-water modeling system tidal constituents toolbox for ADCIRC," Coastal Engineering Technical Note CETN IV-21, U.S. Army Engineer Research and Development Center, Vicksburg, MS.
- [2] Le Provost, C., Genco, M. L., and Lyard, F. (1995). "Modeling and predicting tides over the World Ocean," Quantitative Skill Assessment for Coastal Ocean Models, Coastal and Estuarine Studies, Vol. 47, pp 175-201.

Related Links

- ADCIRC
- Boundary Conditions
- Coverage
- Linear Truncation Error Analysis

(LTEA)

- Meshes
- Spatial Attributes
- Steering

ADCIRC Spatial Attributes

ADCIRC Nodal/Spatial Attributes

Spatial attributes (or nodal attributes) are parameters that can be applied to the entire domain of the simulation in the form of a dataset with values assigned to the nodes of the mesh. ADCIRC supports several such spatially varying input parameters and SMS (starting with version 11.0) includes functionality to assist the user in defining, managing and editing these parameters. With each release of ADCIRC, the list of supported nodal attributes changes. There are several such attributes that are under development and are used by individuals in the ADCIRC community, but are not available in the release version. Therefore, the interface in SMS is a combination of specifically defined attributes to correspond with the standard parameters in ADCIRC and *custom* attributes to allow for customization and support of attributes not yet included in either the ADCIRC release or the standard list in the SMS interface.

The user can define these attributes in a the *Spatial Attributes* dialog under the *ADCIRC* menu. There are a number of predefined attributes that the user can use or they can define their own. Here is a list of the predefined spatial attributes and a brief meaning of each.

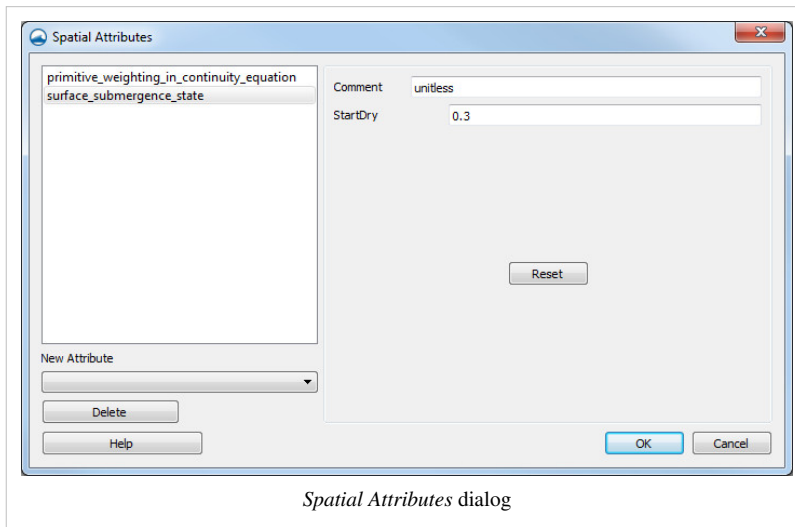
In version 50 of ADCIRC, the following predefined nodal attributes are supported:

- primitive weighting in continuity equation – Tau0
- surface submergence state – StartDry
- quadratic friction coefficient at sea floor – Fric
- surface directional effective roughness length – z0Land
- surface canopy coefficient – VCanopy
- bridge pilings friction parameters – BK, BAlpha, BDeIX, POAN
- Manning's n at sea floor – ManningsN
- Chezy friction coefficient at sea floor – ChezyFric
- sea surface height above geoid – GeoidOffset
- horizontal eddy viscosity – ESLM
- wave_refraction_in_swan – SwanWaveRefrac (Supported in list of *predefined* attributes in SMS 11.2)
- bottom_roughness_length – Z0b_var (Supported in list of *predefined* attributes in SMS 11.2)
- average horizontal eddy viscosity in sea water wrt depth – ESLM
- elemental_slope_limiter – elemental_slope_limiter_grad_max (Supported in list of *predefined* attributes in SMS 11.2)
- advection_state – AdvectionState (Supported in list of *predefined* attributes in SMS 11.2)

Please visit the ADCIRC web site ^[1] to learn more about the meaning of each of the spatial attributes listed above.

How to Use Spatial Nodal Attributes in a Simulation

Spatial nodal attributes are managed in the SMS using editable datasets. The creation and management of attributes is managed through the *Spatial Attributes* dialog. You can access the *Spatial Attributes* dialog by selecting *ADCIRC | Spatial Attributes* from the menu.



You can create a new spatial attribute by selecting the attribute name in the *New Attribute* combo-box. This will bring up the *Spatial Attributes Initial Values* dialog (discussed below).

The top left of the dialog has a list of the currently defined spatial attributes. The currently selected spatial attribute can be modified by the controls on the right side of the dialog or by editing specific dataset values by selecting the specific node(s) and entering a value. Each attribute has an associate comment. The comment is ignored by

the model and is generally used for units information. The value below the comment is intended to represent the default value, or the value used the most in the dataset. Values at nodes that do not use the default value are written by the SMS to the fort.13 file as exceptions. The reset button will bring up the *Spatial Attributes Initial Values* dialog (discussed below).

You can delete the current attribute by clicking on the **Delete** button. For each spatial attribute, SMS creates one or more datasets in the project explorer. However, these datasets cannot be deleted through the project explorer. They must be managed through the *Spatial Attributes* dialog.

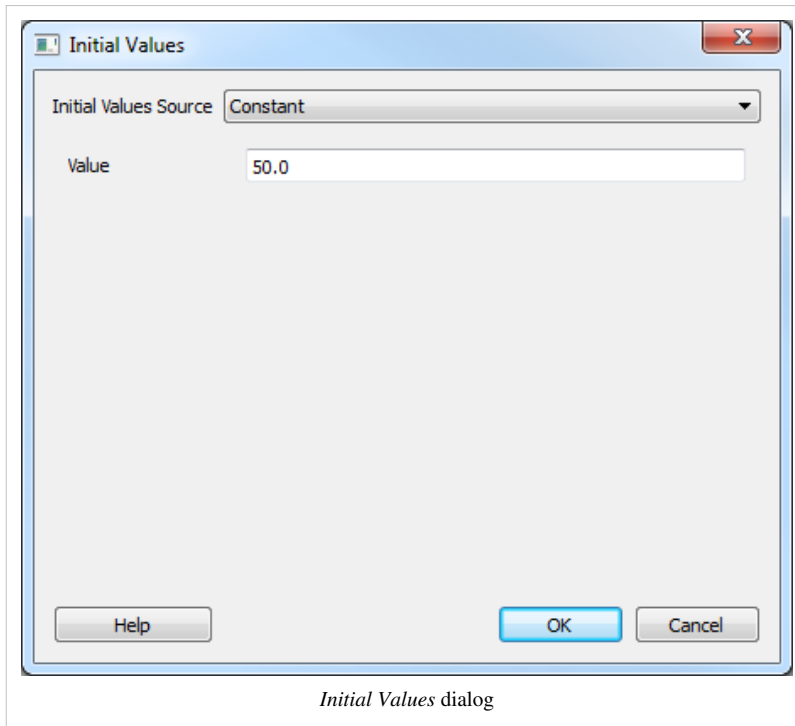
Editing Spatial Attributes

To edit a spatial attribute, open the *Spatial Attributes* dialog. Select the spatial attribute that needs to be edited and select the **Reset** button.

Initial Values Dialog

The *Initial Values* dialog is used to setup the initial values for the dataset(s) used for the spatial attribute.

Every spatial attribute has the option of starting from a constant value or use the values from an existing dataset. In addition, some spatial attributes have an option to populate the spatial attribute based upon other data inside of SMS, using a rule, or using a utility similar to those provided on the ADCIRC website to generate.



Populate Options

A spatial attribute consists of one or more datasets that can include unique values at each of the nodes in the mesh. This is potentially a lot of data and will generally not be specified manually. Two options exist for general population of the dataset. These include:

- **Constant** – the user specifies a single value for all nodes. The user can later select one or more regions of nodes and assign a different value to these nodes using the scalar edit box at the top of the SMS interface.
- **Dataset** – the user selects an existing data set that is copied to the

spatial attribute. The user may use any of the standard methods for defining the source dataset including interpolation and the data calculator. Individual edits can still be made to nodes or regions of nodes by manually selecting them.

In addition to these two default methods of populating data sets, SMS supports some custom population approaches for specific attributes. These are described below.

Primitive weighting

The SMS includes a tool for populating the primitive weighting coefficient for each node based on the node spacing and the node depth. The user specifies a threshold for node spacing (Critical average node spacing) and depth (Critical depth). The defaults for these are 1750 m and 10 m respectively. The user also specifies a values for tau for three conditions including default (0.03), deep (0.005) and shallow (0.02).

If the average distance between a node and its neighbors is less than the critical value then tau0 for that node is set at the tau default. If the average distance between a node and its neighbors is greater than or equal to a critical spacing then tau0 for that node is assigned based on the depth using the deep or shallow value specified.

Quadratic, Manning N, and Chezy friction coefficients

The spatial attributes which reflect bottom friction "Chezy_friction_coefficient_at_sea_floor," "Mannings_n_at_sea_floor," and "quadratic_friction_coefficient_at_sea_floor" can be populated from NLCD land use data. You can obtain NLCD geo-tiffs from the Multi-Resolution Land Characteristics Consortium website (www.mrlc.gov ^[2]). Use the seamless server to download the data in the area you are interested in. To populate using this method you must download the NLCD geo-tiff for the region of interest, load it into the SMS (creating an image) and convert it to a raster (right click on the image). (Refer to the information of rasters for guidance).

When populating any of these spatial attributes, you provide a roughness value for each land-use type and a default roughness value that will be used in areas of the raster that are NULL or outside the bounds of the raster.

The SMS will extract values from all visible raster objects (turned on in the project explorer). For each node in the mesh, the "area of influence" is computed for the node. The area of influence is a polygon that represents the area of

the mesh around a node. This is computed as the area encompassing the node and half the distance to each of his neighboring nodes. All of the raster values within the area of influence are extracted from the visible raster objects. A composite roughness value is computed taking a weighted average of all the raster values. If the area of the extracted values is less than the area of influence around the node (because we passed the extent of the raster objects or encountered null values), the default value is used to compute the composite n using the area not represented in the values.

Surface directional effective roughness

The surface directional effective roughness length composes the contributing vegetation types associated with each of the 12 directions (30 degrees each) around each node in a mesh and assigns the 12 characteristic or composite reduction values to the datasets for the node.

The SMS includes an option to populate these values using NLCD land use data raster objects. These must be loaded before populating this attribute. The user specifies a *Total distance* to be included in the computation of the directional roughness as well as a *Weighted distance* and method of interpolation. The user also specifies a roughness value for each land use type.

This is in accordance with the utility provided on the ADCIRC web site for generation of this attribute. The SMS invokes this utility to compute an effective roughness for each cell for each direction.

Surface canopy coefficient

The surface canopy attribute provides a method for ADCIRC to reduce wind stress at specific locations in the mesh to represent the protection afforded by local vegetation. The SMS includes an option to populate these values based on NLCD land use raster objects (which must be loaded prior to generating this parameter).

The user specifies whether to compute the reduction based only on the nodal location, or based on the area of influence around a node (half the distance to adjacent nodes). If the area of influence method is used, the user can also specify a percentage cover that must exist for a reduction to be applied. Finally, the user specifies a roughness reduction to be applied for each land use (vegetative cover type).

Custom Attributes

The name of a spatial/nodal attribute is key for its use in an ADCIRC simulation. ADCIRC reads the name from the fort.15 file and the corresponding values from the fort.13 file. If the name does not match a supported attribute in that version of ADCIRC that is being used, it is ignored. For this reason, the SMS interface includes the specific names supported by ADCIRC at the time of development. This minimizes the possibility of errors when entering the name of a nodal attribute.

There are two common situations which would require the user to enter a *custom* attribute. The first is to define an attribute for an ADCIRC run for an attribute that is not yet included in the SMS interface. The second is to define an attribute for a developmental feature in a custom version of ADCIRC.

In either case, the user simply selects the *Custom...* option from the *New Attribute* combo box and the *Spatial Attribute – Custom* dialog appears. This dialog allows the user to specify a name for the attribute, which **must** match the expected string in the corresponding version of ADCIRC. The user also specifies a comment, and the dataset(s) name(s) associated with this attribute. By clicking **OK**, the initial values dialog appears to allow the user to populate each dataset.

Related Links

- [ADCIRC](#)
- [Boundary Conditions](#)
- [Coverage](#)
- [Linear Truncation Error Analysis \(LTEA\)](#)
- [Meshes](#)
- [Model Control](#)
- [Steering](#)

LTEA

Linear Truncation Error Analysis (LTEA) was initially presented by Dr. Scott C. Hagen as part of his doctoral research at Notre Dame. Development has continued on the methodology at the University of Central Florida. The LTEA algorithm performs analysis on an existing ADCIRC mesh and its solution to help quantify the error associated with the mesh. Normally, this ADCIRC solution is taken from a "linear ADCIRC" run. This type of run is used to make the process faster and to simplify the LTEA algorithm applied to the unstructured mesh. A second phase of the LTEA process uses the error values at each node to create a relative size function covering the domain called DelX.

The LTEA Toolbox

SMS includes a graphical interface that allows the user to use the LTEA theory to guide the generation of a finite element mesh. The tool requires two inputs which must be loaded into SMS. These include:

- A bathymetry scatter set.
- An ADCIRC coverage having at least one polygon with the boundary conditions assigned.

If you wish to start the mesh generation process from an existing ADCIRC mesh, you must convert the mesh to both a scatter set and ADCIRC conceptual model. Right-clicking on the mesh in the project explorer gives access to commands to perform the basic conversion. The conceptual model must be defined on the arcs created in this way.

The toolbox is accessed through the **Mesh Generation Toolbox...** from the *ADCIRC* menu in the Mesh Module. From the dialog that appears, select the *LTEA* option and click **Run**. The steps that follow include:

- **Step 1 – Linear Run Mesh generation.** In this step the user must specify the scatter set and conceptual model to be used for mesh generation. This step generates a basic mesh from the conceptual model to perform a linear run. This mesh may be saved for future reference. If a mesh already exists that is suitable for the linear run, the option to generate a linear run mesh should be turned off.
 - **Step 2 – Linear ADCIRC Run.** In this step, the toolbox runs ADCIRC in linear mode with the M2 tidal constituent and performs an harmonic analysis on the result. If a run of ADCIRC and harmonic analysis has already been performed, the option to use existing solution data becomes available. In this case the user must specify the datasets that contain the output of the harmonic analysis. If this is not the initial pass through the mesh generation toolbox, or if the user wants to generate a "Size guideline function" from another source, it may be provided. If this is the case, the linear run as well as the next step "LTEA calculations" are bypassed.
 - **Step 3 – LTEA Analysis.** The LTEA algorithm processes the harmonic analysis output and determines the relative error due to node spacing throughout the domain. It then computes a size guideline for generating a new mesh. The guideline is a dataset (a value for each node of the linear run mesh) named "DelX". The user must instruct LTEA to only perform calculations on the interior (no partial molecule) or to approximate the LTEA calculations right up to the boundary. (Note: extra controls exist in the dialog for tools that are under development.)
-

- **Step 4 – Generate Final Mesh.** At this point, SMS has almost all the information required for mesh generation. The user must specify the target size of desired mesh as a number of nodes and a tolerable deviation from that target. The acceptable size transitions are also specified here (see the Smooth Dataset article for a description of this).

The tool can be used repetitively to generate various meshes of the same area with varying resolution. In these cases, the first three steps should be bypassed by entering the input data in step 1 and the "DelX" function in step 3 and then proceeding to step 4.

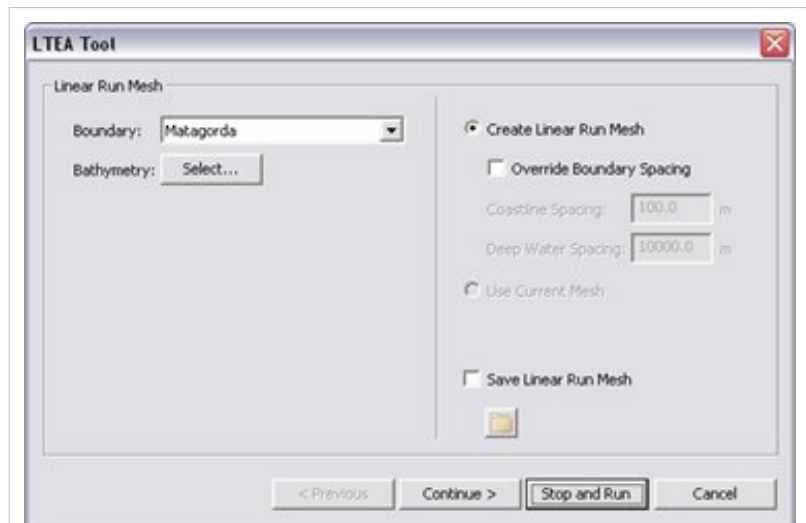
LTEA Tool Overview

A set of standard buttons can be found at the bottom of the *LTEA Toolbox*. They are as follows:

- **Help...** – launches the *Help* dialogue
- **Continue >** – Moves to next step of the *LTEA Toolbox* dialog.
- **< Previous** – Allows user to return to a an earlier step in the *LTEA Toolbox* dialog.
- **Stop and Run** – causes SMS to close the toolbox and run the LTEA analysis. The analysis generates several datasets used as size functions in the mesh generation process.
- **Run** – activates the model engine at the completion of all steps.
- **Cancel** – closes the *LTEA Toolbox* dialog without saving and information entered.

Step 1: Linear Rush Mesh

- Boundary:
- Bathymetry:
- Create Linear Run Mesh
 - Override Boundary Spacing
 - Coastline Spacing
 - Deep Water Spacing
- Use Current Mesh
- Save Linear Run Mesh



Step 1: Linear Rush Mesh

Step 2: IMS-ADCIRC Linear Run

- Run IMS-ADCIRC Linear Run
 - Wave Continuity/Friction Coefficient:
 - Minimum Angle for Tangential Flow:
 - Absolute Convergence Criteria:
 - Ramp Time
 - Run Time:
 - Time Step:
- Provide Harmonic Solutions
 - Eta Amplitude:
 - Eta Phase:
 - Velocity Amplitude:
 - Velocity Phase:
- Provide DelX
 - DelX:

Step 2:IMS-ADCIRC Linear Run

Step 3: LTEA Analysis

- Run Type:
- Value for dc:
- Minimum spacing from:
- Node Number:
- Use Partial molecule
 - Molecule size:

Step 3: LTEA Analysis

Step 4:Generate Final Mesh

- Target number of nodes:
- Element area change limit:
- Redistribute boundaries
- Truncate element sizes
 - Minimum Size
 - Maximum Size

Case Studies / Sample Problems

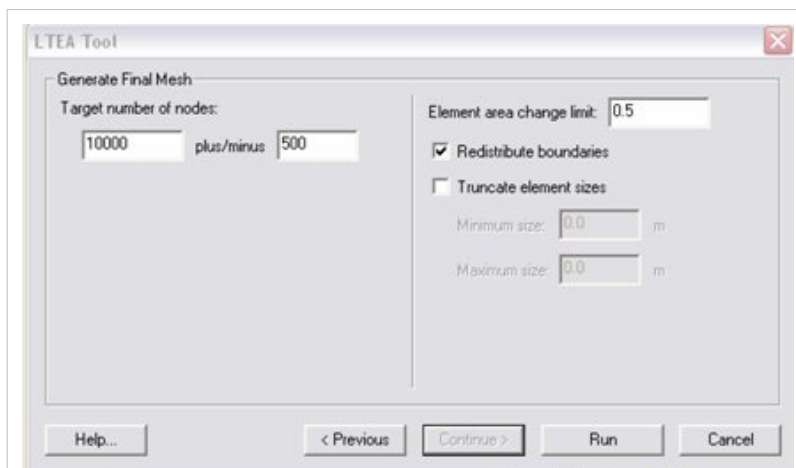
Tutorials

The following tutorials may be helpful for learning to use LTEA in SMS:

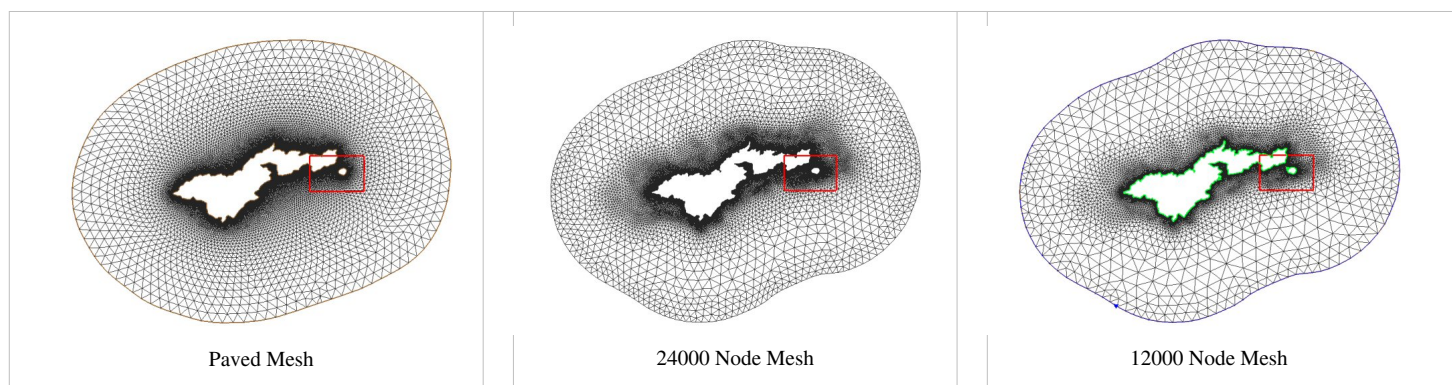
- Models Section
 - ADCIRC LTEA – Uses LTEA to mesh Shinnecock bay and the area around it along Long Island, NY

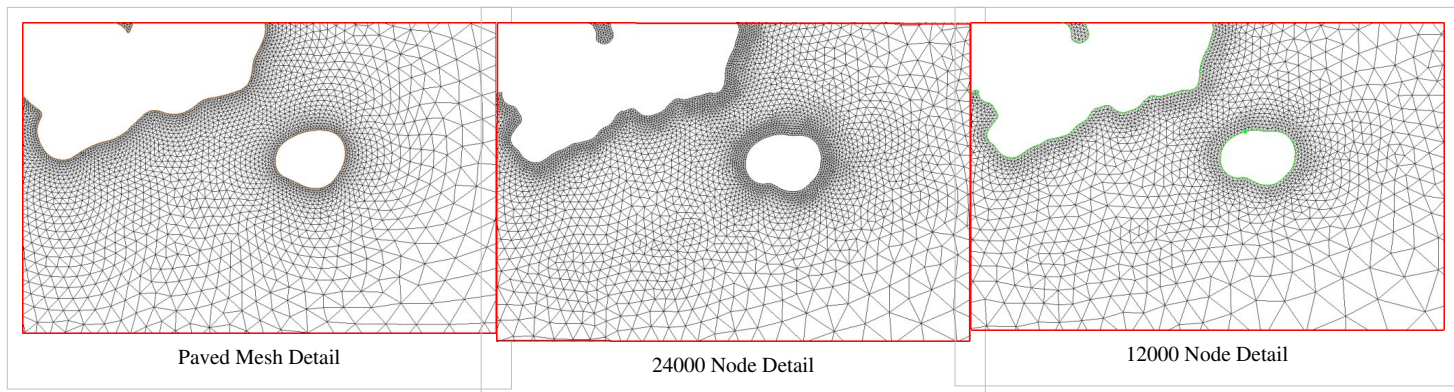
American Samoa

The following images illustrate the results of the LTEA toolbox applied to a domain around American Samoa. The first pair of images illustrate a mesh generated for the domain using the paving method. Density at the coastline was controlled by redistributing the vertices on the arcs representing the coastline and the density varied to a larger ocean boundary density. This mesh consists of 22,576 nodes (43,055 elements). The other images illustrate the varying resolution generated by LTEA to result in constant error with target mesh sizes of 24,000 nodes and 12,000 nodes respectively. The LTEA toolbox created meshes with 24,078 nodes (45,929 elements) and 12,029 nodes (22,543 elements).



Step 4:Generate Final Mesh

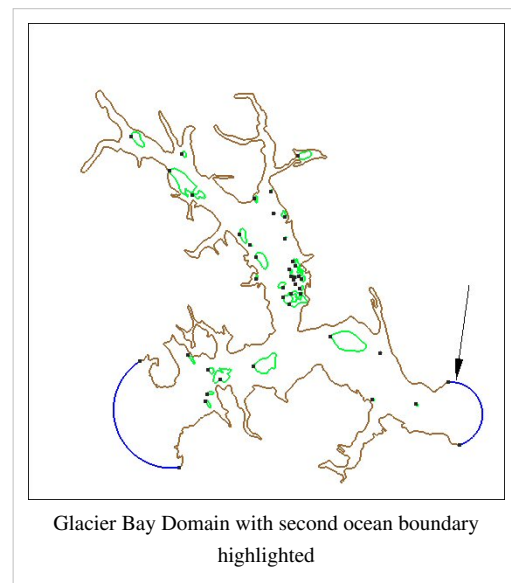
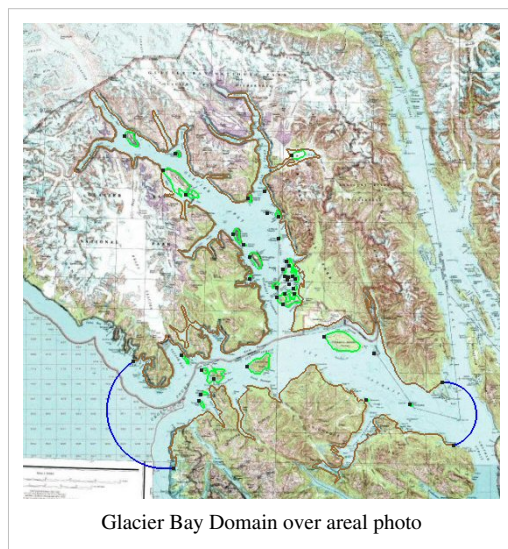




These images illustrate the redistribution of density to increase the density in areas that require additional detail for solution variations, or to reduce the number of nodes in the mesh.

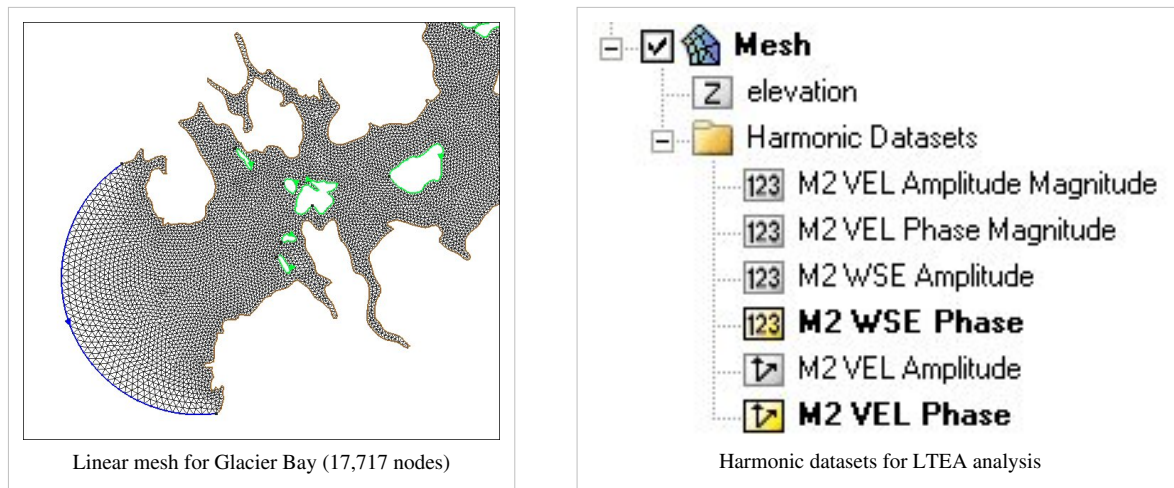
Glacier Bay Alaska

The case of Glacier Bay Alaska ^[13], by Dave F. Hill's ^[1] research group, poses another problem for the LTEA toolbox. This case includes two ocean boundaries. The figures below show three meshes generated for this case and illustrate the large variation in node density that can be produced by the procedure.

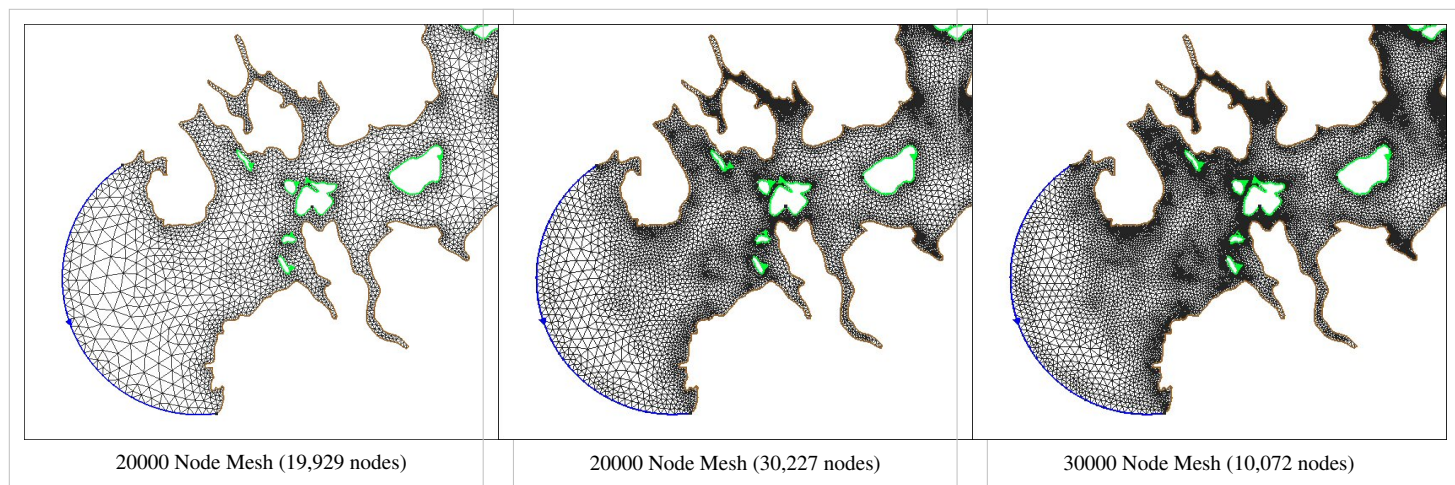


This case includes two ocean boundaries. Currently, the LTEA toolbox makes a sometimes erroneous assumption that only one ocean boundary exists. To work around this problem in the current version of SMS, the following steps are required:

- Change the inland ocean boundary to land
- Run the first step of the LTEA toolbox to generate the "Linear Run Mesh" and then "Stop and Run" to that point.



- Outside of the toolbox, change the second ocean boundary to ocean on the linear run mesh and run a linear run of ADCIRC with harmonic analysis turned on. This will generate the datasets for LTEA calculations.
- Relaunch the toolbox and select the datasets from the linear run to guide the mesh generation process.



Related Topics

- ADCIRC
- Steering

External Links

- Coastal Hydrosience Analysis, Modeling & Predictive Simulations Laboratory (CHAMPS Lab) ^[2]
- Dec 2006 Automatic, unstructured mesh generation for 2D, shelf-based tidal models ^[3]
- Sep 2006 Automatic, unstructured mesh generation for tidal calculations in a large domain ^[4]
- Sep 2006 Resolution Issues in Numerical Models of Oceanic and Coastal Circulation ^[5]
- 2001 Two-dimensional, unstructured mesh generation for tidal models ^[6]
- 2000 One-dimensional finite element grids based on a localized truncation error analysis ^[7]
- 1998 2D Finite Element Grids Based on a Localized Truncation Error Analysis ^[8]
- Glacier Bay Test Case by Dave F. Hill ^[13]

5.1.a. ADCIRC Boundary Conditions

ADCIRC Boundary Conditions

Much of ADCIRC's versatility as a model is due to the large number of different boundary types and boundary conditions available in the model. For detailed description of the ADCIRC boundary conditions, we refer you to the documentation provided by the ADCIRC development group at <http://www.adcirc.org>.

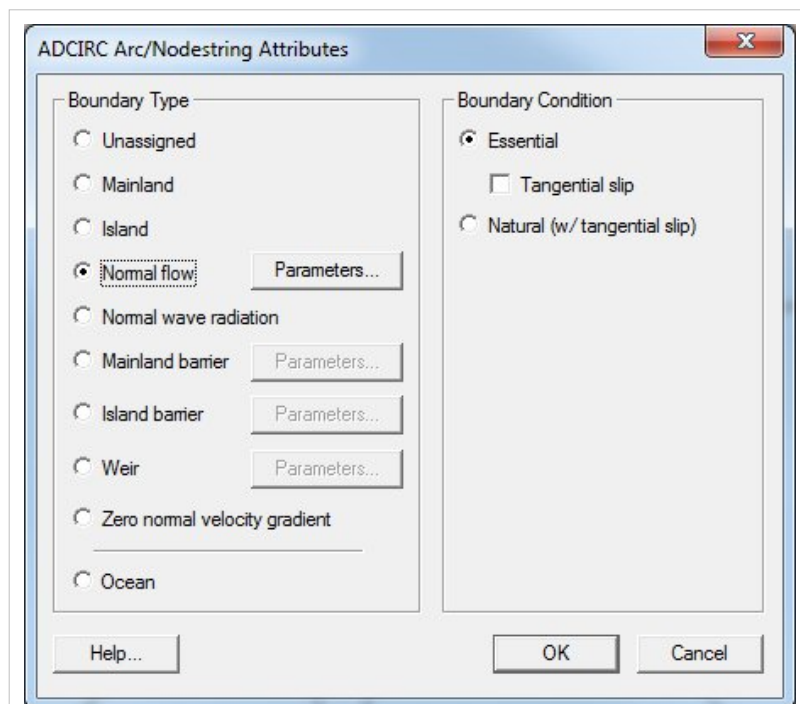
ADCIRC Boundary Conditions Nodestrings

Boundary conditions are assigned by setting attributes to *Feature Arcs* representing boundary arcs in the domain used in the simulation.

The *ADCIRC Arc/Nodestring Attributes* dialog allows the user to define the type of boundary conditions for each feature arc/nodestring.

A feature arc/nodestring can be defined as mainland, island, normal flow, mainland barrier, island barrier, tidal constituents, etc.

To set the boundary types, choose the **Select Feature Arc** tool from the Toolbox and double-click the desired arc to open the *ADCIRC Arc/Nodestring Attributes* dialog; then assign the desired boundary conditions to the arc.

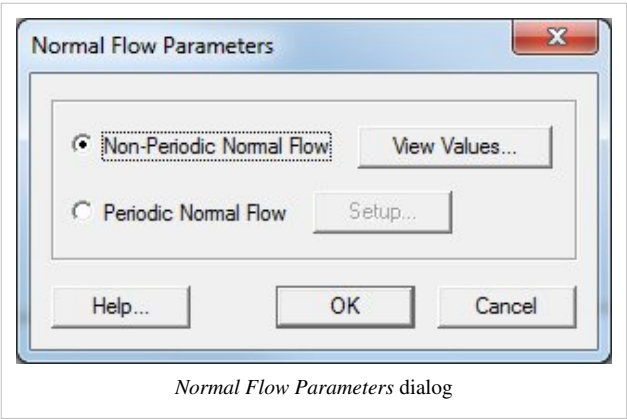


ADCIRC Boundary Conditions are assigned in the *ADCIRC Arc/Nodestring Attributes* dialog

ADCIRC Boundary Types

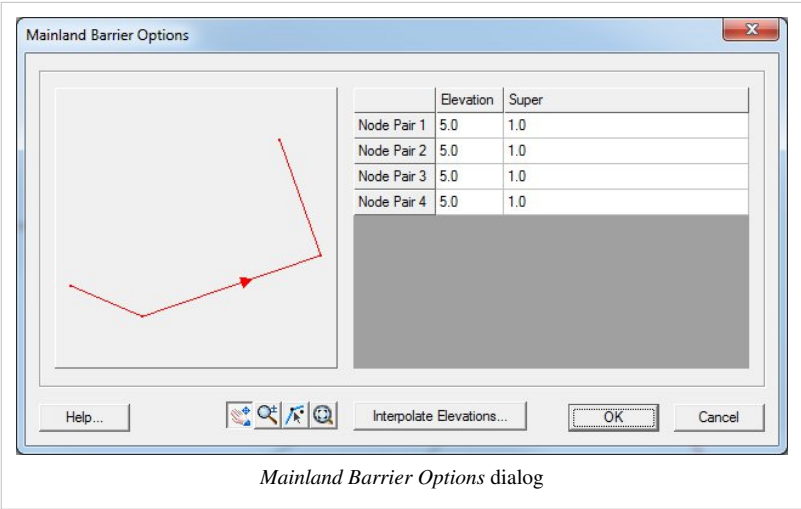
Boundary types are assigned to feature arcs in the conceptual mode or nodestrings on the ADCIRC mesh. Correct boundary type assignments are very important to run a successful ADCIRC project. The following boundary types are available in SMS for ADCIRC:

- **Unassigned** – default, no boundary condition assigned
- **Mainland** – This type of boundary represents a mainland boundary with no normal flow condition and free tangential slip.
- **Island** – This type of boundary represents an island boundary with no normal flow condition and free tangential slip. This can be selected for a closed feature arc.
- **Normal Flow** – This type of boundary represents a river inflow or open ocean boundary with a specified normal flow condition and free tangential slip. Discharges are specified either for harmonic discharge forcing or for time series discharge forcing.



Normal Flow Parameters dialog

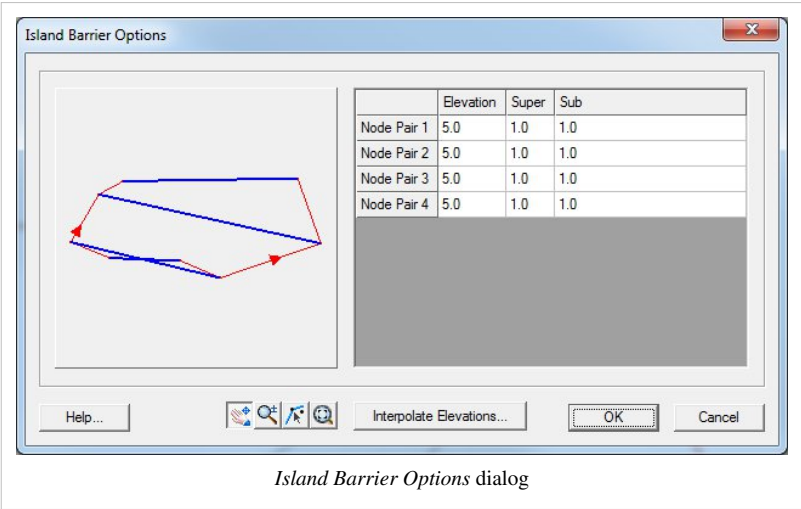
- **Normal Wave Radiation** – This type of boundary represents an open boundary where waves are allowed to propagate freely out of the domain.
- **Mainland Barrier** – This type of boundary represents a mainland boundary comprised of a dike or levee. Non-zero normal flow is computed using a supercritical, free surface weir formula if the barrier is overtopped. Zero normal flow is assumed if the barrier is not overtopped.



Mainland Barrier Options dialog

- **Island Barrier** – This type of boundary represents a dike or levee that lies inside the computational domain. Non-zero normal flow is compute using either subcritical or supercritical, free surface weir formula (based on the water level on both sides of the barrier) if the barrier is overtopped. Zero normal flow is assumed if the barrier is not overtopped. This boundary condition requires two nodestrings with an equal number of nodes. See

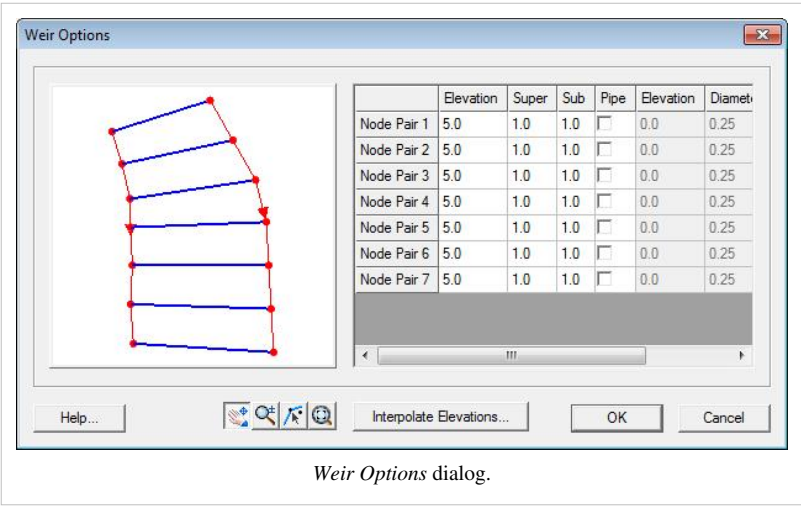
ADCIRC Weirs and Island Barriers for more information.



Island Barrier Options dialog

- **Weir** – This type of boundary is similar to an Island Barrier with the addition of cross barrier flow. Cross barrier flow simulates flow through the barrier simulating pipes or culverts from one side to the other. Flow rate and direction are based on barrier height, surface water elevation on both sides of the barrier, barrier coefficient and the appropriate barrier flow formula. In addition cross barrier pipe flow rate and direction are based on pipe crown height, surface water

elevation on both sides of the barrier, pipe friction coefficient, pipe diameter and the appropriate pipe flow formula. This boundary condition requires two nodestrings with the same number of nodes. See ADCIRC Weirs and Island Barriers for more information.



Weir Options dialog.

- **Zero Normal Velocity Gradient–**
This type of boundary forces flow through the specified nodestring reflective of flow at a fictitious point inside the domain. This is referred to as a weakly reflective boundary in some numerical engines. The fictitious point lies on the inward directed normal to the boundary a distance equal to the distance from the boundary node to its farthest 'neighbor. This should ensure that the fictitious point does not fall into an element that

contains the boundary node. The velocity at the fictitious point is determined by interpolation.

- **Ocean –** This type of boundary represents an open interface for flow with a specified water surface elevation. Elevations are specified either as tidal constituents for harmonic forcing or as time series or water level.

ADCIRC Boundary Conditions Options

Depending upon the specified boundary type, the following boundary conditions are available:

Boundary Condition	Boundary Type
Essential w/ Tangential Slip	Mainland Island Normal Flow Mainland Barrier
Essential w/o Tangential Slip	Mainland Island Normal Flow Mainland Barrier
Natural (w/ Tangential Slip)	Mainland Island Normal Flow Mainland Barrier
Tidal Constituents	Ocean
Curve	Ocean
Extract from Dataset	Ocean

Recording Stations

The SMS interface allows the user to create recording stations at specified nodal locations. At these locations, the ADCIRC model will output specified quantities at a user specified time interface. This allows for comparison of time series with observed buoy data. For example, the global output interval may be 30 minutes or 1 hour, while the recording station output could be at a higher frequency such as 6 minutes. The following recording stations can be assigned to an ADCIRC mesh node:

2D Station Types

- Elevation - ADCIRC will output a times series of computed water surface elevation at this location.
- Velocity - ADCIRC will output a times series of computed velocity magnitude at this location.
- Concentration - ADCIRC will output a times series of constituent concentration at this location.
- Meteorological - ADCIRC will output a times series of wind and pressure variables at this location.

3D Station Types (Only available when running in 3D mode - research)

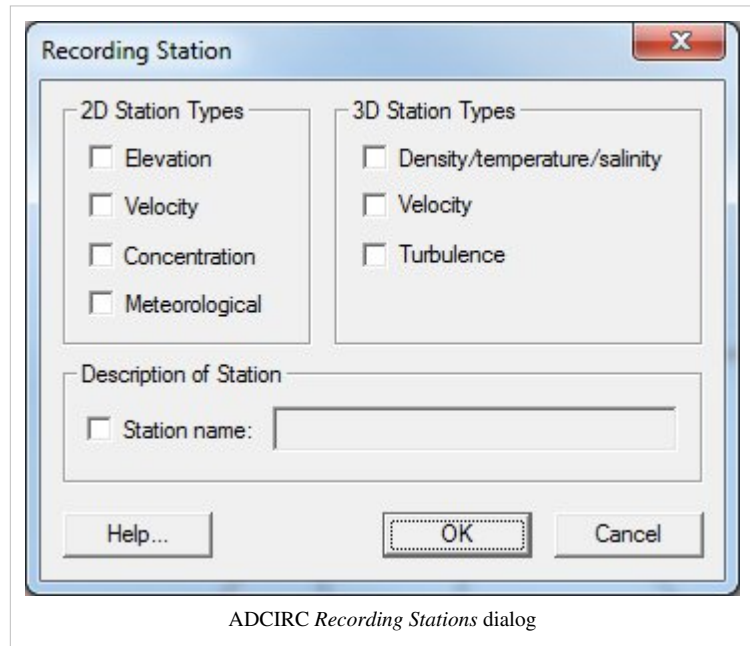
- Density/Temperature/Salinity
- Velocity
- Turbulence

Description of Station

Stations can be assigned a name to make station identification easier. A single point can be a recording station for multiple types of data (i.e. a node can record both elevation and velocity).

Related Links

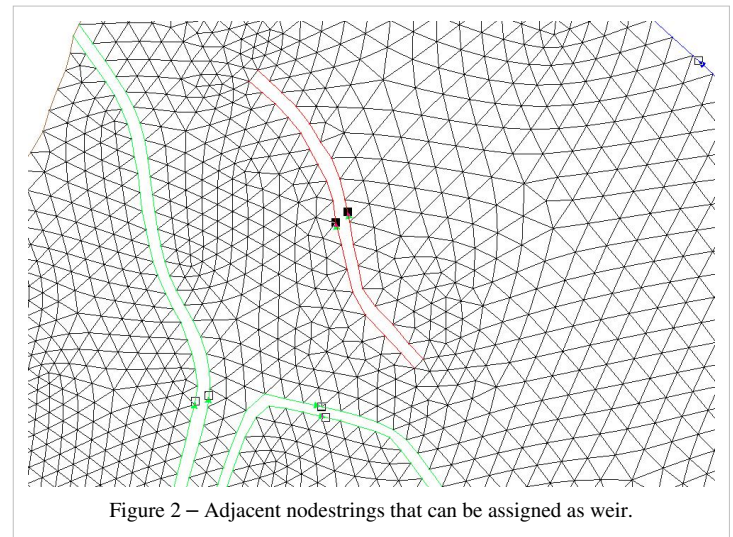
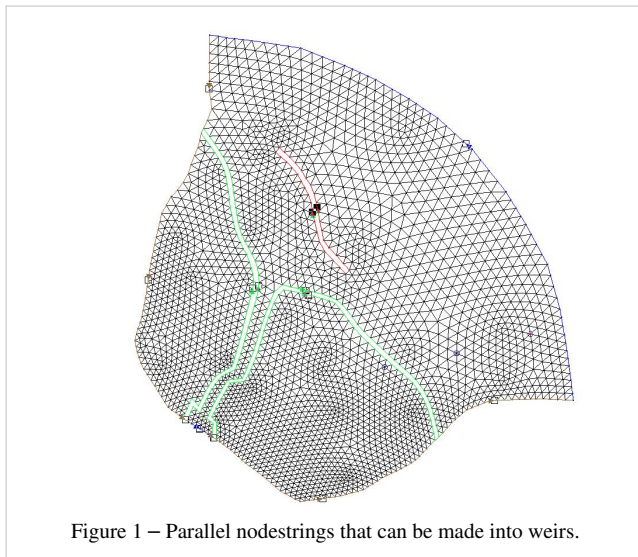
- [ADCIRC](#)
- [Coverage](#)
- [Linear Truncation Error Analysis \(LTEA\)](#)
- [Meshes](#)
- [Model Control](#)
- [Spatial Attributes](#)
- [Steering](#)



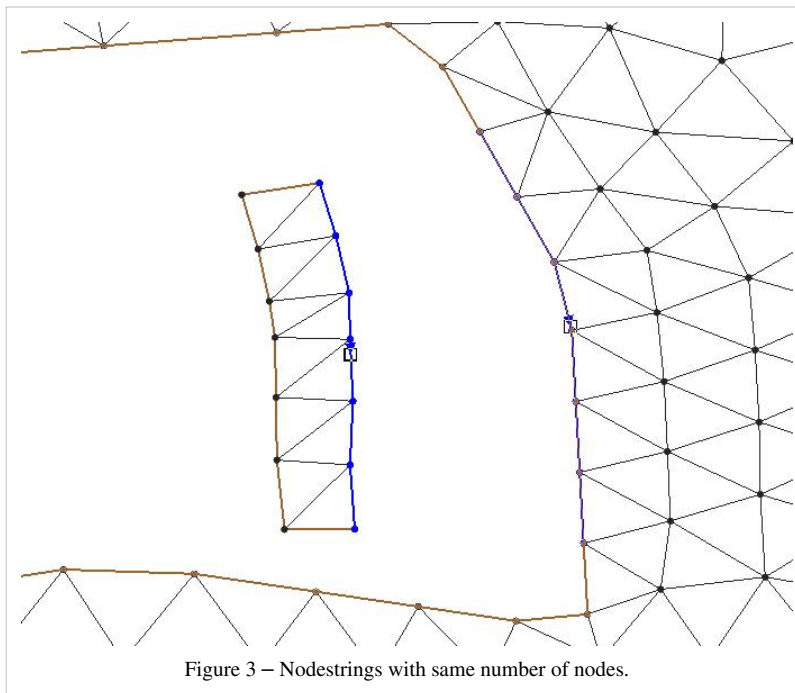
ADCIRC Weirs and Island Barriers

ADCIRC Weirs and Island Barriers

An ADCIRC weir is a boundary type that are assigned to two nodestrings on an ADCIRC mesh. A weir comprises of two nodestrings next to each other with an equal amount of nodes on each nodestrings. In order to add a weir in ADCIRC, there must be two adjacent nodestrings available as shown in figures 1 and 2 below.



The number of nodes on each nodestrings must be of an equal amount or a weir cannot be assigned. See figure 3 below. Each node and it's corresponding node on the parallel nodestring will form a node pair.



Weir Options

Once two nodestrings with same number of nodes exist, they can be assigned as a weir by selecting both weirs and selecting **Assign BC**. The *Nodestring attributes* will open. By selecting *Weir* and then clicking on the **Parameters** button, the *Weir Options* dialog will open. The dialog shows the different Node pairs and elevation for each node pair.

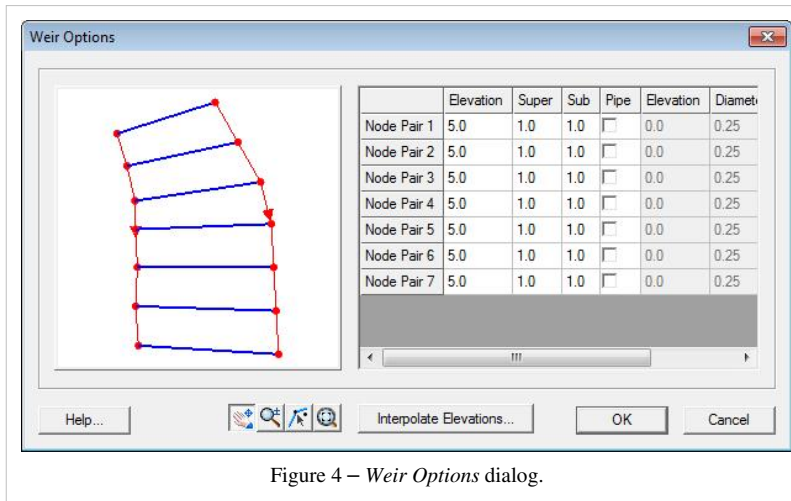


Figure 4 – Weir Options dialog.

Elevation

Elevation of each node pair. The elevation for each node pair can be entered manually or can be interpolated from a dataset

Super

Weir flow coefficients for super critical flow. A typical value for this is 1.0.

Sub

Weir flow coefficients for sub critical flow. A typical value for this is 1.0.

- Pipe
- Elevation = elevation of the pipe
- Diameter = diameter of pipe
- Coefficient = $f(L/D)$

$$f \cdot \frac{L}{D}$$

where

- L is the length of the pipe
- D is the hydraulic diameter of the pipe (for a pipe of circular section, this equals to the internal diameter of the pipe)
- f is a dimensionless coefficient called the Darcy friction factor ^[1]. It can be found from a Moody diagram ^[2] or by solving the Colebrook equation.

Interpolate Elevations Button

Clicking on the **Interpolate Elevations** button opens the *XY Series Editor*. The *XY Series Editor* can be used to generate and edit curves defined by a list of x and y coordinates. The curve can be created and edited by directly editing the xy coordinates using a spreadsheet list of the coordinates. An entire list of curves can be generated and edited with the Editor and curves can be imported from and exported to text files for future use. You can also paste your xy data directly to the spreadsheet.

Extract Elevations

Data entered in as weir or island barrier boundary condition data can be used to create arcs and/or set grid cell elevations. SMS calculates the midpoint between the two nodestrings used to create a weir or island barrier boundary condition. At each of these midpoints, the boundary condition elevation is used. To use this functionality, select the **Extract Weir Elevations...** option from the *Nodestring* menu. This will open the *Extract Weir Elevations* dialog. Here you can choose to use only selected nodestrings or all nodestrings in the mesh. If a coverage is used, SMS will create arcs that represent the weirs on the specified coverage. If one or more grids are used, SMS will set the elevation values for all grid cells that lie underneath the weirs or island barriers by interpolation.

5.2. BOUSS-2D - A Boussinesq Wave Model for Coastal Regions and Harbors

BOUSS-2D

BOUSS-2D	
Model Info	
Model type	Boussinesq Wave Model for Coastal Regions and Harbors.
Developer	Okey George Nwogu, Ph.D. ^[1] Zeki Demirebilek, Ph.D. ^[2]
Web site	BOUSS-2D web site ^[3]
Tutorials	General Section • Data Visualization • Observation Models Section • BOUSS-2D Several sets of sample problems and case studies are available. These include: • Aquaveo sample problems • Model Validation cases from the BOUSS-2D technical report

BOUSS-2D is a comprehensive model for simulating the propagation and transformation of waves in coastal regions and harbors based on a time-domain solution of Boussinesq-type equations. It is based on Boussinesq-type equations derived by Okey Nwogu and has been under development since 1993. The equations are depth-integrated for the conservation of mass and momentum for nonlinear waves propagating in shallow and intermediate water depths.

Functionality

BOUSS-2D computes nearshore wave fields including mean wave heights, mean current direction, mean water level breaking and transient representation of water levels, currents, and wave breaking.

BOUSS-2D is a comprehensive numerical model for simulating the propagation and transformation of waves in coastal regions and harbors based on a time-domain solution of Boussinesq-type equations. The governing equations are uniformly valid from deep to shallow water and can simulate most of the phenomena of interest in the nearshore zone and harbor basins including:

- Reflection/diffraction near structures
- Energy dissipation due to wave breaking and bottom friction
- Cross-spectral energy transfer due to nonlinear wave-wave interactions
- Breaking-induced longshore and rip currents
- Wave-current interaction
- Wave interaction with porous structures

The governing equations in BOUSS-2D are solved in the time domain with a finite-difference method. Input waves may be periodic (regular) or non-periodic (irregular), and both unidirectional or multi-directional sea states may be simulated. Waves propagating out of the computation domain are either absorbed in damping layers or allowed to

leave the domain freely. The SI engineering units are used in BOUSS-2D calculations.

Output Options

The user can instruct BOUSS-2D to create a variety of output files. These may include spatially varied data consisting of a value for each cell in the grid, transient data defining time series at a location, or a combination of these two options (multiple time steps of data that includes a value at each cell). The output options, along with the keyword included in the parameter file to enable these options are shown below.

- Steady-state/single value spatially varied datasets
 - Significant wave height (":HS_FILE")
 - Mean currents (":MEAN_UV_FILE")
 - Mean wave direction (":THETA_FILE")
- Transient spatially varied datasets. Each output includes data from a specified start time, to a specified end time at a specified time step.
 - Water surface elevations (":SAVE_ETA_ANIMATION")
 - Transient currents (":SAVE_UV_ANIMATION")
- Time series output at specified cells (probes). BOUSS-2D saves each type of data (for multiple locations) in a single "*.ts1" file.
 - Water surface elevations (":TS_ETA_FILE")
 - Currents (":TS_U_FILE", ":TS_V_FILE") – This saves the current at a specified elevation up from the bed. Multiple elevations can be monitored.
 - Pressure (":TS_PRESSURE_FILE") – This saves the pressure at a specified elevation up from the bed. Multiple elevations can be monitored.
 - Flow rate (":TS_Q_FILE") – This saves the flow crossing a location and can be used for overtopping.

The spatially varied data may be output in to either BOUSS-2D native files or a eXtensible Data Format File (XMDF). If BOUSS-2D format is specified, the model creates "*.grd" files for each of the single value spatially varied outputs and binary data files for the transient data. The ":SOLUTION_FILE_OPTION" in the par file instructs the model to save the data in BOUSS-2D format (if set to 0), XMDF format (if set to 1) or both formats (if set to 2). When the XMDF option is specified, the ":XMDF_SOLUTION_FILE" record must also be in the parameters file along with the name of the XMDF file to store the datasets in.

Saving BOUSS-2D

When you do a *File | Save As...* the following files get saved in the *.sms

- *.mat referenced to new save location
- *.map referenced to new save location
- Damping files saved to temp folder
- *.par referenced to new save location
- *.sol referenced to original save location unless rerun
- *.h5 referenced to new save location

Using the Model / Practical Notes

BOUSS-2D can be applied to a wide variety of coastal and ocean engineering problems, including complex wave transformation over small coastal regions (1-5 km), wave agitation and harbor resonance studies, wave breaking over submerged obstacles, breaking-induced nearshore circulation patterns, wave-current interaction near tidal inlets, infra-gravity wave generation by groups of short waves, and wave transformation around artificial islands.

As with many numerical models, BOUSS-2D can terminate or crash due to numerical instabilities. These are usually caused by problems related to the grid, the boundary conditions, or model parameters. The following lists describe common causes of instability and methods to correct them.

Instability due to the grid/geometry

- Model stability requires a low Courant number throughout the domain. SMS computes an approximate maximum time step to maintain a Courant number below 0.5. You may want to lower the timestep even more. Additionally, you may want to truncate the computational domain to areas with depth above a specified minimum. Another option is to increase resolution by using smaller computational cells. Either of these options increase run time, so before applying them, you may want to look at the other causes of instability.
- Abrupt changes in elevation from one cell to another in the computational domain could result in instabilities. You may find it helpful to smooth the grid. (A smoothing command is available by right clicking on the grid object in the project explorer in the SMS interface.)
- Computation nodes surrounded on three or four sides by land may be created during the grid creation process. These "isolated" cells may become unstable and generally don't have an impact on the wave climate. They can be converted to land cells.

Instability due to the boundary conditions

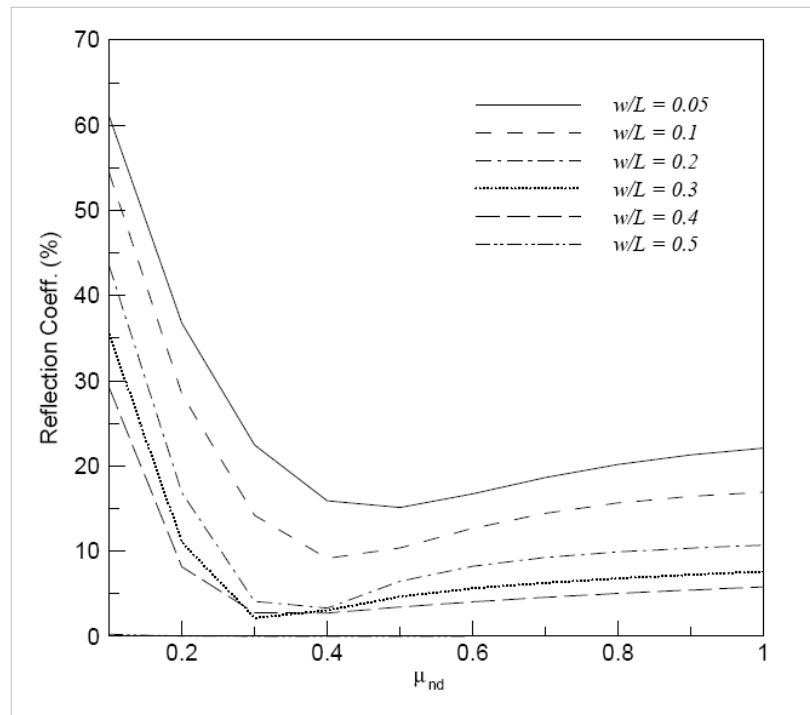
- Generally, avoid placing damping or porosity layers along structures and shorelines.
- Wave makers are more stable on the edges of the domain. Therefore, generally speaking, the wave maker should be placed on the boundary of the domain in constant (or nearly constant) depth water. (The SMS interface offers to extend the grid and transition to constant depth if a wave maker is created in a location with more than 20% variation in depth.) This is especially true in real world applications where reflected waves are of no concern. Also, when simulating large waves, the greater stability of external wavemakers may be required.
- Wave makers should be placed far enough from shore to avoid interaction between the wave maker and reflecting waves. This is because the external boundary behind the wave maker is treated as a vertical wall.
- Exceptions, or applications in which internal wavemakers (i.e. wavemakers placed inside the domain) are recommended include:
 - In applications with significant reflections from structures inside the computational domain. When reflected are caused by coastlines, structures, or bathymetry (reflected wave sources), the simulated seastate will become less uniform spatially, and the simulation may not reach a steady-state condition. The resulting wave field in such simulations will generally consist of nodes and anti-nodes that resemble a standing wave pattern, where waves appear to be bouncing back and forth inside the domain. If reflected waves cannot escape through boundaries of the modeling domain (or are constrained to exit the domain), a steady-state condition technically cannot be reached irrespective of the length of simulation. When reflected waves intercept external wavemakers, the extremes (lows and highs) in the calculated wavefield may keep building and can eventually lead to model instabilities.
 - Internal wavemakers should be used for finite domains and especially for limited area physical modeling studies, and with the above specified guidance.
- If wavemakers are placed on the interior of the domain, they should cross the entire domain to avoid potential "end effects", and have a damping layer placed behind (on the seaward side of) the internal wavemakers to absorb

reflected waves. There should also be a gap (at least one non-damped cell) between the internal wavemaker and the damping layer located offshore.

In the absence of laboratory or field data to calibrate damping and porous layers for an application, users should consider multiple simulations with a range of damping widths and/or coefficients. This graph from BOUSS-2D's technical report illustrates the variation of effective reflectivity given various damping coefficients and damping layer widths.

To use this graph:

- Compute L (the wavelength for the incident wave).
- Select a w/L ratio. Use this ratio to compute w (damping width).
- Select an expected reflection percentage. Follow a horizontal line for this percentage on plot to intersect the graph for selected w/L ratio. Read associated damping coefficient from plot.
- Note that the reflection coefficient is very sensitive to a change in damping coefficient when the coefficient is small (< 0.3) and much less sensitive when the coefficient is larger.
- This process may require the damping parameters be changed when different wave conditions are considered.
- It should be observed that this plot is for normally incident waves. Different reflection coefficients would be obtained for obliquely incident waves.
- Damping layers should be 5-10 cells wide.



Instability due to model parameters

- The model includes a Smagorinsky term to account for subgrid turbulence. If you know turbulence is expected this term can be left at the default (0.0), however, it may be increased to increase stability. (This should be done with caution. Remember, don't suppress the wiggles, they are trying to tell you something.)

External Links

- CHL BOUSS-2D website [3]
- May 2007 ERDC/CHL CHETN-I-73 Infra-Gravity Wave Input Toolbox (IGWT): User's Guide [4]
- May 2005 ERDC/CHL CHETN-I-70 BOUSS-2D Wave Model in SMS: 2. Tutorial with Examples [5]
- Mar 2005 ERDC/CHL CHETN-I-69 BOUSS-2D Wave Model in the SMS: 1. Graphical Interface [6]
- Sep 2001 ERDC/CHL TR-1-25 BOUSS-2D: A Boussinesq Wave Model for Coastal Regions and Harbors [7]

Related Topics

- Cartesian Grid Module
- BOUSS-2D Files
- BOUSS-2D Model Control Dialog
- BOUSS-2D Calculators
- CGWAVE page
- SMS Models page
- Spectral Energy

BOUSS-2D Using the Model

BOUSS-2D Grid

When using the model you will need a BOUSS-2D Cartesian grid. If a grid has already been created or an existing simulation read, the grid object will exist in the Project Explorer and selecting that object will make the Cartesian grid module active.

To create a bouss-2d grid:

1. Right-click on the map data tree item and select new coverage.
2. Select **BOUSS-2D**.
3. Give your coverage a name then click **Ok**.
4. Click out a grid frame using the **Grid Frame** tool.
5. Follow the instructions found at Creating 2D Cartesian Grids.

Boundary Condition data

All numeric models require boundary condition data. In BOUSS-2D, boundary conditions are defined along arcs for porosity/damping, and on nodes for wave-makers.

Damping / Porosity coverage

Damping and porosity coverages contain arcs that represent damping/porosity. The location of the arcs will be the location of the damping/porosity when running a simulation. Assigned to the arcs is the attributes of width and value. Multiple arcs can be created in a single coverage.

To create a damping/ porosity coverage:

1. Right-click on the map data tree item and select new coverage
 2. Select **BOUSS-2D Damping or Porosity**
 3. Give your coverage a name then click **Ok**
 4. Select the newly created coverage
 5. Click on the **Create Feature Arc** tool
 6. Click out an arc over the grid where you want your damping/porosity
 7. Click on the **Select Feature Arc** tool
 8. Click on an arc, then right-click and select **Attributes**
-

Wave Maker coverage

Wave maker coverages contain points that represent a wave maker. The location of the point will be the location of the wave maker when running a simulation. Assigned to the points are the wave maker settings including type and direction.

To create a wave maker coverage:

1. Right-click on the map data tree item and select new coverage
2. Select **BOUSS-2D wave maker**
3. Give your coverage a name then click **Ok**
4. Select the newly created coverage
5. Click on the **Create Feature Point** tool
6. Click out a point over the grid where you want your wave maker
7. Click on the **Select Feature Point** tool
8. Click on an arc, then right-click and select **Attributes**

Related Topics

- BOUSS-2D

BOUSS-2D Simulations

Simulations are available in BOUSS-2D starting in SMS 11.2. A simulation should contain a BOUSS-2D grid, a wave maker coverage, and optional damping and porosity coverages. Right-clicking on the simulation will display the options and dialogs.

To create a new Simulation:

1. Right-click on empty space in the project explorer and select **New Simulation→BOUSS-2D**
2. Drag and drop bouss2d grid, wave maker/damping/porosity coverages under the newly created **Simulation** tree item.

A New tree item will appear

Menu Commands

Right-clicking on the simulation tree item will display a list of menu commands and dialogs that are available. These items are:

- Generating Arcs along land boundary – open a dialog that will create arcs in a damping or porosity coverage from an existing grid or scatterset.
 - Generating Arcs along open boundary – open a dialog that will create arcs in a damping or porosity coverage from an existing grid or scatterset.
 - Calculators
 - Spectral Energy
 - BOUSS-2D ProbesProbe Manager
 - Model Control
-

Output Options

The user can instruct BOUSS-2D to create a variety of output files. These may include spatially varied data consisting of a value for each cell in the grid, transient data defining time series at a location, or a combination of these two options (multiple time steps of data that includes a value at each cell). The output options, along with the keyword included in the parameter file to enable these options are shown below.

- Steady-state/single value spatially varied datasets
 - Significant wave height (":HS_FILE")
 - Mean currents (":MEAN_UV_FILE")
 - Mean wave direction (":THETA_FILE")
- Transient spatially varied datasets. Each output includes data from a specified start time, to a specified end time at a specified time step.
 - Water surface elevations (":SAVE_ETA_ANIMATION")
 - Transient currents (":SAVE_UV_ANIMATION")
- Time series output at specified cells (probes). BOUSS-2D saves each type of data (for multiple locations) in a single "*.ts1" file.
 - Water surface elevations (":TS_ETA_FILE")
 - Currents (":TS_U_FILE", ":TS_V_FILE") – This saves the current at a specified elevation up from the bed. Multiple elevations can be monitored.
 - Pressure (":TS_PRESSURE_FILE") – This saves the pressure at a specified elevation up from the bed. Multiple elevations can be monitored.
 - Flow rate (":TS_Q_FILE") – This saves the flow crossing a location and can be used for overtopping.

The spatially varied data may be output in to either BOUSS-2D native files or a eXtensible Data Format File (X MDF). If BOUSS-2D format is specified, the model creates "*.grd" files for each of the single value spatially varied outputs and binary data files for the transient data. The ":SOLUTION_FILE_OPTION" in the par file instructs the model to save the data in BOUSS-2D format (if set to 0), X MDF format (if set to 1) or both formats (if set to 2). When the X MDF option is specified, the ":X MDF_SOLUTION_FILE" record must also be in the parameters file along with the name of the X MDF file to store the datasets in.

The output options are located in the model control.

- Model Check – The model check will warn you of potential errors that you should consider fixing before running the model.
- Exporting – When the BOUSS-2D files are exported a new directory under the project called "BOUSS-2D" is created. In this directory the grids will be written to. Also a new directory under the "BOUSS-2D" will be created and it will be the name of your simulation. In this directory you will find the *.par file(link to Parameter file)
- Running the model

BOUSS-2D Calculators

Wave Characteristics Calculator

- Input Wave Characteristics
 - Period, T
 - Depth, h
 - Average Wave Height, H
- Calculated Wave Characteristics
 - Wavelength, L
 - Wave Number, k
 - Celerity, C
 - Group Velocity, C_g
 - Angular Velocity
 - Maximum Wave Height
 - Steepness

BOUSS-2D Calculators

Wave Characteristics | 1D Runup and Overtopping

Input Wave Characteristics

Period, T : 0.0 sec

Depth, h : 0.0 m

☐ Average Wave Height, H : 0.0 m

Calculated Wave Characteristics

Wave Property	Value	Unit
Wavelength, L		m
Wave Number, k		
Celerity, C		m/s
Group Velocity, C_g		m/s
Angular Velocity		rad/s
Maximum Wave Height		m
Steepness		m/m

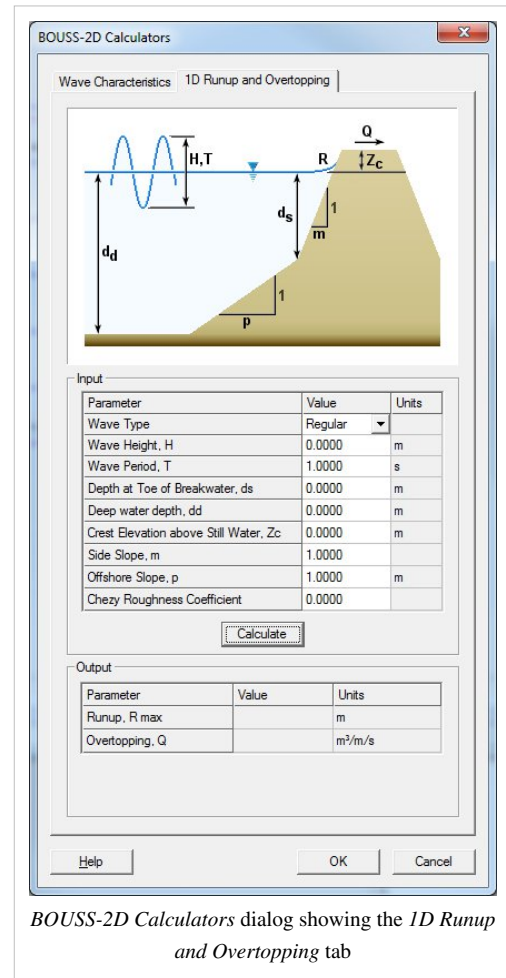
Help OK Cancel

BOUSS-2D Calculators dialog showing the Wave Characteristics tab

Run-up and Overtopping Calculator

To assist the user in the design of coastal structures, the interface includes a one-dimensional wave run-up and overtopping calculator. This utility runs a 1D simulation with BOUSS-1D based on user specified parameters. To access the *1D Run-up and Overtopping* calculator select the **Calculators** menu item in the *BOUSS2D* menu to bring up the *BOUSS-2D Calculators* dialog. The *1D Run-up and Overtopping* calculator is a tab in the *BOUSS-2D Calculators* dialog. The *BOUSS-2D Calculators* dialog is always in the BOUSS-2D interface.

Input to the *1D Run-up and Overtopping* calculator is organized into a spreadsheet. The first row of the input parameters spreadsheet is fixed and will contain the column titles given in the table below. The remaining rows contain the input parameters as shown in the dialog. The names and units columns are read-only. The value column is editable.



BOUSS-2D Calculators dialog showing the 1D Runup and Overtopping tab

Parameter	Value	Units
Wave Type	Choose between "Regular" and "Irregular". Titles of the wave height & period change depending on wave type. Default is "Regular"	-
Wave Height(H)(regular) or Significant Wave Height (Hs) (irregular)	User must specify	(m)
Wave Period(T) (regular) or Peak Period (Tp) (irregular)	User must specify	(sec)
Depth at Toe of Breakwater (ds)	User must specify	(m)
Crest Elevation above Still Water (Zc)	User must specify	(m)
Side Slope (m)	User must specify	-
Offshore Slope (p)	User must specify	-
Chezy Roughness Coefficient	User must specify	m ^{1/2} /sec

Output of the calculator is displayed on the bottom portion of the dialog after the calculate button is clicked. The output is organized into a spreadsheet. When the dialog first comes up, the values in the output parameters spreadsheet are blank. The output parameters calculated are:

1. Run-up, R. This value corresponds to Rmax for regular waves and R2% for irregular waves. The units of run-up are in meters

2. Overtopping, Q. A single overtopping value will be computed. The units for overtopping are (m³/s)/m.

BOUSS-2D Files

Input Files

- Required files
 - Parameter File (*projname.par*)
 - Bathymetry File (*projname_bathy.grd*)
- Optional files
 - Damping File (*projname_damping.grd*)
 - Current File (*projname_current.grd*)
 - Porosity File (*projname_porosity.grd*) (Still under development - be advised to use with caution)
 - Variable Roughness File (*projname_friction.grd*) (Still under development - be advised to use with caution)

Output Files

- Spatial files - values at every active computation point, These can be saved in 5 separate native BOUSS-2D formatted files (three ASCII and two binary) or in a single binary X MDF file (*projname.h5* – this is an HDF5 file). The native format files include:
 - Transient datasets
 - Water Surface File (*projname.eta*) (This also includes breaking index at each cell.)
 - Current File (*projname.uv*)
 - Mean values
 - Significant Wave Height File (*projname_hs.grd*)
 - Mean Current File (*projname_mean_uv.grd*)
 - Mean Water Level File (*projname_mwl.grd*)
- Locational files – These files contain information for specifically selected cells (probes)
 - Water Level at Probes File (*projname_ts_eta.ts1*)
 - U Component of current at Probes File (*projname_ts_u.ts1*)
 - V Component of current at Probes File (*projname_ts_v.ts1*)
 - Pressure at Probes File (*projname_ts_p.ts1*)

Related Topics

- BOUSS-2D
-

BOUSS-2D Graphical Interface

The BOUSS-2D graphical interface includes tools to assist with creating, editing, and debugging a BOUSS-2D model. The BOUSS-2D interface exists in the Cartesian Grid Module.

BOUSS-2D Menu

The following menu commands are available in the BOUSS-2D Menu:

- **Spectral Energy**

Brings up the *Spectral Energy* dialog to define/view wave energy spectra. Generally, BOUSS-2D will generate wave conditions internally, but a spectrum may be input. This command also allows the user to visualize wave spectra that are generated inside of the model.
 - **Assign BC**

This command is used to assign damping, porosity, or wave maker conditions along a selected cell string(s). Using this command will open the *BOUSS-2D Boundary Conditions* dialog.
 - **Assign Cell Attributes**

Selected cells can be defined as land or water
 - **Probe Manager**

Brings up the *Probe Manager* to control time series output from the model
 - **Calculators**

Brings up a pop up menu to access the *Wave Conditions Calculator* (see appendix A) as well as the Run-up/Overtopping Estimator.
 - **Model Check ...**

Under development – check for common problems.
 - **Model Control...**

Brings up the *Model Control* dialog to specify model parameters.
 - **Run BOUSS-2D**

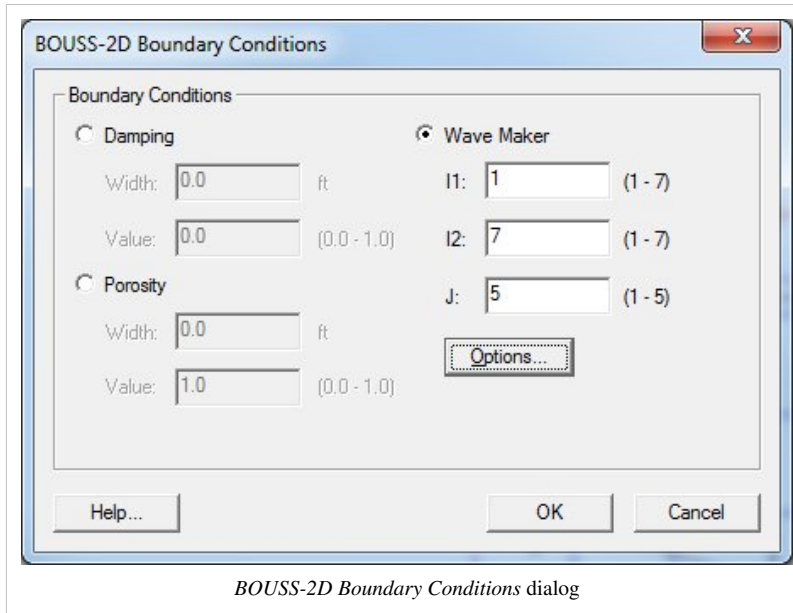
Brings up a dialog that allows the user to check what executable of BOUSS-2D should be run and then runs the model with the currently loaded simulation. As the model runs, a dialog monitors progress of the model and gives the user status messages. When the run is complete, the spatial solutions are read in for analysis and visualization.
-

Model Control

The BOUSS-2D Model Control Dialog is used to setup the options that apply to the simulation as a whole. These options include time controls, run types, output options, global parameters, print options and other global settings.

Boundary Conditions

All numeric models require boundary condition data. In BOUSS-2D, boundary conditions are defined on cell strings. The default boundary condition is a closed boundary (no flow).



BC Cell Strings

When the BOUSS-2D grid is created, SMS creates cell strings around the computational boundaries of the domain. A cell string is a list of contiguous node locations in the grid. The image to the right shows a grid with four cell strings that were automatically created. The cell string on the left includes the cells along the open ocean. This one has been assigned to be a wave-maker. The one on the right defines the interface between ocean and land along the coastline, and the top and bottom define the portion of the grid that are open to the ocean on those sides.

Cell strings can also be created manually to specify the location of structures, wave-makers, and areas where damping and/or porosity layers may be necessary.



Boundary conditions are specified along cell strings in the BOUSS-2D Boundary Conditions dialog, which is accessed by selecting one or more cell strings using the select cell string tool, and then selecting the Assign BC menu item from the BOUSS-2D menu. Normally, the user will select a single cell string and assign a boundary condition. If a boundary condition already exists for the selected cell string, the attributes are displayed. The different options for a cell string include:

1. *Unassigned BC* – When a cell string is created in BOUSS-2D its default boundary condition type is Unassigned. Unassigned cell strings do not influence the model. In fact, unassigned cell strings are not saved as part of the BOUSS-2D input files.
2. *Damping BC* – Waves propagating out of the computational domain are absorbed in damping regions (or damping layers) placed around the perimeter of the computational domain. Damping layers can also be used to model the partial reflection from harbor structures inside the computational area. The user must enter a physical width into the “Width” edit field to specify the size of the damping layer. The damping region extends the width on either side of the cell string. The damping value is a non-dimensional damping coefficient that is allowed to vary from 0.0 to 1.0. No damping will occur when a value of 0.0 is used. Waves will be damped when a value of 1.0 is used along the side boundaries. A typical value for shoreline is 0.1. The default damping value is 1.0. SMS will assign the value specified at the cell string and ramp down to 0.0 at a distance of “width” from the cell string.
3. *Porosity BC* – Porosity boundary conditions are used to simulate partial wave reflection and transmission through surface-piercing porous structures such as breakwaters. Enter a physical width into the “Width” edit field to specify the size of the porous structure. Like with the damping regions, this width is extended on both sides of the cell string. The porosity value is a non-dimensional porosity coefficient that is allowed to vary from 0.0 to 1.0. A value of 0.0 corresponds to an impervious structure, while a value of near 1.0 would correspond to a highly porous structure. Typical porosity for stone type breakwaters is 0.4. The default porosity value is 1.0.
4. *Wave-maker BC* – The wave-maker option is only available when a single cell string is selected and that cell string lies in a single column or row (straight line). Legal cell strings can be created using the *SHIFT* key when creating cell strings, using automatically created cell strings along a grid boundary, or by creating short cell strings. The extent and position of the wave maker can be modified using I,J indices in the dialog. BOUSS-2D generates waves emanating from this cell string. The properties of the waves are defined using the Wave Generator Properties dialog (described below) that is accessed through the Options button. The edit fields are used to position and size the wave maker in the computational domain. The first two values are the Start and End cells of the wave maker along the column or row that is specified by the third value, which is the Offset value. The Start and End values are limited to the number of cells in either the I- or J-direction, and the Offset value is limited to the number of rows or columns.

When the **OK** button is clicked, a check is done to see if the wave maker cell string is at a constant depth. If the depth varies by more than 20% and the wave maker is on the edge of the grid (not internal), the user is asked whether they want to force constant depth along the wave maker cell string or not. If so, the grid is extended to allow the wave maker to be at the deepest elevation along the string, with a maximum slope of 1:10 from the existing grid to the new wave-maker position. A Wave Calculator is provided as part of BOUSS-2D interface in SMS (see Appendix B) to assist users in the preparation of wave input parameters required by the model. Note that the BOUSS-2D | Assign BC menu item is disabled any time multiple wave makers are selected or if a wave maker and one or more other cell strings are selected.

Running the Model

The BOUSS-2D files are written automatically with the SMS project file or can be saved separately using the *File | Save BOUSS-2D* or *File | Save As* menu commands. See BOUSS-2D Files for more information on the files used for the BOUSS-2D run.

BOUSS-2D can be launched from SMS using the *BOUSS-2D | Run BOUSS-2D* menu command. A check of some of the common problems called the *Model Checker* is done each time the model is launched, or by selecting the *BOUSS-2D | Model Check* menu command.

Related Topics

- Create a 2D Cartesian grid

BOUSS-2D Model Control

The **Model Control...** command in the *BOUSS-2D* Menu opens the *Model Control* dialog. This dialog is divided into sections for different types of parameters which are used by the model as it runs. These include:

- *Input Datasets* – These inputs specify spatially varied input parameters for the model. In addition to the bathymetry, BOUSS-2D can utilize spatially varied damping, porosity and currents. In this section, the user defines whether these inputs are used (“None” is selected if not), and if so, how they are defined. The Damping and Porosity values can be specified using boundary conditions on cell strings or selected from the scalar functions. The Current is selected from the available vector functions.

BOUSS-2D Model Control dialog

- *Time Control* – The interface displays the recommended values for Duration and Time step in parentheses. The Courant number (Nwogu and Demirbilek 2001, page 15, 35^[7]) is computed based on the cell size and the specified time step. A value 0.3 to 0.5 is recommended.
- *Laminar and Turbulent Viscous Flow Coefficients for Porosity* – The Turbulent value ranges from 0.0 to 3.6 with a default value of 2.4. The Laminar value ranges from 0 to 1500 with a default value of 800. Stone size represents the characteristic stone size (d50) in meters of the breakwater armor layer and must be greater than zero.
- *Grid Information* – This section displays the attributes of the active grid and cannot be edited from within the Model Control dialog.
- *Parameters* – The Tidal Offset value is the elevation of the water level relative to the still water level. Check the Enable Wave Runup box to enable wave runup calculations. A minimum flooding depth must be specified for

runup calculations with a default value of one-hundredth of the wave height. The Chezy coefficient ranges from 0.0 to 1000.0 (default = 50) and the Smagorinsky coefficient must be greater between 0.0 and 2.0 (default=0.2). The choice for Nonlinear Option is either Strong or Weak. Check the Enable Wave Breaking box to enable simulation of wave breaking and enter a value for the Turbulent Length Scale with a default value equal to the wave height.

- *Variable Roughness* –
 - In the model control options, the user can select whether they want to use a constant Chezy coefficient, or variable roughness. With the option variable roughness, users can create an editable dataset of the roughness for the grid from the model control dialog.
 - A roughness coverage may also be used to create variable roughness in a BOUSS2D Cartesian grid. The roughness coverage is created in the same way as in BOUSS2D Runup/Overtopping. In the coverage, polygons can be created and given a Chezy coefficient. With an active BOUSS2D Cartesian grid created, right-clicking on the roughness coverage will allow the option "Map → Active Grid". This will pop-up a dialog, which will ask for a dataset name and a default Chezy roughness value to use where no polygon overlaps the given grid cell. Clicking **OK** will create an editable roughness dataset for the grid and set the model control option to variable roughness.
- *Spatial Output Options* – In this section of the dialog, the user defines what output files BOUSS-2D should create. The Override default file prefix box allows the user to specify a file name for these solution files. The default file prefix is the filename of the *.par file.
 - By checking Output WSE and/or Output Velocity the user tells the model to output time varying datasets. By default, the model will save data at each time step. This would result in huge files, so normally the user will select the Override Default toggle. This enables the Begin Output, End Output, and Step controls. The Begin & End Output values must be greater than 0.0 and less than the run duration. Also, the begin time must be less than the end time. The Step control determines how often the model will save time-varying output. SMS computes the number of frames and an approximate amount of memory required for the resulting datasets. If snap shots throughout the simulation are desired, that time range can be the entire simulation with a large interval to prevent the number of time steps from becoming excessive. These time steps are then not close enough together to create a coherent animation. Another option is to view only a couple of wave cycles, and make the time interval 5–10% of the wave period to get enough frames to generate a smooth animation of the waves in that time range. As mentioned, the time varied quantities include:
 - Water Surface Elevation and breaking
 - Velocity.
 - BOUSS-2D can output spatial data (1 value for each cell in the grid). These quantities are read by the SMS as stored as datasets on the grid. The options include up to four steady state quantities and up to two transient (time varying) quantities. The steady state options include:
 - Significant Wave Height
 - Mean Currents
 - Average Wave Direction (Theta)

Related Topics

- BOUSS-2D

BOUSS-2D Parameter File

The BOUSS-2D parameter file contains a series of comments and keywords that define the components and control the options of a simulation. Any line that begins with "#" symbol is a comment and is ignored by the model. However, some utilities exist that may utilize comments.

Keywords begin with a ":" and are followed by values associated with the keyword.

The parameter file format is shown in Figure 1.

<pre># ##### # BOUSS-2D Run Parameter File: cirp_ideal.par # Written by: SMS # Creation Date: Tuesday June 22 16:49 2004 #####</pre>	File Header
<pre># # Bathymetric Grid Parameters # :BATHY_FILE filename.grd :TIDAL_OFFSET offset value</pre>	Definition of grid and tidal offset
<pre># # Damping Parameters :DAMPING_FILE filename.grd</pre>	Keywords for damping or porosity spatial input data */
<pre># # Wavemaker #1 parameters # :START_WAVEMAKER : WM_POS_I1 value (column number) : WM_POS_J1 value (row number) : WM_POS_I2 value (column number) : WM_POS_J2 value (row number) : WAVE_TYPE value (Regular/Irreg_Uni/Irreg_Multi) : WAVE_HEIGHT value (meters) : WAVE_PERIOD value (seconds) : WAVE_DIRECTION value (degrees from North) : WAVE_CYCLES value (integer) :END_WAVEMAKER</pre>	The Wave maker definition. The position values must correspond to a single row or column. The values required depend on the wave type and method of defining that wave. The values shown illustrate a regular wave.
<pre># # Simulation parameters # :DURATION value (seconds) :TIME_STEP value (seconds) :CHEZY_COEFF value (unitless default = 50) :SMAGORINSKY_CONST value (unitless default = 0.2) :NONLINEAR_OPTION value (Strong/Weak) :CHECK_WAVE_BREAKING value (Yes/No default - Yes) :TURB_LENGTH_SCALE value (meters) :CALC_WAVE_RUNUP value (Yes/No default - No)</pre>	General model parameters

<pre># # Output Parameters # :OUTPUT_FILE_PREFIX <i>file prefix</i> # # Output File for Significant Wave Height # :HS_FILE <i>file prefix_hs.grd</i> # # Output File for Mean Currents # :MEAN_UV_FILE <i>file prefix_mean_uv.grd</i> # # Output File for Mean Wave Direction # :THETA_FILE <i>file prefix_theta.grd</i></pre>	Single Timestep (Steady State) spatially varied output options
<pre># # Surface Elevation Animation File Output Options # :SAVE_ETA_ANIMATION :ETA_ANIM_FILE <i>file prefix.eta</i> :START_TIME value (seconds) :END_TIME value (seconds) :SAVE_TIME_STEP value (seconds) :SAVE_FULL_GRID value (Yes/No default = Yes) :END_SAVE_ETA_ANIMATION # # Velocity Animation File Output Options # :SAVE_UV_ANIMATION :UV_ANIM_FILE <i>file prefix.uv</i> :START_TIME value (seconds) :END_TIME value (seconds) :SAVE_TIME_STEP value (seconds) :SAVE_FULL_GRID value (Yes/No default = Yes) :END_SAVE_UV_ANIMATION # # Time Series Output File Names # :TS_ETA_FILE <i>file prefix_ts_eta.ts1</i> :TS_U_FILE <i>file prefix_ts_u.ts1</i> :TS_V_FILE <i>file prefix_ts_v.ts1</i> :TS_PRESSURE_FILE <i>file prefix_ts_pressure.ts1</i> :TS_Q_FILE <i>file prefix_ts_q.ts1</i> :TS_RUNUP_FILE <i>file prefix_ts_runup.ts1</i></pre>	Multiple Timestep (transient animations) spatially varied output options Header for time series output options - must be included to create TS1 files

# # Time Series Output Options # :SAVE_TIMESERIES value (id) :SAVE_TS_ETA value (Yes/No) :SAVE_TS_UV value (Yes/No) :SAVE_TS_PRESSURE value (Yes/No) :SAVE_TS_Q value (Yes/No) :TS_X value (meters) :TS_Y value (meters) :TS_Z_UV value (meters above bed) :END_SAVE_TIMESERIES #	Time Series specification block. Repeated as many times as needed.
# # Time Series Runup Output Options # :SAVE_RUNUP value (id) :TS_X value (meters) :END_SAVE_RUNUP #	Runup Series specification block. Repeated as many times as needed. Currently only supported for 1D mode
# # Output File for XMDf Solution File # :SOLUTION_FILE_OPTION value (0 - BOUSS-2D Native, 1 - XMDf, 2 - Both) :XMDf_SOLUTION_FILE <i>file prefix_sol.h5</i>	Output format selection

Figure 1. BOUSS2D Parameter File Format.

BOUSS-2D Probes

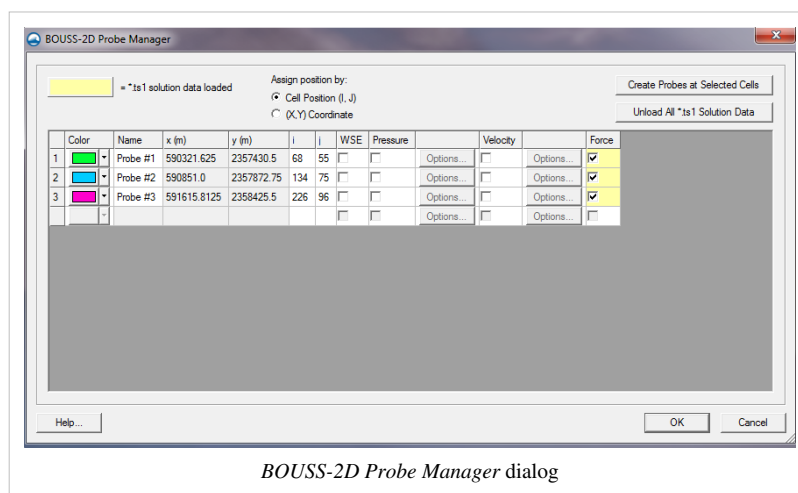
BOUSS-2D Probe Manager

BOUSS-2D can output the histories of the computed water-surface elevation, velocities, force, and pressure at every grid point and at every time-step. However, due to the number of data points in the domain, this is usually done at intervals of 15-30 min.

In order to provide a more complete temporal representation of the results of the calculation, the model allows the user to specify probes.

At a probe location (x,y,z) the user can specify what data should be saved and at what temporal resolutions. The options include water surface, velocity, force, and pressure.

The probe manager allows creating, editing, and deleting probes. This dialog is only accessible when a BOUSS-2D grid exists. The properties associated with probes are color, name, location, water surface elevation (WSE), pressure, force, and velocity.



BOUSS-2D Probe Manager dialog

Related Topics

- BOUSS-2D
- BOUSS2D Runup / Overtopping

BOUSS-2D Test Cases

Test 1 ^[1]

Simple test demonstrating the use of an internal wavemaker.

Test 2

Coming soon.

Test 3

Coming soon.

Related Topics

- BOUSS-2D

5.3. BOUSS-2D Runup/Overtopping

BOUSS2D Runup / Overtopping

The data from various coverages in SMS is combined to create a BOUSS-2D Run-up / Overtopping simulation. The different coverages are as follows:

Transects Coverage

Transects are made by creating arcs in the transects coverage. Each transect represents the 1-d grid used for a run-up simulation. It is recommended that these transects be linear. To assign elevations to the transects, use the right-click option, "Extract elevations". This will prompt you to select a dataset to use for extracting. This must be done before launching a run-up simulation.

Wavemaker Coverage

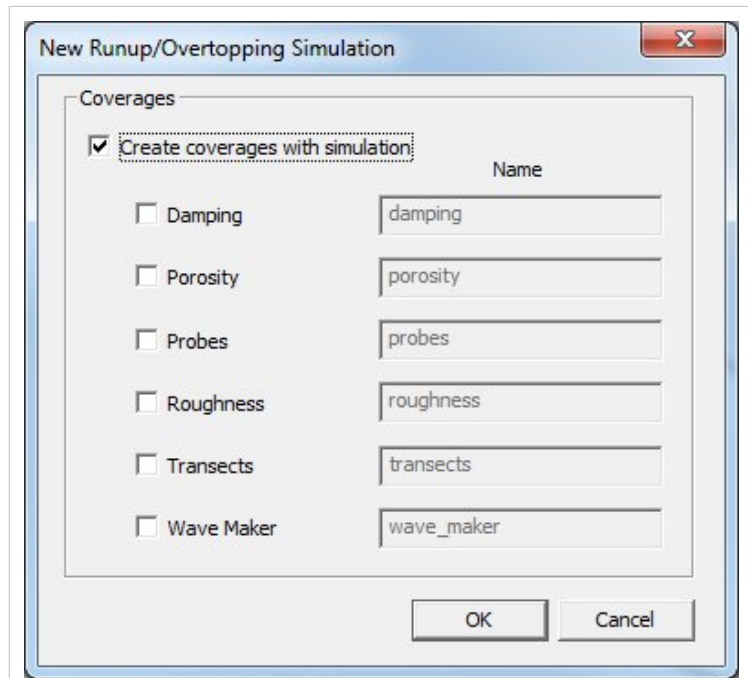
Wavemakers are made by creating arcs in this coverage. The wave maker properties can be edited by double-clicking on the arc or by right-clicking and selecting "Attributes." Each wavemaker can have multiple sets of wave parameters. Each set of wave parameters will be run in a separate simulation. It is necessary that each wavemaker arc in a simulation has the same number of wave parameters. The location of the wave makers on the transect is determined by the intersection of the transect arc and the wavemaker arc. Each transect arc may have only one wavemaker.

Probes Coverage

Probes are also made by creating arcs. The location of each probe on the transect is determined by the intersection of the transect arc and the probe arc. In the arc attributes, you can set the type of probe, which can be any of the following: Velocity, Pressure, Water Surface Elevation and Force. To create a run-up probe, simply create a polygon in the desired location. The run-up probe will be the portion of the transect arc that is inside of the polygon.

Auto-Create Probes

The *properties* dialog for the probes coverage can be used to automatically create probes.



BOUSS2D Runup/Overtopping New Runup/Overtopping Simulation dialog

Runup Probes

To automatically create a runup probe, define the following variables:

- **Minimum z** – probes will not be created below this elevation
- **Maximum z** – probes will not be created above this elevation
- **Delta z** – the difference in elevation that must occur for a probe to be created

When auto-creating these probes, SMS will traverse the transect arc, starting from the land and going seaward. Once a local maximum that is below "maximum z" is found, this point will be the starting point of the probe. SMS continues to traverse the arc until it reaches a point that is a distance of "delta z" below the starting point. This point will be the end point. However, if SMS reaches another local maximum with a higher elevation than the starting point, this point with the higher elevation will replace the original starting point.

Other Probes

To automatically create other types of probes, you need to set the following parameters:

- **Elevation (m)** – this is the elevation (along the transect) at which the probe should be placed
- **Location** – this can be set to "First" or "Last". This is used only when the defined elevation occurs more than once on the transect arc. If "First" is selected, the probe will be placed at the first qualifying location on the transect arc. If "Last" is selected, the probe will be placed at the last location.
- **Probe Types** – the available probe types are flowrate, WSE, force, pressure, and velocity. You can set which type(s) should be created for this rule. For pressure and velocity, you must define elevation(s) above the seabed where they should be placed. These can be entered by clicking the "Define..." button next to their checkboxes.

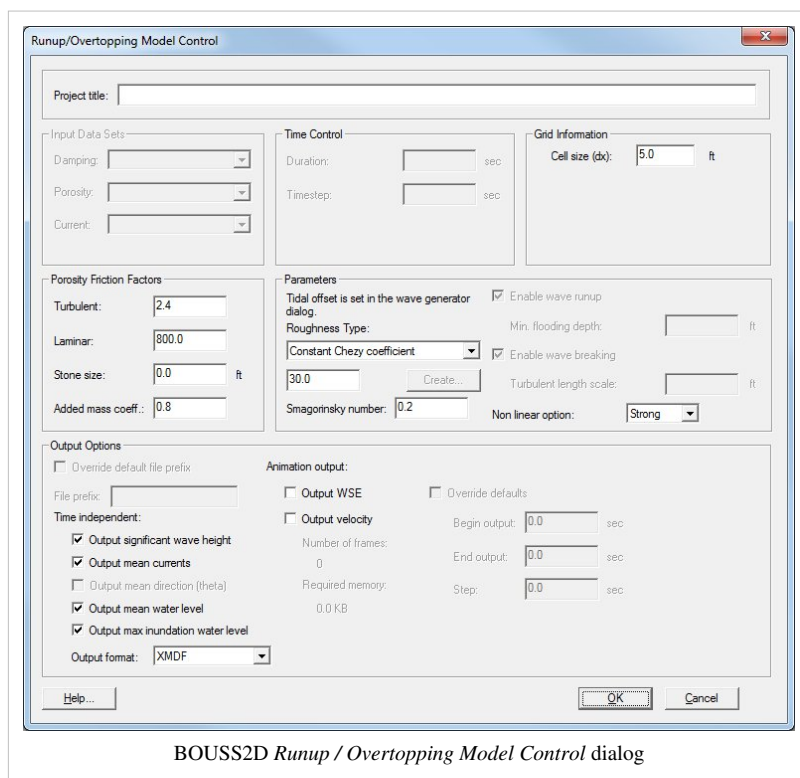
Damping / Porosity Coverage

Damping and porosity attributes are created by making arcs. The type of attribute (damping or porosity) and its coefficient and width are set in the arc properties.

Roughness Coverage

If there is varying roughness throughout a transect arc, roughness polygons can be used to define the varying roughness. Each polygon that is created will be assigned a Chezy roughness value (set in the polygon attributes). The portion of the transect arc that is inside of the polygon will be assigned that value. Otherwise, the default Chezy coefficient (defined in the model control), will be used. Note: It is helpful to turn on the display of inactive coverages while creating the wavemakers, probes, damping arcs, porosity arcs, and roughness polygons. This enables you to see the location of the transect arcs to ensure that you are intersecting the wavemakers, probes, etc., in the desired locations.

Runup / Overtopping Model Control



BOUSS2D Runup / Overtopping Input Files

The following files are used by BOUSS-2D in a run-up simulation:

- *.par
- bathy.grd
- porosity.grd (optional)
- damping.grd (optional)
- friction.grd (optional)

Each of these can be read/written by SMS. The prefix for each file will follow this format: "coveragename_arcId".

Parameter File

SMS will write a parameter file for each set of wave parameters on each transect. This file contains model control information, 1-d grid data, and the wave maker parameters. Each parameter file represents a unique simulation.

Bathymetry File

This file defines the elevations along the 1-d grid (transect arc).

Porosity File

A porosity file will be written for each transect arc that is overlapped by a porosity arc. This file defines the porosity values along the 1-d grid. If no porosity arcs are defined, the model uses a default value of 1.0 along the entire grid.

Damping File

A damping file will be written for each transect arc that is overlapped by a damping arc. This file defines the damping values along the 1-d grid. If no damping arcs are defined, the model uses a default value of 0.0 along the entire grid.

Friction File

A friction file will be written for each transect arc that is overlapped by a roughness polygon. This file defines the roughness values along the 1-d grid. If no roughness polygons are defined, the model uses the Chezy coefficient defined in the model control along the entire grid.

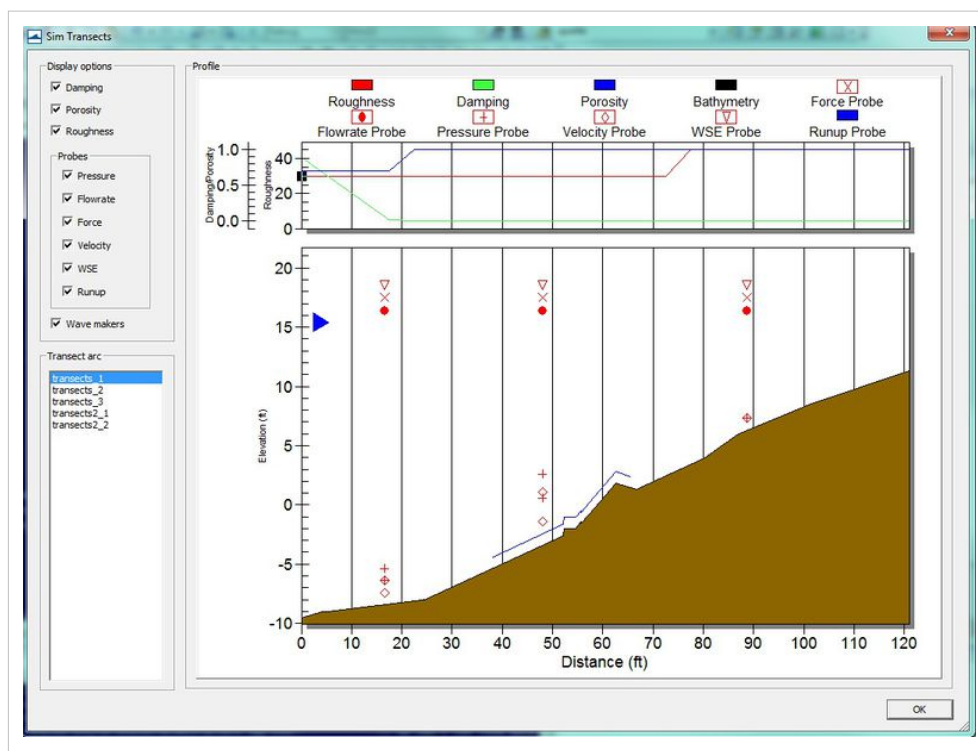
Related Topics

- BOUSS2D Runup / Overtopping

BOUSS2D Runup / Overtopping Viewing Data

Transect Profile

By using the plot wizard or by right-clicking on the simulation icon in the project explorer, you have the option to view the profile of an individual transect. Each arc in the simulation can be viewed by selecting the desired arc in the Transect arc list on the bottom left portion of the dialog. The bathymetry is displayed with the locations of probes and wavemakers. On a separate plot, the damping, roughness, and porosity can be viewed. This plot is for viewing purposes only.

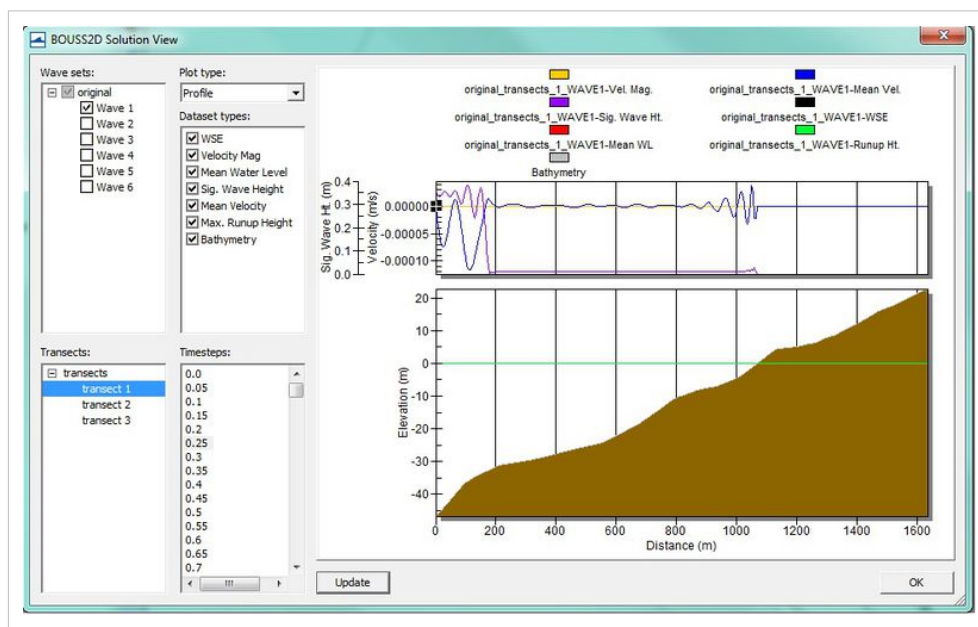


Solution View

After running a simulation, the data can be viewed in the *Solution* dialog. This dialog can be accessed through the *Plot Wizard*. There are two different views in this dialog: *Profile* and *Time Series*.

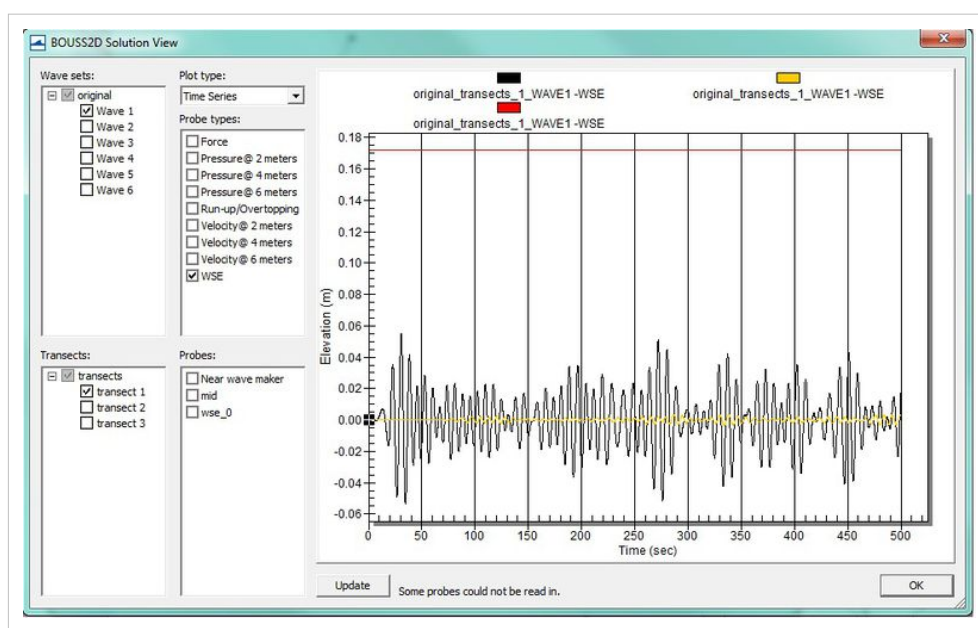
Profile

This view is used to display the bathymetry with the following datasets if they were turned on in the model control: WSE, velocity magnitude, mean water level, significant wave height, mean velocity, and maximum runup height. You can view the data for multiple wave sets for a single transect at the same time. Once the desired data is selected, clicking the **Update** button will refresh the plot window to show the selected data.



Time Series

The time series view displays the probe data from the simulation. In this view, you can display the data for any combination of transects, wave cases, and probes.



Statistics

If a simulation has been run, the "Statistics" option will be available in the right-click menu for the simulation. This dialog will display various statistics for velocity, pressure, force, and water surface elevation probes on each transect. The time series statistics display the following:

- Minimum – the minimum value in the time series
- Maximum – the maximum value in the time series
- Mean – the mean value for the time series
- Standard Deviation – the standard deviation for the time series

The "Eta Zero-Crossing" option displays:

- HAV – Average value of all peaks.
- H13 – Value exceeded by 1/3 of the peaks.
- H110 – Value exceeded by 1/10 of the peaks.
- HMAX – Maximum value of all peaks.
- TAV – Average period.
- T13 – Period exceeded by 1/3 of the peaks.
- T110 – Period exceeded by 1/10 of the peaks.

The "Runup Statistics" option displays:

- RMAX – Maximum value of all peaks.
 - R2 – Value exceeded by 2% of peaks.
 - R10 – Value exceeded by 10% of peaks.
 - R33 – Value exceeded by 1/3 of peaks.
 - RMEAN – Average of all peaks.
-

5.4. CGWAVE

CGWAVE

CGWAVE	
Model Info	
Model type	General-purpose wave prediction model for simulating the propagation and transformation of ocean waves in coastal regions and harbors, and appropriate for modeling the most significant physical processes in channels, inlets and harbors, open coastal regions, around islands and structures.
Developer	Vijay Panchang, Ph.D. ^[1] Zeki Demirbilek, Ph.D. ^[2]
Web site	CGWAVE web site ^[2]
Tutorials	General Section • Data Visualization • Mesh Editing • Observation Models Section • CGWAVE Several sets of sample problems and case studies are available. These include: • Aquaveo Sample Problems • Model Validation cases from the CGWAVE website ^[3]

The model CGWAVE (Demirbilek and Panchang 1998) is a two-dimensional wave transformation model that can be used to predict wave properties (wave heights, velocities, pressures, radiation stresses) in domains of complex shape and depth variations when an input wave condition (amplitude, direction, and period; or a spectral combination of these) is provided.

Graphical Interface

SMS provides a graphical interface that is designed to allow users to visualize the projects they are creating, easily modify project parameters, and view the solutions produced by the CGWAVE model. See CGWAVE Graphical Interface for more information.

The CGWAVE Graphical Interface contains tools to create and edit an CGWAVE simulation. The simulation consists of a geometric definition of the model domain (the mesh) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the 2D Mesh Module and setting the current model to CGWAVE. If a mesh has already been created for a CGWAVE simulation or an existing simulation read, the mesh object will exist in the Project Explorer and selecting that object will make the 2D Mesh module active and set the model to CGWAVE. See Building a Mesh for more information.

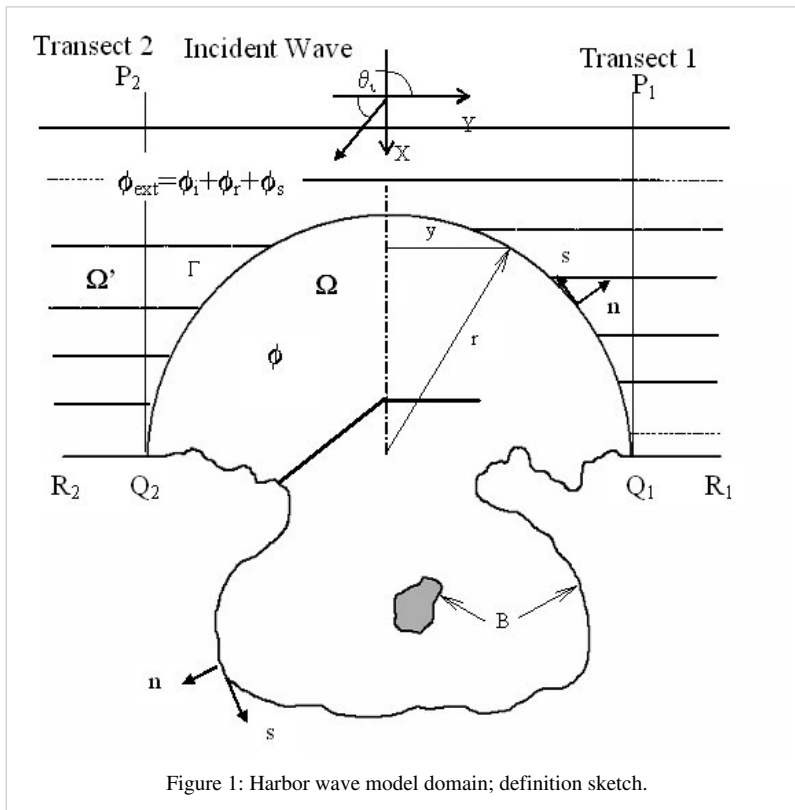
The interface consists of the 2D Mesh Module Menus and tools augmented by the *CGWAVE* menu. See CGWAVE Graphical Interface for more information.

Functionality

The model is based on extensions of the “combined refraction-diffraction” equation, which is applicable to both long and short waves and hence finds wide application in coastal engineering and harbor resonance studies. Being elliptic, the equation represents a boundary value problem, which can accommodate internal non-homogeneities (islands, structures, etc.) and boundaries. It hence forms a well-accepted basis for performing wave simulations in regions with arbitrarily-shaped (manmade or natural) boundaries and arbitrary depth variations without limitations on the angle of wave incidence or the degree and direction of wave reflection and scattering that can be modelled. In essence, it represents the complete two-dimensional wave-scattering problem for the non-homogeneous Helmholtz equation. Irregular wave conditions may be simulated using (1) by superposition of monochromatic simulations (e.g. Chawla et al. 1998; Panchang et al. 1990; Zhao et al. 2001.)

The wave phenomena that can be simulated with CGWAVE are: bathymetric refraction, diffraction by structures (e.g. breakwaters) and the bathymetry, reflection (from structures and natural boundaries (seawalls, coastlines, etc) as well as from bed slopes), friction, breaking, and floating (fixed) docks. The model uses a triangular finite-element formulation with grid sizes varying throughout the domain based on the local wavelength; the grids can be efficiently generated using the SMS graphical interface when a bathymetry file is provided. The model allows one to specify the desired reflection properties along the coastline and other internal boundaries. It is therefore particularly well-suited for simulating waves in harbors. While the basic equation is intended for monochromatic waves, irregular (i.e. spectral) wave conditions are simulated in CGWAVE through a linear superposition of monochromatic simulations (e.g. Panchang et al. 1990; Zhao et al. 2001.) Typically, simulations involving a domain containing hundreds of thousands of finite element nodes can be performed in a few minutes on a PC.

For harbor applications, the model also uses a semi-circle (as an open boundary) to separate the model domain from the outer sea. A typical CGWAVE model domain is shown in Fig. 1. The input conditions are provided at the offshore ends of two one-dimensional cross-shore sections. (In practice, the input condition is known at the end of one of the transects. The condition at the offshore end of the other transect is obtained by appropriate phase translation.) A combination of the incident and reflected waves is computed along these transects using a one-dimensional version of the governing equation; this partial solution is then mapped on to the semicircle to force the two-dimensional model. The remainder of the solution on the boundary consists of a scattered wave that emanates from within the domain; this component is allowed to radiate out through the use of an impedance boundary condition. In the model interior, a finite-element grid is used to represent the depth field, and the coastlines (denoted by B in Fig. 1) are assigned a reflection coefficient. For open ocean applications, all variations must be included inside a circle, outside which the depths are assumed constant (Panchang et al. 2000).



Using the Model / Practical Notes

Methodology

When beginning a new project, first a domain must be created to perform the calculations. This domain comes from a combination of the coastline (land side of the domain) and the bathymetric survey. Begin by reading the survey into SMS and making sure it is an adequate and accurate representation of the study area. Ensure that the data covers the entire domain, and that the existing (or natural) coastline is adequately defined. If the survey includes topographic data, the coastline can be extracted as a contour of constant depth. Since CGWAVE does not

support calculation of wave run-up, this should always be a positive depth (0.5 m or so is recommended). Follow the mesh generating techniques demonstrated in the tutorial problem for CGWAVE.

After creating the finite element mesh, it may be useful to make an initial run of CGWAVE in linear mode with fully absorbing coastlines and an incident wave condition. You may want to include both a normally incident wave run, and a separate obliquely incident wave. When CGWAVE is run in linear mode, the solution is unlimited and technically wave heights coming from the solution can grow uncontrolled. Numerically, there is no limit on wave heights. This is not realistic since waves in nature are finite in height (we know wave heights are depth-limited in shallow-water depths). However, the resulting solutions, especially the phase diagrams, can be used to perform visual quality control, since forward propagating waves are the easiest for making intuitive assessments (e.g. spacing between crests should be smaller and wave heights generally increasing in shallower water; bending of phase-lines for oblique incidence to approach the shore in a normal fashion, high wave heights in very shallow water because breaking was not applied, etc).

Repeating the incident runs with full reflection should lead to some standing wave patterns (large and small wave heights; rapid change of phase). A visual examination of the results will enhance confidence in later simulations.

Domain Shape

A mesh for use with CGWAVE must match one of two domain shapes. For cases involving a continuous coastline, the ocean boundary should be semi-circular. For cases examining waves around an island, the domain should be circular. The interface includes tools to build rectangular domains along coastlines, however, this is for research purposes only and should not be applied in projects with the current versions of CGWAVE. SMS includes tools in the Map Module to create a network with a valid domain shape (see the tutorial for CGWAVE). Therefore, it is strongly recommended that the Map Module be the principal method for creation of new networks. The Mesh Module may be used to create a mesh, however in the case of CGWAVE the Mesh Module primarily is useful for network editing and assigning model parameters required by CGWAVE. These model parameter commands are

grouped in the *CGWAVE* menu.

To the extent possible, it is necessary to have the coastlines outside the semicircle be as straight as possible, and ideally, parallel to one of the axes (as shown in Fig 1). The semicircle should be so located that the depths outside vary, to the extent possible, in a 1-d (cross-shore) direction only in the exterior. These depths are introduced in the model through the two 1-d sections. The user must specify a reflection coefficient for the exterior coastline.

Of course, these conditions are not encountered in practice in an ideal manner, but it is important to understand that the model works under these assumptions. V-shaped coastlines do not meet these criteria. Also, not all real domains have 1-dimensionally varying exterior depths. Regions with 2-d depth variations should preferably be incorporated inside the semicircle and it may be necessary to enlarge the domain so that solutions in the area of interest (hopefully) are not affected by the coastline shape. It is impossible to accommodate all real topographic variations of the model's exterior domain. Since a part of what happens outside the semicircle influences the solution within (i.e. how the backscattered waves behave), some simplified representation of the exterior is needed. The exterior representation used in *CGWAVE* is more sophisticated than that used in earlier models (which assumed collinear and fully reflecting coastlines and constant exterior depths).

A full circle option is available for open sea problems where the exterior is of constant depth; the theoretical ideal for this is the Fourier Bessel Series (which works for unit wave input only); the parabolic mode may also be used (Panchang et al. 2000) with little loss of accuracy.

Domain Size

The size of the domain is also governed by the wavelength. Typically, the minimum radius for the semicircle should be about 2 or 3 times the wavelength, which also dictates the overall number of grids. It is important that the modeler first estimate the number of grid points needed, at least in an approximate sense. This can be done by estimating a nominal wavelength L (based on the period and depth), and using $L/10$ as a rough measure of the grid size. Obviously, large domains with short period waves will lead to hundreds of thousands of nodes. This is not a problem physically, but can be very time consuming to simulate. For spectral conditions, the domain size should be dictated by the longer wavelengths L_1 (say radius = $3L_1$), but the resolution for the shorter wavelengths (L_2) would be, say $L_2/10$, which may be excessively fine for the longer wavelengths. Obviously, some care (and perhaps compromise) is needed in designing the grid and also selecting the input spectral components. If some components have little energy, it may be best to delete them from the modeling.

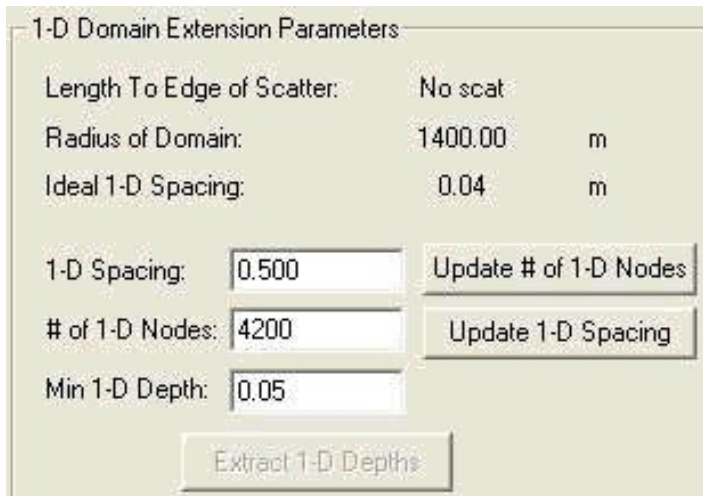
Boundary Conditions

The input wave condition (amplitude, period, direction) must be specified at the end of the 1-d cross-section. Simulations are less reliable (not necessarily wrong) if the waves approach the coast at a glancing angle (say within 20° of the coastline). The 1-d sections **MUST** reach out further than the outermost point of the semicircle. The domain should have no zero or negative depths; the land-water interface must be demarcated using a small water depth (say 0.5 m, but the limit depends on the overall water depths and wave periods and the problem being solved). Note that making this too shallow may result in a large number of grids in this area which may not be necessary for all applications).

For basic specifications, the incident wave angle specification is as follows. Waves going in the +x axis (i.e. to East) are specified as incident angle = 0° . Waves going in +y direction (i.e. to North) are specified as incident angle = 90° , to West = 180° , to South = 270° . Incident wave direction is used in degrees. Convergence tolerance is typically assigned 10^{-8} , output frequency is 100 every iterations.

For the boundary condition specification, the *parabolic plus two 1d sections* option (Option 3 in *.cgi) should be chosen for most harbor problems, with a semicircle open boundary; in earlier versions, it is possible to use the option with just one 1d section (Option 2 in *.cgi) but it really provides no benefit over Option 3. The other options consist of the *parabolic approximation* (Option 1 in *.cgi) which is suitable if the exterior region is of constant depth. The

Fourier Bessel Series (Option 0 in *.cgi) must be used if its limitations, described above in Section 2.2 (equations 15-17), are met; typically it is sufficient to specify 15 terms in the infinite series. For harbor problems, SMS will create the 1d sections based on depths based on depths on semicircle, and the desired grid spacing (used to solve the 1d equation (20)), based on (approximately) $L/10$ must be specified in the dialog box:



The dialog box titled "1-D Domain Extension Parameters" contains the following fields and buttons:

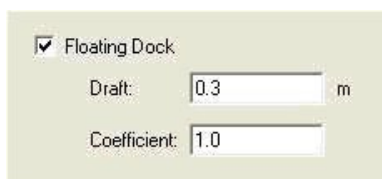
- Length To Edge of Scatter: No scat
- Radius of Domain: 1400.00 m
- Ideal 1-D Spacing: 0.04 m
- 1-D Spacing: 0.500 (with an "Update # of 1-D Nodes" button)
- # of 1-D Nodes: 4200 (with an "Update 1-D Spacing" button)
- Min 1-D Depth: 0.05
- An "Extract 1-D Depths" button at the bottom.

Wave Breaking

There are two methods for applying wave breaking. For practical problems, we recommend making a non-breaking simulation and then applying an H/d limit (generally between 0.6 and 0.85) to alter the calculated wave heights to the limit (say $0.78d$). The other, more rigorous approach is to run the model in the nonlinear breaking mode where breaking properties are continuously recalculated based on prior solutions. This requires several rounds of iterations and is recommended for special problems and/or research applications. For nonlinear runs, the maximum number of nonlinear iterations is generally specified as 15 with a tolerance of 10^{-6} . For spectral simulations, obviously the former approach is recommended.

Docks

For incorporating docks, the actual "without-dock" water depth is originally specified while generating the grid; areas covering the docks must then be highlighted, and a draft and coefficient as shown in the dialog below.



The "Floating Dock" dialog box contains the following fields:

- ☒ Floating Dock
- Draft: 0.3 m
- Coefficient: 1.0

The draft depth is a physical quantity and can be measured on an existing structure or designed. The coefficient, which we will call α , is a numerical term that impacts the conveyance of energy under the dock. It is a function of the wave number of the wave passing the dock (k), the characteristic structure size (a), the depth (h) and the draft (d) of the dock. A review of the theory can be found in Tsay and Liu (1983).

To compute an appropriate coefficient, the following process can be followed:

- Compute ka , kh and d/h
- Limit value of kh
 - if $(kh > 4)$ $kh = 4$
 - if $(kh < 0.1)$ $kh = 0.1$
- Compute initial value of α as:
 - if $(kh \leq 2)$ $\alpha = 0.97 - (0.65*kh) + (0.1*kh*kh)$

- if ($kh > 2$) $\alpha = 0.09 - 0.02*kh$
- Adjust α for d/h
 - $\alpha = \alpha*(1.4-0.8*(d/h))$
- Adjust α for ka
 - if ($ka < 1$) $\alpha = 0.8*\alpha$

Visualizing Solutions

Solutions generated by CGWAVE can be visualized in SMS in two ways. The most straightforward method is to read the solution file. When SMS reads this solution file, it translates the data and creates datasets for wave amplitude, direction and phase. If additional functions are desired from the wave characteristics, they must be generated using a CGWAVE translating utility. An example of this is included in the CGWAVE reference manual. The translating utility creates generic dataset files that can be imported using the Data Browser.

Mesh Generation

CGWave computes properties that illustrate the characteristics of waves in a coastal area. To adequately perform this approximation, the mesh must contain several elements per wavelength. Steps for generating a mesh for CGWAVE are spelled out in the description of the Graphical Interface.

Saving CGWAVE

When you do a *File | Save As...* the following files get saved in the *.sms

- *.mat referenced to new save location
- *.map referenced to new save location
- *.cgi referenced to new save location
- *.h5 referenced to new save location
- *.cgo referenced to old save location unless rerun then referenced to new save location

What does the error message “Number of Nodes Exceeded the Dimension” mean?

This message indicates your mesh has more elements than the program is dimensioned for. Running with more nodes/elements than the program is dimensioned for is not recommended. CGWAVE can be compiled for larger dimensions if necessary for a specific case. Contact tech support for more information.

Memory Requirements and Notes

Currently the CGWAVE engine is written in FORTRAN. All memory management in the program is handled with static arrays. An effort is currently underway to rewrite the code to use dynamic arrays. The executable distributed with SMS has been compiled to handle meshes with up to approximately 480,000 nodes. The number is not exact because some of the arrays dimensions are set based on the number of nodes on the open boundary and the coastline, which are very domain specific. In order to obtain an executable that will handle a larger number of nodes, you should contact technical support at Aquaveo. Larger executables are often posted on their ftp site.

It should be noted that due to the limitation of static arrays, the size of model that can be evaluated on a Windows PC is limited to a maximum of around 2,500,000 nodes. Users should be warned that running such a large model on a PC may take an extended period of time (possibly days).

With the availability of 64-bit machines, it is possible to access larger amounts of memory. However, this will not be an option for CGWAVE until the code is reworked to support dynamic arrays and tested on 64-bit machines.

Theoretical Basis / Mathematical Details

- Governing Equations
- Boundary Conditions
- Numerical Solution

Additional Documents:

- Simulation of Waves in Harbors Using Two-Dimensional Elliptic Equation Models [4]
- Solution of the Mild-Slope Wave Problem by Iteration [5]
- A Finite Element Model for Wave Refraction and Diffraction. Tsay, T.-K. & P.L.-F. Liu (1983). Appl. Ocean Res., v5, 1, 30-37.
- Simulation of Wave Breaking Effects in Two-Dimensional Elliptic Harbor Wave Models [6]
- Simulation of Waves at Duck (North Carolina) Using Two Numerical Models [7]
- Incorporating Rubble Mound Jetties in Elliptic Harbor Wave Models [8]
- Exterior Reflections in Elliptic Harbor Wave Models [9]
- Exterior Bathymetric Effects in Elliptic Harbor Wave Models [10]
- Improved Coastal Boundary Condition for Surface Water Waves [11]
- Outgoing Boundary Conditions for Finite-Difference Elliptic Water-Wave Models [12]

External Links:

- CHL CGWAVE website [2]
- May 2007 ERDC/CHL CHETN-I-73 May 2007 Infra-Gravity Wave Input Toolbox (IGWT): User's Guide [4]
- Mar 2004 ERDC/CHL CHETN-I-68 How to Use CGWAVE with SMS:An Example for Tedious Creek Small Craft Harbor [13]
- Jun 2003 ERDC/CHL CHETN-I-67 Tedious Creek Small Craft Harbor:CGWAVE Model Comparisons Between Existing and Authorized Breakwater Configurations[14]
- Aug 1998 Technical Report CHL-98-xx CGWAVE: A Coastal Surface Water Wave Model of the Mild Slope Equation [15]

Related Topics

- SMS Models page
-

CGWAVE Overview

CGWAVE

Overview

The model CGWAVE (Demirbilek and Panchang 1998) is a two-dimensional wave transformation model that can be used to predict wave properties (wave heights, velocities, pressures, radiation stresses) in domains of complex shape and depth variations when an input wave condition (amplitude, direction, and period; or a spectral combination of these) is provided.

Functionalities

- CGWAVE Flies
- CGWAVE Graphical Interface
- CGWAVE Model Checker
- CGWAVE Model Control
- Saving CGWAVE files

Project Explorer

Menus

The following menu commands are available in the CGWAVE Menu:

- **Assign BC**
This command is used to assign boundary conditions along a selected nodestring(s).
- **Material Properties**
Material properties can be assigned and defined.
 - *Bottom friction* – The bottom friction can be specified for the element(s) and material selected in this field.
 - *Floating dock* – Represent an object anchored in place, but floating in or on the water and thereby obstructing wave fields. Elements will be treated as floating barriers when the simulation is saved.
- **Model Check**
Check for common problems. The *model checker* performs the generic mesh checking along with optionally checking to insure:
 - that all boundaries mesh boundaries are assigned as land with a reflection coefficient or as open ocean.
 - that all water depths are positive.
- **Model Control**
Brings up the *Model Control* dialog to specify model parameters.
- **Reset 1D Spacing**
...
- **Run CGWAVE**
Brings up a dialog that allows the user to check what executable of CGWAVE should be run and then runs the model with the currently loaded simulation. As the model runs, a dialog monitors progress of the model and gives the user status messages. When the run is complete, the spatial solutions are read in for analysis and visualization.

Obsolete Commands

The following commands are no longer in current versions of SMS.

- **Spectral Energy**

Brings up the *Spectral Energy* dialog to define/view wave energy spectra. This command also allows the user to visualize wave spectra.

How do I?

To learn more about how to use the CGWAVE Model go to the Tutorials section.

Numeric Engine Background

- CGWAVE Math Details: Governing Equations
- CGWAVE Math Details: Boundary Conditions
- CGWAVE Math Details: Numerical Solution

CGWAVE Math Details: Boundary Conditions

Domains on which the elliptic eq. (5) is solved are enclosed by closed boundaries (represented by coastlines and surface-penetrating structures like pier walls or pier legs, breakwaters, seawalls, etc.) and open boundaries (which represent an artificial boundary between the area being modelled and the sea region outside. A separation between the model domain and an outer water area from where no waves enter the model domain (e.g. a creek or tributary at the backbay or down wave end of the domain) may be considered to be a fully-absorbing closed boundary. An open boundary is considered to be one where an incident wave is specified (and may contain other radiated waves). Along these boundaries, appropriate conditions must be specified to solve (1); however, even in the best of circumstances, only approximate boundary conditions can be developed (e.g. see Dingemaans, 1997).

Closed Boundary Conditions

Along coastline and surface-protruding structures, the following boundary condition has traditionally been used (e.g. Berkhoff 1976; Tsay & Liu 1983; Tsay et al. 1989; Oliveira and Anastasiou 1998; Li 1994a)

$$\frac{\partial \phi}{\partial n} = \alpha \phi \quad (6)$$

where n is the outward normal to the boundary and α is related to a user-specified reflection coefficient as follows:

$\alpha = ik \frac{1 - K_r}{1 + K_r}$ ⁽⁷⁾ K_r varies between 0 and 1 and specific values for different types of reflecting surfaces have been compiled by Thompson et al. (1996).

It may be verified that (6) is strictly valid only for fully-reflecting boundaries (i.e. $K_r = 1$). For partially reflecting boundaries, it is valid only if waves approach the boundary normally. For other conditions, (6) is approximate, and may produce distortions in the model solutions. These limitations may be eliminated by describing the solution at the boundary more fully as a sum of incident and reflected waves:

$$\phi = A \{ \exp[ik(n \cos \theta + s \sin \theta)] + K_r \exp[ik(-n \cos \theta + s \sin \theta + \beta)] \} \quad (8)$$

where A is the amplitude of the approaching waves, θ is the direction at which they intersect the boundary ($\theta = 0$ for normally incident waves), s is the coordinate along the tangent to the boundary, and β is a phase shift between the incident and the reflected wave. (8) leads to the following boundary condition.)

$$\frac{\partial \phi}{\partial n} = ik \cos \theta \frac{1 - K_r \exp(ik\beta)}{1 + K_r \exp(ik\beta)} \phi \quad (9)$$

Unfortunately, θ and β are not known a priori inside the model domain, and must be estimated by approximation. For fully absorbing boundaries ($K_r = 0$), Li and Anastasiou (1992) and Li et al. (1993) have used (9) after estimating θ from Snell's Law and the deep-water incident wave angle. Alternatively, Isaacson and Qu (1990) estimated θ as follows:

$$\theta = \arctan\{(\partial\chi/\partial s)/(\partial\chi/\partial n)\} \quad (10)$$

where χ is the argument of the complex quantity θ (i.e. the phase of θ). For implementation, they first used (6) as a boundary condition, obtained χ from the results, determined θ from (10), used (9) as a boundary condition to perform a second iteration of the model, recalculated χ and θ , performed a third model iteration using (9), and so on. Like Pos (1985), they assumed $\beta = 0$ while using (9), based on limited numerical tests that showed little sensitivity to β . Clearly, like the Snell's Law approach, (10) is valid only for $K_r = 0$ (although problems with non-zero K_r were also considered). To include the effect of the reflected waves (i.e. the second term in the right hand side of (8)), Isaacson et al. (1993) suggested estimating θ as follows:

$$\theta = (1/k) \arcsin\{\partial\chi/\partial s\} \quad (11)$$

Again an iterative method with repeated model calculations were needed. Steward and Panchang (2000) analyzed these methods and noted difficulties with convergence of the above iterative methods and with the quality of the solutions obtained with (10) and (11). They were able to eliminate these difficulties by estimating θ from the following expression:

$$\tan \theta = \frac{\partial\chi}{\partial s} \bigg/ \left(\frac{\partial\chi}{\partial n} + \frac{2K_r k (\cos(k\beta) + K_r)}{1 + 2K_r \cos(k\beta) + K_r^2} \cos(\theta) \right) \quad (12)$$

(12) is a generalization of (10) that allows non-zero K_r and β . For a detailed comparison of results, see Steward and Panchang (2000). Fig. 2 shows a simulation with waves propagating into a rectangular harbor area. Clearly, solutions obtained with (12) are qualitatively superior to those obtained with (10).

Despite the increasing sophistication seen progressively in (6) and (9) and in the various ways of estimating θ , some fundamental problems remain. The most important one is inherent in (8), i.e. the assumption that the total wave field near the boundary can be represented either by one set of plane waves (in the case of (10) or the Snell's Law approach of Li et al. (1993)) or by two sets of plane waves (in the case of (11) and (12)) propagating in constant depth. In domains of complex shapes (as in Fig. 1) with arbitrary bathymetry and boundaries with varying reflectivities, a complex pattern of waves can result; simple wave trains are not easily discernible and, as noted by Isaacson and Qu (1990), the definition of a single θ (and β) can become meaningless. Further, even when there is a well-defined train of waves near the boundary (justifying the use of the above methods), precise estimation of K_r and β is still problematic. Values of K_r provided by Thompson et al. (1996) certainly do not cover full range of reflecting surfaces that the modeller encounters, nor do they cover the dependence of these parameters on the incident wave frequency. Efforts to incorporate the work of Dickson et al. (1995) and Sutherland and O'Donoghue (1998) pertaining to β in models such as the one described here are lacking.

In some ways it may be best to recognize these difficulties at the outset and use the simplest expressions (6) and (7) by combining all the uncertainties noted above into a single parameter α , which may be regarded as a tuning parameter. This is the approach followed in CGWAVE.

Open Boundary Conditions

Along the open boundary, an incident wave ϕ_i must be specified. Along this boundary, however, waves backscattered from within the domain will also exist, and their magnitude is generally not known. In the context of simple rectangular domain models, with one side (aligned, say, in the y direction) constituting the open boundary, Panchang et al. (1988, 1991), Li (1994a, b), and Oliveira and Anastasiou (1998) have used the following condition

$$\frac{\partial \phi}{\partial x} = ik(2\phi_i - \phi) \quad (13)$$

(13) is obtained by assuming that the incident and backscattered components along this boundary can be described by $\phi_i = A_i \exp(ikx)$ and $\phi = B \exp(-ikx)$ respectively (where A_i is the (specified) amplitude of the incoming wave and B is an unknown), adding the two components, and differentiating. Obviously, this is valid only if the incident and backscattered waves near the boundary are plane waves propagating in the $\pm x$ direction.

For more complex domains involving multidirectional scattering, (13) is inappropriate. Harbor applications generally use model domains such as that described in Fig. 3, where the semicircle is used to separate the model area from the open sea. In the exterior domain Ω' the potential ϕ is comprised of three components:

$$\phi = \phi_i + \phi_r + \phi_s \quad (14)$$

where ϕ_i = the incident wave that must be specified to force the model, ϕ_r = a reflected wave that would exist in the absence of the harbor, and ϕ_s = a scattered wave that emanates as a consequence of the harbor and must satisfy the Sommerfeld radiation condition. With appropriate descriptions for these components, a boundary condition can be developed along the semicircle.

In traditional harbor models (Mei 1983; Tsay and Liu, 1983; Thompson et al. 1996; Chen and Houston, 1987; Xu and Panchang, 1993; Demirbilek and Panchang, 1998), the exterior wave conditions are described as follows:

$$\phi_i = A_i [ikr \cos(\theta - \theta_i)] \quad (15), \text{ which is the specified input}$$

$$\phi_r = A_i [ikr \cos(\theta - \theta_i)] \quad (16)$$

$$\phi_s = \sum_{n=0}^{\infty} H_n(kr) (A_n \cos n\theta + B_n \sin n\theta) \quad (17)$$

where (r, θ) denotes the location of a point in polar coordinates, H_n is the Hankel function of the first kind and order n , and A_n and B_n are unknown coefficients.

For the specified incident wave field given by (15), equations (16) and (17) result from the solution of the relevant eigenvalue problem in the traditional method. As demonstrated by Xu et al. (1996), however, this eigenvalue problem, in which ϕ_s and ϕ_r are coupled, may be solved only under the following conditions:

- (i) The exterior region must have a constant depth, (ii) the exterior coastlines A1D1 and A2D2 must be fully reflecting and collinear.

These requirements usually cannot be met in practice where the exterior geometry varies arbitrarily, and the unrealistic bathymetric representation used perforce by the modeller invariably has an adverse influence on the solution. In field applications, the exterior bathymetry is irregular and the depth generally increases in the x -direction. Condition (i) is thus violated, causing two problems as demonstrated by Panchang et al. (2000). First, the modeller must arbitrarily select a representative “constant” depth and test the sensitivity of the solutions to these depths. This can be extremely time-consuming. Second, the effect of reflections from the sloping exterior bathymetry is ignored. These effects are often significant, especially for long periods that are of interest in harbor resonance studies. Condition (ii) is also problematic. Exterior coastlines are not always fully reflecting for all wave conditions, and imposing full reflection in such cases yields extremely large amplification factors and rapid variations in the wave pattern in the outer regions of the domain. (See examples in Xu et al. (1996), Demirbilek et al. (1996), and in Thompson et al. 1996)). One may of course enlarge the interior region in the hope that these effects do not contaminate the results in the area of interest; however, there is no guarantee that these effects are confined to

specific regions. In addition, the extra memory requirements and grid-generation for a larger domain are usually exceedingly demanding. Thus, while (16)-(17) constitute rigorous solutions of the eigenvalue problem, their use renders the application of harbor wave models problematic in practice. (One consequence of the above is that many of the models in this category cannot correctly simulate fairly simple phenomena like waves approaching a sloping beach. Investing confidence in model results when applied to field situations is therefore difficult.)

An effective alternative, followed in CGWAVE, is to use a “parabolic approximation” to describe ϕ_s :

$$\frac{\partial \theta_s}{\partial r} = -p\phi_s - q\frac{\partial^2 \phi_s}{\partial \theta^2} \quad (18)$$

where

$$p = ik_0 + \frac{1}{2r} - \frac{i}{8k_0 r^2}, \frac{d}{dx}(CC_g \frac{d\psi}{dx}) + kCC_g(k \cos^2 \theta + iW)\psi = 0 \quad (18a)$$

where r and θ represent the polar coordinates of a point on the open boundary. (p and q are not unique, and alternative forms, each obtained with an appropriate rationale, have been investigated by Givoli (1991), Xu et al. (1996), Panchang et al. (2000)). The parabolic approximation (18) allows the scattered waves to exit only through a limited aperture around the radial direction. Unlike (17), therefore, it does not rigorously satisfy the Sommerfeld radiation condition. However, using this formulation decouples ϕ_s from the other components. These components (ϕ_i and ϕ_r) may be obtained by making a compromise between a detailed exterior bathymetric representation (which as noted earlier, is difficult) and the constant depth representation (which is unrealistic). A one-dimensional representation, where the depths vary in the cross-shore direction only (Fig. 1), may be selected. This is reasonable, since in general, this is often the direction in which the depths vary the most. If natural variations do not permit the representation of the exterior depths by only one section, a second one-dimensional section, shown as transect 2 in Fig. 1, may be constructed. For transects 1 and 2 with varying depths, no simple analytical expression (such as (16)) can be found for the reflected wave (since ϕ_i and ϕ_r are coupled). However, the quantity:

$$\phi_0 = \phi_i + \phi_r \quad (19)$$

may be obtained by the solution of the one-dimensional version of (5), since the depths along these transects vary in one direction only. This one-dimensional equation is (Schaffer and Jonsson, 1992; Panchang et al. 2000):

$$\frac{d}{dx}(CC_g \frac{d\psi}{dx}) + kCC_g(k \cos^2 \theta + iW)\psi = 0 \quad (20)$$

where, for one-dimensional geometry,

$$\phi_0 = \psi(x) \exp(iky \sin \theta) \quad (21)$$

(20) is an elliptic ordinary differential equation requiring two boundary conditions. It may easily be solved via a simple finite-difference scheme. (For the present, the dissipation factor W is considered to be prespecified). Assuming that transect 1 extends out to a region of constant depth (or deep water), a condition at P_1 may be obtained by combining a specified incident wave

$$\phi_i(P_1) = A_i \exp(ikx \cos \theta_i + iky \sin \theta_i) \quad (22)$$

(where A_i is a given input wave amplitude) and an unknown reflected wave:

$$\phi_r(P_1) = B \exp(-ikx \cos \theta_i + iky \sin \theta_i) \quad (23)$$

Without loss of generality, the point P_1 may be located at $x = 0$, which allows elimination of B to yield

$$\frac{\partial \psi}{\partial x} = ik \cos \theta_i (2A_i - \psi) \quad (24)$$

At the coastal boundary point Q_1 , the partial reflection boundary condition (9) may be used in the following form:

$$\frac{\partial \psi}{\partial x} = \frac{i\sqrt{k^2 - k^2 \sin^2 \theta}(1 - K_r)}{1 + K_r} \psi \quad (25)$$

where K_r is the reflection coefficient for the exterior coastline (i.e. near Q_1) and $k \sin \theta$ is constant for the one-dimensional problem.

The solution of (20) using boundary conditions (24) and (25) along with (21) produces ϕ_0 along transects 1 and 2. These solutions are denoted by ϕ_{01} and ϕ_{02} . The desired ϕ_0 along the semicircle may be obtained by laterally translating ϕ_{01} and ϕ_{02} via interpolation between transects 1 and 2 as follows:

$$\phi_0 = (1 - m)\phi_{01} \exp(-ik(r - y) \sin \theta) + m\phi_{02} \exp(ik(r + y) \sin \theta) \quad (26)$$

where we have set $y = 0$ at the center of semicircle, the interpolation function $m = (r - y)/2r$, r is the radius of the semicircle, y is the lateral coordinate of the open boundary node relative to the origin of semicircle (Fig.3).

The boundary condition for φ along the semicircle Γ may be obtained by using the continuity of the potential (equations 14 and 19) and its derivative along with (18) and (26):

$$\frac{\partial \psi}{\partial x} = \frac{i\sqrt{k^2 - k^2 \sin^2 \theta}(1 - K_r)}{1 + K_r} \psi \quad (27)$$

Thus, the solution of (20) provides ϕ_0 along the 1-dimensional transects. These values can be translated laterally and substituted into (27) to obtain the open boundary condition for the two-dimensional equation (1). Zhao et al. (2000) and Panchang et al. (2000) have demonstrated that this procedure provides extremely satisfactory solutions for a large number of test cases.

In CGWAVE, if non-linear breaking is implemented, a version of (21) is solved by iteration since W depends on the (unknown) amplitude.

Related Topics

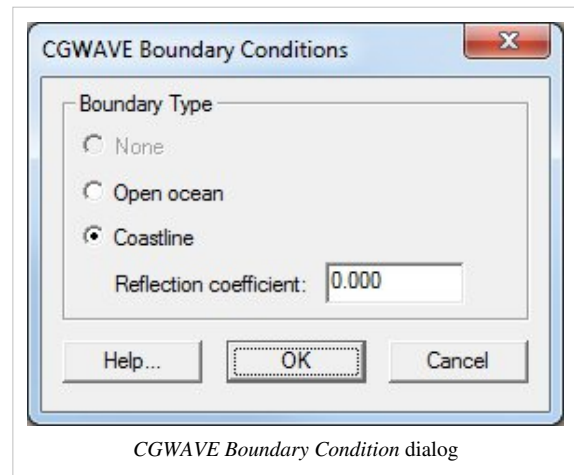
- CGWAVE
- Governing Equations
- Numerical Solution

CGWAVE Boundary Conditions Dialog

In the CGWAVE model, a wave direction, amplitude and frequency must be specified at open boundaries and reflection coefficients are defined for all closed boundaries. In addition to these exterior boundaries, the model also includes the capability to simulate interior islands, and floating barriers.

Assign Boundary Condition

The *CGWAVE Boundary Conditions* dialog is used to assign boundary conditions to individual nodestrings. This dialog is invoked with the **Assign BC** command in the *CGWAVE* menu. Before assigning boundary conditions to nodestrings, at least one nodestring must be selected using the  **Select Nodestring** tool. To assign boundary conditions to the selected nodestring(s), select one of the boundary condition options.



Boundary Types

- **None**
- **Open Ocean** – Delineate the region where waves will enter the domain. The attributes of the waves that enter the domain are defined as incident wave characteristics in the *CGWAVE Model Control* dialog. These values are propagated from an offshore location to the open ocean boundaries.
- **Coastline** – Represent a region where the wave is obstructed. At these locations, the Reflection coefficient should be set. The coastline reflection term defines to what degree a section of coastline reflects incoming waves. Legal values vary from 0.0 for a gradual sandy incline to 1.0 for solid vertical rock wall. This boundary condition may be assigned to either exterior regions of the domain, or interior holes which represent islands.
 - **Reflection coefficient**
 - **Floating Barrier**

Related Topics

- CGWAVE
- CGWAVE Graphical Interface
- CGWAVE Boundary Conditions
- Feature Objects Menu

CGWAVE Files

Input Files

The *.cgi File

CGWAVE always requires an input file to run. This input file is created by SMS with an extension of *.cgi (CGwave Input). It is an ASCII file that defines the model parameters, the boundary definition, the nodal locations and the connectivity. Lines that begin with a "%" indicate the line is a comment. These are terminated with a "&" character. There is a comment for each value on the data line, so five comment lines indicate the following data line will have five values. This format is generally followed throughout. Some utilities available for processing CGWAVE data files require that all the "standard" comments exist. The model parameter data is sequentially read, and follows the following format:

%number of characters in title &

```
%number of terms in the fourier-bessel series solution (default = 35) &
%output echo frequency to screen &
%maximum number of iterations for convergence &
%max no of iters. for nonlinear mecha (fri,breaking,dispersion) &
      30           35           100           500000           10
```

Title of this Run, jul 10 2007

```
%type of application &
%type of open boundary condition &
%bottom friction &
%wave breaking &
%nonlinear dispersion relation &
%choice of solver &
```

```

0          3          1          1          0          2
%number of components &
1
%Incident Wave Angle || Wave Period(s) || Incident Wave Amplitude &
225.0000000000000000E+00  30.000000000000000  1.000000000000000
%exterior reflection &
%bottom friction coefficient &
%wave breaking parameter &
%tolerance for the equations &
%tolerance for nonlinear mechanisms &
0.00E+00  0.12E+00  0.15E+00  0.10E-09  0.10E-06

```

After the model parameter data comes the boundary definition. This includes two blocks of data marked with "&C" to indicate a boundary open to have waves enter, and "&B" for sections of coastline and islands (land boundaries). There can only be one "Open" boundary (&C). Its type will be 0 or 1 indicating semi-circle or full circle. The full circle is applied when modeling waves around an island. Multiple closed boundaries (&B) may be included. One for each segment of the coastline (changing reflection coefficients), and at least one for each island in the domain.

%Open boundary flag

```

%type of open boundary &
%number of nodes on it &
%center of the circle - x &
%center of the circle - y &
%initial angle &
%semicircle orientation angle (ccw from +x axis, E=0, N=90, W=180, S=270) &
%node id, ... ...

```

&C

```

0  220  0.000000000E+00  0.000000000E+00  0.000000000E+00  0.230036700E+02
      220      219      218      217      216      215
      214      213      212      211      210      209
      208      207      206      205      204      203
      .
      .
      .
      .
      .

```

```

%Coastline boundary flag
%number of nodes on it &
%reflection coefficient &
%node id, ... ...

```

&B

```

360  5.00000000000000003E-02
      1      221      440      659      880      881
1103      1329      1559      1793      1794      2036
2285      2538      2798      2799      2800      3070

```

•
•
•
•
•

Following the boundary definition, the file includes the locations of the computational nodes with a "&N" identifier:

```
%Node coordinates and Depth &
%number of nodes &
```

&N

```
236211
-0.452985572E+04 0.259162993E+04 0.215026303E+00 -0.452888341E+04 0.259197240E+04 0.242541197E+00
-0.452784808E+04 0.259233707E+04 0.271839587E+00 -0.452671893E+04 0.259273479E+04 0.303792731E+00
-0.452546078E+04 0.259317794E+04 0.334700922E+00 -0.452409490E+04 0.259365905E+04 0.361240031E+00
•
•
•
•
•
```

Following the nodal locations comes the material definitions with a "&M" identifier. Material types define two important options for CGWAVE. First, roughness is specified as a material. Second, floating breakwaters are defined in the material block. Each type of breakwater, and each roughness zone will require it's own material type. The materials are then assigned to each element in the element block below. In the example below, two materials are defined. The first has a roughness of 0.5 and is type 0, indicating it is not a floating breakwater. The second has the same roughness and is of type 1, indicating it is a floating breakwater. For the floating breakwater, two additional parameters are included, a draft and a coefficient of transmission.

```
%Materials
%number of materials &
%material id &
%bottom friction &
%floating dock &
%draft &
%coefficient.
```

&M 2 1 0.5000 0 0.0000 0.0000 2 0.5000 1 0.0000 1.0000

Following the material definitions comes the element connectivity with a "&E" identifier. Each line includes the three node identifiers that make up an element and the material type that fills that element.

```
%Elements
%number of elements &
```

&E

```
456220
1 221 2 1
221 222 2 1
222 3 2 1
```

```

.
.
.
.
.

```

The *.cg1 File

When running CGWAVE from for varied sloped, semi-circular boundaries, it is recommended that the two "Semi 1D Lines" option be used. This option transforms the input wave conditions from the offshore depth to a variety of depths as those conditions propagate shoreward. The transformed conditions are mapped to the variable depth boundary of the semi-circle (open boundary). SMS extracts depths from the surrounding survey if it exists. Otherwise the user must create a profile from the deep water location where the wave conditions are sampled, through to the coastline. The file format includes two such profiles. One marches down the left side of the semi-circle, and the other down the right. The format of each profile includes the distance between sample points and the number of sample points. Two other values are included as space holders for now. SMS always writes out "0.0 1" for these values. The file then includes one line for each sample point including a depth and an optional roughness value at that depth as follows:

```

0.5000 4912 0.0 1
73.394101 0.120000
73.223910 0.120000
73.053719 0.120000
.
.
.
.
9.177065 0.120000
7.433389 0.120000
5.914170 0.120000

```

Output Files

CGWAVE creates a single ASCII solution file. It includes a block of data for each input wave condition. The file includes a complex number that defines the wave amplitude and phase at each node for each wave condition. Upon trying to read this file into SMS, the CGWAVE – Trans post-processor will be executed in order to turn the *.cgo solution file into an XMDF file that can be easily read by SMS.

CGWAVE Math Details: Governing Equations

Governing Equations

In its basic form, the methodology is based on solving the following two-dimensional elliptic equation:

$$\nabla \bullet (CC_g \nabla \phi) + k^2 CC_g \phi = 0 \quad (1)$$

where

$\phi(x, y)$ = complex surface elevation function ($= \phi_1 + i\phi_2$)

$i = \sqrt{-1}$

σ = wave frequency under consideration

$C(x, y)$ = phase velocity $= \sigma / k$

$C_g(x, y)$ = group velocity $= \partial \sigma / \partial k$

$k(x, y)$ = wavenumber ($= 2\pi / L$), related to the local depth $d(x, y)$ through the dispersion relation:

$$\sigma^2 = gk \tanh(kd) \quad (2)$$

The wave height H can be obtained from complex surface elevation function ϕ as follows:

$$H = \partial \phi / \partial k \quad (3)$$

Essentially (1) represents an integration over the water column of the three-dimensional Laplace equation used in potential wave theory. The integration, originally described by Berkhoff (1976) and Smith and Sprinks (1975), is necessary because the solution of the three-dimensional problem is computationally difficult for harbors with a characteristic length that is several times the wavelength. The integration is based on the assumption that the vertical variation of the wave potential is largely the same as that for a horizontal bottom, i.e.

$$\phi(x, y, z) \approx \frac{\cosh k(d+z)}{\cosh kd} \phi(x, y) \quad (4)$$

This approximation is obviously valid for a "mild slope", characterized by $|\nabla d|/kd \ll 1$, a criterion that is usually met in practice. (Extensions to steep slopes are described later). Unlike "approximate" mild slope wave models (e.g. REFDIF and RCPWAVE described by Dalrymple et al. 1984; Kirby, 1986; and Ebersole, 1985), there are no intrinsic limitations on the shape of the domain, the angle of wave incidence, or the degree and direction of wave reflection and scattering that can be modeled with (1). While (1) is valid for a monochromatic (single incident frequency-direction) wave condition, irregular wave conditions may be simulated using (1) by superposition of monochromatic simulations.

As noted earlier, (1) incorporates the effects of refraction, diffraction, and reflection induced by any nonhomogeneity in the model domain. We now provide extensions of (1) that include, in addition, dissipative effects (friction and wave breaking), steep-slope effects, and floating docks.

Dissipation

To include dissipative effects, we consider the following extended form of (1):

$$\nabla \bullet (CC_g \nabla \phi) + (k^2 CC_g + iC_g \sigma W) \phi = 0 \quad (5)$$

in which a dissipation term (with W) has been included. By separating the real and imaginary parts of (5), Booij (1981) has shown that (5) satisfies the energy balance equation in the presence of dissipation. The term W may represent breaking and/or friction and is described later.

In (5), W represents the combined effects of friction and breaking, which may be separated as follows:

$$W = w/C_g + y \quad (6)$$

where w is the friction coefficient defined by Dalrymple et al. (1984) and g is a breaking factor. These coefficients are empirical, and parameterizations for these have been described by Dalrymple et al. (1984), Tsay et al. (1989), and Chen (1986) for friction and by Battjes and Janssen (1978), Dally et al. (1985), Massel (1992), Chawla et al. (1998), and Isobe (1999) for breaking. Some of these parameterizations have been extensively validated against field data (e.g. Larson 1995; Kamphuis 1994). The parameterization used in CGWAVE is based on the formulation by Dalrymple et al. (1984).

Published studies demonstrating the effects of friction in harbor models (e.g. Chen 1986; Tsay et al. 1989; Demirebilek and Panchang 1998; Kostense et al. 1986) have estimated w on the basis of the incident wave amplitude. It is then easy to pre-specify w while solving (5). These studies appear to show that friction can change the magnitude of resonant peaks in harbor models quite substantially; at other frequencies, the effect seems to be minimal.

As to breaking, Zhao et al. (2000) applied a finite element wave model to several tests involving breaking. These tests involved a sloping beach, a bar-trough bottom configuration, shore-connected and shore-parallel breakwaters on a sloping beach, and two field cases in the North Sea and Ponce de Leon Inlet (Florida). Five breaking formulations, given by Battjes and Janssen (1978), Dally et al. (1985), Massel (1992), Chawla et al. (1998), and Isobe (1999) were examined. In general, they found that the formulations of Battjes and Janssen (1978) and Dally et al. (1985) were the most robust from point of view of incorporation into an elliptic model based on (5) and provided excellent results compared to data. CGWAVE includes the Battjes and Janssen (1978) formulation.

Unlike friction the breaking coefficient is a function of the wave amplitude which is unknown *a priori* inside the domain, and its inclusion makes the problem nonlinear and requires iteration. For the first iteration, W is set equal to 0 and (1) is solved (e.g. non-breaking solutions are obtained). The resulting wave heights are used to estimate W via the Battjes and Janssen (1978) parameterizations and (5) is solved again. The process is repeated until convergence is obtained. This can be a very time-consuming process. For many practical applications, it is therefore suggested that simple non-breaking simulations be performed, and an H/D breaking limit be then applied to artificially cut off the excessively large waves. This option is available in CGWAVE.

Steep-slope effects

Unlike the nonlinear mechanisms described above, the “mild slope” requirement discussed in Section 1 is relatively easy to eliminate. Massel (1993), Porter and Staziker (1995), Chamberlain and Porter (1995), and Chandrasekera and Cheung (1997) developed extensions of (1) to include steep-slope effects. Their extensions may be described by the following equation:

$$\nabla \bullet (CC_g \nabla \phi) + (k^2 CC_g + d_1 (\nabla h)^2 + d_2 \nabla^2 h) \phi = 0 \quad (35)$$

where d_1 and d_2 are functions of local depths. Reference may be made to these publications for the various definitions of d_1 and d_2 ; in general, though, differences in the proposed definitions of these functions impact model results to a very small extent. The steep-slope terms are fairly straightforward to include in the model because they are linear. Further, they have the advantage of being “automatic”, i.e. they have little contribution for mild slopes and do not change the solution technique; when they are significant, the additional computational demand is negligible. However, steep slopes lead to breaking and model performance in the vicinity of steep slopes (e.g. see the work of Massel and Gourlay (2000) that include breaking and steep-slope effects near coral reefs). CGWAVE uses the formulation by Chandrasekera and Cheung (1997).

Floating Docks

One problem frequently encountered by engineers when using models based on (1) pertains to the presence of floating structures in the modeling domain (e.g. floating breakwaters or docks in marinas). These structures of course violate the “free-surface” requirement of (1). The problem near the dock is 3-dimensional, whereas the model (eq. 1) is solved in a 2-dimensional framework. CGWAVE uses an approximate method suggested by Tsay & Liu (1983) for tackling floating structures in the context of 2-d harbor wave models. This approach merely calls for a suitable modification to the second term on the left-hand side of (1). (Tsay and Liu (1983) examined suppressing this term). As a consequence, the method is extremely simple to implement with existing finite element models. A model grid is first generated as usual with no regard to the floating structure, grid elements covering the floating structure (in plan view) are selected, they are assigned a depth value equal to the under-keel clearance, and the coefficient of the second term in (1) is set to zero for these elements. Clearly, this is an ad-hoc method intended for convenience in engineering practice, and although Tsay & Liu (1983) provided heuristic arguments in support of this approach, their testing of this procedure was rather limited. Li et al. (2005) found, however, that the method produces results which deviate considerably from the solution of the Laplace equation, and hence developed a simple modification to the original Tsay & Liu (1983) approximation. This involves adjusting the under-keel depth by a factor $\alpha = A \ln(ka) + B$, where a = half width of the structure, and A and B are given in Figure 2 (and Table 1) for different values of relative submergence (defined by draft/water-depth = d/h). The modified approximation yields improved results, when compared to both laboratory data and theoretical results, for a wide range of conditions. By way of practical demonstration, simulations in Douglas Harbor (Alaska) was described by Li et al. (2005) for examining the effects of proposed floating dock configurations. The factor α must be provided to CGWAVE along with the draft depths during grid generation.

Related Topics

- CGWAVE
- Boundary Conditions
- Numerical Solution

CGWAVE Graphical Interface

The CGWAVE Graphical Interface includes tools to assist with creating, editing, and debugging a CGWAVE model. The CGWAVE interface exists in the 2D Mesh Module.

Model Construction Steps

There is a very consistent method that can be used to apply the CGWAVE model. The steps to this process include:

1. Load bathymetry – This data can come from LIDAR surveys, digital elevation maps (DEMs), previous grids or a variety of other sources. They must be referenced to the same datum the wave data will reference, and must have positive values represent depths. SMS includes functionality to convert datums, reverse directions and smoothen filter data.
2. Limit bathymetry to positive values – CGWAVE does not handle wetting/drying or runup processes. All the nodes in the model must have a positive depth. This limiting process can be handled later in the mesh generation process, or the bathymetry itself can be modified (in a copied dataset) using the data calculator. For example, if you want to limit the domain to areas of at least one meter of depth use the following equation in the data calculator $\max(d1, 1.0)$ (where d1 is the label for the depth dataset).
3. Compute the wave length – This is also done in the dataset toolbox, using the *Wave Length and Celerity* tool. Enter the smallest wave period of interest (to generate the highest needed resolution).
4. Create a size function – Typically this is simply a scaled version of the wave length dataset. For example, a basic rule would be to have at least six elements per wave length. Some experts recommend at least 10 elements per wave length. The real issue is that you need enough resolution to represent the wave shape.
 1. To create a "Size" function with "N" elements per wave length, go to the *data calculator* and enter the equation $d3/N$ (where d3 is the label for the wave length dataset).
 2. An alternative that may be needed with large domains is to create a spatially varied scale for the wave length function. For example, you may want to have 15 elements per wave length in the shallow region or your model (i.e. depths less than 3 meters), but only have 7 elements per wave length in the deep regions (i.e. depths greater than 200 meters). To create this size function, you would still use the data calculator and enter the equation $d3/\max(\min(15+(d1-3)/(200-3)*(7-15), 7))$ (where d1 is the label for the depth function and d3 is the label for the wave length dataset).
6. Define the coastline or land edge of the domain – This can be done using a contour of the bathymetry or reading a coastline vector file. It should be stored in SMS as a coastline arc in a CGWAVE coverage in the map module.
7. Define the ocean boundary – This is also an arc in the CGWAVE coverage. It must be either a semi circle or circle and can be defined in SMS by selecting the coastline (or extreme locations on the coastline) and issuing the *Feature Objects | Define domain...* command in the map module.
8. Build polygons in the map module and assign polygon attributes for the polygon to use the depth and size functions for bathymetry and size respectively in the mesh generation process. Then generate the mesh.
9. With this mesh constructed, the rest of the graphical interface, defined below can be used to control the numerical simulation.

Model Control

The *CGWAVE Model Control* dialog is used to setup the options that apply to the simulation as a whole. These options include time controls (steady state/dynamic), run types, output options, global parameters, print options and other global settings.

Boundary Conditions

All numeric models require boundary condition data. In CGWAVE boundary conditions are defined on nodestrings. The default boundary condition is a closed boundary (no flow). See CGWAVE Boundary Conditions Dialog for more information.

Material Properties

Each element is assigned a material type. Material properties describe the hydraulic characteristics of each material type.

- **Bottom friction** – The bottom friction can be specified for the element(s) and material selected in this field.
- **Floating dock** – Represent an object anchored in place, but floating in or on the water and thereby obstructing wave fields. Elements will be treated as floating barriers when the simulation is saved.

Running the Model

The CGWAVE Files are written automatically with the SMS project file or can be saved separately using the *File | Save CGWAVE* or *File | Save As* menu commands.

CGWAVE can be launched from SMS using the *CGWAVE | Run CGWAVE* menu command. A check of some of the common problems called the Model Checker is done each time the model is launched, or by selecting the *CGWAVE | Model Check* menu command.

CGWAVE Menu

The following menu commands are available in the *CGWAVE* Menu:

Command	Functionality
Spectral Energy	Brings up the <i>Spectral Energy</i> dialog to define/view wave energy spectra. This command also allows the user to visualize wave spectra.
Assign BC	This command is used to assign boundary conditions along a selected nodestring(s).
Material Properties	Material properties can be assigned and defined. • <i>Bottom friction</i> – The bottom friction can be specified for the element(s) and material selected in this field. • <i>Floating dock</i> – Represent an object anchored in place, but floating in or on the water and thereby obstructing wave fields. Elements will be treated as floating barriers when the simulation is saved.
Model Check	Check for common problems. The <i>model checker</i> performs the generic mesh checking along with optionally checking to insure: • that all boundaries mesh boundaries are assigned as land with a reflection coefficient or as open ocean. • that all water depths are positive.
Model Control	Brings up the <i>Model Control</i> dialog to specify model parameters.
Reset 1D Spacing	...
Run CGWAVE	Brings up a dialog that allows the user to check what executable of CGWAVE should be run and then runs the model with the currently loaded simulation. As the model runs, a dialog monitors progress of the model and gives the user status messages. When the run is complete, the spatial solutions are read in for analysis and visualization.

Processing Solutions

CGWave creates a single output file (normally including *.cgo extension when run with SMS). This file can be brought into SMS to graphically view the results. As SMS reads the file, it translates (using an embedded version of the CGWAVE "trans" code) the complex numbers representing wave heights and phases for each wave component into spatial and temporal datasets including:

- Steady State
 - Wave Height
 - Wave Phase
 - Direction of Maximum Particle Velocity
- Time Varying (through a single wave period. SMS breaks the period into 20 time steps)
 - Pressures (at surface, mid depth and bed)
 - Particle Velocities (at surface, mid depth and bed)
 - Sea Surface Elevation
 - Wave Velocity.

When the solution is read in, SMS allows the user to limit the wave heights in the solution. This only applies to linear runs of CGWAVE to allow the heights to be adjusted to be more realistic. This is accomplished by applying a factor, whose value ranges from 0 to 1 (defined as H/d). (Because wave height cannot exceed the value of depth or factor=1). SMS recommends a range of 0.4 to 0.8 and defaults to 0.64. The user may enter another factor dependent on that users knowledge of wave mechanics, type of problem, etc.

Related Topics

- CGWAVE
- SMS Models

CGWAVE Incident Wave Conditions

The *Incident Wave Conditions* section of the *CGWAVE Model Control* dialog allows the user to specify wave conditions to be simulated in a run. In simplistic terms, these are the boundary conditions for the simulation.

Each condition (term) is defined as a wave direction, period and amplitude. The wave direction is specified in degrees measured from the positive x-direction in a counter clockwise direction. This means that waves propagating from West to East have a direction of 0°, and waves propagating from South to North have a direction of 90°. The amplitude is specified in meters, and the period in seconds. These values are used as boundary conditions at a location *out to sea* and are propagated to the open ocean boundaries on the domain using one-dimensional wave propagation. A set of values (one direction, frequency and period) can be selected by clicking in the window. The selected set can be changed via the edit boxes above the window. The buttons below the window can be used as follows:

- **New**

The **New** button creates a new condition or wave term below the selected term. The values are defaulted to zero.

- **Copy**

The **Copy** button copies the selected condition or wave term as a new set in the list. The new set is placed directly below the selected set. (Note: it does not make sense to include the same term twice since the two terms are equal to a wave of double the amplitude in the specified direction at the specified frequency. This option is provided for convenience in generating terms with common parameters.)

- **Delete**

The **Delete** button deletes the selected condition or wave term. The rest of the terms are unaffected.

- **Import**

The **Import** button allows the user to select a file containing Incident Wave Conditions data. SMS prompts the user to replace or append to existing data. See the CGWAVE documentation for the Incident Wave Conditions file format.

- **Clear**

The **Clear** button deletes all of the Incident Wave Conditions data.

Wave Condition Generation

The SMS interface for CGWAVE allows generation of multiple wave conditions from a directional spectrum or to represent long wave or infra-gravity wave conditions. These are accessed using the buttons below the incident wave conditions.

- **Generate From Spectrum** – Wave conditions that make up a spectral wave condition can be added by selecting the spectrum to be simulated. A spectrum can be generated or read in using the **Spectral Energy** command CGWAVE menu.
 - **Long Wave Input Toolbox** – The long wave toolbox generates infra-gravity wave terms using a 1 dimensional Boussinesq model run and user specified parameters.
-

CGWAVE Model Control

The CGWAVE model requires several user-specified parameters to control the analysis. The **Model Control** command from the *CGWAVE* menu opens the *Model Control* dialog. This dialog contains parameters that control the execution of CGWAVE. The parameter description for each field is displayed in SMS using the interactive help messages.

Controls include:

- Title
- Incident Wave Conditions

The *Incident Wave*

Conditions section allows the user to specify wave conditions to be simulated in a run. See CGWAVE Incident Wave Conditions for more information.

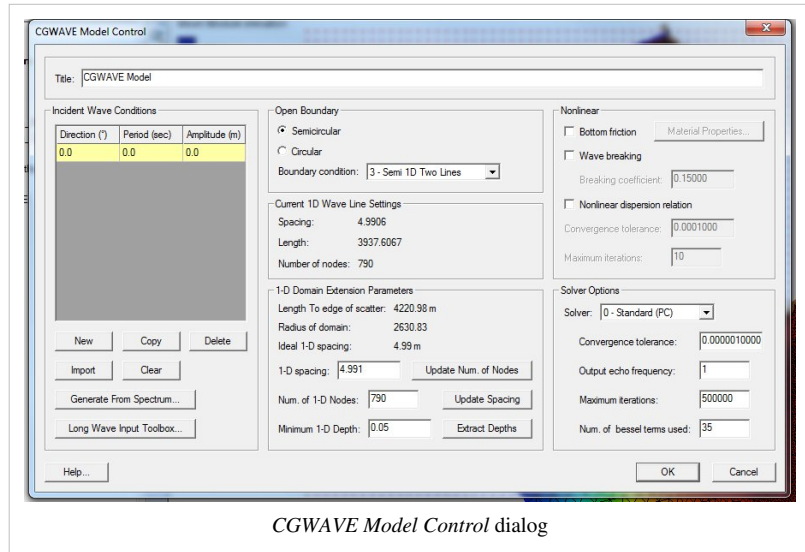
- Open Boundary

The *Open Boundary* section allows the user to specify the open boundary type. This should match the type selected when generating the domain in the Map Module. If the mesh has been generated using other methods, the domain shape must match the *Open Boundary* type.

- Semicircular
- Circular
- Boundary condition:
- 1-D Options

The 1-D section allows the user to specify the number of nodes and the nodal spacing for the one-dimensional data that is used to distribute incident wave data to the open boundary. The file format is described in the CGWAVE user manual[2]. The **Compute 1-d Length** button calculates the 1-d Spacing variable such that the specified number of nodes will extend to the limits of the bathymetry data. This bathymetry data is defined by a scattered dataset. Therefore, a scatter dataset must exist in order for this button to be active. If no scatter data exists, the user must generate an appropriate file containing the one-dimensional depth values.

- Current 1D Wave Line Settings
 - Spacing:
 - Length:
 - Number of nodes:
- 1D Domain Extension Parameters
 - Length to edge of scatter:
 - Radius of domain:
 - Ideal 1D spacing:
 - 1D spacing
 - Num. of Nodes
 - Minimum 1D Depth



CGWAVE Model Control dialog

- Update Num. of Nodes
- Update Spacing
- Extract Depths
- Solver Options

CGWAVE support two separate numerical solvers for robustness. Users select which solver to use via the *Solver* radio group. The *Standard solver* should be utilized first. If CGWAVE fails to converge, the *Modified* method can be utilized.
- Solver:
 - Convergence tolerance:
 - Output echo frequency:
 - Maximum iterations:
 - Num. of bessel terms used:
- Iteration Control

CGWAVE allows the user to specify the maximum number of iteration to be performed by the numerical solver. This number is specified in the *Maximum Iterations* edit field. The solver will terminate after performing the maximum number of iterations or when the change in the solution is less than the convergence tolerance specified in the *Convergence Tolerance* edit field. It is recommended that the convergence value be between 1.0e-6 and 1.0e-9. CGWAVE will print a progress report on the tolerance at the user specified report interval entered in the *Convergence Interval* edit field.
- General Parameters

CGWAVE includes options to model bottom friction, wave breaking and nonlinear dispersion. These options may be enabled using the appropriate toggle box. See the CGWAVE user manual for more information about these parameters.
- Bottom friction
- Wave breaker
 - Breaking coefficient
 - Nonlinear dispersion relation
 - Convergence tolerance
 - Maximum iterations

Related Topics

- CGWAVE Graphical Interface

CGWAVE Math Details: Numerical Solution

Equation (5) is generally solved using the boundary element method, the finite-difference method, or the finite element method. In general, finite-difference discretizations are not well-suited to represent the complex domain shapes described, for example, in Fig. 1. Not only are the boundaries distorted, but the number of uniformly spaced grids may also be excessively large. (Adequate resolution, typically 10 points per wavelength, demands that the spacing be determined from the smallest wavelength.) Most studies with the finite-difference method have been limited to largely rectangular domains (e.g. Li 1994a, 1994b; Panchang et al. 1991; Li and Anatasious 1992). Boundary element models can handle arbitrary shapes and require minimal storage since only the boundaries are discretized; however, they are limited to subdomains with constant depths only (e.g. Isaacson and Qu 1990; Lee and Raichlen 1972; Lennon et al. 1982). Finite element models, on the other hand, allow the construction of grids with variable sizes (based on the local wavelength) and give a good reproduction of the boundary shapes. Most finite element models (e.g. Tsay and Liu 1983; Tsay et al. 1989; Kostense et al. 1988; Demirbilek and Panchang 1998; Panchang et al. 2000) have used triangular elements, and modern graphical grid generating software permits efficient and accurate representation of harbors with complex shapes. The Surface Water Modeling System can be used to conveniently generate as many as 1,000,000 elements of varying size, based on the desired (user-specified) resolution, and to specify the desired reflection coefficients on various segments of the closed boundary. The solution of (1) by the finite element method is described in detail by Mei (1983) and by Demirbilek and Panchang (1998) when different types of open boundary conditions are used.

Whether one uses finite differences or finite elements for discretization, the numerical treatment of (1) with appropriately chosen boundary conditions leads to system of linear equations:

$$[A][\phi] = [B]^{(28)}$$

where $[\phi]$ represents the vector of all the unknown potentials. For solving (5), a similar system results as long as W is prespecified. The matrix $[A]$ is usually extremely large. In earlier models (e.g. Tsay and Liu 1983; Tsay et al. 1989; Chen, 1990; Chen and Houston, 1987) the solution of (28) was accomplished by Gaussian Elimination, which requires enormous memory and is prohibitive when the number of wavelengths in the domain is large (i.e. short waves or a large domain). Pos and Kilner (1987) were able to alleviate this difficulty somewhat by using the frontal solution method of Irons (1970).

In recent years, the solution of (28) has been obtained with minimal storage requirements for $[A]$. This is due to the development by Panchang et al. (1991) and Li (1994a) of iterative techniques especially suited for (1). These techniques, based on the conjugate gradient method, guarantee convergence and have been found to be extremely robust in a wide variety of applications involving both finite differences and finite elements for several kinds of boundary conditions. For a review of other methods, see Panchang and Demirbilek (2000). Options based on the work of both papers, viz. Panchang et al. (1991) and Li (1994a), are available in CGWAVE. It is found that the latter often leads to faster convergence, but in an oscillating fashion. The former leads to a monotonically decreasing error which can be more reassuring while the iterations are in progress.

Related Topics

- CGWAVE
- Governing Equations
- Boundary Conditions

CGWAVE Practical Notes

Domain Shape: To the extent possible, it is necessary to have the coastlines outside the semicircle be as straight as possible, and ideally, parallel to one of the axis (as shown in Fig 1). The semicircle should be so located that the depths outside vary, to the extent possible, in a 1-d (cross-shore) direction only in the exterior. These depths are introduced in the model through the two 1-d sections. The user must specify a reflection coefficient for the exterior coastline.

Of course, these conditions are not encountered in practice in an ideal manner, but it is important to understand that the model works under these assumptions. V-shaped coastlines do not meet these criteria. Also, not all real domains have 1-dimensionally varying exterior depths. Regions with 2-d depth variations should preferably be incorporated inside the semicircle and it may be necessary to enlarge the domain so that solutions in the area of interest (hopefully) are not affected by the coastline shape. It is impossible to accommodate all real topographic variations of the model's exterior domain. Since a part of what happens outside the semicircle influences the solution within (i.e. how the backscattered waves behave), some simplified representation of the exterior is needed. The exterior representation used in CGWAVE is more sophisticated than that used in earlier models (which assumed collinear and fully reflecting coastlines and constant exterior depths).

The size of the domain is also governed by the wavelength. Typically, the minimum radius for the semicircle should be about 2 or 3 times the wavelength, which also dictates the overall number of grids. It is important that the modeler first estimate the number of grid points needed, at least in an approximate sense. This can be done by estimating a nominal wavelength L (based on the period and depth), and using $L/10$ as a rough measure of the grid size. Obviously, large domains with short period waves will lead to hundreds of thousands of nodes. This is not a problem physically, but can be very time consuming to simulate. For spectral conditions, the domain size should be dictated by the longer wavelengths L_1 (say radius = $3L_1$), but the resolution for the shorter wavelengths (L_2) would be, say $L_2/10$, which may be excessively fine for the longer wavelengths. Obviously, some care (and perhaps compromise) is needed in designing the grid and also selecting the input spectral components. If some components have little energy, it may be best to delete them from the modeling.

The input wave condition (amplitude, period, direction) must be specified at the end of the 1-d cross-section. Simulations are less reliable (not necessarily wrong) if the waves approach the coast at a glancing angle (say within 20° of the coastline). The 1-d sections **MUST** reach out further than the outermost point of the semicircle. The domain should have no zero or negative depths; the land-water interface must be demarcated using a small water depth (say 0.5 m, but the limit depends on the overall water depths and wave periods and the problem being solved). Note that making this too shallow may result in a large number of grids in this area which may not be necessary for all applications).

For basic specifications, the incident wave angle specification is as follows. Waves going in the +x axis (i.e. to East) are specified as incident angle = 0° . Waves going in +y direction (i.e. to North) are specified as incident angle = 90° , to West = 180° , to South = 270° . Incident wave direction is used in degrees. Convergence tolerance is typically assigned 10^{-8} , output frequency is 100 every iterations.

For the boundary condition specification, the "parabolic plus two 1d sections" option (Option 3 in *.cgi) should be chosen for most harbor problems, with a semicircle open boundary; in earlier versions, it is possible to use the option with just one 1d section (Option 2 in *.cgi) but it really provides no benefit over Option 3. The other options consist of the "parabolic approximation" (Option 1 in *.cgi) which is suitable if the exterior region is of constant depth. The Fourier Bessel Series (Option 0 in *.cgi) must be used if its limitations, described above in Section 2.2 (equations 15-17), are met; typically it is sufficient to specify 15 terms in the infinite series. For harbor problems, SMS will create the 1d sections based on depths based on depths on semicircle, and the desired grid spacing (used to solve the 1d equation (20)), based on (approximately) $L/10$ must be specified in the dialog box:

1-D Domain Extension Parameters

Length To Edge of Scatter: No scat

Radius of Domain: 1400.00 m

Ideal 1-D Spacing: 0.04 m

1-D Spacing: 0.500

of 1-D Nodes: 4200

Min 1-D Depth: 0.05

A full circle option is available for open sea problems where the exterior is of constant depth; the theoretical ideal for this is the Fourier Bessel Series (which works for unit wave input only); the parabolic mode may also be used (Panchang et al. 2000) with little loss of accuracy.

It is useful to perform initial sample runs, with no breaking and with fully absorbing coastlines, for a normally incident and obliquely incident wave input. The resulting solutions, especially the phase diagrams, can be used to perform visual quality control, since forward propagating waves are the easiest for making intuitive assessments (e.g. spacing between crests should be smaller and wave heights generally increasing in shallower water; bending of phase-lines for oblique incidence to approach the shore in a normal fashion, high wave heights in very shallow water because breaking was not applied, etc). Repeating the above runs with full reflection should lead to some standing wave patterns (large and small wave heights; rapid change of phase). A visual examination of the results will enhance confidence in later simulations.

There are two methods for applying wave breaking. For practical problems, we recommend making a non-breaking simulation and then applying an H/d limit (generally between 0.6 and 0.85) to alter the calculated wave heights to the limit (say $0.78d$). The other, more rigorous approach is to run the model in the nonlinear breaking mode where breaking properties are continuously recalculated based on prior solutions. This requires several rounds of iterations and is recommended for special problems and/or research applications. For nonlinear runs, the maximum number of nonlinear iterations is generally specified as 15 with a tolerance of 10^{-6} . For spectral simulations, obviously the former approach is recommended.

For incorporating docks, the actual “without-dock” water depth is originally specified while generating the grid; areas covering the docks must then be highlighted, and a draft and coefficient, selected from Fig. 2 (Li et al. 2005) or Table 1, must then be specified for these areas. The coefficient = $\alpha = A \ln(ka) + B$. For $kh < 0.1$ and $kh > 4$, we recommend using the A and B values corresponding to these thresholds. For convenience, the numerical values corresponding to Figure 8 are given in Table 1. (Note, a = half width of the structure, relative submergence = draft/water-depth = d/h).

☒ Floating Dock

Draft: 0.3 m

Coefficient: 1.0

For shallow water, α is roughly equal to unity for shallow draft, and $\alpha \sim 0.7$ for deep draft. For intermediate and deep water, α is not constant but shows an increasing trend with ka . In very deep water, the mismatch is large. Note that for short waves (relative to submergence), T tends to 0. This requires us to create a high level of wave blockage, which can be accomplished by $\alpha \sim 0$ (Li et al. 2005). Of course this result would not hold if d/h were much smaller than the smallest value investigated by Li et al. (2005), which is 0.25 (typical in engineering practice).

It is worthwhile to refine the grid in these areas since the model uses the under-keel clearance multiplied by the coefficient as the depth for calculation. This modified depth is smaller than the original depth (and hence the resolution should be finer).

Finally, it is noted an extensive list of test-cases are provided in the appendix. CGWAVE has been validated against these tests (which represents possibly the most rigorous testing for wave models). The model results are compared to lab data or analytical model results. The input and output files are provided. The user is highly encouraged to perform these simulations, alter parameters, etc. so that an examination of the results may help understand what can be expected. At a minimum, we recommend a visual inspection of the results provided. Often real-life problems have complex solutions which are difficult to explain or even anticipate. The fact that the model reproduces the correct result in so many cases may enhance the user's confidence in his/her results, assuming the modeling was performed with due diligence. The test-cases can also be used as a teaching tool.

Related Topics

- CGWAVE

CGWAVE Test Cases

The following test cases are available with grids, input, and output files for the user to familiarize himself/herself with.

Test cases are available here with grids, input, and output files for the user to familiarize himself/herself. An extensive list of test-cases are also provided by the Coastal and Hydraulics Laboratory [3]. CGWAVE has been validated against these tests (which represents possibly the most rigorous testing for wave models). The model results are compared to lab data or analytical model results. The input and output files are provided. The user is highly encouraged to perform these simulations, alter parameters, etc., so that an examination of the results may help understand what can be expected. At a minimum, we recommend a visual inspection of the results provided. Often real life problems have complex solutions which are difficult to explain or even anticipate. The fact that the model reproduces the correct result in so many cases may enhance the user's confidence in his/her results, assuming the modeling was performed with due diligence. The test cases can also be used as a teaching tool.

The following tutorials may also be helpful for learning to use CGWAVE in SMS:

- General Section
 - Data Visualization
 - Mesh Editing
 - Observation
- Models Section
 - CGWAVE

Tests 1 & 2^[1]

These tests involve monochromatic wave propagation over the shoal-slope bathymetry of Berkhoff et al. (1982). Grid has 15 points per wavelength. Parabolic approximation open boundary condition. Test 1 – input amplitude = 1 meter (linear). Test 2 – input amp = 0.0232 meter (nonlinear). Resulting amplification factors along Transect 5 are shown – they match results and data in Demirbilek and Panchang (1998). Wave direction and phases diagram shows largely progressive waves except near the shoal where the waves become multidirectional.

All runs involve no breaking.

Tests 3 & 4^[2]

Wave propagation over flat bottom and a shoal, after Vincent and Briggs (1989, JWPCOE). Monochromatic ($T = 1.3$ s) and broad-directional spectral (BI) input based on Panchang et al. (1990, JWPCOE). For spectral simulation, input consists of 29 directional components in the $\pm 60^\circ$ bandwidth and 5 frequency components. All runs involve no breaking. Results match numerical and experimental data described in Demirbilek and Panchang (1998), Vincent and Briggs (1989), and Panchang et al. (1990)

Test 5^[3]

Wave propagation ($T=5.05$ s, $d = 0.25$ meter) over a flat bottom surrounded by infinite ocean. **Depth = 0.25 meter.** Test 5 – using Bessel-Fourier boundary conditions (this is the most accurate boundary condition for the problem as specified although the exterior conditions are unrealistic in practice). See Xu et al. (1996, JWPCOE) for details. Note the Bessel-Fourier boundary condition works only for input amplitude = 1 meter. For other input amplitudes, solution should be appropriately scaled by user.

Test 6^[4]

As in test 5, but with friction which seems to become effective for long waves. Test 6 – the circular domain is assigned $f = 0.5$ everywhere, waves propagate in from the right. Friction leads to smaller wave heights.

Test 7^[5]

As in Test 5, but only the central area has a non-zero friction.

Test 8^[6]

Waves propagating towards a coastline on a flat seabed, parabolic and one-dimensional open boundary condition. No breaking or friction used.

Test 9^[7]

As in Test 8, but with parabolic boundary condition only. This is used to demonstrate correctness of this boundary condition since solution for constant exterior depth is known.

Test 10^[8]

As in Test 8, but friction $f = 0.5$ for the whole domain. Note wave height input = 2 m. at the end of one-dimensional section which extends beyond the semicircle. Wave heights decrease in shoreward direction due to friction. No breaking used.

Test 11^[9]

As in Test 8, but with $f = 0.5$ in a central square region (can be seen on .cgi file). $f = 0$ elsewhere. No breaking.

Test 12^[10]

As in Test 10, but with central square area indicated as a “floating dock”.

Test 13 - Circular Island/Shoal^[11]

Long wave propagation past the circular island/shoal combination of Homma (1950). Bessel-Fourier open boundary condition. Input $T = 240$ sec. Results match analytical solution given in Demirbilek & Panchang (1998). No breaking.

Test 14^[12]

As in Test 5, but with a circular pile in the domain. Input $T = 10$ s and constant depth = 15.03 meters. Results match analytical solution (see Panchang et al. 2000, ASCE JWPCOE).

Test 15^[13]

As in 14, but the pile is off-center. Parabolic open boundary condition. Results match analytical solution (Panchang et al. 2000, JWPCOE).

Test 16^[14]

Long wave ($T = 260$ s) propagation up a sloping beach. Parabolic and one-dimensional boundary condition. Solution is completely one-dimensional. Results match those in Panchang et al. (2000).

Test 17 ^[15]

Oblique wave incidence on uniformly sloping beach. Results match analytical solution of Radder (1979) given in Panchang et al. (2000, JWPCOE).

Test 18 ^[16]

Propagation of obliquely incident waves (incidence angle $= 20^\circ$) past a shore-perpendicular thin fully-reflecting breakwater on a sloping beach (beach is fully absorbing). Parabolic and one-dimensional open boundary condition. Results match analytical results given in Kirby (1986) and Panchang et al. (2000).

Tests 19 & 20 ^[17]

As in Test 18, but with nonlinear breaking on and off. Test pertains parameters in Zhao et al., (2000, Coastal Engineering).

Test 21 ^[18]

As in Test 19, but with shore-parallel breakwater. Parameters and results as in Zhao et al. (2000). Results are for no breaking.

Test 22 ^[19]

As in Test 19, but with shore-parallel breakwater. Parameters and results as in Zhao et al. (2000). Results are for nonlinear breaking.

Test 23 ^[20]

Wave propagation/resonance in a rectangular harbor. Results match analytical solution plotted in Demirebilek & Panchang (1998). With friction $f = 0.12$, the resonant peak amplification reduces substantially for $kl = 1.4$ ($T = 1.0447$ s).

Test 24 ^[21]

Wave propagation around a floating square platform in circular domain. While developing the 2-d grid, the area covering the dock is also filled with finite elements; each node is assigned a depth equal to the local under-keel clearance times the correction factor a . The parameters in the simulations are $2a = 2$ m, $h = 1$ m, $d/h = 0.5$ and $ka = 2$ (corresponding to the cases described by Tsay and Liu (1983). So the depth used for calculation $= \alpha \cdot d = \alpha \cdot (0.5) = 0.04$. Correction factors such as $a = 0.08$ are given in Li et al. (2005, Canadian J. of Civil Engr). Results are similar to 3d results given in Tsay & Liu (1983).

Test 25 ^[22]

Wave propagation around a circular shoal in a circular domain. This is intended to show the effects of the “steep slope” terms. The test is based on Fig. 7 in Chandrasekhara and Cheung (1997, JWPCOE).

Test 26 ^[23]

Radiation Stress calculations. Waves propagating towards a coastline on a flat seabed, parabolic and one-dimensional open boundary condition. No breaking or friction used. Five folders are given. Three separate monochromatic cases (260 degrees, 270 degrees, and 280 degrees, i.e. normal incidence and 10 degrees off-center incidence) each of amplitude 0.5 m and period $T = 1$ s, for fully absorbing coastline. For 270 degrees, results for fully reflecting coastline are also given. Results match theoretical solution (eq. 6-9 and eq. 54 in Copeland, 1985, Coastal Engg).

For spectral tests, the same 3 waves were added to form the input spectrum. Radiation stresses for the spectrum are an integration of individual components (eq. 1 in Feddersen 2004, Coastal Engg). The spectral results can be used also to check the mean wave direction (which should be 270 degrees) and the mean frequency ($= 6.28$ radians/s).

Test 27 ^[24]

Wave propagates (incidence angle $=0^\circ$) over a rectangular friction region in a constant-depth domain. With friction $f = 2.122$, input $T = 20\text{s}$, $H = 6.1\text{m}$ and constant depth $= 15.2\text{m}$. Results match solution obtained by Dalrymple et al. (1984).

Test 28 ^[25]

Obliquely incident wave propagates in a rectangular channel with the fully-reflecting side walls. Input $T = 12\text{s}$ and constant depth $= 8\text{m}$. Results match analytical solution plotted in Dalrymple and Martin (2000, JWPCOE).

Related Topics

- CGWAVE

Long Wave Input Toolbox

Infragravity (IG) waves are surface gravity waves with frequencies lower than the wind waves (hence the reference to them as *long* waves). IG waves consist, among others, of long-period oceanic waves generated along continental coastlines by nonlinear wave interactions of storm-forced shoreward-propagating ocean swells. They differ from normal oceanic gravity waves which are created by wind acting on the surface of the sea.

While normal wind waves typically have periods in the order of 20 seconds (or less), these waves can interact with coastlines which filter out the energy and convert some of the energy to subharmonic frequencies (ranging from 50 - 350 seconds). These are the IG waves. IG waves could also refer to phenomena such as tides and oceanic Rossby waves.

Motivation

When modeling harbors, it is possible energy in the IG range could result in resonance in the harbor. Therefore, it may be useful to simulate these terms in a CGWAVE analysis of a harbor.

IG waves may also influence navigation, coastal inlets, and coastal structural design projects.

The Toolbox

To define wave conditions consistent with infra-gravity waves, the interface in SMS includes the infra-gravity wave input toolbox. This tool utilizes a one dimensional Boussinesq model to compute wave conditions for input to CGWAVE.

Input

The IGWT requests for the following inputs:

- An input wave spectrum (from file or from SMS) – This defines the wave conditions at a deep water point.
 - When coming from a file, this refers to a frequency spectrum or *.spf file. These are directly input to the one dimensional Boussinesq model. The file format is defined by the developer of BOUSS1D and BOUSS2D.
 - When coming from a spectrum, the user chooses a spectra that has been loaded into or created by the SMS spectra generator. SMS creates a series of frequencies and energy densities from this spectrum that are fed into the one dimensional Boussinesq model.
 - Offshore water depth – This parameter refers to the depth in meters at the deepwater buoy (or site). If this is deeper than the Boussinesq limit, the depth will be set to the limit.
 - Nearshore water depth – This refers to the depth in meters at the predominant breaking location. This value can be approximated as twice the incident significant wave height.
-

- Minimum/maximum long wave period – This refers to the cutoff periods for the IG wave spectrum. Typical values are from 15 to 600 sec.
- Number of components – using this parameter the user specifies how many wave components will be generated **in each** specified direction for CGWAVE. This term is referred to as 'N' in the discussion below.
- Maximum oblique angle – The one dimensional Boussinesq model ignores wave direction. In fact, the directional bins from the input spectrum are ignored as they are converted to a frequency spectrum. This value defines a total variation (in degrees) for the directions to be considered. They are centered around the shore normal direction of the CGWAVE mesh.
- Number of angles – This value should be a positive integer. The toolbox will create N different wave components for each of this number of directions. If this number is 1, all the components will be generated in a shore normal direction. If this number is 2, then two sets of components will be generated, each half the oblique angle either side of the shore normal direction.

Process

The IGWT is actually setting up and running a one dimensional Boussinesq model as well as a pair of utilities to compute wave components to be applied on the CGWAVE mesh. Since this process is not always successful, it can be useful for the user to understand the process and in some cases, work through the process manually. As the tool is exercised and modified (feedback is welcome), the need for manual operation should be reduced, but understanding the process is still valuable. The three executables used in this process are distributed with SMS and can be found in the models area of the installation in the BOUSS2D folder. Each can be run in a DOS prompt in interactive mode if needed.

The 1D Boussinesq Run

The IGWT uses the model executable *bouss1d.exe* which is a one dimensional version of the Boussinesq equations. SMS creates a one dimensional profile starting at the specified **deep** water condition, maintaining that depth for five wavelengths and then transitioning up a gentle slope (1:50) to a depth of zero (0.0). The IGWT uses a grid spacing of 1/20 of the wavelength computed from the provided frequency spectrum.

The one dimensional run creates a number of output files at the specified **near shore** depth. These including significant wave height, mean water level, runup and a time series of water surface. The IGWT will use the water surface file (**_ts_eta.001*) as input to the spectral analysis utility in the next step.

Spectral Analysis

The IGWT uses the utility *spec_anal.exe* to process the water surface time series at the near shore depth and create a spectrum. This analysis can also be run manually at a DOS prompt after the one dimensional Boussinesq run generates the time series file. The utility will prompt the user for the name of an output file (qspec.001 by default). This output file is used as input in the next utility to generate wave components.

Wave Component Extraction

The IGWT uses the utility *export_v1.exe* to process the frequency spectrum at the near shore location and extract wave components at regular frequency spacing from the minimum to the maximum defined range. The utility creates a text file (out.txt) that lists these components.

Once the three processes have been run, the IGWT reads the out.txt file and distributes the wave components over the specified directions (centered around shore normal).

Output

The IGWT creates a series of wave components for each of the directions specified in the input. If the number of angles is 0, then there will be N components distributed through the frequency range. In the number of angles is 1, the toolbox will generate 3*N components (one direction offset from shore normal in each direction).

It is possible that zero energy components may be added. The user should verify the generated components.

Approach

The IGWT utilizes a one-dimensional version of the BOUSS-2D model to transform wave spectrum from the "deep-water" limit of the Boussinesq model ($H < L/2$). A constant 1:50 slope is assumed between the offshore and nearshore water depths. If complex offshore topography exists, use BOUSS-2D to bring the waves to the nearshore.

External Links:

- May 2007 ERDC/CHL CHETN-I-73 May 2007 Infra-Gravity Wave Input Toolbox (IGWT): User's Guide [4]

5.5. Coastal Modeling System

CMS

The Coastal Modeling System

The Coastal Modeling System (CMS) has been a research and development area of The Coastal Inlets Research Program (CIRP) at the United States Army Corps of Engineers – Engineering Research and Development Center (USACE-ERDC), Coastal and Hydraulics Laboratory (CHL) since 2006. It was build from a group of numerical models that have been under development since 2002. Information on the CIRP and publication on the CMS can be found at [1]

The system is a coordinated system of major multidimensional numerical models integrated to simulate waves, currents, water level, sediment transport, and morphology change in the coastal zone. Emphasis is on navigation channel performance and sediment exchanges between the inlet and adjacent beaches in the coastal zone. The CMS has been verified with field and laboratory data.

System Components

- CMS-Flow
- CMS-Wave

Steering

In order to combine the capabilities of the two main numeric engines of Flow and Waves, the user must pass information from one engine to the other. In the case of CMS-Flow, this means reading in wave data from CMS-Wave. Information passed to CMS-Flow includes reading radiation stress gradients that directly impact currents (wave driven currents) and height fields, wave directions and breaking data which enter into the sediment transport rate formulations. In the case of CMS-Wave, the option exists to read in currents and simulate their transformation by the current. For either situation, the data fields must be interpolated onto the native domain (interpolate wave data onto the flow grid and/or flow data onto the wave grid).

This may be done interactively using the tools in SMS. However, it is much more efficient to read or define simulations for both engines, and invoke the steering module from the Data menu. This tool runs the engines separately, but interpolates the output and passes it to the other engine automatically.

External Links:

- CIRP Wiki – online help database for CIRP information/publications and CMS modeling software [2]
 - US Army Engineer Research and Development Center – Ongoing Research [3]
 - Presentations
 - Coastal Modeling System (CMS) for Integrated Calculation of Waves, Flow, Sediment Transport, and Morphology Change [4]
 - Introduction to CMS-Wave [5]
 - Additional information on CMS-Flow capabilities [6]
 - CMS-Wave Demonstrations to Louisiana (Levees & Muddy coast) [7]
-

- Future of the CMS [8]

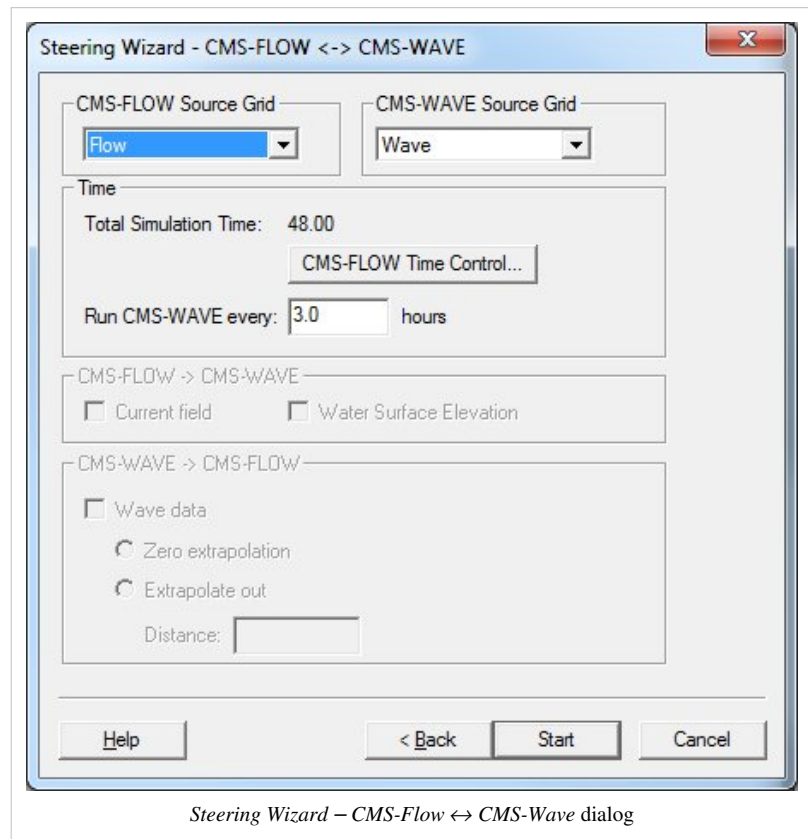
CMS-Flow/CMS-Wave Steering

Steering CMS INLINE page – Provides the ability to perform CMS-FLOW ↔ CMS-WAVE steering with data stored in memory. The steering is controlled by the cms-flow model executable and not by SMS.

The *Steering Wizard – CMS-Flow ↔ CMS-Wave* dialog is reached by going to the *Data* menu and selecting the **Steering Module** command. From the *Steering Wizard* dialog, select *CMS INLINE* and click on **Next**. The following dialog is the *Steering Wizard – CMS-Flow ↔ CMS-Wave* dialog. This dialog has the following options:

- CMS-FLOW Source Grid option – Select the cms-flow grid to be used in the steering.
- CMS-WAVE Source Grid option – Select the cms-wave grid to be used in the steering.
- Time options – Select the time parameters from the cms-flow model control interface.
 - Total Simulation Time:
 - CMS-Flow Time Control – this button opens the *CMS-Flow Model Control* dialog.
 - Run CMS-Wave every: – this field lets the intervals in which SMS will run CMS-Wave
- CMS-FLOW → CMS-WAVE options – Disabled. These options are set in the cms-flow model executable.
 - Current Field
 - Water Surface Elevation
- CMS-WAVE → CMS-FLOW options – Disabled. These options are set in the cms-flow model executable.
 - Wave Data
 - Zero extrapolation
 - Extrapolate Out – selecting this option causes the *Distance* field to become active where the distance can be specified.

Instructions for CMS-Flow/CMS-Wave steering can be found on the CIRP Wiki ^[1].



Related Topics

- Steering
- CMS-Flow
- CMS-Wave

External Links

- CIRP Wiki for CMS-Flow and CMS-Wave Model Coupling ^[1]

CMS-Flow/Save Points

Creating Save Points

Save points are cell-specific output that is used for CMS-FLOW simulations. To create save points do one of the following:

1. Create Save Points from an existing CMS_FLOW grid.
 - Click on the **Select Grid Cell** tool on the active grid.
 - Right-click on one or more selected cells and select **Add Save Point(s)...**
The CMS Save Point Manager is displayed showing the new cell(s). If no save points existed prior to this, a new save point coverage is created and a link will appear under the grid in the tree item data.
– OR –
3. Right-click on *Map Data* and select **New Coverage**. Find *CMS-Flow Save Points* under *CMS-FLOW* and click **Ok**. A new save point coverage is created. Click on the new coverage so its active, then click the **Create Feature Point** tool. Click out some feature points. When finished, right-click on the new coverage and select **Properties**. The *CMS Save Point Manager* is displayed showing the new cells.

Save Point Manager Dialog

The *CMS Save Point Manager* dialog is a spreadsheet showing individual save point cells. By default the "Name" column will display the grid cell id (if the save point was created from a grid). Attributes such as hydro, sediment salinity and waves and be turned on/off by toggling the values in the column. If the save points were loading into sms from a *.sp file, the cell row will be highlighted to indicate this.

Storing Save Points in the *.cmcards File

When saving the *.sms project, the save points get stored in the *.cmcards file. An example looks like this:

```

!Save Points

HYDRO_OUTPUT_INTERVAL    5.0 MINUTES
SEDIMENT_OUTPUT_INTERVAL 5.0 MINUTES
SALINITY_OUTPUT_INTERVAL  5.0 MINUTES
WAVE_OUTPUT_INTERVAL      5.0 MINUTES

SAVE_POINT "6, 61" -2867.5 2022.0 HYDRO SEDIMENT WAVE
SAVE_POINT "20, 59" -2517.5 1966.0 HYDRO
SAVE_POINT "10, 31" -2767.5 1182.0 SEDIMENT
SAVE_POINT "11, 31" -2742.5 1182.0 SEDIMENT
SAVE_POINT "10, 32" -2767.5 1210.0 SALINITY
SAVE_POINT "11, 32" -2742.5 1210.0 WAVE

```

The SAVE_POINT is formatted [name][x location][y location][hydro (if on)][sediment (if on)][salinity(if on)][wave(if on)]

***.sp/*.spx File for Reading in Save Point Information**

Simulation output data gets stored in *.sp and *.spx file. The *.sp file is an individual output file. The *.spx file simply identifies all of the *.sp files that should be loaded for a simulation. An *.sp file looks something like this:

```

SAVE_POINT_OUTPUT eta
REFERENCE_TIME      2001/01/01 00:00 -0000 GMT
CREATION_DATE       2012/05/28 09:02 -0600 GMT
CMS_VERSION         4.00.11
TIME_UNITS          HOURS
OUTPUT_UNITS        'm'
NUMBER_POINTS       2

NAME_BEGIN
    '6, 61'
    '20, 59'
NAME_END

```

XY_BEGIN

-2867.5000 2022.0000

-2517.5000 1966.0000

XY_END

SCALAR_TS_BEGIN

0.0000 0.0000E+00 0.0000E+00

0.0333 1.7545E-09 1.7434E-09

0.0667 1.4042E-08 1.3971E-08

0.1000 4.6679E-08 4.6524E-08

0.1333 1.1143E-07 1.1117E-07

Sms will read in a *.sp file and create a cms-flow save point coverage with the points.

Time Series Plot Wizard

*.sp files that are read into sms can be plotted into a time-series curve. To view the curve do the following:

1. Load a *.sp file into SMS
2. Right-click on **Display→Plot Wizard** from the menu dialog. Select *CMS Save Points* from the plot type then click next. Change the coverage to *Specified object(s)*. Now select the points you wish to plot. Now select the *Solutions and Datasets* then click **Ok**. A time series plot is created.

5.5.a. CMS-Flow

CMS-Flow

CMS-Flow	
Model Info	
Model type	Hydrodynamic model intended for local applications, primarily at inlets, the nearshore, and bays
Developer	Christopher W. Reed, Ph.D. Alejandro Sanchez Mitchell E. Brown
Web site	[1]
Tutorials	General Section • Data Visualization • Observation Models Section • CMS - CMS-Flow

CMS-Flow is a component of the Coastal Modeling System (CMS). Until 2007, it was developed under the name M2D. At that point in time, it was revised, file formats were updated for better flexibility and expandability, and it was incorporated into the CMS suite.

The model developers at the United States Army Corps of Engineers maintain a wiki specifically for the numerical engine. It can be viewed at: cirp.usace.army.mil/wiki/CMS-Flow ^[1]. For more information on the model itself, refer to the users manual ^[2] published by USACE-ERDC.

CMS-Flow is a finite-volume numerical engine which includes the capabilities to compute both hydrodynamics (water levels and current flow values under any combination of tide, wind, surge, waves and river flow) sediment transport as bedload, suspended load, and total load, and morphology change.

The interface in SMS allows the user to set up and edit computational grids, specify model parameters, define interaction of this model with the wave counterpart (CMS-Wave), launch the model and visualize the results.

The model is intended to be run on a project-scale, meaning the domain should only be on the order of 1-100 kilometers in length and width. The following sections describe the interface and make recommendations for application of the model.

Graphical Interface

The CMS-Flow Graphical Interface is contained in the Cartesian Grid Module as well as the Map Module and includes tools to create and edit a CMS-Flow simulation. The simulation consists of a geometric definition of the model domain (the grid) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the Cartesian Grid Module and setting the current model to CMS-Flow. If a grid has already been created for a CMS-Flow simulation or an existing simulation read, the grid object will exist in the Project Explorer and selecting that object will make the Cartesian grid module active and set the model to CMS-Flow. See Creating 2D Cartesian Grids for more information.

Using the Model / Practical Notes

For new simulations, users will create the CMS-Flow grid based on a conceptual model. The conceptual model includes:

- **Grid Generation** – We recommend that you generate a CMS-Flow grid using the conceptual model and a CMS-Flow Coverage. This coverage has attributes associated with a two-dimensional Cartesian grid and the model parameters associated with CMS-Flow. The grid position and extents are defined in the coverage using a grid frame, which you can define with three clicks of the mouse (recommendation is to click the lower left corner, lower right corner and then upper right corner, but the position, orientation and size can all be edited during the grid generation process. The coverage also defines the location of land and water in the grid using one of three methods:
 - **Land/Water cells defined based bathymetric values** – CMS-Flow uses depths, so positive depth indicates water, negative depth indicates land. Cells with depth less than the negative value of the water surface are dry. This option requires a geometric survey that includes both the bathymetric area and the areas that could potentially be flooded. This is the most intuitive option and the preferred method if geometric data is available.
 - **Land/Water interface defined by coastline arcs** – This option allows the user to define, read or import arc definitions that delineate the water area. These arcs include an orientation. To the left of the arc is land, to the right is water. The user can select an arc and swap its orientation. All the area inside the grid frame on the "water" side of the arc must have elevations defined either from a survey, or by specification. Cells created on the "land" side of the arc will never be included in calculations (they are permanently dry). These arcs also include an attribute defining how cells spanning this interface are to be classified. They may be forced to be water (ocean preference), forced to be land (land preference) or split based on the percentage of the cell on each side of the arc (percent preference).
 - **Land/Water interface defined by polygons** – This option also requires the user to define arcs delineating the extents of the computational area. However, these arcs must be closed into polygons. Each polygon is specified to enclose land or water and cells are classified accordingly.
- **Model Output** – The numerical engine consists of several components. The base engine computes hydrodynamics. To this, sediment transport and salinity can be enabled as well. Each process produces spatially varied solutions (values for each wet cell) that SMS can display as spatial datasets. Additional observation cells can be created to view output at a higher temporal resolution.

Case Studies / Sample Problems

The following tutorials may be helpful for learning to use CMS-Flow in SMS:

- Models Section
 - CMS – CMS-Flow

Related Links

- SMS Models page
 - CMS-Wave
-

External Links

- Sep 2008 Modeling of Morphologic Changes Caused by Inlet Management Strategies at Big Sarasota Pass, Florida [3]
- Jul 2007 ERDC/CHL CHETN-IV-69 Tips for Developing Bathymetry Grids for Coastal Modeling System Applications [4]
- Aug 2006 ERDC/CHL TR-06-9 Two-Dimensional Depth-Averaged Circulation Model CMS-M2D: Version 3.0, Report 2, Sediment Transport and Morphology Change [2]
- Feb 2006 ERDC/CHL CHETN-IV-67 Frequently-Asked Questions (FAQs) About Coastal Inlets and U.S. Army Corps of Engineers' Coastal Inlets Research Program (CIRP) [5] Updated FAQ Website [6]
- May 2005 ERDC/CHL CHETN-IV-63 Representation of Nonerrodible (Hard) Bottom in Two-Dimensional Morphology Change Models [7]
- May 2004 ERDC/CHL TR-04-2 Two-Dimensional Depth-Averaged Circulation Model M2D: Version 2.0, Report 1, Technical Documentation and User's Guide [8]
- Dec 2003 ERDC/CHL CHETN-IV-60 SMS Steering Module for Coupling Waves and Currents, 2: M2D and STWAVE [9]

CMS-Flow Cell Attributes

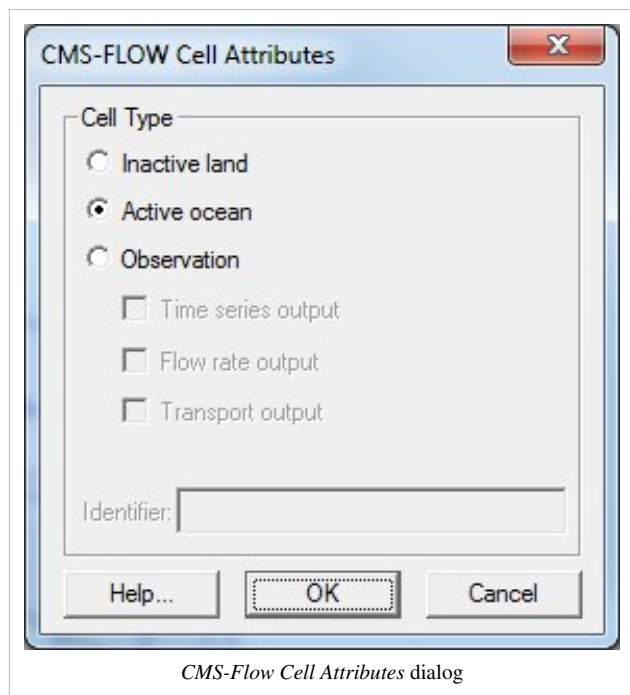
Individual cells of a CMS-Flow grid can be of four attributes which define the cells' computational inclusion into the simulation and controls specific computation output.

The cell attributes are as follows:

- Inactive land – This cell only exists within SMS, it will not be saved as part of the CMS-Flow computational domain. If this cell is adjacent to an active ocean cell, then interface between the two cells will be treated as a wall.
- Active ocean – This is a computational cell.
- Observation – This is a computational cell which also represents an observation station. CMS-Flow can output calculations of time series (water elevation and velocities), flow rate and transport at the cell.

This window is accessed from the *CMS-Flow* | **Assign Cell Attributes...** menu item or the *Cell Attributes...*

right-click menu associated with the **Select Grid Cell** tool with at least one cell selected.



Dialog Description

This dialog mainly consists of the *Inactive land*, *Active ocean*, *Inactive ocean*, or *Observation* radio group. Only one cell type may be chosen. If *Observation* option is selected then the observation output type check boxes and *Identifier* edit field are enabled. At least one output type must be selected for a valid observation station.

If multiple cells with differing cell types are selected, the window will not display a selected cell type and the **OK** button will be disabled. This is to protect the previous assignment from being overwritten by mistake. The **OK** button will be enabled when an option is selected. If multiple cells with differing observation output types are selected, the window will show *Observation* selected, the differing output type check boxes will be in a tri-state mode (checked with the check mark dimmed) and the **OK** button will be enabled. This is to represent that some cells are of the type and the rest are not. By clicking on the check box in tri-state mode, the new assignment (on or off) will be applied to all selected cells when **OK** is selected. A check box cannot be returned to tri-state mode once changed, select *Cancel* to omit assignment. A check box in tri-state mode when **OK** is selected means that that specific output type will not be changed, though other assignments (fully checked check boxes or identifier text) may be applied. The *Identifier* edit field works similarly, however, a note ("*Not the same for all cells") will appear when the identifying text is not unique for all selected cells. In this case, the edit field will display the last non-blank identifier found while checking the selected cells. If the note is present, then assignment of identifier text will not occur. The note will disappear if the text is changed, and therefore the text assignment will be applied to all selected cells when **OK** is selected.

Related Topics

- CMS-Flow

CMS-Flow Graphical Interface

The CMS-Flow graphical interface includes tools to assist with creating, editing, and debugging a CMS-Flow model. The CMS-Flow interface exists in the Cartesian Grid Module. The Map Module also provides flexibility when creating the grid.

CMS-Flow Menu

The interface consists of tools to create the grid, a *CMS-Flow* menu with model specific commands and visualization tools for analyzing the datasets computed by the model. The menu includes the following commands:

- **Assign BC...** (Boundary Condition) – This command allows the user to assign a boundary condition to the cell string through the *CMS-Flow Boundary Conditions* Dialog.
- **Delete BC...** – This command allows the user to delete the boundary condition assigned to that string.
- **Cell Attributes...** – This command allows the user to set an individual cells status.
- **Merge Cells** – This command allows the user to merge consecutive rows or columns into one.
- **Model Check...** – The CMS-Flow **Model Check** performs a number of checks on the simulation to ensure a valid model.
- **Model Control...** – This command allows the user to view and edit the current parameters that affect how CMS-Flow runs and what options are to be included in the current simulation.
- **Run CMS-Flow...** – This command launches the CMS-Flow model for the current simulation.

See CMS-Flow Menu for additional information.

Setting Up/Editing a Model

Grid Creation

A grid for CMS-Flow consists of varying sized rectangular cells. Each cell defines the depth from a fixed datum to the bottom of the water column at the centroid of the cell. CMS-Flow computes the water level at the cell centroid, and the flow velocity across each cell face. A grid may be created in one of two ways including:

- Through the Cartesian Grid module – This option allows for the creation of a simple grid consisting of cells of uniform size (they may be rectangular, but all the same size). See Creating 2D Cartesian Grids for more information. This grid may be edited in the Cartesian Grid module (see Editing 2D Grids).
- Using a CMS-Flow Coverage in the Map Module – This method results in a grid that can be edited with all of the tools available in the Cartesian Grid module, but also allows flexibility in two ways:
 - Refine points may be created to specify cell resolution in one or both directions at specific locations inside the domain. This allows areas of high concern, and high variability, such as in the mouth of an inlet to be defined with specified high resolution, which transitions to lower resolution in areas away from these regions.
 - The grid frame, with the grid definition, is saved and can then be copied and edited to create variations of grid resolution and coverage.

Boundary Conditions

All numeric models require boundary condition data. In CMS-Flow, boundary conditions are defined on cell strings.

Cell Strings

For CMS-Flow, a cell string consists of a series of contiguous or adjacent cells. This means the cells share at least a corner and usually an edge. When SMS creates a grid, it also creates cell strings around the computational domain. This consists of open ocean cells along the edge of the domain as well as the last ocean cells adjacent to the land. SMS will create one cell string for each continuous section of ocean along the grid boundary, one cell string for each section of coastline, and one cell string for each island. Each cell string may be assigned an attribute to describe the type of boundary condition applied at that location. If no specification is made, the default boundary condition is a closed boundary (no flow is allowed across this boundary). This is typically the case for coastline and island cell strings. These cell strings are actually not even required for CMS-Flow, but are useful when using CMS-Flow output in a Particle Tracking Model (PTM) simulation.

Cell strings may be split into multiple cell strings by selecting the cell where the break is desired and selecting the **Split** command from the *Cellstring* menu.

Cell strings may be merged by selecting two (or more) cellstrings that connect in a continuous line, and selecting the **Merge** command from the *Cellstring* menu.

Assigning Boundary Conditions

The user assigns a boundary condition by activating the **Select Cellstring** tool, selecting the cell string, and then selecting the **Assign BC...** command from the menu. This invokes the *CMS-Flow Boundary Conditions* dialog and allows the user to define the conditions along the cell string. Multiple cell strings may be selected and assigned at the same time, but this is not common. Typically, the boundary conditions will only be applied to cell strings along the edge of the domain, but may be applied to cell strings on the interior if a portion of the domain is to be excluded from computation (such as cutting off a portion of a bay) for computational efficiency.

Boundary condition types supported by CMS include:

- Land – This is the default condition and indicates that no flow will cross this boundary.

- Flow rate-forcing – This allows the user to specify an inflow rate (flow in cubic meters/second at each cell). This can be used to represent a river flowing into the domain.
- WSE-forcing – This allows the user to specify the water surface elevation as a function of time for the cellstring. Options include specifying a single curve (water level -vs- time) and all the cells will have the same water level at the specified time and extracting individual curves for each cell either from a regional tidal database (ADCIRC database) or from a regional (larger) circulation model.
- Tidal constituent-forcing – This allows the user to specify the water surface as a function of the principal tidal constituents around the domain. When this option is selected, the user must specify which constituents (along with their amplitudes and phases) in the Model Control dialog. All cells in the tidal cell strings will be forced with the same water levels.
- WSE and Velocity-forcing – This option allows the user to extract both water levels and velocity values for each cell in the cellstring from a regional model. This requires a regional model to extract from.

Cell Attributes

The user may select cells and assign attributes to them using **Select Cell** tool and two commands from the *CMS-Flow* menu.

If a cell is selected, its bathymetric depth may be edited using the "Z" edit field at the top of the screen.

For any selected cells, the **Assign Cell Attributes** command allows the user specify the cell as water or land, and if it is water, specify that it is an observation cell. This option instructs the model to output data at a higher frequency at this location for analysis after the simulation is complete.

When entire adjacent rows or columns are selected, they may be merged to decrease resolution in this area of the domain. (Warning, this operation deletes all cell strings, so all boundary condition specification would be lost if this operation is performed. Make sure you do this type of operation during grid construction.)

Model Check

Whenever the user issues the **Model Check** command, and just before launching an analysis, SMS performs a check on the model. If conditions are detected that are out of the ordinary or recommended range, warnings are given with suggestions to correct the problem. If serious problems are detected, errors are reported, again with suggestions.

Model checks include checking on the aspect ratio of the cells, ensuring that compatible boundary conditions are specified, and ensuring that model parameters are within acceptable ranges.

The model check is not all inclusive. Just because a model passes the model check does not guarantee the model will run and produce viable solutions.

Model Control

The *CMS-Flow Model Control* dialog is used to setup the options that apply to the simulation as a whole. These options include time controls, run types, output options, global parameters, print options and other global settings.

Running the Model

The CMS-Flow files are written automatically with the SMS project file or can be saved separately using the *File | Save CMS-Flow* or *File | Save As* menu commands. See the CMS-Flow publications from ERDC for more information on the files used for the CMS-Flow run.

CMS-Flow can be launched from SMS using the *CMS-Flow | Run CMS-Flow* menu command. A check of some of the common problems called the Model Checker is done each time the model is launched, or by selecting the *CMS-Flow | Model Check* menu command.

Related Topics

- CMS-Flow
- Hard Bottom
- Observation Cells
- Cell Attributes
- Advanced Cards
- Salinity
- Create a 2D Cartesian grid

CMS-Flow Grid Modification Flag

As CMS-Flow runs, especially when coupled with CMS-Wave, water levels increase and decrease and bed morphology changes the bottom elevations. Some cells may change state from dry to active. When this occurs, the datasets associate with grid change. The grid modification flag was created to indicate that the grid status has changed so that the model knows that geometric relationships (such as neighbor files) need to be updated.

The grid modification flag will start at 0 and increment by 1 when one of the following occurs:

- Merging Rows and columns
- Splitting rows and columns
- Dragging row or column edges
- Duplicating a grid
- Making an active ocean cell inactive or vice versa

The flag is written out to the *.cmcards file with the GRID_MODIFICATION_NUMBER card. Below is an example:

```
GRID_MODIFICATION_NUMBER 4
```

Related Topics

- CMS-Flow
-

CMS-Flow Hard Bottom

CMS-Flow includes a Hard Bottom morphologic constraint that "provides the capability to simulate mixed bottom types within a single simulation," (ERDC/CHL TR-06-9 Two-Dimensional Depth-Averaged Circulation Model CMS-M2D: Version 3.0, Report 2, Sediment Transport and Morphology Change [2], p. 33). Hard bottom is a cell depth constraint applied to areas of the grid which represent exposed or covered non-erodible material such as bedrock. This specification limits the erosion of sediment to the hard bottom depth and therefore constrains water depth. During sediment transport calculations, exposed hard bottom cells may become covered through deposition.

By default, CMS-Flow cells are fully-erodible cells with no specified hard bottom depth (inactive cells; denoted by the CMS-Flow null value of -999.0).

When specified, cell hard bottom depths will appear in the Project Explorer as a scalar dataset beneath the CMS-Flow grid. This dataset can not be deleted, though it can be edited like any other dataset. A CMS-Flow simulation must contain the hard bottom dataset (even if it is not specified) so SMS will create a defaulted (inactive cells) dataset if it does not already exist when saving the simulation.

Hard bottom should be specified only for computational (ocean) cells.

The hard bottom dataset can created, edited, viewed and verified using the following SMS interface features.

Model Control

Within the CMS-Flow Model Control window, the hard bottom dataset can be created from the *Sediment* tab. If the dataset does not exist, it can be created using the **Create Dataset** button. If a dataset exists (created using the Data Calculator) which represents the intended hard bottom specifications, the **Select Dataset...** button can be used to select such dataset and copy the values to the hard bottom dataset.

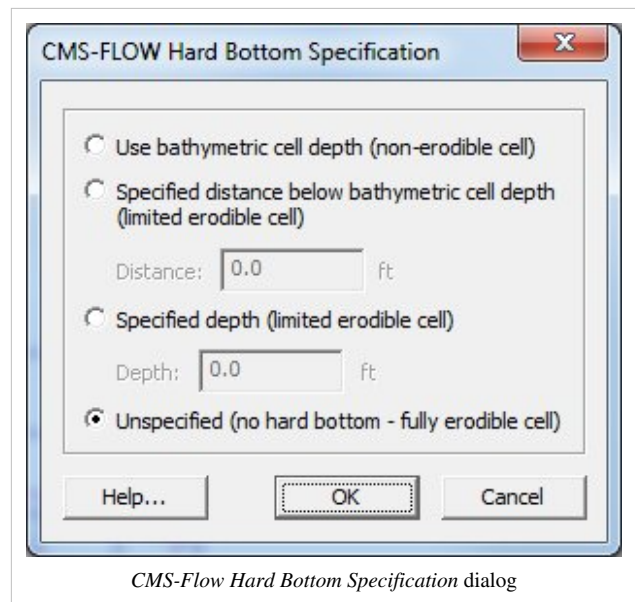
Hard Bottom Specification

Although the hard bottom dataset can be edited (when its the active dataset) by selecting a cell (or group of cells) and changing the scalar (S) value in the *Edit Window*, an user-friendly window exists which provides specification options. With the **Select Grid Cell** tool active, make a selection, right-click to bring up the tool menu and choose the **Specify Hard Bottom...** option.

This will open the *CMS-Flow Hard Bottom Specification* window.

The following options are provided in the *Hard Bottom Specification* window:

- **Use bathymetric cell depth** – Sets the cell hard bottom depth to be the cell geometry value thereby creating an exposed non-erodible condition. If multiple cells were selected, then each cell will use its respective bathymetric depth.
- **Specified distance below bathymetric cell depth** – Sets the cell hard bottom depth to be the cell geometry value plus the specified distance thereby creating a sediment-covered non-erodible condition. The distance is limited to positive values to ensure the hard bottom depth is greater than the geometry value. The cell can provide sediment for transportation, however, the amount of erosion is limited. If multiple cells were selected, then each cell will



use its respective bathymetric depth.

- **Specified depth** – Sets the cell hard bottom depth to the specified depth thereby creating a sediment-covered non-erodible condition similar to specified distance. The depth is limited to greater than the geometry value. If multiple cells were selected, then the depth is limited to greater than the largest geometry value and all cells will have the same value.
- **Unspecified** – Resets to an inactive hard bottom condition. The cell hard bottom depth is set to the CMS-Flow null value. If multiple cells were selected, then all cells will be reset.

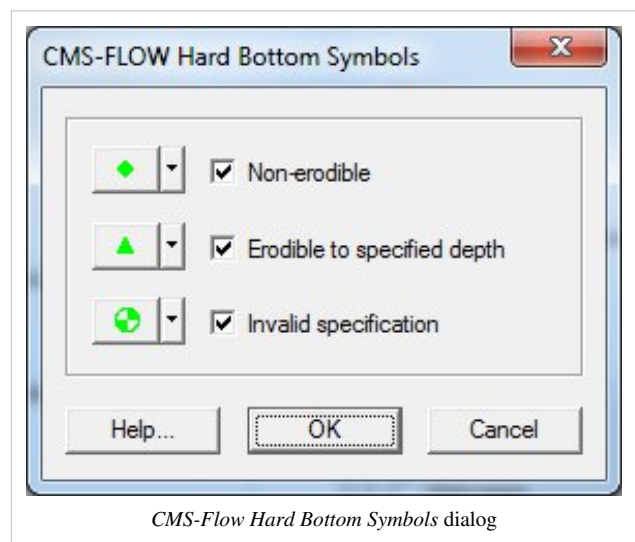
If no cells are selected when opening the *Hard Bottom Specification* window, then all computational (ocean) cells will be used. If a selection of only non-computational cells, then specification cannot occur. If a selection contains computational and non-computational cells, then the specification will only apply to the computational cells.

If multiple computational cells with differing specifications are selected, the window will not display a selected specification type and the **OK** button will be disabled. This is to protect the previous specifications from being overwritten by mistake. The **OK** button will be enabled when an option is selected. The minimum hard bottom depth of the multiple computational cells selected will be displayed in the *Depth* edit field and the minimum hard bottom depth minus the maximum geometry depth of the multiple computational cells selected will be displayed in the *Distance* edit field.

Display Options

The hard bottom dataset (when its the active dataset) will only display the cells with hard bottom specified if the *Ocean cell* display option is turned on. Inactive hard bottom cells are not displayed.

CMS-Flow includes hard bottom symbols to differentiate specifications. On the *Cartesian Grid* page of the *Display Options* window (when CMS-Flow is the active model), the *Hard bottom symbols* check box controls the display of symbols that will appear in hard bottom cells (even if the hard bottom dataset is not active). If this is turned on, then the user must be aware of the individual symbol settings accessed by clicking on the **Options...** button. The **Options...** button displays the *CMS-Flow Hard Bottom Symbols* window.



CMS-Flow Hard Bottom Symbols dialog

Hard bottom symbols can be selected for three hard bottom specification types:

- **Non-erodible** – Displayed in exposed hard bottom cells (cell hard bottom depth is equal to cell bathymetric depth).
- **Erodible to specified depth** – Displayed in sediment-covered hard bottom cells (cell hard bottom depth is greater than cell bathymetric depth).
- **Invalid specification** – Displayed in hard bottom cells where the hard bottom depth is less than cell bathymetric depth (the geometry is below the erosion limit).

If the *Hard bottom symbols* check box is turned off, no symbols will be displayed and the individual settings cannot be accessed, however, the individual settings will not be changed.

Model Check

The CMS-Flow Model Checker, accessed from the *CMS-Flow* | **Model Check...** menu item, includes a check to ensure that no invalid hard bottom specifications exist in the grid. An invalid specification may be created, for example, by setting an infeasible hard bottom scalar value in the Edit Window or adjusted the grid's geometry without updating the hard bottom. It is suggested that the model checker be used prior to running CMS-Flow.

Related Topics

- CMS-Flow

CMS-Flow Hot Start

In the CMS-Flow *Model Control*, the user can specify a previously saved hot start file to be used as initial conditions or instruct CMS-Flow to save hot start files for future use.

To create a hot start file, either select *Write Hot Start output file* and select an output time, or select *Automatic recurring Hot Start file* and choose an output interval.

- Choosing to write the hot start file at a specific output time will create the following file:
 - *hot_start.h5* This file has the simulation data including elevations and velocities.
- Choosing automatic recurring hot start files will create the following files:
 - *HOTSTART.INFO* This file records what time the hot start file was written and which hot start file is the most recent.
 - *HOTSTARTx.H5* (where "x" is a counter) These files have the simulation data including elevations and velocities.

Once the hot start files are created, they can be read into CMS-Flow. Open the *CMS-Flow Model Control* and check the "initial conditions file" check box, then select the hot start file you wish to use.

When using a hot start file, the following parameters should be changed as follows:

- *Start Date*: no change
- *Start Time*: no change
- *Simulation Duration*: decrease by the duration of the hot start file
- *Boundary Conditions*: no change

Note: For a simulation using a hot start file, the first time step of the solution will be the start time plus the value of one time step plus the duration of the hot start file.

Related Topics

- CMS-Flow
-

CMS-Flow Menu

The CMS-Flow menu includes commands for editing and running a simulation. This includes viewing and assigning boundary conditions and cell attributes, checking the simulation for common errors, and specifying model parameters. Commands from the menu include:

- **Assign BC...** – This command is inactive unless a cell string is selected. It allows the user to assign a boundary condition to the cell string through the *CMS-Flow Boundary Conditions* dialog. Depending on the boundary condition type, curves may be created or extracted to define the boundary condition. When appropriate, the file browse button allows the user to load an already-existing time series for use with one of the time series boundary conditions. Boundary conditions specified within this dialog include:
 - **Land** – When a cell string is created, the string receives this assignment. This defines a boundary as closed and no flow will cross this boundary.
 - **Flow Rate-forcing** – A time series must be defined (by clicking the **Define Curve...** button) that contains a list of times and flow rates in m³/sec along the boundary.
 - **WSE-forcing** – A time series must be defined for either the string or each cell in the string. The time series associated with this boundary contains a list of times and water levels in meters along the boundary. An option to extract time series curves or each cell can be applied to regional models loaded into SMS or create time series curves from tidal constituents.
 - **Tidal Constituent-forcing** Uses up to eight tidal constituents to force with. Tidal constituents are specified within the *CMS Tidal Constituent* dialog accessed by clicking **Global Attributes..** in the *Options* section of the *Boundary Conditions* dialog. The *CMS-Flow Tidal Constituents* dialog allows the user to specify the tidal constituents to force with along any tidal constituent-forcing boundaries that have been defined within the M2D grid. The user must specify an amplitude and phase for each constituent that will be used to force with. In SMS 11.1 Up to 37 constituents may be used: M2, S2, N2, K1, M4, O1, M6, MK3, S4, MN4, NU2, S6, MU2, 2N2, OO1, LAM2, S1, M1, J1, MM, SSA, SA, SF, MF, RHO, Q1, T2, R2, 2Q1, P1, 2SM2, M3, L2, 2MK3, K2, M8, MS4. Earlier versions us up to eight constituents may be used: M2, S2, N2, K2, K1, O1, M4, and M6.
 - **WSE and Velocity-forcing** – This option adds the requirement to specify currents at each cell to the *WSE-forcing* option. These must be extracted from a regional model.
 - **Harmonic WSE-forcing** – With this condition, the user can specify the components of the forcing curve after selecting **Global Attributes...** in the *Options* section of the *Boundary Conditions* dialog. The specification of terms of the time series consist of amplitude (m), frequency (cycles/hr) and phase (deg). Values are written out in the *.cmcards.file with the HARMONIC_BEGIN and HARMONIC_END cards. Assigned cellstrings are written as HARMONIC_CELLSTRING in the same file.
- **Delete BC** – This command is inactive unless a cell string is selected. It allows the user to delete the boundary condition assigned to that string.
- **Assign Cell Attributes** – This command allows the user to set an individual cells status. A cell can be active (water), inactive (land), or observation. Observation cells are assumed to be active. The user also specifies a roughness value for the cell. In future versions a material type will be specified instead of a roughness value directly. Observation cells can be assigned where CMS-Flow can write time series and/or flow rate output files.
- **Merge Cells** – This command allows the user to merge consecutive rows or columns into one. The select row or column tools must be used to perform this operation.
- **Model Check...**

The CMS-Flow Model Check performs a number of checks on the simulation to ensure a valid model. These checks include:

 - All water boundaries have boundary conditions specified.

- Cell aspect ratio is no greater than 2.
- Max time step size is suggested according to an internal calculation made within SMS.
- Make sure model control has been set up correctly (i.e. if a tidal boundary exists, make sure that tidal constituents have been defined to force with).
- Ensure that CMS-Flow has been set up to output the specific data for any existing observation station and that observation stations exist for any specified output data. Observation is not required to run CMS-Flow so this is a reminder that something may have been intended but forgotten.
- Ensure that no invalid hard bottom specifications exist in the grid. An invalid specification may be created, for example, by setting an infeasible hard bottom scalar value in the *Edit Window* or adjusted the grid's geometry without updating the hard bottom.

The CMS-Flow Model Checker is accessed from the *CMS-Flow* | **Model Check...** menu item. It is suggested that the model checker be used prior to running CMS-Flow.

- **CMS-Flow Model Control...** – The *CMS-Flow Model Control* dialog allows the user to view and edit the current parameters that affect how CMS-Flow runs and what options are to be included in the current simulation. The dialog includes six tabs which partition the parameters into related groups. The tabs and their related parameters include:
 - General – General model parameters and options.
 - Flow – General model parameters and options.
 - Wind – Wind datasets. These are optional spatially varied input to the engine.
 - Wave – Wave datasets. These are optional spatially varied input to the engine.
 - Output – Parameters associated with global output.
 - Advanced – This tab allows those working with model developers to work with experimental options in the model.
- **Run CMS-Flow ...** – This command launches the CMS-Flow model for the current simulation.

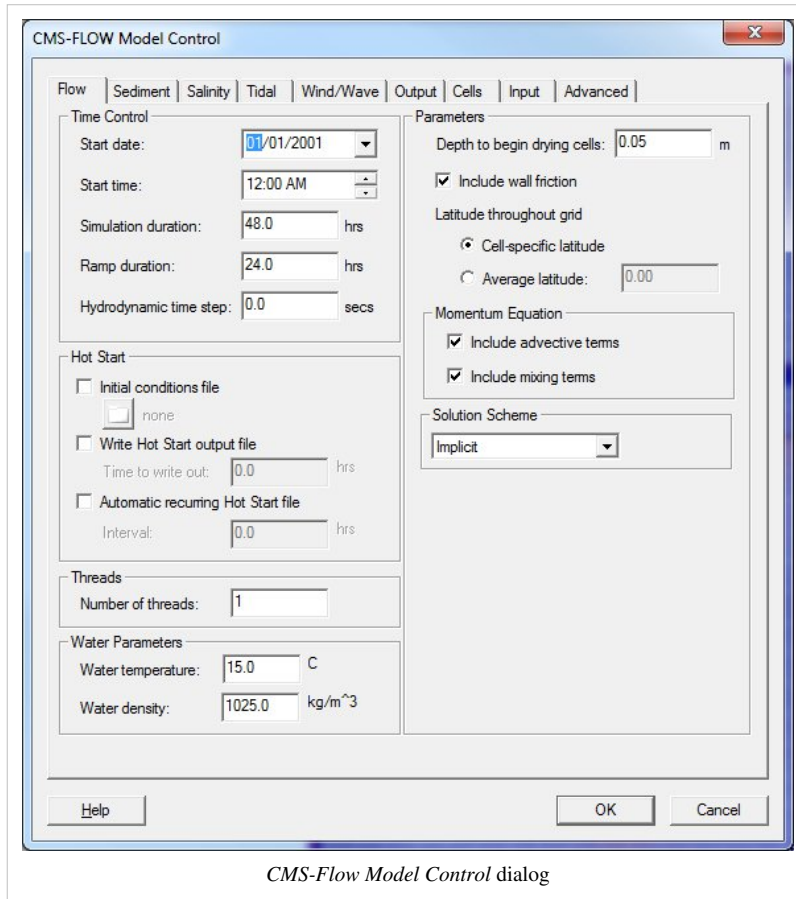
Related Topics

- CMS-Flow

CMS-Flow Model Control

The *CMS-Flow Model Control* dialog allows the user to view and edit the current parameters that affect how CMS-Flow runs and what options are to be included in the current simulation. The dialog includes nine tabs which partition the parameters into related groups. The tabs and their related parameters include:

Flow



CMS-Flow Model Control dialog

This dialog allows the user to specify general flow model parameters, controlling which model options the simulation will employ. The controls include:

Time Control

In this section the user can set the starting time, duration and hydraulic time step. The Ramp duration defines the length of an incremental loading portion at the beginning of the simulation. Controls include:

- Start date and time
- Simulation duration or end date and time
- Ramp duration

Hot Start

These controls allow the user to specify a previously saved hot start file to be used as initial conditions or

instruct CMS-Flow to save hot start files for future use.

- Initial conditions file
- Write Hot Start output file
- Time at which Hot Start output file should be written
- Automatic recurring Hot Start file
- Interval for writing recurring Hot Start file

See CMS-Flow Hot Start for more information.

Threads

CMS-Flow can take advantage of multiple processors using Open-MP parallelization technology. This control allows the user to specify the maximum number of threads that the engine should occupy.

Water Parameters

- Water temperature
- Water density

Parameters

- Depth to begin drying cells
- Include wall friction
- Latitude throughout grid
 - Cell specific latitude
 - Average latitude

Momentum Equation

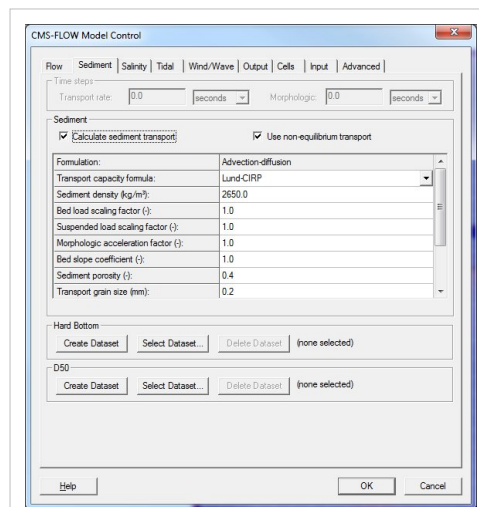
- Include advective terms
- Include mixing terms

Solution Scheme

CMS-Flow now has two solution schemes. These include the traditional "explicit" scheme. This method requires that flow be tracked through each element, resulting in hydrodynamic time steps in the order of one to two seconds. The new option is an "implicit" scheme which uses finite volume technology, supports much larger time steps resulting in fewer time steps (shorter run times) and is identically mass conserving. When the implicit scheme is being used, one of the following matrix solver types should be specified.

- GMRES
 - GAUSS-SEIDEL
 - GAUSS-SEIDEL-SOR
 - BICGSTAB
-

Sediment



CMS-Flow Model Control dialog showing the Sediment tab

- Time steps
 - Transport rate
 - Morphologic
- Sediment
 - Calculate sediment transport
 - Use non-equilibrium transport
 - Formulation
 - Advection-diffusion
- Hard Bottom
 - Create Dataset
 - Select Dataset
 - Delete Dataset
- D50
 - Create Dataset
 - Select Dataset
 - Delete Dataset

Salinity

This tab allows the user to specify model parameters relating to salinity modeling in CMS-Flow. The controls include:

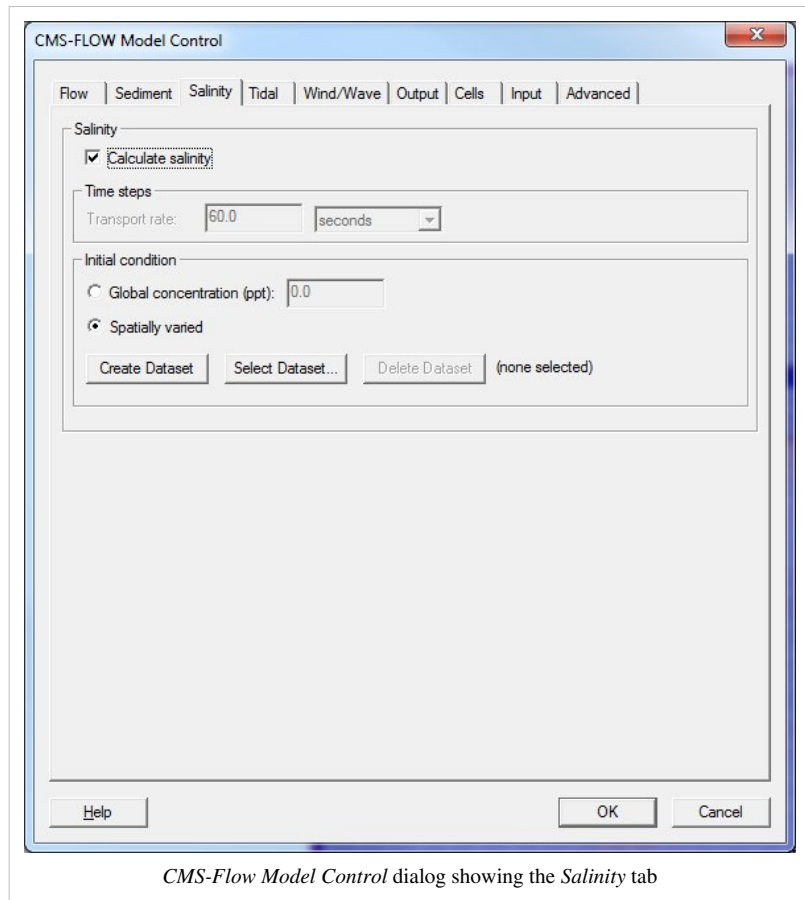
Salinity

CMS-Flow includes the capability to simulate the diffusion of saline water in fresh water. This section includes controls to enable and initialize these calculations.

- Calculate salinity – Turn on the calculation of salinity concentrations in the CMS-Flow simulation.

Time Steps

- In addition to the time step specified in the *general parameters* tab for hydrodynamic calculations, the explicit model also allows the specification of longer time step for the salinity dispersion process.
- Transport rate – If salinity is enabled in this tab, and the solver option is set to "explicit" in the *Flow* tab, the user must specify a this time step. It must be larger than the hydrodynamic time step and is generally at least a factor



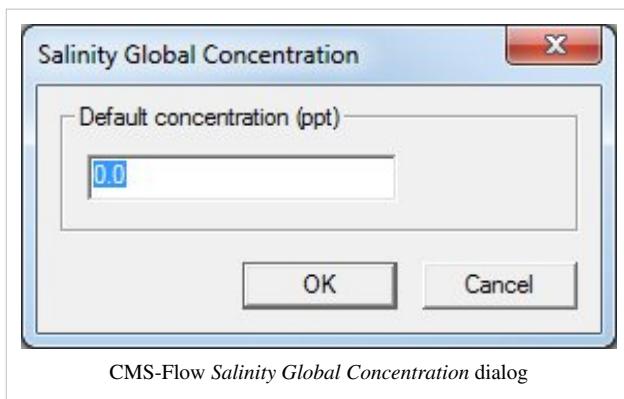
CMS-Flow Model Control dialog showing the Salinity tab

of 20. The default transport rate time step is 60 seconds.

Initial Condition

Salinity transport requires that each cell have an initial value for salinity concentration (measured in ppt). CMS-Flow allows this value to start as a constant for each cell, in which case, during the initial (or ramp time) of the simulation the concentrations are distributing to natural values.

- Global concentration – This control allows the user to specify an initial concentration for the entire domain. The value 0.0 ppt (fresh water) is a commonly used value.
- Spatially varied – these controls allow the user to create or select a spatially varied dataset that will be used by CMS-Flow as the initial concentration at each cell. This dataset is editable by the user.
- Create Dataset

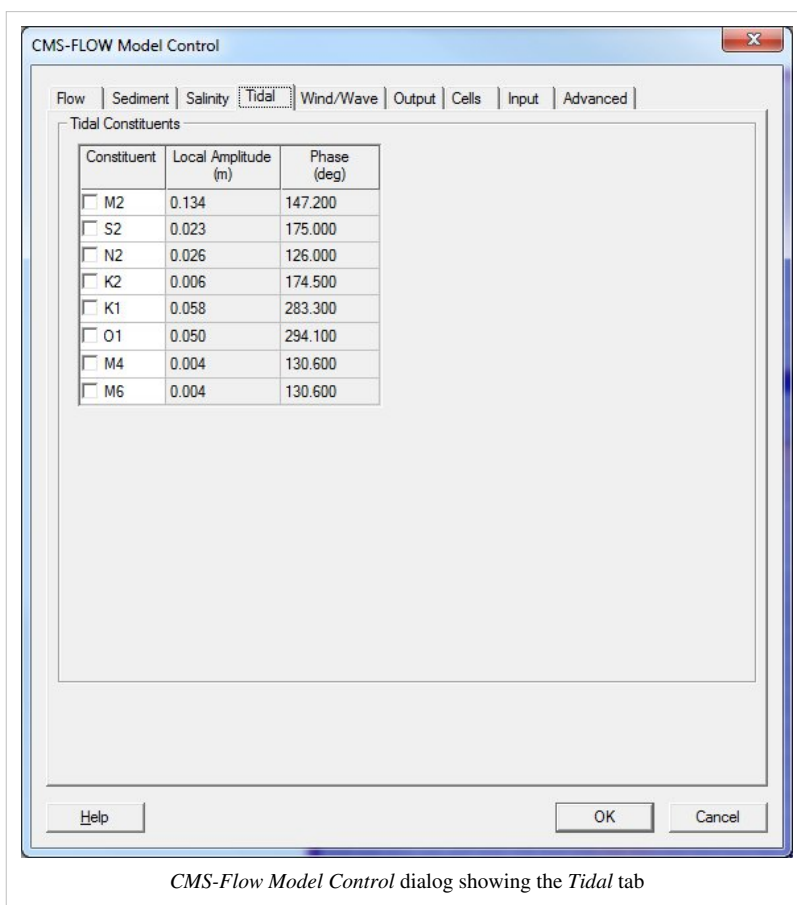


- Select Dataset
- Delete Dataset

External Documentation

- Two-Dimensional Depth-Averaged Circulation Model CMS-M2D: Version 3.0, Report 2, Sediment Transport and Morphology Change[1]

Tidal



Flow

This dialog allows the user to specify general hydrodynamic or flow model parameters, controlling which model options the simulation will employ. The controls include:

Hydrodynamic Time Step

This control allows the user to set the hydrodynamic time step for the model. If the explicit solution scheme is being used, a recommended timestep will be calculated based on grid cell size and depths.

Water Parameters

This section allows the user to specify various general parameters to be used by the simulation. This include:

- Water temperature
- Water density

- Depth to begin drying cells: This is a minimum depth of water before the model considers the cell to be dry. Shutting off cells when the depth drops below a tolerance avoids flutter of insignificant transfers of water. The recommended default value is 0.05 meters.
- Include advective terms
- Include mixing terms
- Include wall friction: Normally friction at the closed boundaries are ignored. The model can be instructed to include friction along these boundaries as if a wall exists there.
- Include stokes velocities

Bottom Friction Dataset

Friction datasets can be created and specified to define a spatially variable bottom roughness to provide a resistance to flow. This value has little impact in deep ocean applications but can be important in shallow regions. This is a user editable dataset. The available dataset types are:

- Mannings N
- Bottom friction coefficient
- Roughness height

A bed-slope friction coefficient toggle gives the option to specify a wave-current bottom friction type and a coefficient.

Turbulence Parameters

CMS-Flow now supports turbulence parameters.

- Model: Subgrid, Falconer, Parabolic, Mixing length
- Base value
- Current bottom coefficient
- Current horizontal coefficient
- Wave bottom coefficient
- Wave breaking coefficient

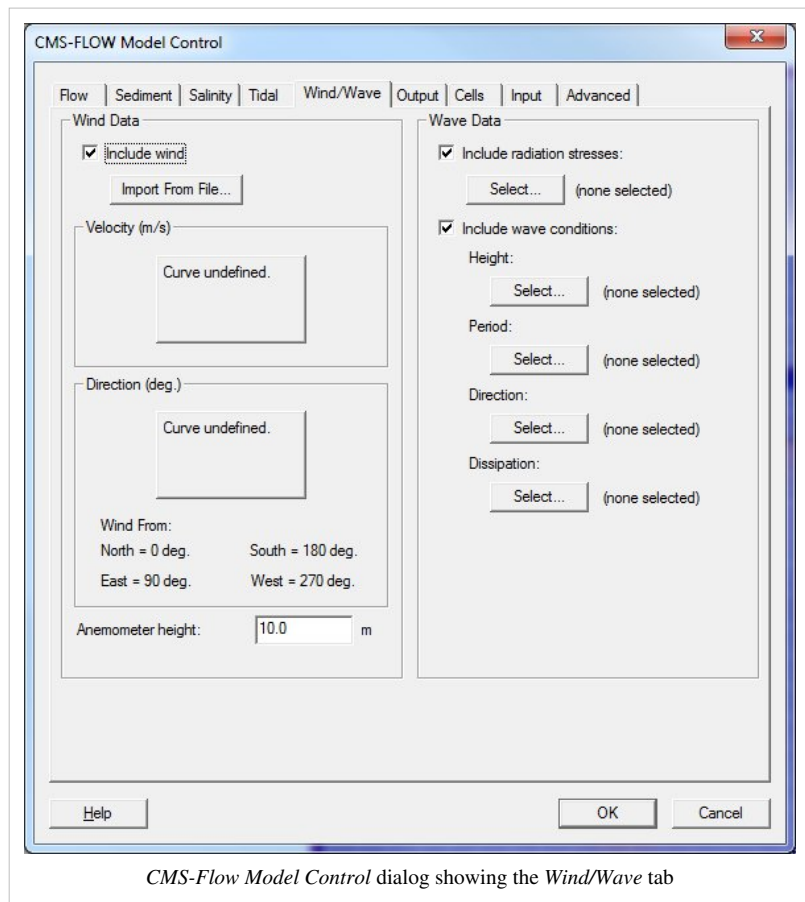
Wind/Wave

This dialog allows the user to specify wind conditions. These are optional inputs to the engine.

Wind Data

The user can turn on the option to tell CMS to include wind calculations in the simulation. Wind is simulated in CMS-Flow as a spatially constant, but temporally varying quantity. The model does not currently support tropical cyclonic winds.

- Spatially constant
 - Import from File...: Button allowing the user to import wind data from a file. The file must be in an "m2w" wind format.
 - Velocity (m/s): Button which invokes a time series editor to specify the variation of wind velocity through time in the simulation.
 - Direction (deg): Button which invokes a time series editor to specify the variation in direction of the wind through time in the simulation.
 - Anemometer height (m)
- Meteorologic stations – This option allows the user to insert data for multiple locations
 - Import from File...: Button allowing the user to import wind data from a file. The file must be in an "m2w" wind format.
 - Coordinates (X,Y): Location of meteorologic station
 - Velocity (m/s): Button which invokes a time series editor to specify the variation of wind velocity through time in the simulation.
 - Direction (deg): Button which invokes a time series editor to specify the variation in direction of the wind through time in the simulation.
 - Anemometer height (m)
- Temporally and spatially varying from file



CMS-Flow Model Control dialog showing the Wind/Wave tab

- Navy fleet numeric with pressure
- OWI/PBL
- Single ASCII file

Navy fleet and ASCII files require the following input:

- Number of X values
- Number of Y values
- Minimum X location
- Maximum Y location
- Time increment
- Distance between X values
- Distance between Y values

OWI/PBL requires three different files:

- Oceanweather Wind File
- Oceanweather Pressure File
- Oceanweather XY File

For more wind file information, see: [http:// cirp. usace. army. mil/ wiki/ CMS-Flow_Wind_Forcing#Spatially_Variable_Wind_and_Atmospheric_Pressure](http://cirp.usace.army.mil/wiki/CMS-Flow_Wind_Forcing#Spatially_Variable_Wind_and_Atmospheric_Pressure) ^[2].

Wave Data

CMS-Flow includes three options on how to use wave data.

- None – no wave data is imputed into the model.
- Single wave condition – the effects of waves are input to the model in the form of spatially varying datasets. These datasets must already exist on the grid. The user simply selects which dataset to use
 - Height
 - Period
 - Direction
 - Dissipation
 - Stress gradients (vector dataset)
 - Surface roller stress gradients (optional)
- Inline steering
 - CMS-Wave file. Location of the grid file or the simulation file
 - Steering Interval or automatic value
 - Specify extrapolation distances. If checked this will write out FLOW_EXTRAPOLATION_DISTANCE and WAVE_EXTRAPOLATION_DISTANCE cards.
 - Wave Water Level – Option to select Tidal, Last, or Tidal plus variation (default value).

Output

This dialog allows the user to specify global output options for the simulation. This controls the datasets created by the engine which consist of spatially and temporally varied quantities (values per cell at each output time value).

These output datasets are stored in an X MDF solution file.

Output Times Lists

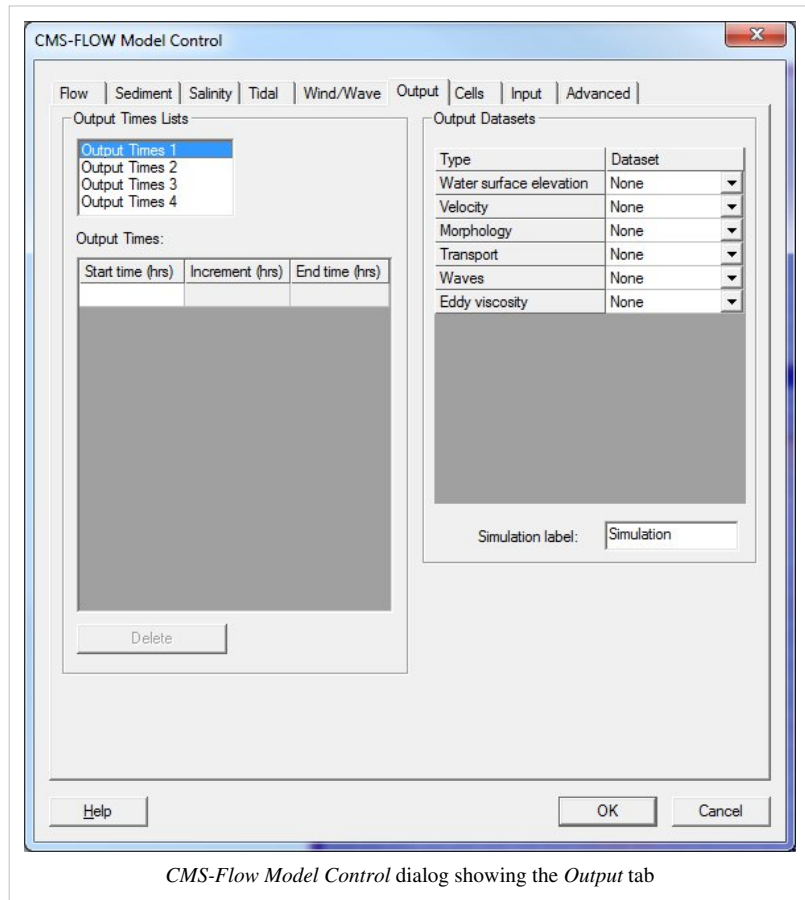
Due to the potentially long simulation times, CMS-Flow does not output at a constant interval during the simulation. Instead, the user defines one or more lists of times. For each dataset output by the engine, the user selects one time list. The selected quantity will be saved at each time in the time list.

- Lists: CMS-Flow supports up to four different and individual sets of time values.
- Output Times: The times included in the selected time list are displayed in this window.
- Type: The user adds and removes times to/from a time list using the tools in the time type section.

Output Datasets

CMS-Flow can output each of the following datasets. For each group, a time list must be specified. The optional datasets may be turned off/on by clicking on their checkboxes in the output dataset tree.

- Water surface elevation
- Velocity
 - Current Magnitude (optional)
 - Current Velocity
- Morphology
 - Depth (through time)
 - Morphology Change (optional)
- Transport
 - Sediment Total-Load Concentration (optional)
 - Sediment Total-Load Capacity (optional)
 - Fraction Suspended (optional)
 - Total Sediment Transport
 - Salinity Concentration (optional)
- Waves
 - Wave Height



CMS-Flow Model Control dialog showing the Output tab

- Wave Period
- Wave Height Vector
- Wave Dissipation (optional)
- Wind
 - Wind Speed (optional)
 - Wind Speed Vector
- Eddy Viscosity

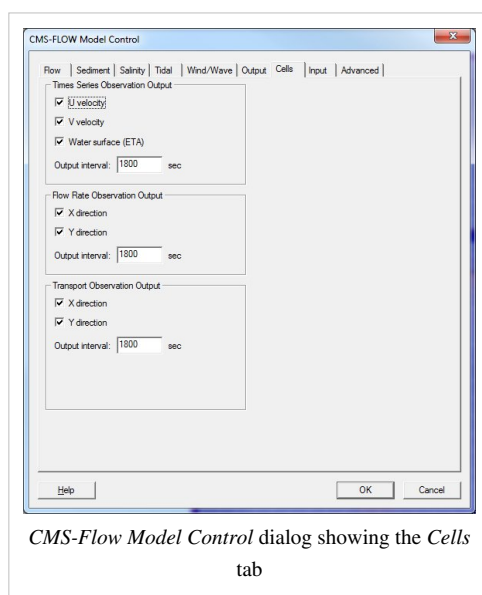
Statistical Output

Statistical data can be written out for hydrodynamics, sediment transport, and salinity. This option is turned on by toggles, and a start time, increment, and end time must be specified.

Output Options

Output can be written out in ASCII format and using XMDf file compression. These options are turned on by toggle boxes. All datasets created by the model will be labeled with a simulation label and quantity label. An edit field allows specification of the simulation label.

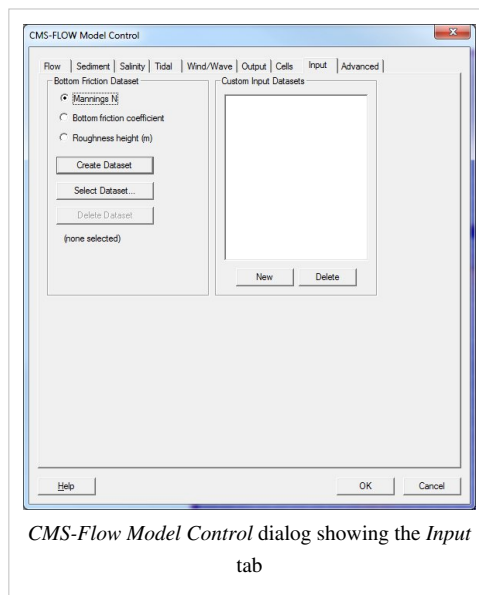
Cells



CMS-Flow Model Control dialog showing the Cells tab

- Time Series Observation Output
 - U velocity
 - V velocity
 - Water surface (ETA)
 - Output interval
- Row Rate Observation Output
 - X direction
 - Y direction
 - Output interval
- Transport Observation Output
 - X direction
 - Y direction
 - Output interval

Input



- Bottom Friction Dataset
 - Manning's n
 - Bottom friction coefficient
 - Roughness height (m)
 - Create Dataset
 - Select Dataset
 - Delete Dataset
- Custom Input Datasets

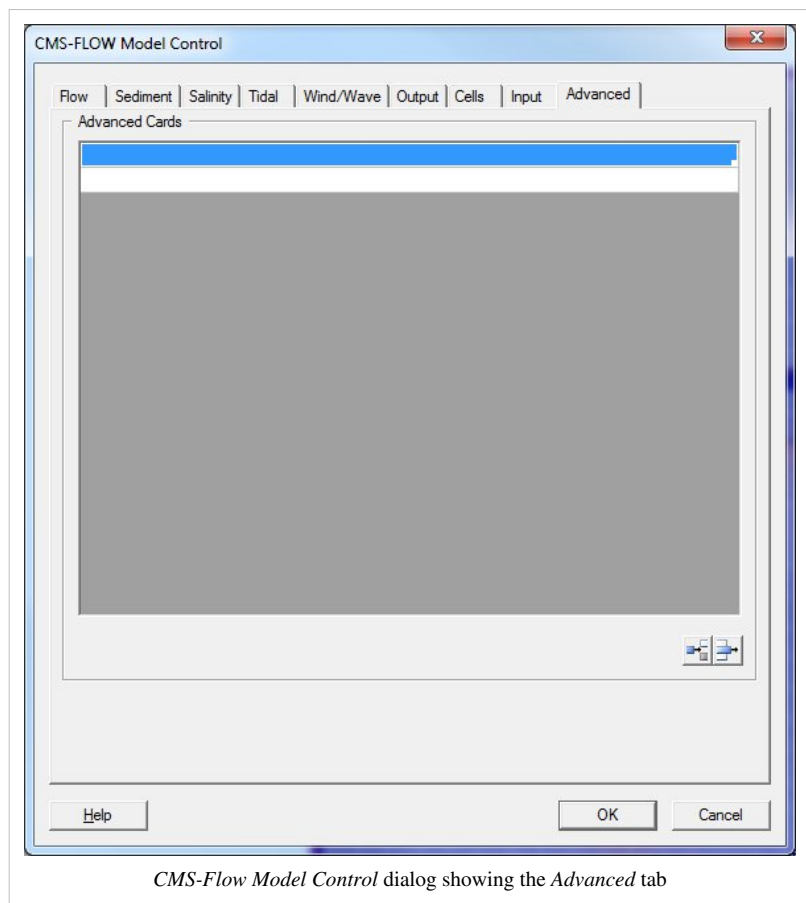
Advanced Options

This dialog allows the user to specify experimental options when working with the model developers.

Advanced Cards

Model options not available in the interface can be specified in the CMS-Flow model control under the advanced tab. Each row in the spreadsheet will be saved as a different line in the *.cmscards file. The cards can be inserted and deleted using the insert and delete rows.

- The lines will be written out exactly as typed.
- The lines will be written out in the order in which they appear in the dialog.
- Any lines encountered in a *.cmscards file that is not recognized will appear in this list.
- Lines beginning with a "!" character are treated as comment lines and will be skipped over during a read.
- Comment lines are automatically generated and will not be preserved.



- All of these advanced cards will be grouped together at the bottom of the file regardless of where they appeared during reading.

See CMS-Flow Advanced Cards

Related Topics

- CMS-Flow

CMS-Flow Observation Cells

CMS-Flow includes observation cells which can be used to output calculations at specific locations. Within SMS, there are two steps to output observation data. First select locations (cells) and specify the cell types as observation stations in the Cell Attributes window. Second, specify what type of data CMS-Flow will output and at what frequency, on the *Cells* tab of *Model Control*.

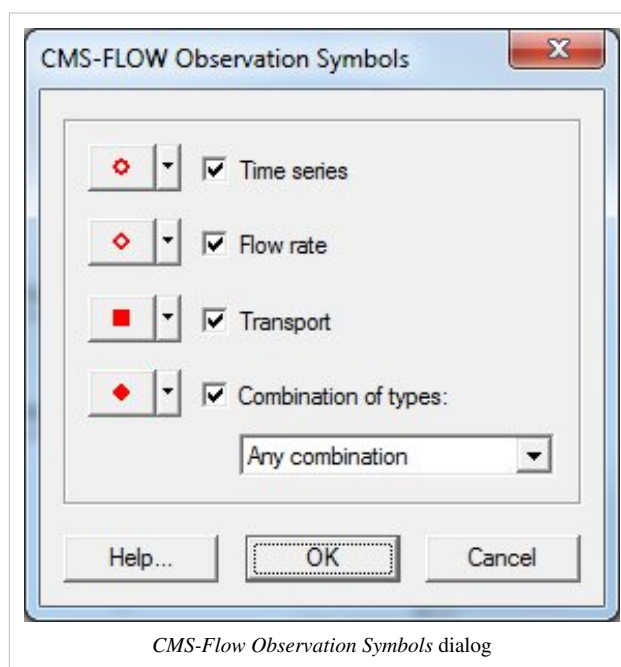
Model Control

Within the *CMS-Flow Model Control* window, the type and frequency of observation output to be written can be specified on the *Cells* tab. For each type (time series, flow rate, and transport), individual calculations can be selected and the *Output interval* will be enable if at least one calculation type is selected. *Water surface (ETA)* is calculated at the center of the cell, while all vectors (U and V, X and Y) are calculated at the center of the left face (for horizontal) and the center of the bottom face (for vertical) of the cell.

Display Options

CMS-Flow includes observation symbols to differentiate specifications. On the Cartesian Grid page of the *Display Options* window (when CMS-Flow is the active model), the *Observation symbols* check box controls the display of symbols that will appear in observation cells. If this is turned on, then the user must be aware of the individual symbol settings accessed by clicking on the **Options...** button. The **Options...** button displays the *CMS-Flow Observation Symbols* window.

Observation symbols can be selected for the three observation output types (Time series, Flow rate, Transport) and a combination of types. The specific output type symbols are displayed in an observation cell that only includes this type or, depending on the state and choice of *Combination of types* option, also includes this type. Since observation cells can include multiple output types, it is possible to display overlapping symbols. The *Combination of types* combo box specifies the qualification for the combination symbol to be displayed in an observation cell. If the combination symbol is displayed in a cell, then no other observation symbol will also be displayed in that cell. The *Combination of types* can be used to quickly highlight specific observation stations as follows:



CMS-Flow Observation Symbols dialog

- **Any combination** – Any observation cell that includes at least any two output types.
- **Time series and flow rate** – All cells that only include both time series and flow rate output types.
- **Time series and transport** – All cells that only include both time series and transport output types.
- **Flow rate and transport** – All cells that only include both flow rate and transport output types.
- **All types** – All cells that include all three output types.

Observation cells of only one type, such as only time series, can be found by turning on the specific type (*Time series*), turning off the other two types (*Flow rate* and *Transport*), and turning on and setting *Combination of types* to *All types*. The time series only observation cells will be highlighted with time series symbol (ignore the combination symbols).

If the *Observation symbols* check box is turned off, no symbols will be displayed and the individual settings cannot be accessed, however, the individual settings will not be changed.

Model Check

The *CMS-Flow Model Checker*, accessed from the *CMS-Flow* | **Model Check...** menu item, includes a check to ensure that CMS-Flow has been set up to output the specific data for any existing observation station and that observation stations exist for any specified output data. Observation is not required to run CMS-Flow so this is a reminder that something may have been intended but forgotten. It is suggested that the model checker be used prior to running CMS-Flow.

Related Topics

- CMS-Flow

CMS-Flow Output Control

Output files specified here are associated with observation cells that have been assigned within the grid. For example, if a time series observation cell exists, an output file will be written out by M2D for every file type that is checked within this dialog. The same holds true for flow rate observation cells. All observation cell output files are given the file extension of “.m2o”. A prefix is specified for all time series and flow rate output files. The user must also specify the time step increment (in seconds) at which to write to the output files. This increment should be a multiple of the simulation time step.

A brief explanation of the information that each of the following observation cell output file types contains is given (* = prefix):

Time Series Output Files:

- U Output (*_u.m2o): Velocity in the x-direction
 - V Output (*_v.m2o): Velocity in the y-direction
 - ETA Output (*_h.m2o): Water level
 - U DETA/DX ($u \frac{\partial}{\partial x}$) (*_udhdx.m2o):
 - V DETA/DY ($v \frac{\partial}{\partial y}$) (*_vdhdy.m2o):
 - ETA DU/DX ($\frac{\partial}{\partial x} u$) (*_hdudx.m2o):
 - ETA DV/DY ($\frac{\partial}{\partial y} v$) (*_hdvdy.m2o):
 - X-Momentum Advection U DU/DX ($u \frac{\partial u}{\partial x}$) (*_xmomu.m2o): U component of the momentum advection term in the x-direction
-

- Y-Momentum Advection $U \frac{DV}{DX}$ ($u \frac{v}{\Delta x}$) (*_ymomu.m2o): U component of the momentum advection term in the y-direction
- X-Momentum Advection $V \frac{DU}{DY}$ ($v \frac{u}{\Delta y}$) (*_xmomv.m2o): V component of the momentum advection term in the x-direction
- Y-Momentum Advection $V \frac{DV}{DY}$ ($v \frac{v}{\Delta y}$) (*_ymomv.m2o): V component of the momentum advection term in the y-direction
- X Bottom Friction (*_xbfric.m2o): X component of the bottom friction
- Y Bottom Friction (*_ybfric.m2o): Y component of the bottom friction
- X Wind Stress (*_xwnd.m2o): X component of the wind stress
- Y Wind Stress (*_ywnd.m2o): Y component of the wind stress

Flow Rate Output Files:

- X Direction (*_qx.m2o): Flow rate in the x-direction
- Y Direction (*_qy.m2o): Flow rate in the y-direction

Related Topics

- CMS-Flow
- CMS-Flow Model Control

CMS-Flow Spatial Datasets

Spatially varied data, or spatial datasets are integral to the use and application of the CMS-Flow numerical engine. The engine uses spatial data for input and each process simulated by the engine creates spatial datasets that can then be visualized and used for post processing in the SMS.

Input Datasets

CMS-Flow has several spatially variable input fields. The inputs are sorted by process and labeled with units below.

Hydrodynamics

Variable	Units	Symbol	Description
Bottom Friction	NA	(none)	Roughness of the bottom used to simulate resistance to flow. For example, this could be based on a Manning n value.

Sediment transport/Morphology change

Variable	Units	Symbol	Description
D50	millimeters	(mm)	Median grain size for particles in each cell of the grid domain
Hard Bottom	meters	(m)	Constraint for maximum erosion for each cell of the grid domain

Salinity

Variable	Units	Symbol	Description
Initial Concentration	parts per thousand	(ppt)	Initial salinity concentration for each cell of the grid domain

Output Datasets

Hydrodynamics

Variable	Units	Symbol	Description
Water Surface Elevation	meters	(m)	Elevation above mean sea-level computed at the centroid of each cell.
Current Velocity	meters per second	(m/sec)	Velocity components in the u and v directions computed at the top and right side of each cell. The SMS interface interpolates these values over the cell, converts them to global world (x, y) values and displays these as vectors.
Flow Rate	cubic meters per second	(m^3/sec)	Flux over each cell.

Sediment transport

Variable	Units	Symbol	Description
Concentration Capacity	kilogram per meter cubed	(kg/m^3)	Total transport capacity for flow in the cell. Computed for LUND-CIRP, van Rijn, and Watanabe transport equations.
Sediment Concentration	kilogram per meter cubed	(kg/m^3)	Computed suspended sediment concentration at the centroid of each cell
Sediment Transport Rate	cubic meters per second per meter	($m^3/sec/m$)	Computed transport rate across the right and top faces of each cell. Displayed as a vector quantity in SMS

Morphology change

Variable	Units	Symbol	Description
Bed depth	meters	(<i>m</i>)	Depth below mean sea level to current floor of the model (positive values measured downwards)
Change in depth	meters	(<i>m/sec</i>)	Change in the depth at each cell since the start of the simulation

Salinity

Variable	Units	Symbol	Description
Salinity Concentration	parts per thousand	(<i>ppt</i>)	Concentration at each cell centroid

Related Topics

- CMS-Flow

CMS-Flow Time Control

The *CMS-Flow Time Control* dialog allows the user to specify the starting day and time for the simulation.

Other options include:

- Start Time – The hour at which to begin the model. This will be the first time step listed in any solution files.
- Simulation Duration – The length of the simulation in hours.
- Ramp Duration – The period of time in days at which to allow the model to “build up” or “ramp up” to a legitimate solution. Aids in the convergence of the model.
- Time Step Size – The time increment at which M2D makes internal calculations. Running the **CMS-Flow Model Check** from the *CMS-Flow* menu will provide an initial estimation for the maximum time step size that will produce a convergent result.

Retlated Topics

- CMS-Flow Model Control
-

CMS-Flow Transport Control

The *CMS-Flow Transport Control* dialog has been combined with the *CMS-Flow Model Control* dialog in SMS 11.0 and later.

The *CMS-Flow Transport Control* dialog allows the user to view and edit the sediment and salinity parameters that affect how CMS-Flow runs and what options are to be included in the current simulation. The dialog includes two tabs which partition the parameters into related groups. The tabs and their related parameters include:

Sediment

Sediment Transport – This tab allows the user to specify model parameters relating to both salinity and sediment transport modeling in CMS-Flow. These are two separate options. The tab also includes controls to allow specification of spatially varied data that impact sediment transport. The controls include:

Time Steps

In addition to the time step specified in the general parameters tab for hydrodynamic calculations, the explicit model also allows the specification of longer time steps for processes that do not vary as rapidly. These include the salinity and sediment transport.

- Transport rate – If either salinity or sediment transport are enabled in this tab, the user must specify a "this time step". It must be larger than the hydrodynamic time step and is generally at least a factor of 20. The default transport rate time step is 60 seconds.
- Morphologic – If sediment transport is enabled for a simulation the user specifies the morphologic time step. This controls how frequently the bottom elevations of the domain are updated in the hydrodynamic calculations. This time step is generally measured in hours because the change in bed elevations do not occur rapidly enough to impact the hydrodynamics.

Sediment

The Sediment section of this tab turns on the calculation of sediment transport and specifies the formula that CMS-Flow will employ for sediment transport.

- Calculate sediment transport
- Use non-equilibrium transport

Calculate Morphology Change During Ramp Period

This can be toggled on or off.

Avalanching

This can be toggled on or off

Hard Bottom

CMS-Flow includes the option to limit sediment transport on a cell by cell basis to simulate the existence of hard bottom. This section of the dialog allows the user to select, or create a dataset to represent these limiting depths.

- Create Dataset – By default, when SMS creates a "hard bottom" dataset, an inactive value (-999) will be assigned for each cell. This value indicates that no limit to scour exists. The user may select cells and assign limiting depths.

- **Select Dataset** – If survey data exists that defines the limiting depths of scour, these can be interpolated to the grid, creating a spatially varied dataset. The select button allows the user to select such a set to use as the hard bottom dataset. SMS copies this dataset to an editable set allowing further modification to these values.

D50

CMS-Flow relies on a spatially varied dataset of median particle size to compute the shear stresses used in sediment transport and morphology calculations. This section of the dialog allows the user to create/select the dataset that defines these particle diameters. The buttons operate similar to the buttons for Hard Bottom described above.

- **Create Dataset** – When creating a dataset, SMS prompts the user for the median particle size that will be assigned as the initial value for all cells.
- **Select Dataset**

Sediment Standard Deviation

- **Create Dataset** – When creating a dataset, SMS prompts the user for the median particle size that will be assigned as the initial value for all cells.
- **Select Dataset**

Salinity

CMS-Flow includes the capability to simulate the diffusion of saline water in fresh water. This section includes controls to enable and initialize these calculations.

- **Calculate salinity** – Turn on the calculation of salinity concentrations in the CMS-Flow simulation.
- **Global concentration** – This control allows the user to specify an initial concentration for the entire domain. Generally, a constant value is chosen, with 0.0 ppt (fresh water) being a commonly used value. Since this is not a realistic condition for salinity studies, the first portion of the simulation is often ignored (as with a hydrodynamic ramp) while the model computes spatially varied salinity concentrations. Hot start files can be used to skip this portion of a simulation for subsequent runs.

External Documentation

- Lund Cirp and Watanabe Formula
 - Two-Dimensional Depth-Averaged Circulation Model CMS-M2D: Version 3.0, Report 2, Sediment Transport and Morphology Change[1]
-

CMS-Flow User-Editable Datasets

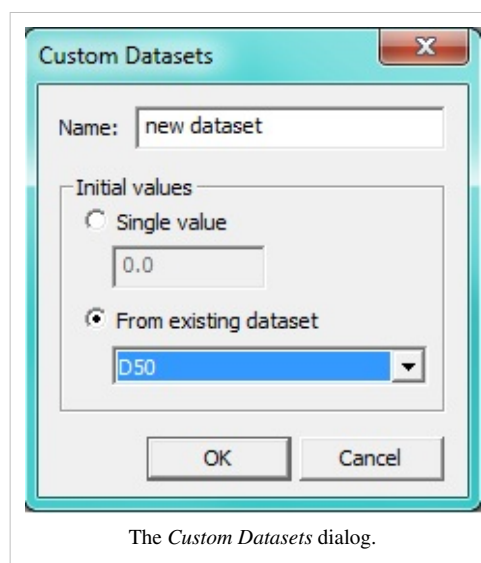
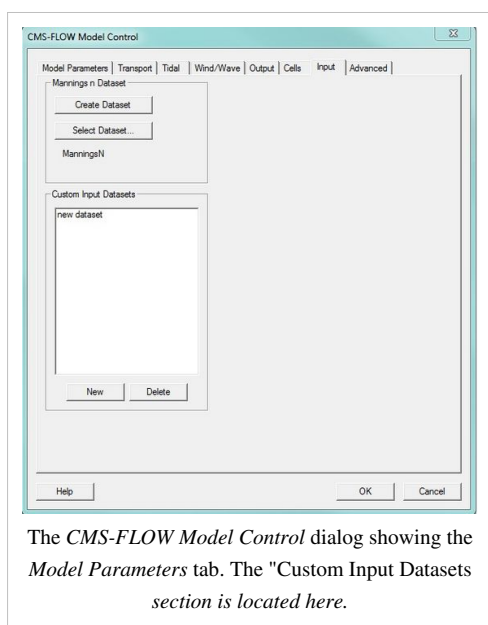
SMS now has the ability to create editable datasets for CMS-Flow simulations. The datasets can be created using a single value or an existing dataset as the initial values.

Editable datasets are managed in the *Custom Input Datasets* portion of the *CMS-Flow Model Control*.

Selecting the **New** button will bring up the *Custom Datasets* dialog. Here the user has the options to use a single value or an existing dataset as initial values. The list of datasets in the combo box contain all of the scalar datasets that already exist in the project.

These datasets are saved to an HDF5 file and referenced in the cms_cards file. a new boundary condition, cross-shore, was added. This condition allows water to flow in either direction over the boundary.

Note: This is primarily a developer tool.



Related Topics

- CMS-Flow

Lund Cirp and Watanabe Formula

The Lund Cirp and Watanabe formula can be found on page 16 of the Two-Dimensional Depth-Averaged Circulation Model CMS-M2D: Version 3.0, Report 2, Sediment Transport and Morphology Change TR^[1].

Transport Slope Coefficient

The Transport Slope Coefficient can be found on page 32 of the Two-Dimensional Depth-Averaged Circulation Model CMS-M2D: Version 3.0, Report 2, Sediment Transport and Morphology Change TR^[2]. D_s = empirical slope coefficient with typical range of 5 to 30.

The Transport Slope Coefficient can vary site by site and even within a single site domain in that some areas have constraints with naturally occurring steep bed slopes (e.g., channels) and other areas have gentle slopes (e.g. beach profiles, or tidal flats). It is a diffusion coefficient for increasing downhill transport or decreasing uphill transport (if D is >1) This is a good parameter to use as a morphology change calibration factor (along with the scalesus and scaled coefficients). One thing to note is that what may calibrate well for one area will not calibrate well for another so an average value may be necessary.

Additional Information

Karambas, T.V. 2003. "Nonlinear wave modeling and sediment transport in the surf and swash zone," Advances in Coastal Modeling, V.C. Lakhan (ed.), Elsevier Oceanography Series, 67, Amsterdam, The Netherlands, 267-298

Larson, M., Hanson, H., and Kraus, N.C. 2003. "Numerical modeling of beach topography change," Advances in Coastal Modeling, V.C. Lakhan (ed.), Elsevier Oceanography Series, 67, Amsterdam, The Netherlands, 337-365

Chapter 4 of

Horikawa, K. 1988. (ed.) "Nearshore dynamics and coastal processes. Theory, measurement, and predictive models," University of Tokyo Press, Tokyo, Japan

Related Topics

- CMS-Flow
- Q&A CMS-Flow

References

- [1] Aug. 2006 - Two-Dimensional Depth-Averaged Circulation Model CMS-M2D: Version 3.0, Report 2, Sediment Transport and Morphology Change TR (<http://cirp.usace.army.mil/Downloads/PDF/TR-06-9.pdf>)
 - [2] Aug. 2006 - Two-Dimensional Depth-Averaged Circulation Model CMS-M2D: Version 3.0, Report 2, Sediment Transport and Morphology Change TR (<http://cirp.usace.army.mil/Downloads/PDF/TR-06-9.pdf>)
-

CMS-Flow Advanced Cards

CMS-Flow Projection Cards

Below are two examples of CMS Flow Projection Cards:

HORIZONTAL_PROJECTION_BEGIN	!Optional	
DATUM	NAD83	!NAD27 NAD83 LOCAL
SYSTEM	UTM	!UTM STATE_PLANE GEOGRAPHIC LOCAL
UNITS	METERS	!METERS FEET DEGREES
ZONE	15	!Only if necessary
HORIZONTAL_PROJECTION_END		
VERTICAL_PROJECTION_BEGIN		
DATUM	LOCAL	!NGVD29 NAVD88 LOCAL
UNITS	METERS	!METERS FEET
OFFSET	2.0 m	!Positive is upwards
VERTICAL_PROJECTION_END		

Related Topics

- CMS-Flow

M2D

M2D support in the SMS interface ended with SMS version 9.2. This was because M2D became the hydrodynamic/sediment transport model in the Coastal Modeling System (CMS). To reflect the involvement in the "system", the name of the model was changed from M2D to CMS-Flow.

The Coastal Modeling System (CMS) is a system of interacting wave, circulation, sediment transport, and bottom-morphology change models centered around a two-dimensional (2D), depth-averaged finite difference circulation model. The CMS was designed for application at coastal inlets where the tide, wind, rivers, and waves may force water and sediment motion. This project-level model is easy to set up and runs quickly as compared to other comparable models. A 3D option is now under testing.

Please see the CMS-Flow wiki page for more information about this model within SMS 10.0+

Telescoping Grids

CMS-Flow can use a grid with varying resolution which is referred to as a telescoping grid. Telescoping grids have a base grid with a specified cell size. The cells in the child grid of the base grid have half the cell size (half in both I and J directions). This means each cell is divided into 4 cells.

Telescoping grids are created by creating polygons where you want the different levels of refinement in a CMS-Flow coverage. Each polygon is assigned the maximum allowable size for cells in this area. When you do **Map→2D Grid**, you specify the size of the base grid. SMS will build a telescoping grid with as many levels as required to satisfy the constraints imposed by the polygons.

Note: Telescoping grids in SMS 11.0 are introduced with limited capability and no support. There are known issues such as problems with shading that will not be addressed in SMS 11.0.

5.5.b. CMS-Wave

CMS-Wave

CMS-Wave	
Model Info	
Model type	Designed for accurate and reliable representation of wave processes affecting operation and maintenance of coastal inlet structures in navigation projects as well as in risk and reliability assessment of shipping in inlets and harbors.
Developer	Lihwa Lin, Ph.D. ^[1]
Web site	http://cirp.usace.army.mil/
Tutorials	General Section • Data Visualization • Observation Models Section • CMS - CMS-Wave (pending)

CMS-Wave (formerly known as WABED) is one of the principal components of the Coastal Modeling System (CMS). WABED stood for "Wave-Action Balance Equation Diffraction" model. The model is a 2-D wave spectral transformation (phase-averaged). The term "phase-averaged" means that it neglects changes in the wave phase in calculating wave and other nearshore processes. This class of wave models represent changes that occur only in the wave energy density. It was originally build to represent theoretically developed approximations for both wave diffraction and reflection in a nearshore domain.

The SMS interface to CMS-Wave includes tools for creating input files as well as post-processing capabilities.

Graphical Interface

The CMS-Wave Graphical Interface contains tools to create and edit a CMS-Wave simulation. The simulation consists of a geometric definition of the model domain (the grid) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the Cartesian Grid Module and setting the current model to CMS-Wave. If a grid has already been created for a CMS-Wave simulation or an existing simulation read, the grid object will exist in the Project Explorer and selecting that object will make the Cartesian grid module active and set the model to CMS-Wave. See Creating 2D Cartesian Grids for more information.

The interface consists of the Cartesian grid menus and tools augmented by the CMS-Wave Menu. See CMS-Wave Graphical Interface for more information.

Model Data Files

Four input files are required for a CMS-Wave simulation. Up to six optional input files may also be utilized depending on the processes being modeled and the selected model parameters. Depending on which options are selected for the simulation, CMS-Wave will generate from one to six output files. See CMS-Wave Files for more information.

Sample Simulations

Simulations illustrating the model capabilities will be posted here soon.

Model Development History

- July 2006 - Initial CHETN announcing the release of WABED is published. Interface is available through the STWAVE interface in SMS.
- December 2007 – Four new features were recently added to CMS-Wave. These include:
 - wave run-up – The wave run-up calculation includes both wave setup and maximum vertical swash that enter, for example, in beach erosion during storms.
 - wave transmission and overtopping at structures – The calculation of wave transmission and overtopping is possible for either vertical wall or rubble- mound jetties and breakwaters, and for submerged reefs. These wave run-up and wave overtopping structure calculations are necessary, for example, in the study of overwash and flanking of beaches adjacent to jetties.
 - card-format for convenient model control
 - option for running in a Fast Mode – With the new *Fast-Mode* option, CMS-Wave calculates the spectral transformation on five directional bins (each 30-deg angle for a broad-band input spectrum) or on seven directional bins (each 5-deg angle for a narrow-band input spectrum, or on 25- deg angle for wind input) to minimize execution time. In Fast Mode, the wave model is at least five times faster than in the Normal Mode, which operates on 35 directional bins in the half plane. Fast Mode is recommended for users who need rapid calculation in reconnaissance or test applications. Wave information estimated in the Fast Mode is expected to be less accurate than in the Normal Mode because the calculation is based on fewer directional bins. Normal Mode should be specified for final runs.

Related Topics:

- SMS Models Page
- CMS-Flow

External Links:

- August 2008 ERDC/CHL TR-08-13 CMS-Wave: A Nearshore Spectral Wave Processes Model for Coastal Inlets and Navigation Projects [2]
- May 2007 ERDC/CHL CHETN-I-74 WABED Model in the SMS: Part 2. Graphical Interface [3]
- Jul 2006 ERDC/CHL CHETN-III-73 Wave-Action Balance Equation Diffraction (WABED) Model: Tests of Wave Diffraction and Reflection at Inlets [4]
- Jul 2007 ERDC/CHL CHETN-IV-69 Tips for Developing Bathymetry Grids for Coastal Modeling System Applications [4]
- Feb 2006 ERDC/CHL CHETN-IV-67 Frequently-Asked Questions (FAQs) About Coastal Inlets and U.S. Army Corps of Engineers' Coastal Inlets Research Program (CIRP) [5] Updated FAQ Website [6]

- Sep 2008 Modeling of Morphologic Changes Caused by Inlet Management Strategies at Big Sarasota Pass, Florida [3]

CMS-Wave Cell Attributes Dialog

The *CMS-Wave Cell Attributes* dialog is used to assign a cell type to selected Cells. It is opened using the menu command *CMS-Wave | Assign Cell Attributes*. The following cell types can be assigned:

Default

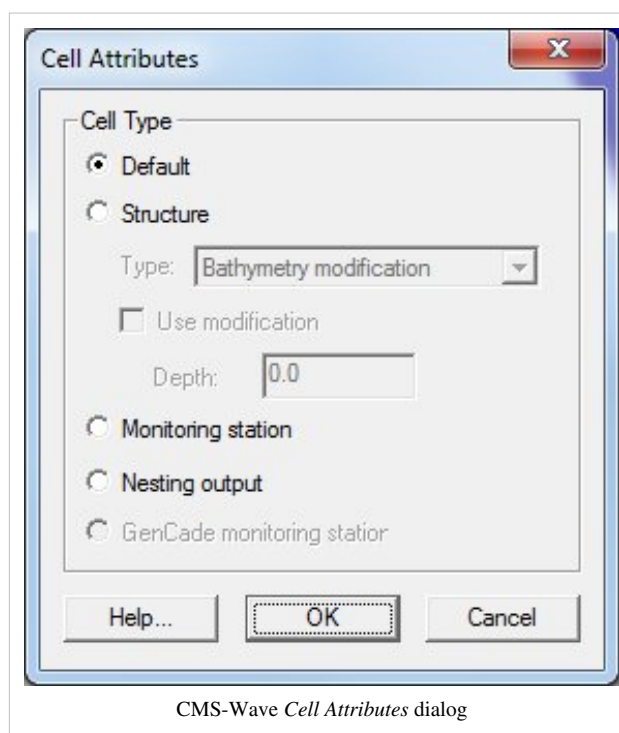
The elevation values of these cells will be used in order to determine whether flow exists in each cell or not.

Structure

The cell attributes dialog of CMS-Wave supports various structure types. Each type of structure can specify an attribute value. The following structure types are supported:

- **Bathymetry modification** – for adding alternative feature or structure (immersed or exposed) without modifying the input depth
- **Wave runup** – for calculation of wave runup and overwash on beach face or structure, and adjacent land
- **Floating breakwater** – for calculation of transmitted waves of a floating breakwater
- **Wall breakwater** – for a vertical wall breakwater
- **Rubble-mound** – for a composite or rubble-mound breakwater
- **Piers/docks** – High permeability; for a pier or dock
- **Rubble-mound breakwater** – Low permeability; for a rubble-mound breakwater

Each of the structures can provide an optional attribute value. The value specified is as follows:



CMS-Wave Cell Attributes dialog

CMS-Wave Structure Attributes

Structure Type	Attribute Name	Description	If Not Provided
Bathymetry modification	Depth	Feature structure depth	Assumed land if not provided
Wave runup	Elevation	Beach/structure elevation above mean water level (no effect if < 0.0)	Assumed input depth
Floating breakwater	Draft	Floating breakwater draft (no effect if < 0.05 m)	No effect
Wall breakwater	Elevation	Beach/structure elevation above mean water level (immersed if < 0.0)	Assumed input depth
Rubble-mound	Elevation	Beach/structure elevation above mean water level (immersed if < 0.0)	Assumed input depth
Piers/docks – High permeability	Porous layer thickness	See permeable structures below.	

Rubble-mound breakwater – Low permeability	Porous layer thickness	See permeable structures below.	

Permeable Structures

CMS-Wave treats cells designated as "Piers/Docks" and "Rubble Mound Breakwater" as if they have a structure that fills the entire water column. If the water level rises, the assumption is made that the structure is still all the way to the water surface (and beyond if needed). CMS-Wave ignores the specified depth for these cells when the structure type is specified. The layer thickness defines the thickness that will convey wave energy. Physically, this is like the distance from the mean sea level down to the footing of the pier group or dock structure. For a rubble mound, it is the distance below mean sea level where the mound sits on solid material (non-permeable).

Monitoring Station

Spectral output is generated for cells designated as monitoring station cells.

Nesting output

Spectral output is generated for nesting output cells to be used as input for a nested child grid.

Displaying Cell Attributes

The symbols and colors used to identify the attributes assigned to cells can be changed in the STWAVE Display Options.

Related Topics

- CMS-Wave
- STWAVE Menu

CMS-Wave Graphical Interface

The CMS-Wave graphical interface includes tools to assist with creating, editing, and debugging a CMS-Wave model. The CMS-Wave interface exists in the Cartesian Grid Module.

Model Control

The *CMS-Wave Model Control* dialog is used to setup the options that apply to the simulation as a whole. These options include time controls, run types, output options, global parameters, print options and other global settings.

Running the Model

The CMS-Wave files are written automatically with the SMS project file or can be saved separately using the *File | Save CMS-Wave* or *Save As* menu commands. See CMS-Wave Files for more information on the files used for the CMS-Wave run.

CMS-Wave can be launched from SMS using the *CMS-Wave | Run CMS-Wave* menu command. A check of some of the common problems, called the Model Checker, is done each time the model is launched, or by selecting the *CMS-Wave | Model Check* menu command.

Visualizing Results

Select the spectral grid as the grid to use in the *Spectral Energy* dialog. This will open the spectral grid for viewing in the *Spectral Energy* dialog. Make sure to set the original grid back before leaving the dialog to ensure the model does not change.

CMS-Wave Menu

See CMS-Wave Menu for more information.

Related Topics

- CMS-Wave
- Create a 2D Cartesian grid

CMS-Wave Menu

The following menu commands are available in the CMS-Wave Menu:

- **Assign Cell Attributes**
Opens the CMS-Wave Cell Attributes Dialog, allowing the user to specify the cell type.
- **Nest Grid**
Opens the CMS-Wave Nesting Options Dialog, allowing the user to specify nesting options.
- **Merge Cells**
Merge multiple rows or columns into a single row or column.
- **Model Check...**
Check for common problems.
- **Model Control...**
Opens the CMS-Wave Model Control dialog to specify model parameters.
- **Run CMS-Wave**
Launches the CMS-Wave model using the currently loaded simulation. As the model runs, a dialog monitors progress of the model and gives the user status messages. When the run is complete, the spatial solutions are read in for analysis and visualization.

Obsolete Commands

The following commands are no longer available in the current version of SMS.

- **Spectral Energy**
Opens the *Spectral Energy* dialog to define/view wave energy spectra. Generally, CMS-Wave will generate wave conditions internally, but a spectrum may be input. This command also allows the user to visualize wave spectra that are generated inside of the model.

Related Topics

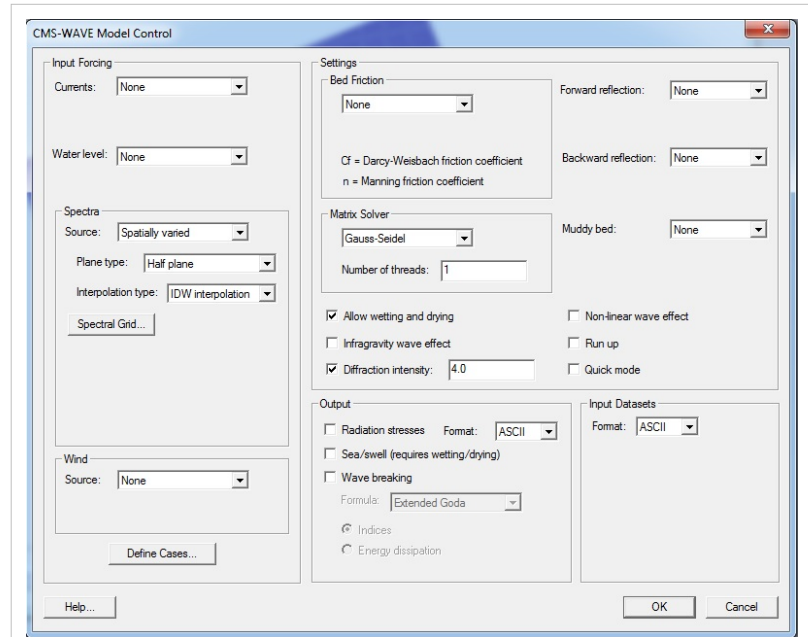
- CMS-Wave

CMS-Wave Model Control

The **Model Control...** command in the *CMS-Wave* Menu opens the *CMS-Wave Model Control* dialog. This dialog is divided into sections for different types of parameters which are used by the model as it runs. These include:

Input Forcing

- **Currents** – A vector dataset can be used to define the currents for the simulation
- **Water Level** – A constant value or a scalar dataset can be used to define the water level for the simulation. If “constant” is selected, the constant value will be defined in the *CMS-Wave Case Definition* dialog for each case.
- **Spectra**
 - **Source:**
 - **Spatially Varied** – this option uses a spectral coverage with spectral data defined at one or more locations. The spectral coverage is assigned to the model in the *CMS-Wave Case Definition* dialog.
 - **Parent Grid** – Nesting output from a parent grid will be used to drive the simulation. The parent grid is specified by clicking on the **Select...** button and selecting the desired grid. The spectral grid to be used for the simulation will match the spectral grid for the parent simulation.
 - **Plane Type** – this option is only used if the *Spatially Varied* source option is selected.
 - Half plane
 - Full plane
 - Full plane with input reverse spectra – this option allows spectral input to be used on two opposite boundaries.
 - **Interpolation Type** – sets the method for STWAVE to use when interpolating between spectra
 - Average spectra
 - IDW interpolation
 - **Spectral Grid**
 - **Create Nesting Points** – if the *From parent grid* source option is selected and a parent grid has been specified, clicking on this button will allow the user to specify which points should be used from the parent grid for creating the nesting file
- **Wind** – a constant value or a vector dataset can be used to define the wind for the simulation. If constant is selected, the wind direction and magnitude will be specified in the *CMS-Wave Case Definition* dialog for each case.
- **Define Cases**



CMS-Wave Model Control dialog

Settings

The settings section is used to specify model input options including:

- Bed friction
- Matrix Solver
 - Gauss-Seidel
 - ADI
- Allow wetting and drying
- Infragravity wave effect
- Diffraction intensity
- Forward reflection – A constant value for the entire simulation or spatially varying data using a scalar dataset can be used to define the forward reflection for the model.
- Backward reflection – A constant value for the entire simulation or spatially varying data using a scalar dataset can be used to define the backward reflection for the model.
- Muddy bed – Spatially varying data using a scalar dataset can be used to define the muddy bed (values??) for the model.
- Non-linear wave effect
- Run up
- Quick mode

Output

This section is used to specify option output from the model. The output includes:

- Radiation stresses
- Sea/Swell
- Wave Breaking

Input Datasets

Here the user can specify the format for SMS to write out the input datasets. They can be written in either ASCII or XMDF format.

Related Topics

- CMS-Wave

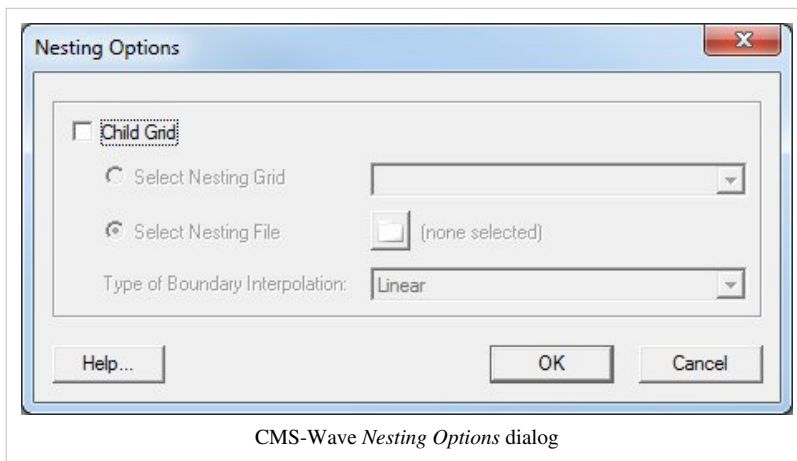
CMS-Wave Nesting Options Dialog

The *Nesting Options* dialog can be reached by using the **Nest Grid** command in the *CMS-Wave* menu. The dialog has the following options.

- Child Grid
 - Select Nesting Grid
 - Select Nesting File
 - Type of Boundary Interpolation

Related Topics

- CMS-Wave



5.5.b.1. CMS-WAVE Files

CMS-Wave Files

A CMS-Wave simulation consists of:

- A Cartesian grid which defines the bathymetric depths over the model domain
- A set of spectra that represent the input wave conditions to be modeled
- Model parameters controlling what options should be used in this simulation
- Optional spatially varied input fields over the same domain such as wind datasets, surge (water level) datasets, and current datasets

SMS stores all of this information internally in its own formats. This includes:

- An XMDF (HDF5) file to store the bathymetry and other datasets on the grid
- An XMDF (HDF5) file to store the spectral grids and wave states entering the domain from the boundary
- Model parameters in the project (sms) file.

In order to run the numeric model, the data must be exported into model native file formats. SMS exports these files when instructed to do so from either the *CMS-Wave* menu or by right-clicking on the CMS-Wave grid. SMS includes three commands in either location. These include:

1. **Export CMS-Wave Files** – this command creates the native files described below for the simulation. If the data has not been previously saved to establish the name of the simulation, SMS prompts the user for a simulation name.
2. **Launch CMS-Wave** – this command uses the saved files to invoke the simulation
3. **Save project, export and launch CMS-Wave** – this command, as described saves the SMS project and then replicates the functionality of the other two commands.

The model native files are referenced by a simulation file, which includes all the names of the other files used as input for a simulation, or that will be created by CMS-Wave when it runs.

SMS stores a simulation name associated each CMS-Wave grid loaded into a session. If a new grid is created, the name is blank and SMS will prompt the user for a name when it is needed.

If the **Launch CMS-Wave** command is issued without having first saved the model native files, SMS will bring up a message indicating that the files must first be saved.

To change the name of the simulation associated with a grid, the user can select the *File|Save as...* command and select the *CMS-Wave simulation* option. SMS will then prompt the user for the new simulation name to be associated with the active CMS-Wave grid. We recommend that this option be utilized only after duplicating the CMS-Wave grid, since the link to the old simulation files is lost when the new simulation name is specified.

Input Files:

- Required files
 - *projname.sim*
 - *projname.std*
 - *projname.dep*
 - *projname.eng*
- Optional files
 - *projname.cur*
 - *projname.eta* (uses the same format as the *.dep file)
 - *projname.struct*
 - *projname.fric* (uses the same format as the *.dep file)
 - *projname.fref* (uses the same format as the *.dep file)
 - *projname.bref* (uses the same format as the *.dep file)
- Auxilliary files
 - *projname.txt*

Output Files

- Spatially varied data (value for each cell)
 - *projname.wav*
 - *projname.brk*
 - *projname.rad*
- Full spectra at selected cells (have the same format as the *.eng file)
 - *projname.obs*
 - *projname.nst*
- Additional files
 - *selhts.out* – text file of wave height, period and direction at specified observation locations.
 - *setup.wave* – text file of wave runup/setup at specified structure locations.

Related Topics

- CMS-Wave

CMS-Wave Control File

CMS-Wave parameter files have been reworked for SMS 11.1 and the associated version of CMS.

File Format

The new CMS-Wave parameters file (also referred to as an options or std file) begins with an identifier line. This line includes the text CMS_WAVE_STD and a version number.

Each subsequent line on the file includes a value, followed by a comment defining the option. The design was intended to replicate a card based ability to omit lines if the default value is to be used.

Sample File

```
CMS_WAVE_STD 1 !free format

1 !iprp
0 !icur
0 !ibreak
0 !irs
0 !kout

0 !ibnd
0 !iwet
0 !ibf
0 !iark
0 !iarkr

0 !iwvbk
0 !nonlin
0 !igrav
0 !irunup
1 !imud
1 !iwnd

4.0000 !akap
0.025 !bf
0.5000 !ark
0.3000 !arkr

0 !isolve
0 !ixmdf
1 !iproc
0 !iview
0 !inest
```

Old File Format

The old parameter file always contain a single line of data defining the parameter options for the simulation. This line will include either 6, 10 or 15 values. SMS writes all 15 values to the input files it creates. The other formats are supported for backward compatibility.

If only the first six values are present, the other nine values will be assigned default values. The default for the diffraction intensity factor is weak diffraction ($akap = 1.0$). The default for the other eight parameters is 0 (defined in table below).

If ten values exist in the file, bottom reflection is defaulted to off, forward reflection is defaulted to 50% and reverse reflection is defaulted to 30%.

After this single line, additional lines may be present to define the location of observation and nesting cells. CMS-Wave will save additional output (spectra) at these cells into the specified *.obs and *.nst files.

A sample file and table of the parameters is defined below.

```
iprp icur ibrk irs kout ibnd iwet ibf iark iarkr akap bf arc arkr iwvbk
kout rows (each row includes the I and J indices of a selected output cell
nestout
nestout rows (each row includes the I and J indices of selected nesting output cells
```

Sample Old File

```
1 0 0 0 2 0 0 0 0 4.000000 0.005000 0.500000 0.300000 0
10 15
23 41
4
28 7
27 8
28 9
29 10
```

Parameter definitions

Parameter	Description
iprp	-1, for fast-mode simulation
	0, for wave generation and propagation (uses wind input and spectra)
	1, for propagation only (neglect wind input)
	2, for wind only (implies a zero energy spectra)
icur	0, no current file read
	1, read current file with a dataset for each input spectra
	2, read single data set from current file for all spectra

ibrk	0, no breaking file will be created
	1, breaking file will be created with breaking index for each cell
	2, breaking file will be created with energy dissipation flux saved for each cell
irs	0, no radiation stress gradient file will be created
	1, radiation stress gradient file will be created
	2, radiation stress gradient file and wave setup/maximum water level files will be created
kout	0, no selected output data files will be created
	n, output spectra (.obs) and parameters (selhts.out) files will be created at n specified cells
ibnd	0, simulation will use single spectra (*.eng) file along offshore boundary
	1, simulation will use nesting file (.nst) with linear interpolation of boundary input
	2, simulation will use nesting file (.nst) with morphic interpolation of boundary input
iwet	0, normal wetting/drying (based on water level input)
	1, no wetting/drying (water level input ignored)
ibf	0, no bottom friction
	1, for bottom friction with constant Darcy-Weisbach type coefficient (bf)
	2, for bottom friction with variable Darcy-Weisbach type coefficient (friction.dat)
	3, for bottom friction with constant Manning coefficient (bf)
	4, for bottom friction with variable Manning coefficient (friction.dat)
iark	0, no forward reflection
	1, with forward reflection
iarkr	0, no backward reflection
	1, with backward reflection
akap	diffraction intensity factor (0 for no diffraction, 4 for strong diffraction)
bf	constant bottom friction coefficient
ark	constant forward reflection coefficient (0 for no reflection, 1 for maximum forward reflection)
arkr	constant backward reflection coefficient (0 for no reflection, 1 for maximum backward reflection)
iwvbk	0, for extended Goda wave breaking (Sakai et al. 1989)
	1, for extended Miche wave breaking (Battjes 1972; Mase et al. 2005b)
	2, for Battjes and Janssen wave breaking (1978)
	3, for Chawla and Kirby wave breaking (2002)

Related Topics

- CMS-Wave Files
- File Formats

CMS-Wave Depth File

CMS-Wave requires a depth at each cell in the domain. In SMS terminology, the depth values comprise a dataset on the grid.

The depths are passed into CMS-Wave in the form of an ASCII data file. There is no header or identifier as to what the data in the file is, other than the default extension of "*.dep". However, that is not even required.

The first line of the file contains the dimensions of the grid (number of columns, number of rows) and the default size of a grid cell (in the I direction and then the J direction). If the grid does not consist of cells of constant size, the size for the J direction will be written as 999. In this case, the cell dimensions (width of each column and height of each row) are included at the bottom of the depth file.

The values in the file are organized in row major format, starting with the "top" row (farthest away from the grid origin) and ending with the "bottom" row (closest to the origin and following the I direction).

SMS writes files with five values per row.

Sample File

```
25 12 10.000 999.000      // number of columns, number of rows, cell dimensions
46.65 42.56 38.85 35.47 32.40 // first 5 depths of cells on the top row
29.61 27.07 24.77 22.67 20.76 // next 5 values, second row will continue on same line
...                      // remaining depth values
7.97 7.25 6.55 5.99 5.44   // first five column widths
4.95 4.50 4.09 3.72 3.38   // next 5 widths, heights will follow on same line
...                      // remaining widths and heights
```

Related Topics

- CMS-Wave Files
- File Formats

CMS-Wave Simulation File

CMS-Wave simulation files contain a list of file names that are part of the simulation. This file is created by SMS when the simulation is exported. The file begins with a keyword that identifies this as a CMS-Wave simulation file, followed by the world origin and orientation of the grid.

The origin positions the grid in a projection (such as UTM or State Plane). It generally corresponds to the location of the corner of the grid offshore. The orientation is the CCW angle from East to the direction of the grid that goes from offshore to on shore. (For example, a coast that opens due West would have an orientation of 0.0, an coast line open to the South would have an orientation around 270 (or -90) degrees.)

Each line begins with a key word identifying the file name. The order can be changed and only the DEP, OPTS and WAVE files are strictly required. The OPTS file includes the options that have been enabled for this simulation. If the OPTS file indicates that a file is needed, the line must appear in the SIM file. If a file is missing, the simulation will not perform as expected. If files are not needed, the presence of the line in the SIM file is ignored. For this reason, SMS writes all files to the SIM file every time.

Sample File

```
CMS-Wave 902500.0      /* Key word, grid origin, grid orientation*/
4086600 135

DEP      grid.dep      /* Name of input file containing depth values for each cell */

OPTS     grid.std      /* Name of input file containing model parameters */

SPEC     grid.eng      /* Name of input file containing energy density spectra (required in most cases) */

OBSE     grid.obs      /* Name of output file to save full spectra at specified monitoring locations */

NEST     grid.nst      /* Name of output file to save full spectra at specified nesting cells */

BREAK    grid.brk      /* Name of output file to save wave breaking indices or energy disipations at each cell. Follows the format
of the DEP file */

SPGEN    grid.txt      /* Name of spectral parameter file, contains parameters used to generate spectra */

RADS     grid.rad      /* Name of output file to save wave radiation stress gradients at each cell. Follows the format of the CUR
file */

STRUCT   grid.struct   /* Name of input file containing structure flags for each cell */

SELHTS   grid.out      /* Name of output file where CMS-Wave will write the bulk wave parameters for selected monitoring cells
*/

ETA       grid.eta      /* Name of optional input file containing water levels for each cell. Follows the format of the DEP file.
This replaces the SUR card. Both are supported */

FRIC     grid.fric      /* Name of input file containing manning's n value for each cell. Follows the format of the DEP file */

FREF     grid.fref      /* Name of input file containing forward reflection value for each cell. Follows the format of the DEP file
*/

BREF     grid.bref      /* Name of input file containing reverse reflection value for each cell. Follows the format of the DEP file */

MUD       grid.mud      /* Name of input file containing mud(absorption) coefficient value for each cell. Follows the format of the
DEP file */

WIND     grid.wind      /* Name of input file containing wind components for each cell. Follows the format of the CUR file */

WAVE     grid.wav      /* Name of output file for spatial wave conditions (height, direction) for each cell */

CURR     grid.cur      /* Name of optional input file containing current values (vx and vy) for each cell */
```

Related Topics

- CMS-Wave Files
- File Formats

CMS-Wave Spectral Energy File

CMS-Wave energy files contain a the definition of a half plane spectral grid (5 degree directional bins and user specified frequency bins) and energy density datasets as boundary conditions for each wave condition to be simulated in a CMS-Wave analysis.

File Format

The first line of the file contains the number of frequency bins and directional bins. Typically, 20-30 frequency bins are used. The model requires 5 degree directional bins, so the number of directional bins must be set to 35.

Following the grid dimensions, the next lines of the file specify the frequencies for the model spectra, starting at the lowest frequency. There must be the specified number of values. They are read in free format and may occupy as many lines as needed. These frequencies should span the range where significant wave energy is contained in the spectrum. This can be estimated by inspecting the input spectrum or estimating the peak period expected using the wave growth curves in the Coastal Engineering Manual. A rule of thumb is that the spectral peak should fall at about the lower one-third of the frequency range (e.g., if the peak frequency is 0.1 Hz, the range may be 0.01 to 0.3 Hz). Wave frequencies higher (or periods shorter) than the highest frequency bin or lower than the lowest frequency bin will not be resolved by the model. Typically, frequency increments are on the order of 0.01 Hz, but the increment need not be constant. West Coast applications will tend to require finer resolution focused at lower frequencies because of long wave periods. Gulf Coast or Great lakes applications will tend to require coarser resolution covering a broader range of frequencies because of shorter wave periods.

Following the specification of the frequency and direction bins is a header line containing a spectrum identifier, wind information, peak frequency and water elevation correction. This line is read in free format.

After the header line, the file contains the energy densities in the units of meters squared/hertz/radian. The spectrum is read starting with the lowest frequency and reading all the directions (-85 to 85), then reading energy densities for the next lowest frequency etc.

The file can contain multiple spectra.

Sample File

```
30 35
0.04000 0.05000 0.06000 0.07000 0.08000 0.09000 0.10000 0.11000 0.12000 0.13000
0.14000 0.15000 0.16000 0.17000 0.18000 0.19000 0.20000 0.21000 0.22000 0.23000
0.24000 0.25000 0.26000 0.27000 0.28000 0.29000 0.30000 0.31000 0.32000 0.33000
5 27.000000 0.000000 0.070000 0.000000
0.00000 0.00000 0.00000 0.00000 0.00000 0.00002 0.00006 0.00019 0.00048 0.00107 0.00209 0.00365 0.00576
.
.
.
```

Related Topics

- CMS-Wave Files
- File Formats

CMS-Wave Spectral Table File

Spectral Table Files contain the parameters for generated wave spectra. This file is created by SMS when the *Spectral Energy* dialog is used to create wave spectra. If wave spectra are imported from buoys or another source, this file will not exist. When this file exists, SMS can use the values in the table to populate the table in the *Spectral Energy* dialog to allow the user to edit the spectra being used.

SPECTRAL TABLE	Values
Method Option Time Index Angle Hs(1) Tp(1) Gamma (1) Hs (1) Tp(1) Gamma(1) Hs(2) Tp(2) Gamma(2) Wind Fetch nn StdDev Depth	Headers
0 -1 999.0 None 25.0 1.0 20.0 8.0 999.0 999.0 999.0 999.0 999.0 30.0 999.0 0.001	1st row of values
0 -1 999.0 None 30.0 1.0 16.0 8.0 999.0 999.0 999.0 999.0 999.0 30.0 999.0 0.001	2nd row of values

Sample File

```
ACE/vis drogue path file          /* Title */
5                                /* Number of Time Steps */
                                /* Current Time Step and Number of Particles */
7200.0000  199

                                /* xy values and id */
-0.766993322E+02  0.346589454E+02  1
-0.766986001E+02  0.346616775E+02  2

.
.

                                /* Next Time Step and Number of Particles */
8000.0000  199

.
.

/* EOF */
```

The number of particles must be the same for each time step.

Related Topics

- CMS-Wave Files
- File Formats

CMS-Wave STD

CMS-Feature enhancements – June 15, 2010 Support the following mode parameters that appear on the first line of the std file (there are a total of 23 of them)

Sample of first line from *.std: 0 0 0 0 0 0 0 1 0 4.000000 0.005000 0.500000 0.300000 0 0 1 0 0 0 0 0 0

These parameters represent the following variables in the code:

```
iprp icur ibreak irs kout ibnd iwet ibf iark iarkr akap bf ark arkr iwvbk nonln igrav irunup imud iwnd isolv ixmxf iproc
```

The first 15 parameters should already be supported in the SMS interface and are documented at: xmswiki.com/xms/SMS:CMS-Wave_Control_File^[1] The new ones (16th to 23th) and their possible values include:

```
nonln (nonlinear wave-wave effect): 0 = none (default), 1 = present
igrav (infragravity wave effect): 0 = none (default), 1 = present
irunup (run-up): 0 = none (default), 1 = automatic, 2 = field file input (*.runup or runup.dat)
imud (muddy bed): 0 = field file input (*.mud or mud.dat, default), 1 = none
iwnd (wind field input): 0 = field file input (*.wind or wind.dat - same format as *.cur, default), 1 = none
isolv (Matrix solver): 0 = ADI (default), 1 = Gauss-Seidel (allow multiple processors)
ixmxf (ixmxf output format): 0 = ascii (default), 1 = output in xmdf format, 2 = input/output in xmdf
iproc (multiple processor): 0 = one-processor (same as 1, default), n = n processors (only for isolv = 1)
isea (swell and local sea output):
    0 = no additional outputs,
    1 = output additional swell.wav (for swell) and sea.wav (local wave) component files,
        (These have the same format as the total wave field output file *.wav)
    2 = output the total.wav (identical to the default *.wav),
        (This is for the steering run that will merge all individual *swsteer.wav files.)
    3 = output the total.wav, swell.wav and sea.wav.
```

The following new files need to be added in the *.std

```
*.fric    *.fref    *.bref    *.mud,    *.wind    *.runup
```

These are the same as friction.dat, forward.dat, backward.dat, mud.dat, wind.dat and runup.dat

Related Topics

- CMS-Wave Files

CMS-Wave Structure File

CMS-Wave structure files contain a line defining the number of cells that should be treated as structures in the model simulation, and then a list of these cells with special features assigned to them. These are specified in the SMS interface by selecting the cell and assigning cell attributes (CMS-Wave menu or right click on selected cell). These cells can represent:

- a floating breakwater
- a bottom mound breakwater
- a beach segment and the land adjacent to it
- jetties or seawalls
- underwater features such as reefs or submerged structures

A trench, submerged mound or structure can be added to the bed as features without modifying the input depth file. Each feature cell is described by four parameters in a line in the structure file. A sample file and table of the parameters is defined below.

File Format

number of structure cells

one row for each structure cell (each row includes the I and J indices of a selected output cell, the structure type and the modifier value

Sample File

```
4 /* there are 5 structure cells in this sample file */
10 15 1 3 /* cell (10,15) will be modified to have a depth of 3 meters */
23 41 2 0.5 /* model will calculate runup and overwash at cell (23,41). It will be assigned an elevation of 0.5 meters above sea level */
15 7 3 1.5 /* model will treat cell (15,7) as a floating breakwater with a draft of 1.5 meters to calculate transmission of waves under the structure */
16 22 4 3 /* model will treat cell (16,22) as a breakwater/seawall/mound with an elevation of 3 meters above sealevel */
```

Parameter definitions

kstruc	Description	Modifier value (cstruc)
1	add alternative depth without modifying input depth file	specify the new structure depth (assumed land if no depth specified)
2	for calculation of wave runup and overwash on beach face or structure and adjacent land	specify the elevation above mean water level for the cell. Input depth used in no value specified. Has no impact if elevation < 0)
3	for calculation of transmitted wave under a floating breakwater	specify the floating breakwater draft (skipped if not provided or value is < 0.05 m)
4	for a vertical wall breakwater	specify breakwater/structure elevation (input depth value used in not provided; immersed if value < 0.0)
5	for a composite or rubble-mound breakwater	

Related Topics

- [CMS-Wave Files](#)
 - [File Formats](#)
-

5.6. CSHORE

CSHORE

This article is for a model that is under development by Aquaveo and its partners. The model is **not available** to the public at this time. It is not known when or if the model will be released.

CSHORE is a transect morphology model which computes sediment transport along a specified profile/transect for a set of storm conditions.

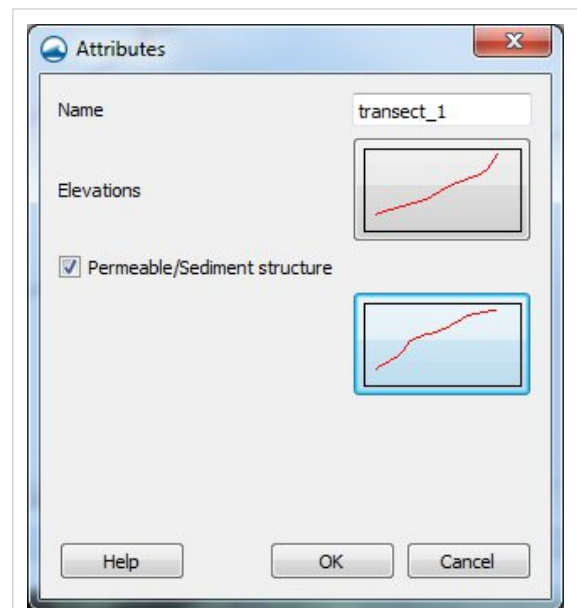
Creating a Simulation with Transect Coverages

Create a new CSHORE Simulation by:

1. Right-click in a blank section of the project explorer.
2. Then select **New Simulation→CSHORE**.
3. This brings up the coverage dialog. Select and name coverages you wish to be created (keep the transect coverage selected). Press **OK**.
4. This creates a simulation along with a transect coverage (roughness coverage would also be created if the option was on).

Working with Transects

1. In the transect coverage, select the **Feature arc** tool.
2. Then click out an arc (this creates the transect).
3. Then select the **Select Feature arc** tool. Double-clicking on the arc will display the attributes in a dialog.
4. The *attributes* dialog contains an elevation curve and also a permeable structure curve if turned on.
5. If the transect arc was created over a scatter set, the curves can automatically be created by right-clicking on the transect coverage, then selecting **Extract Elevations**. Then select the curve to extract to elevations or permeable structure.

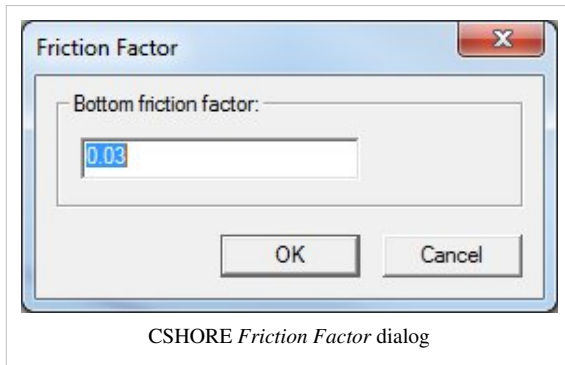


CSHORE Arc Attributes dialog

Working with Roughness

The roughness coverage allows you to create different roughness zones defined within polygon boundaries.

1. In the roughness coverage, select the **Feature arc** tool.
2. Click out multiple arcs that form a closed region.
3. Then select *Feature Objects* | **Build Polygons** from the menu.
4. Switch to the **Select Polygon** tool.
5. Double-clicking on the created polygon will bring up the *Friction Factor* dialog allowing the roughness to be specified.

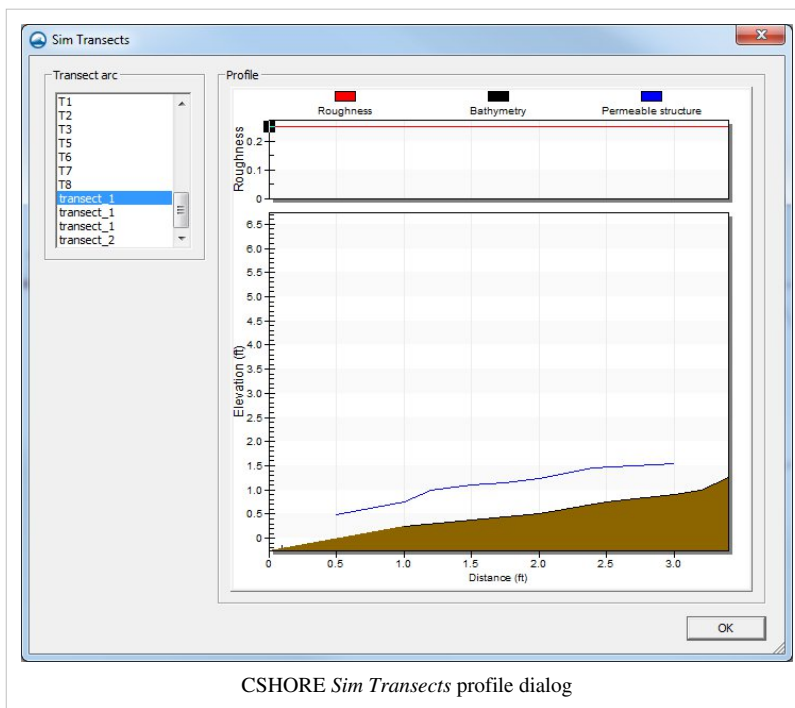


CSHORE Friction Factor dialog

Viewing Transects Profile Plot

Once curves have been established on the transects, you can view the transects profile plot. This is done by:

1. Right-click on the transects coverage and select **Extract elevations** from the right-click menu.
2. Right-clicking on *Simulation*, then selecting **View transects profile**.



CSHORE Sim Transects profile dialog

Model Parameters

The model parameters can be viewed by:

- Right-clicking on *Simulation*, then selecting **Model Control**.

The *Model Control* dialog has the following tabs and options:

Computations

- Comments
- Moveable bottom
- Limited sediment availability
- Permeable bottom
- Stone
 - Porosity n_p
 - Nominal diameter
- Wave overtopping
- No standing water
- Wave/current interactions
- Wind effects
- Roller effects
 - Runup wire height
- Constant nodal spacing
- Empirical breaker ratio parameter
- Default bottom friction factor f_b

The screenshot shows the 'Model Control' dialog box with the 'Computations' tab selected. The 'Comments' field is empty. The 'Moveable bottom' checkbox is checked. The 'Limited sediment availability' checkbox is checked. The 'Permeable bottom' checkbox is checked. The 'Stone' section has 'Porosity n_p ' set to 0 and 'Nominal diameter' set to 0.1 m. The 'Wave overtopping' checkbox is checked. The 'No standing water' checkbox is checked. The 'Wave/current interactions' checkbox is checked. The 'Wind effects' checkbox is checked. The 'Roller effects' checkbox is checked. The 'Runup wire height' is set to 0.02 m. The 'Constant nodal spacing' is set to 2 m. The 'Empirical breaker ratio parameter' is set to -1. The 'Default bottom friction factor f_b ' is set to 0.01. The 'Help', 'OK', and 'Cancel' buttons are at the bottom.

CSHORE Model Control showing the Computations tab

Sediment

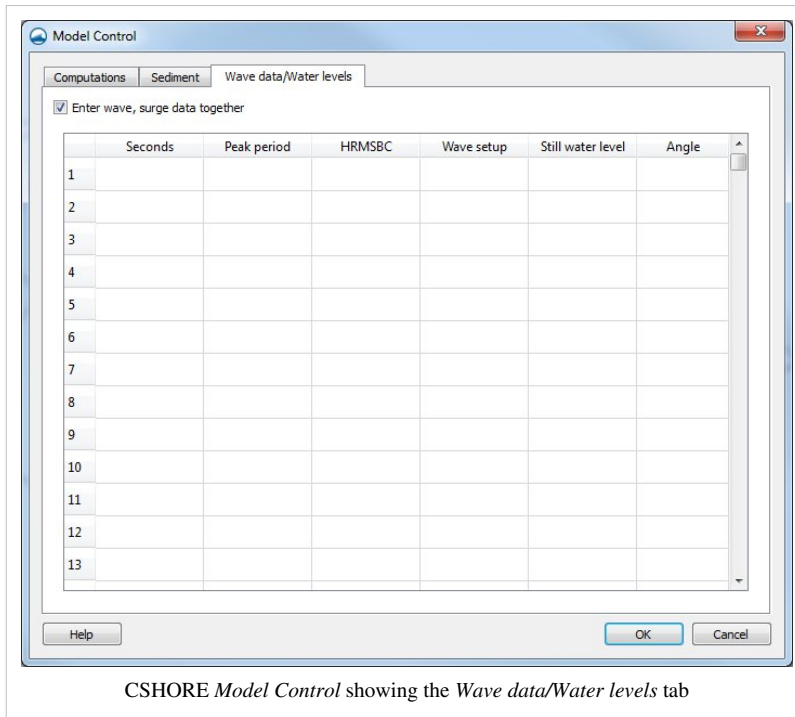
- Median diameter
- Fall velocity
- Specific gravity
- Efficiency e_b
- Efficiency e_r
- Suspended load parameter a
- Suspended load parameter a_o
- Limiting(maximum) slope $\tan$
- Bedload parameter

Wave data/Water levels

- Enter wave, surge data together
- Data table
 - Seconds
 - Peak period
 - HRMSBC
 - Wave setup
 - Still water level
- Angle

The screenshot shows the 'Model Control' dialog box with the 'Sediment' tab selected. The 'Median diameter' is set to 0.3 mm. The 'Fall velocity' is set to 0.0448 m/s. The 'Specific gravity' is set to 2.65. The 'Efficiency e_b ' is set to 0.005. The 'Efficiency e_r ' is set to 0.01. The 'Suspended load parameter a ' is set to 0.5. The 'Suspended load parameter a_o ' is set to 0.1. The 'Limiting(maximum) slope $\tan$ ' is set to 0.63. The 'Bedload parameter' is set to 0. The 'Help', 'OK', and 'Cancel' buttons are at the bottom.

CSHORE Model Control showing the Sediment tab



Running the Model

Running the model is accomplished by:

- Right-clicking on *Simulation*, then selecting **Save, Export and Launch CSHORE**.
- This writes out the necessary input files for CSHORE in a folder called CSHORE in the same directory as the project.
- The model will run, creating text file output of the CSHORE results.
- Upon exiting the *model wrapper* dialog, the CSHORE results will be converted into a different format for SMS.

Solution View

To view the Solution View:

- Click on the *Plot Wizard* located on the bottom toolbar.
- Select **CSHORE Solution View**.
- Click **Next**.
- In the dialog there is a mode for *Profile* and *Time series* that can be chosen from the combo box on the top left side of the dialog. *Time series* may not be chosen if wave overtopping and/or moveable bottom are turned off for all of the simulations.
- In *Profile* mode, the user may select multiple simulations and datasets, along with a single transect and time step.
- In *Time series* mode, the user may select multiple simulations, datasets, and transects.
- After the options are chosen, the user can hit "Update" in order to see a plot of the data on the right side of the dialog.

5.7 CSTORM

CSTORM-MS

This article is for a model that is under development by Aquaveo and its partners. The model is **not available** to the public at this time. It is not known when or if the model will be released.

Create HPC scripts

CSTORM-MS is a set of models organized into a system that work together to create models relative to *coastal storms*. Many of the models in the system are very compute intensive models and may run for long periods of time. Often it is desirable to run these models in a HPC (supercomputer) environment to take advantage of many processors.

Some users may not be as familiar with the HPC environment or want a system to help regenerate scripts as needed. The **Create HPC Scripts** tool within SMS helps users create scripts from parameters. The parameters may be changed and the scripts regenerated as

necessary. The parameters can be saved with the project, exported/loaded into other projects, and stored as the default parameters for future CSTORM-MS simulations.

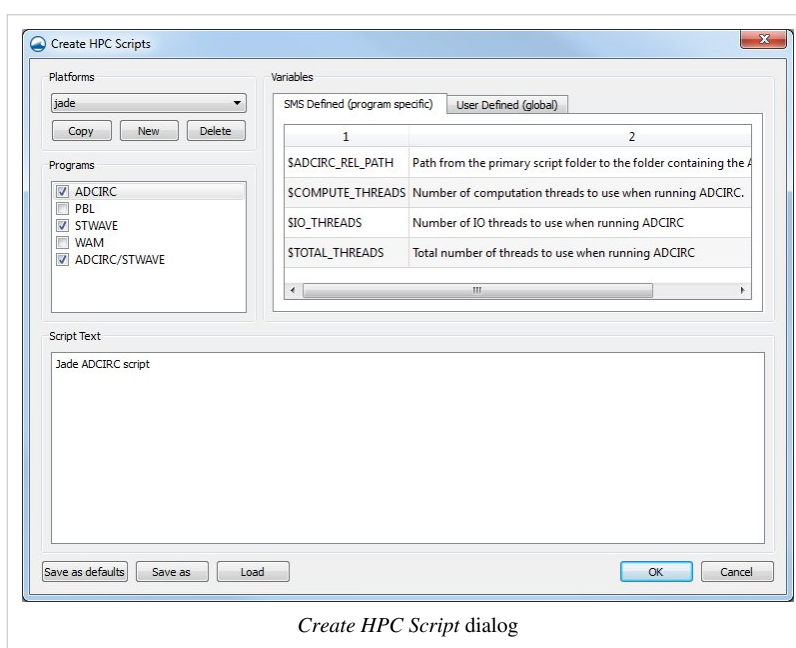
The **Create HPC Scripts** tool is accessed by right-clicking on the CSTORM-MS simulation and selecting the **Create HPC Scripts** command.

Platforms

Platforms are the term used for a collection of script parameters because often the scripts will be created to target a specific HPC platform. However, a user could create multiple sets of scripts for the same platform if there are different run parameters for different types of simulations. The user can create, delete, or copy the scripts from one platform to a new platform.

Programs

Each of the programs within the CSTORM-MS system has its own script within the platform. Each program has its own run requirements which are handled through SMS defined variables (see below).



Variables

Variables are used to modify the script text for export. Variables can make it easier to change a script by identifying the items that should be changed in a script and giving a description for the variable. When the script is generated, every instance of the variable within the script will be replaced with the text in the 'value' column. It is recommended but not required that variables use a naming convention similar to the SMS defined variables for easy identification.

SMS defined variables (program specific)

SMS defined variables provide information to scripts based upon simulation parameters. These parameters are specified elsewhere but may need to be incorporated into individual scripts. One example of this type of variable is the number of processors to use for ADCIRC. SMS knows how many processors ADCIRC wants to use and can automatically make this part of the script.

User defined variables (global)

User defined variables allow scripts to be generated generically and give descriptions for the script parameters needed. User defined variables are global (shared between all programs/platforms). If you wish to have a similar variable specified differently for separate programs, add the program name to your variables. For example, if PBL and ADCIRC both needed a walltime but it should be different use \$PBL_WALLTIME and \$ADCIRC_WALLTIME rather than \$WALLTIME within your scripts.

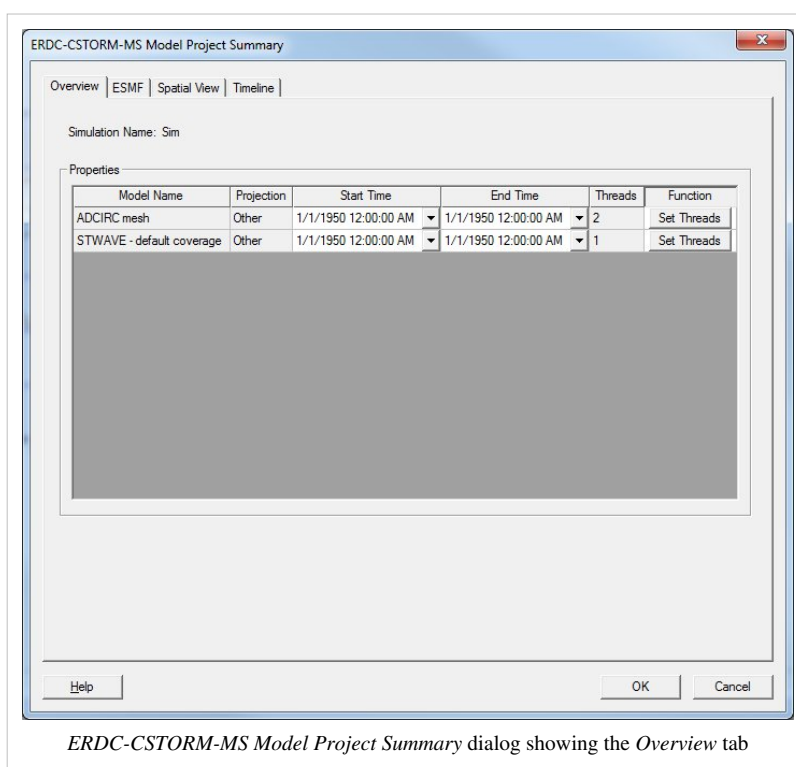
Model Project Summary

The *ERDC-CSTORM-MS Model Project Summary* dialog is accessed by right-clicking on the CSTORM-MS simulation and selecting **Properties** from the menu. The dialog has several tabs and options as follows:

Overview

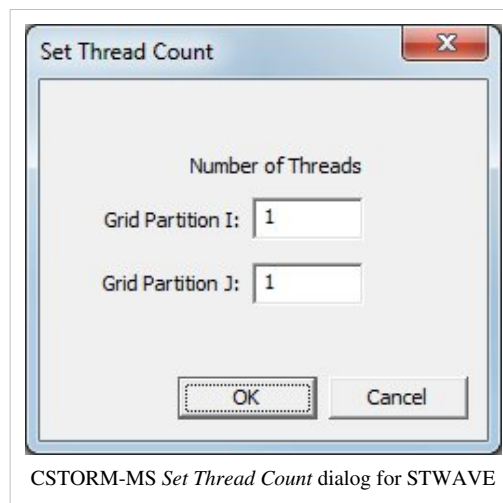
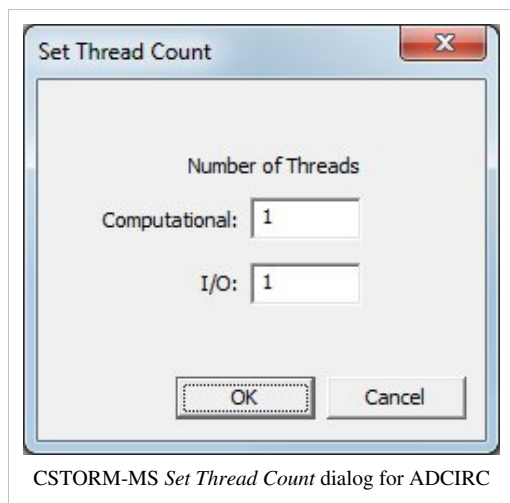
The dialog gives an overview of the components of the Hurricane simulation. It also allows the user to designate the number of processors to be use by each model in the simulation.

- Simulation Name – Name of the Hurricane simulation as set in the project explore
- Properties – Properties of the Hurricane simulation
 - Model Name – Name of models found in the simulation
 - ADCIRC <mesh name> (1 required) – Name of the adcirc mesh used in the hurricane simulation.
 - STWAVE <grid name> (1 required) – Name of the stwave cgrid used in the hurricane simulation.
 - PBL <pbl cov name> (optional) – Name of pbl wind coverage used in the hurricane simulation.



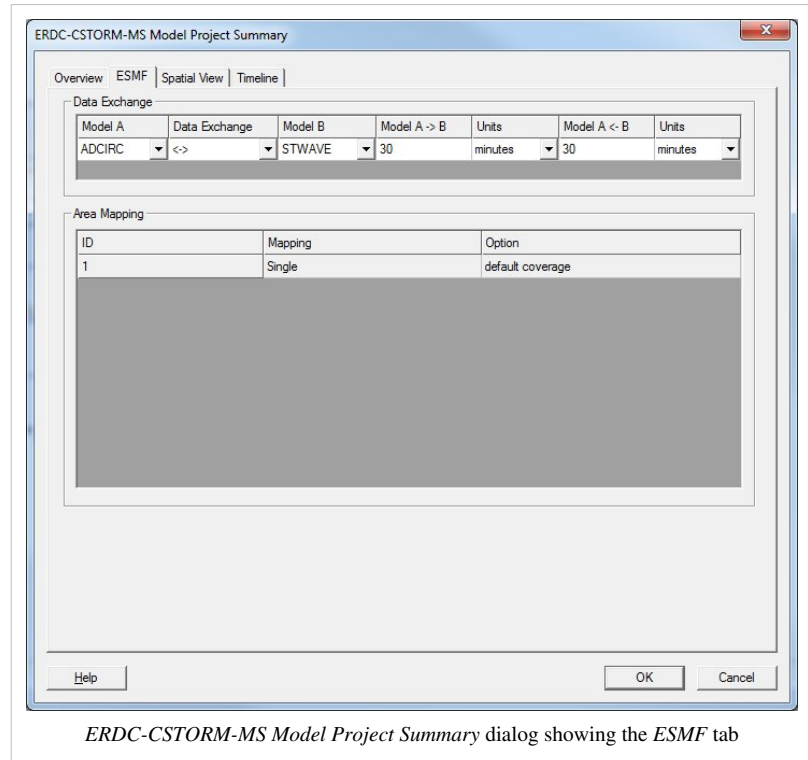
ERDC-CSTORM-MS Model Project Summary dialog showing the Overview tab

- WAM <wam sim name> (optional) – Name of wam simulation used in the hurricane simulation.
- Projection – Projection type of model.
 - Geo – Geographic (Latitude/Longitude)
 - STPL <#> – State Plane number
 - Other – Other type of projection
- Start Time – Temporal starting time of model.
- End Time – Temporal ending time of model.
- Threads – Number of processor threads used by model for computation or I/O processing.
- Function – Function used to adjust the processor threads used by each model.
 - Set Threads – Set the number of processor threads for:
 - Computational or I/O processing (ADCIRC)
 - Grid Partition I and J processing (STWAVE)
 - NONE – WAM and PBL use only one processor thread.



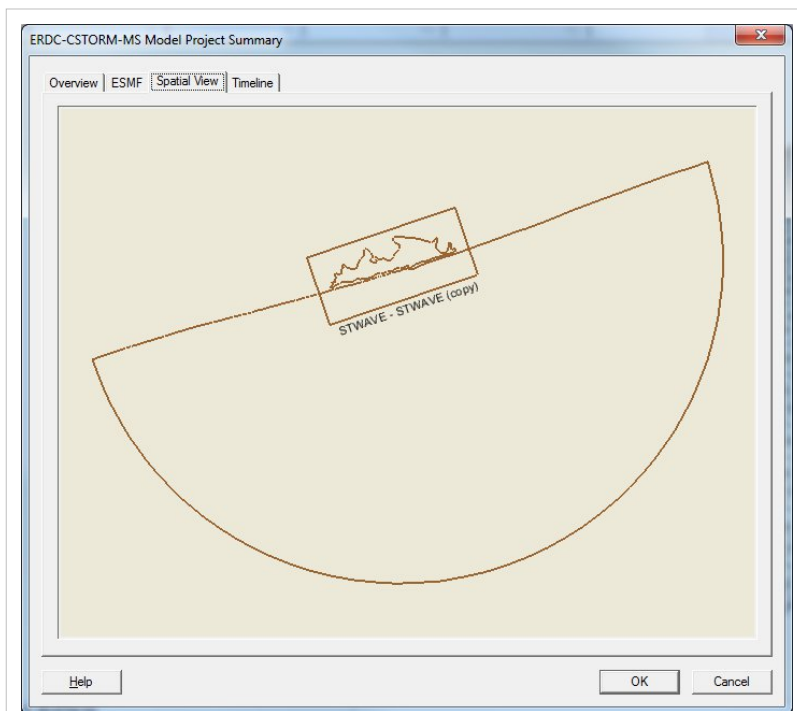
ESMF

- Set the ESMF simulation options in the *Data Exchange* and *Area Mapping* sections of the dialog
- Data Exchange
 - Model A – The meshed based model to be use (hard-coded to ADCIRC)
 - Data Exchange – Controls how data is exchanged between the two models ($\leftrightarrow$ bi-directional, $\rightarrow$ uni-direction A to B, $\leftarrow$ uni-directional B to A)
 - Model B – The grid based model to be use (hard-coded to STWAVE)
 - Model A $\rightarrow$ B – How frequently the results of Model A are passed to Model B
 - Units – Frequency units for A to B exchange (days/hours/minutes/seconds)
 - Model A $\leftarrow$ B – How frequently the results of Model B are passed to Model A
 - Units – Frequency units for B to A exchange (days/hours/minutes/seconds)
- Area Mapping
 - ID – Polygon identifier correlated to the ids visible in the main graphics window.
 - Mapping
 - Single – Model A exchanges data with a single instance of Model B
 - Combined – Model A exchanges data with multiple instances of Model B. Model A can receive either the average or maximum values from the instances of Model B.
 - Option – If the "Mapping" field is set to "Single" and multiple grids overlap the identified polygon, this field allows the user to select which grid will be used. If the "Mapping" field is set to "Combined", this field allows the user to select how the data from the grids will be combined (Average or Maximum.)



Spatial View

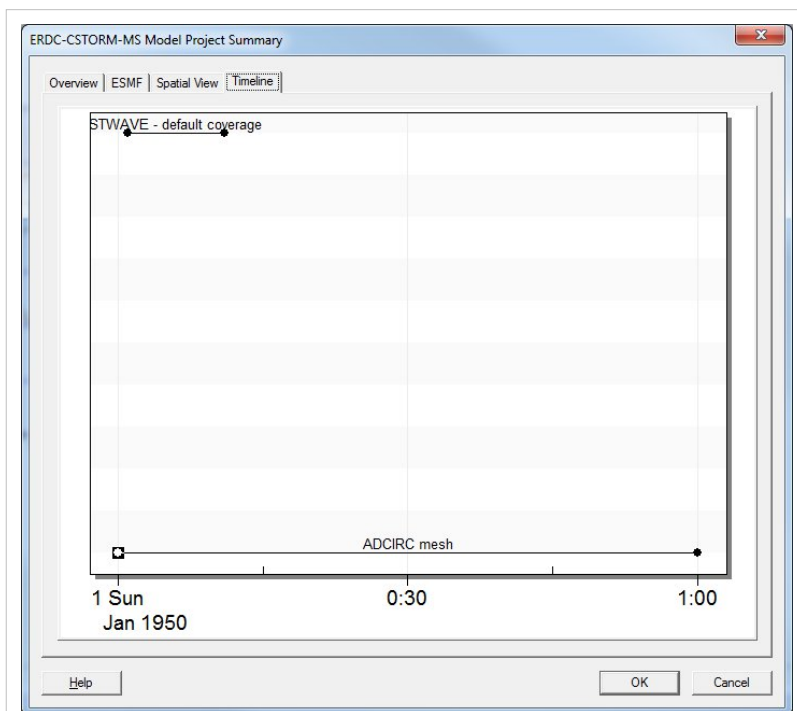
This dialog provides a view of any grid boundaries (WAM and STWAVE), grid frames (PBL), and the ADCIRC mesh boundary associated with the simulation.



ERDC-CSTORM-MS Model Project Summary dialog showing the *Spatial View* tab

Timeline

This dialog displays the timelines for each simulation (WAM, STWAVE, PBL, and ADCIRC) that are part of the hurricane simulation. The start and end times need to be set for each simulation individually.



ERDC-CSTORM-MS Model Project Summary dialog showing the *Timeline* tab

Related Topics

- ESMF - Earth System Modeling Framework

5.8. GenCade

GenCade

The GenCade model is a next generation combination of previous long-term planform evolution of a beach models GENESIS (GENERalized Model for SIMulating Shoreline Change[1]) and Cascade[2].) and Cascade.

GenCade is a regional model for calculating coastal sediment transport, morphology change, and sand bypassing at inlets and engineered structures. GenCade is developed by the U.S. Army Engineer Research and Development Center (USACE-ERDC), Coastal and Hydraulics Laboratory (CHL). The developers maintain documentation ^[3] for the model, comments on the interface ^[4] and steps for a sample application ^[5].

GenCade Coverage

The GenCade coverage allows the user to define the geographical location of the characteristics and structures of a GenCade simulation. One or more coverages can be mapped to the simulation.

Functionality

GenCade simulates shoreline change relative to regional morphologic constraints upon which these processes take place. The evolution of multiple interacting coastal projects and morphologic features and pathways, such as those associated with inlets and adjacent beaches may also be simulated. The model supports responses to imposed wave conditions, coastal structures, and other engineering activity (e.g., beach nourishment).

Typical longshore extents and time periods of modeled projects can be in the ranges of one to 100 km and one month to multiple decades, respectively, and almost arbitrary numbers and combinations of groins, detached breakwaters, seawalls, jetties, and beach fills can be represented. GenCade simulates shoreline change produced by spatial and temporal differences in longshore sand transport. Shoreline movement such as that produced by beach fills and river sediment discharges can also be represented. The main utility of the modeling system lies in simulating the response of the shoreline to structures sited in the nearshore. Shoreline change produced by cross-shore sediment transport as associated with storms and seasonal variations in wave climate cannot be simulated; support of cross-shore processes are being considered for future versions of the model.

Capabilities of GenCade:

- Almost arbitrary numbers and combinations for groins, jetties, detached breakwaters, beach fills, and seawalls
 - Compound structures such as T-shaped, Y-shaped, and spur groins
 - Bypassing of sand around and transmission through groins and jetties
 - Diffraction at detached breakwaters, jetties, and groins
 - Coverage of wide spatial extent
 - Offshore input waves of arbitrary height, period, and direction
 - Multiple wave trains (as from independent wave generation sources)
 - Sand transport due to oblique wave incidence and longshore gradient in height
 - Wave transmission at detached breakwaters
-

Practical Notes For Using GenCade

Shoreline change models are not applicable to simulating a randomly fluctuating beach system in which no trend in shoreline position is evident. In particular, GenCade is not applicable to calculating shoreline change in the following situations that involve beach change unrelated to coastal structures, boundary conditions, or spatial differences in wave-induced longshore sand transport:

- Beach change inside inlets or in areas dominated by tidal flow
- Beach change produced by wave-generated currents
- Storm-induced beach erosion in which cross-shore sediment transport processes are dominant
- Scour at structures

Limitations of GenCade:

- No wave reflection from structures
- No direct provision for changing tide level
- Basic limitations of shoreline change modeling theory

Case Studies / Sample Problems

- Tutorials for learning to use GenCade in SMS are under development.

Related Topics

- 1D Grid Module
- GenCade modeling process
- GenCade graphical interface

External Links

- US Army Corps of Engineers Coastal & Hydraulics Laboratory GENESIS website [6]
 - GENESIS, Report 1, Technical Reference [7]
 - GENESIS, Report 2, Workbook and System User's Manual [8]
 - User's Guide to the Shoreline Modeling System [9]
 - A History of GENESIS Updates [10]
 - Mar 2002 ERDC/CHL CHETN-II-45 Wave Transmission at Detached Breakwaters for Shoreline Response Modeling [11]
 - Mar 1990 CETN-II-21 Computer Program: Genesis Version 2 [12]
 - Cascade User Guide: [13]
 - Cascade Theory and Model Formulation: [14]
 - Inlet Reservoir Model (sub-model within Cascade) [15]
-

GenCade Files

Input

A GenCade simulation requires the following input files:

- *.gen – this is the simulation file and contains links to other input files as well as the definition of all the structures and events and other model parameters.
- *.shi – this contains the initial shoreline position geometry
- *.wave – there must be one wave file for each wave gage. The file contains the wave events for the associated gage for all time spanning the simulation.

The following are optional input files:

- *.shr – this contains the regional contour geometry
- *.shdx – this contains the grid density values for a non-uniform grid

Output

The following files are created by GenCade during a simulation:

- *.prt – this is a summary printed output file that reports on the status of the run. It is for user review.
- *.slo – this file contains the output shoreline positions through the simulation.
- *.off – this file contains the offset from the original shoreline positions. This is the same as the positions from the .slo file minus the initial shoreline positions.
- *.qtr – this file contains the transient, spatially varied transport rates at the specified output times during the simulations.
- *.mql/*.mqr/*.mqn – this file contains the annual averaged transport rates to the left, to the right and net. This is an integration of the components that make up the qtr.
- *.irv – One irv file will be created for each inlet in the simulation. It contains the time varying volumes of each of the shoals related to the inlet.

Related Topics

- [GenCade](#)
-

GenCade Modeling Process

A GenCade model is set up in a globally referenced projection (such as UTM or State Plane).

Required Input Data

The model requires the following data for input:

- Grid frame – This is a geometric object that defines the domain of the simulation. The modeling engine operates on a straight line (1D grid). The grid frame defines the origin, orientation and extent of the simulation region. The grid frame should be oriented to align to the principal shoreline direction. Generally the grid frame is positioned just to the landward side of the shoreline. When looking along the shoreline in a direction parallel (or close to parallel) to the direction of the grid frame, the ocean should be but on the left and the shore to the right. Positions in the model (such as shoreline values) are referenced to this datum. The graphical interface will generate a grid that lies along the grid frame. h.
- Initial shoreline definition – This is series of points ((x,y) or (lat,lon)) that define a reference position for the coastline at the beginning of a simulation (in the model projection). SMS stores these points as an arc in a GenCade coverage.
- Wave conditions: The model supports multiple wave inputs at locations throughout the domain. These are mapped to a single cell in the grid. Wave conditions include the wave parameters (height, period, direction) at each time step in the simulation. The direction of each wave condition must be provided to the model in local shore normal coordinates.
- Other structure definitions to represent the structures (seawalls, groins and inlets) in the domain.
- Dredging/nourishment and bypass events as warranted by the events during the period being simulated.

Directions

The model performs all calculations in a local direction defined by the grid. This direction can be visualized relative to several reference points. Two references are defined here:

- The model references the grid direction as an azimuth angle from North. An azimuth of zero (0.0) directs the local grid to go from South to North (land to the East). The azimuth is measured in a clockwise direction, so an azimuth of 90 degrees defines a grid that goes from West to East (land on the South).
- The SMS interface relies more on graphical positioning, since the grid frame is defined interactively and displayed with an origin. However, the system uses a Cartesian angle reference with the X axis (East) equal to 0 degrees. The direction for this convention follows the right hand rule (CCW = positive). Therefore, a Cartesian angle of zero corresponds to the West to East (land on the South) azimuth of 90 degrees. Similarly, a Cartesian angle of 90 degrees is the same as an azimuth of 0 degrees.

The following steps illustrate the standard modeling process.

1. Define a GenCade coverage.
 2. Create a grid frame defining the domain extents.
 3. Define refinement points as desired to control grid resolution.
 4. Define feature points at the location of wave buoys. Read or enter the wave data at each of these points.
 5. Define arcs representing the various structures in the domain. Assign attributes to these arcs.
 6. Define arcs representing the events in the simulation. Assign attributes to these arcs.
 7. Map the conceptual model to a numeric model.
 8. In the 1D Grid Module, assign the model parameters in the *GenCade Model Control* dialog.
 9. Save the simulation and run the model.
 10. Read solution files to visualize results.
-

Related Topics

- [GenCade](#)

5.8.a. GenCade Graphical interface

GenCade Graphical Interface

SMS provides a graphical interface that is designed to allow users to visualize the projects they are creating, easily modify project parameters, and view the solutions produced by the GenCade model. The GenCade interface exists in the 1D Grid Module.

A GenCade simulation consists of a geometric definition of the model domain (the grid) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model. To set up a simulation, the modeler can work directly with the 1D grid, but it is much more convenient to work with the conceptual model using a GenCade coverage.

Work with an existing simulation by selecting the 1D Grid Module. If a grid has already been created for a GenCade simulation or an existing simulation read, the grid object will exist in the Project Explorer and selecting that object will make the 1D Grid module active and set the model to GenCade.

The interface consists of the conceptual model tools in the Map Module and the grid specific options in the 1D Grid Module.

Map Module – Conceptual Interface

In the map module, the user creates a GenCade coverage (or coverages) that include the information defining the grid and structures in the desired simulation.

Grid Frame

The grid frame defines the orientation and extent that will be occupied by the 1D grid.

Feature Points

In a GenCade coverage, a feature point can be assigned to be:

- A generic place holder
- A wave gage – The attributes of this point include a time series of wave data.
- A refine point – This point will allow the user to control the grid density in the area of the point.

Feature Arcs

The arcs in a GenCade coverage can be assigned a variety of attributes including:

- Geometric object – shoreline and contours.
- Structure – jetty, inlet, sea wall, or breakwater and the associated parameters.
- Event – bypass or beach nourishment events and their time ranges and parameters.

The conceptual model is converted to a numeric model using the **Map → 1D grid** command. This command can overwrite an existing simulation or simply add additional structures to an existing simulation.

1D Grid Module – Numerical interface

The tools in the grid module allow the user to select individual structures graphically and change their parameters. The *GenCade* menu allows the user to bring up tables of all structures of a specific type, view their parameters, and make edits as needed.

GenCade Menu

- Global Section
 - **Edit Grid** – Opens a *Grid Frame* dialog.
 - **Edit Water Level Data** – Opens the *Water Level Events* dialog.
 - **Edit Wave Data** (New for multiple wave inputs) – Opens the *Wave Gages* dialog.
 - **Filter Wave Data** (This tool was part of the GENESIS interface, but is currently not functional for GenCade)
- Shore Parallel Structures Section
 - **Breakwaters** (Detached breakwaters are temporally invariable at this time) – Opens the *Detached Breakwaters* dialog.
 - **Seawalls** – Opens the *Seawalls* dialog.
- Shore Normal Structures Section
 - **Groins** (was combined Groins/Jetties) – Opens the *Groins* dialog.
- Shore Feature Section
 - **Inlets** (New – includes an option to specify a jetty at the inlet) – Opens the *Inlets (Resvoir Model and Jetties)* dialog.
- Events Section
 - **Beach Fills** – Opens the *Beach Fill* dialog.
 - **Bypassing** – Opens the *Bypassing* dialog.
- Management Section
 - **Model Control** – Launches the *GenCade Model Control* dialog.
 - **Run GenCade**

GenCade Tools

GenCade makes use of the tools in the SMS:1D Grid Module

Running the Model

The GenCade Files are written automatically with the SMS project file or can be saved separately using the *File | Save GenCade* or *File | Save As* menu commands. See GenCade Files for more information on the files used for the GenCade run.

GenCade can be launched from SMS using the *GenCade | Run GenCade* menu command.

Related Topics

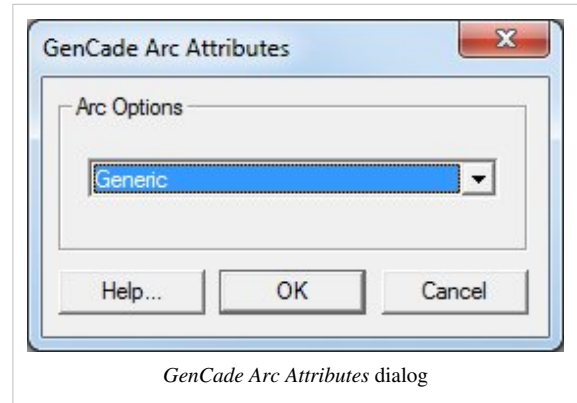
- GenCade Model Control Dialog
- GenCade Result Visualization

GenCade Arc Attributes

The attributes that can be associated with for feature arcs in a GenCade coverage are specified through the *GenCade Arc Attributes* dialog. This dialog can be reached by selecting the arc and either right-clicking and selecting **Attributes** from the right-click menu or selecting **Attributes** command from the *Feature Objects* menu.

Attributes that can be specified for each arc include:

- Arc Type
 - *Generic* – place holder in the conceptual model. Ignored by the simulation.
 - *Initial Shoreline* – the geometry of this arc defines the starting shoreline position.
 - *Reference Line* – this type is not currently used.
 - *Regional Contour* – the geometry of this arc defines the regional geometric shape.
 - *Breakwater* – detached structure, generally offshore. In addition to location, the user specifies depths at the ends of the structure and transmission attributes.
 - *Seawall* – the geometry defined by this arc delineates the furthest landward point the shoreline can erode to in that region.
 - *Groin* – this arc defines a shore perpendicular structure. The user also specifies the structure permeability and its diffractive attributes.
 - *Inlet* – this arc actually defines the location of the mouth of the inlet. For each inlet the user defines an inlet name, initial and equilibrium volumes of the shoals associated with that inlet, bypass coefficient and defines dredging events that occur during the simulation. Several other arcs can define other components of the inlet.
 - *Left Jetty of Inlet* – this arc defines a groin that is part of an inlet complex.
 - *Right Jetty of Inlet* – this arc defines a groin that is part of an inlet complex.
 - *Inlet Attachment Bar* – this arc defines the location of the attachment bar that will form related to an inlet complex.
 - *Bypass Event* – this arc defines the beginning and ending points of a bypass event. The user defines the time ranges and the bypass rates that will occur during the simulation between these points.
 - *Beach Fill Event* – this arc defines the location of a beach nourishment event. The user defines the time ranges over which material will be added to the beach and the amount of berm width that will be added with each time range.



Related Topics

- GenCade
- Feature Objects Menu

GenCade Events

The GenCade simulation can also include timed events that impact the shoreline. These include dredging events at an inlet, beach fill (nourishment) events, and bypassing events.

Beach Fill

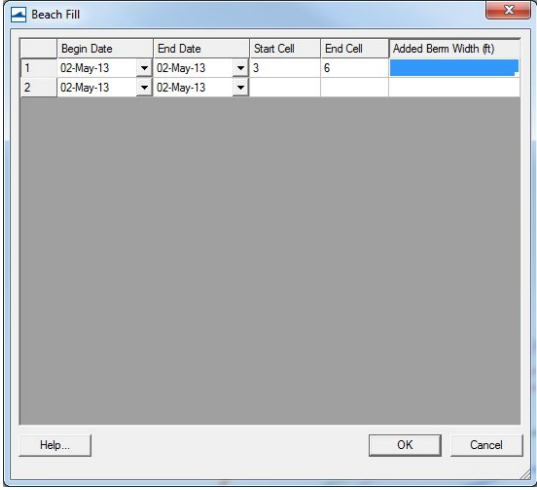
Parameters for characterizing beach fills include begin and end time, location, and width. To define a beach fill, proceed with the following steps:

1. Select the arc associated with the beach fill or select the menu option *GenCade* | **Edit Beach Fill**.
2. The GenCade *Beach Fill* dialog will appear allowing the entry of beach fill parameters. At this point, the user should specify beach fill parameters

Bypassing Event

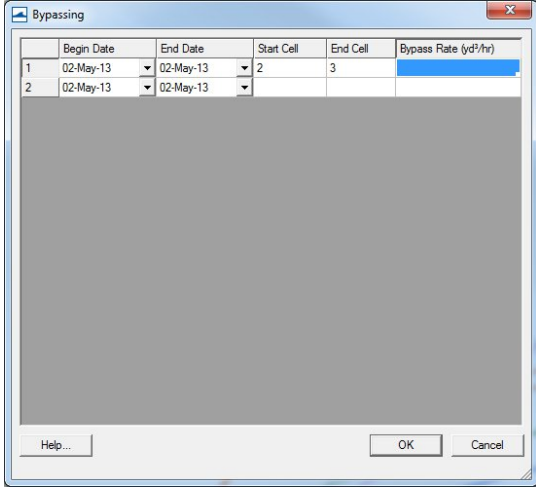
Parameters for characterizing bypassing events include begin and end time, location, and bypassing rate. To define a bypassing event, proceed with the following steps:

1. Select the arc associated with the bypass or select the menu option *GenCade* | **Edit Bypassing**.
2. The GenCade *Bypassing* dialog will appear allowing the entry of bypassing parameters. At this point, the user should specify parameters



	Begin Date	End Date	Start Cell	End Cell	Added Berm Width (ft)
1	02-May-13	02-May-13	3	6	
2	02-May-13	02-May-13			

GenCade *Beach Fill* table



	Begin Date	End Date	Start Cell	End Cell	Bypass Rate (yd³/hr)
1	02-May-13	02-May-13	2	3	
2	02-May-13	02-May-13			

GenCade *Bypassing* table

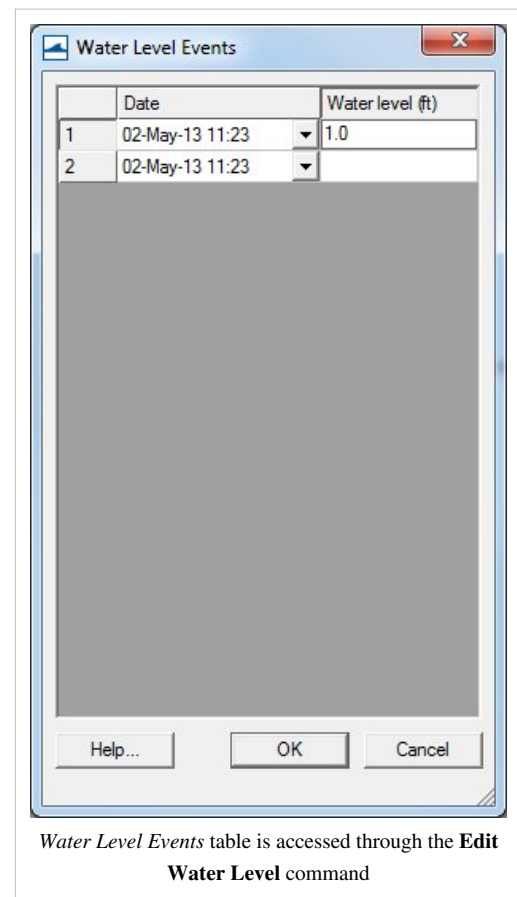
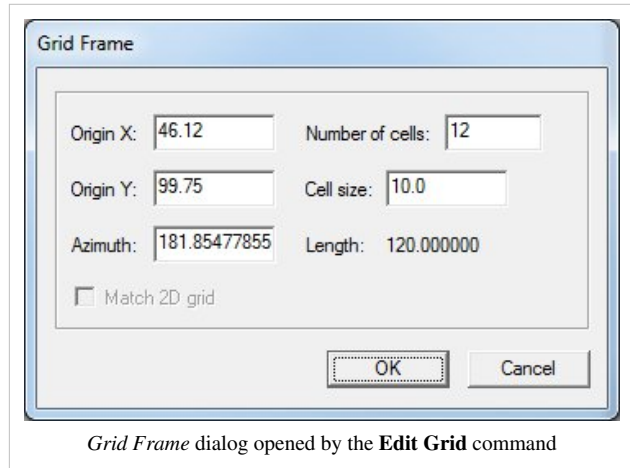
Related Topics

- GenCade

GenCade Menu

The *GenCade* Menu has the following commands:

- **Edit Grid...**
Opens a *Grid Frame* dialog.
- **Edit Water Level...**
Opens the *Water Level Events* table.
- **Edit Wave Data...**
Opens the *Wave Gages* table.
- **Filter Wave Data...**
(This tool was part of the GENESIS interface, but is currently not functional for GenCade)
- **Edit Breakwaters...**
Opens the *Detached Breakwaters* table.
- **Edit Seawalls...**
Opens the *Seawalls* table.
- **Edit Groins**
Opens the *Groins* table.
- **Edit Inlets**
Opens the *Inlets (Resvoir Model and Jetties)* table.
- **Edit Beach Fills...**
Opens the *Beach Fill* table.
- **Edit Bypassing...**
Opens the *Bypassing* table.
- **Model Control...**
Can be used to set beach conditions, lateral boundary conditions and general simulation options. See SMS:GenCade Model Control Dialog For more information.
- **Run GenCade...**
This command will start the GenCade Model.



Related Topics

- GenCade
- GenCade Events
- GenCade Structures
- Wave Gages

GenCade Model Control Dialog

The *GenCade Model Control* dialog is used to set beach conditions, lateral boundary conditions and general simulation options. This document highlights the more commonly used options. Refer to the GenCade web site ^[3] for a more detailed description of how these parameters affect the model results.

Model Setup Tab

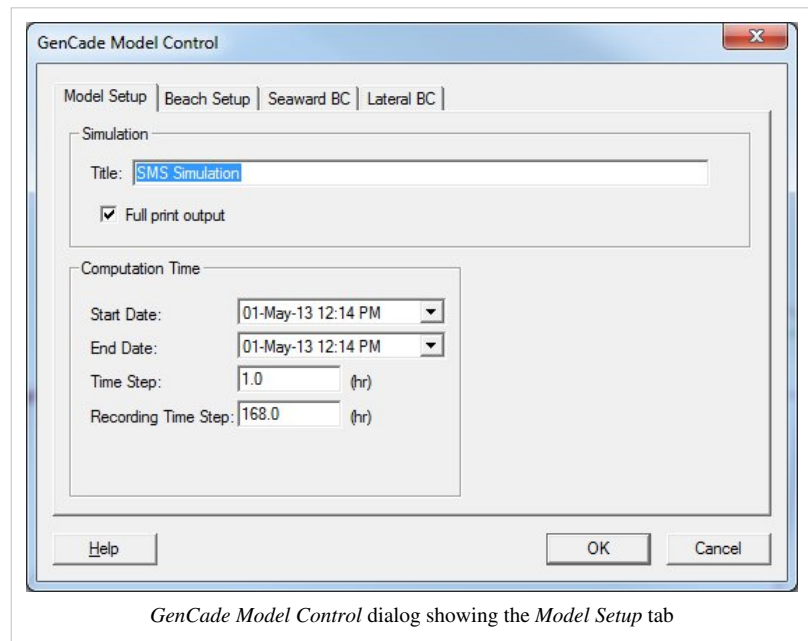
The following parameters are specified in the *model setup* tab:

- Simulation Title
- Simulation start and end time
- Simulation time step
- Recording time step

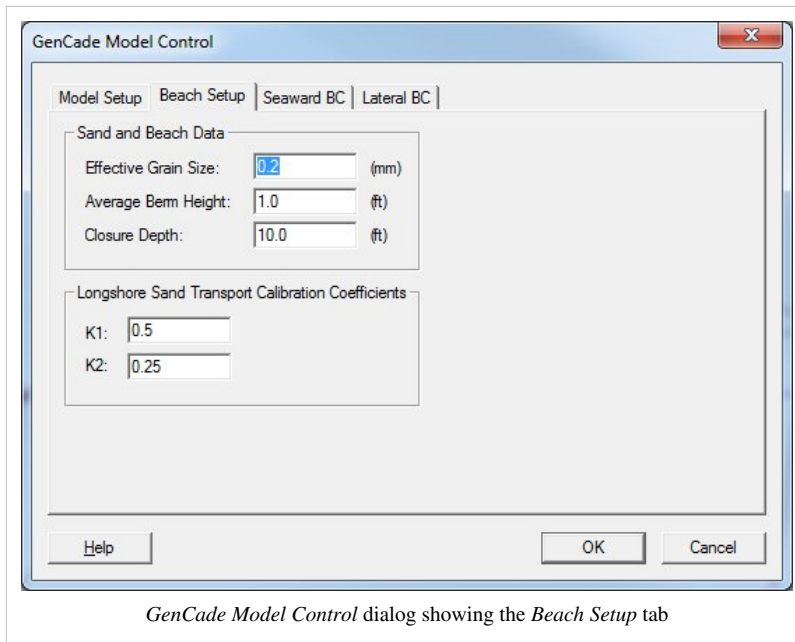
Beach Setup Tab

The following parameters are specified in the *beach setup* tab:

- Sand and Beach Data
 - Effective Grain Size
 - Average Berm Height
 - Closure Depth
- Longshore Sand Transport Calibration Coefficients
 - K1
 - K2



GenCade Model Control dialog showing the *Model Setup* tab



Seaward BC Tab

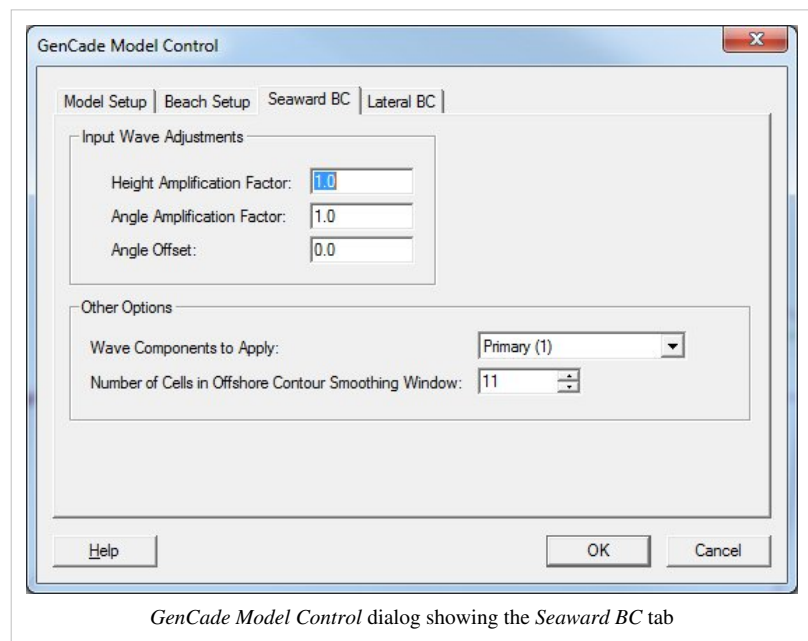
The following parameters are specified in the *seaward boundary condition* tab:

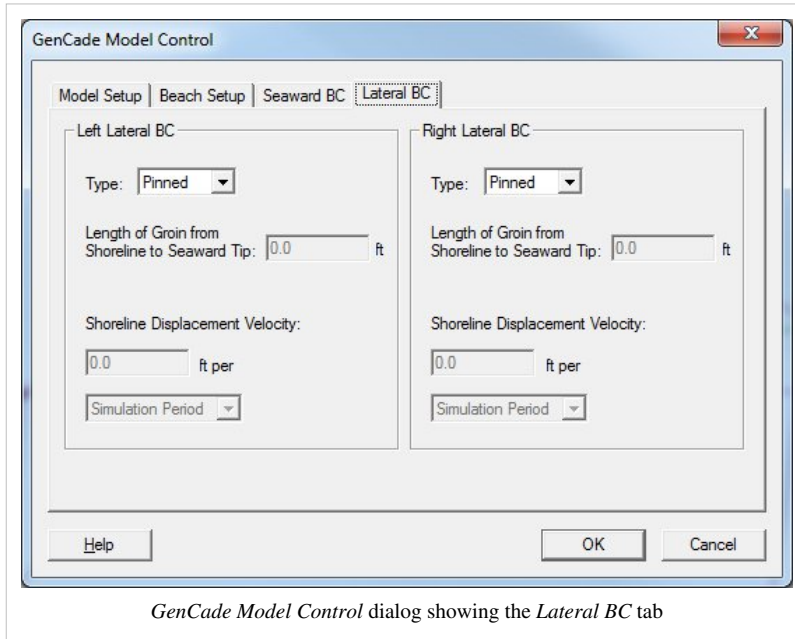
- Input Wave Adjustments
 - Height Amplification Factor
 - Angle Amplification Factor
 - Angle Offset
- Wave Components to Apply (Primary or Primary & Secondary)
- Number of Cells in Offshore contour Smoothing Window

Lateral BC Tab

The following parameters are specified in the *lateral boundary condition* tab for the left and right lateral boundary condition:

- Type
 - Pinned
 - Moving
 - Shoreline Displacement Velocity
 - Gated
 - Length of Groin from Shoreline to Seaward Tip





Related Topics

- [GenCade](#)

GenCade Result Visualization

Visualization of Results

The interface supports three types of plots to visualize the results of the model. These include:

- Shoreline plots on the grid. These plots are available in the system after reading in the shoreline output (*.slo) file. The offset file (*.off) contains the same data as a displacement quantity and can be read and viewed just like the shoreline output file. The user selects the desired dataset which is then plotted in the graphics window right over the initial shoreline. Display options also allow the display of the minimum and maximum shoreline positions during the simulation. When this file is read, SMS adds several datasets to the project explorer representing:
 - The shoreline position throughout the simulation time. This dataset includes a time step for each output time at each cell in the grid.
 - The change in shoreline position from the initial shoreline (since the beginning of the simulation). This dataset is also transient.
 - The average rate of change of the shoreline during the simulation (single value at each cell of the grid).
- Two dimensional plot of spatially variable datasets. In addition to the *.slo solution file, the user may also instruct SMS to load the sediment transport rate (*.qtr) solution file and the mean transport rate files (*.mqr, *.mql and *.mqn). The sediment transport rate file contains the net transport computed for each cell in the grid at each time step. The mean transport rates contain annual average transport rates in the right, left, and net directions. This dataset, along with any of the other spatially variable datasets may be visualized in their own plot window. Display options allow the user to select one or more of the datasets to be plotted. The user may also instruct SMS to plot the "active" dataset which results in the plot being updated as the active dataset is changed in the project explorer. In addition, multiple time steps may be selected (or the plot can use the "active" time step). SMS will assign a symbol to each dataset and cycle through line styles for each displayed time step. To create this plot:
 1. Load at least one solution file into the project.
 2. Click on the **Plot Wizard** tool.

3. Select the *Shoreline* option and click the *Next* button.
 4. Choose the plot options to select the dataset and time steps to be plotted.
- Two dimensional plot of the Inlet Volume transport. The GenCade model outputs a solution file for each inlet defined in the domain. These solution files (*.irv files), contain sediment volumes and transport rates (in and out) for each of the structures/shoals related to the inlet including the ebb shoal, the flood shoal, and attachment and bypass bars both upstream and downstream of the inlet. The SMS package will search the directory for inlet solution files matching the specified inlet names and populate a list of functions that can be plotted for the selected inlet. Each function will be assigned a unique color to be plotted. The user also selects a time range to plot the volume (or transport rates). To create this plot:
 1. Run a simulation that includes an inlet to generate at least one inlet solution file (*.irv).
 2. Click on the **Plot Wizard** tool.
 3. Select the *Inlet TS* option and click the **Next** button.
 4. Choose the plot options to select the inlet, functions and time range to be plotted.

The user may interact with the plot by dragging a zoom box over an area or right clicking on the plot. Right clicking invokes a menu to select the plot options, frame the data, set the plot attributes, or export the plot data in tabular or image form.

Related Topics

- GenCade

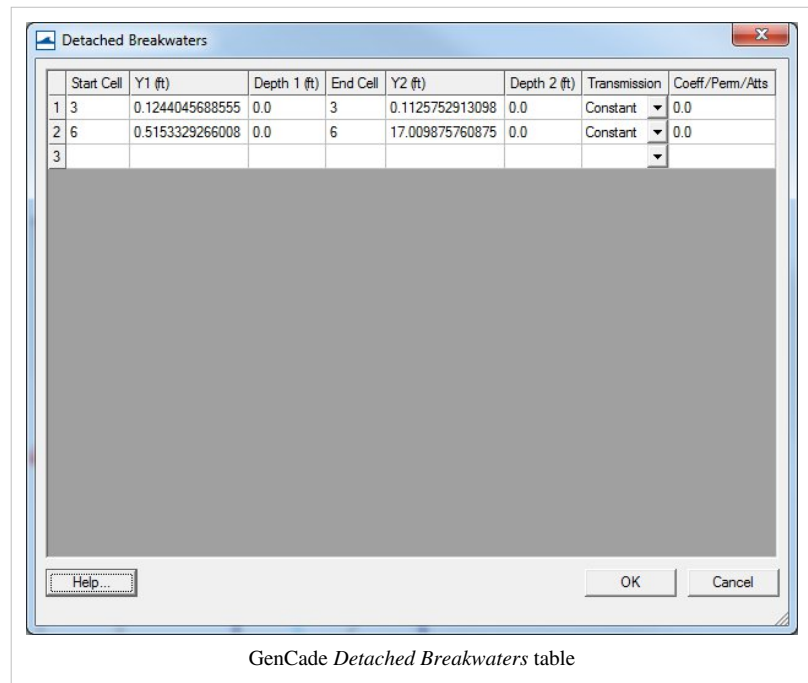
GenCade Structures

GenCade has the capability to model the structures listed below. The *GenCade* menu includes commands to view a table of the structures of a specific type included in a simulation. Some structures can also be selected interactively if they have a geographic attribute such as length.

Breakwaters

Parameters for characterizing breakwaters include start and end location, depth, and transmission coefficient. To define a breakwater, proceed with the following steps:

1. Select the **Create Breakwater** tool and click with the left mouse button to specify the start and end location.
2. Select the menu option *GenCade* | **Edit Breakwaters**. The GenCade *Detached Breakwaters* dialog will appear allowing the entry of breakwater parameters. At this point, the user should verify that the parameters are correct. Further information concerning the selection of breakwater parameters is provided in the GenCade model documentation.



Seawalls

Parameters for characterizing seawalls include start and end location. To define a seawall, proceed with the following steps:

1. Select the **Create Seawall** tool and click with the left mouse button to specify the start and end location.
2. Select the menu option *GenCade* | **Edit Seawalls**. The GenCade *Seawalls* dialog will appear allowing the entry of seawall parameters. At this point, the user should verify that the parameters are correct.

Seawalls

	Start Cell	Y1 (ft)	End Cell	Y2 (ft)
1	4	-0.148428168059	6	-10.40102474531
2				

Help...

OK

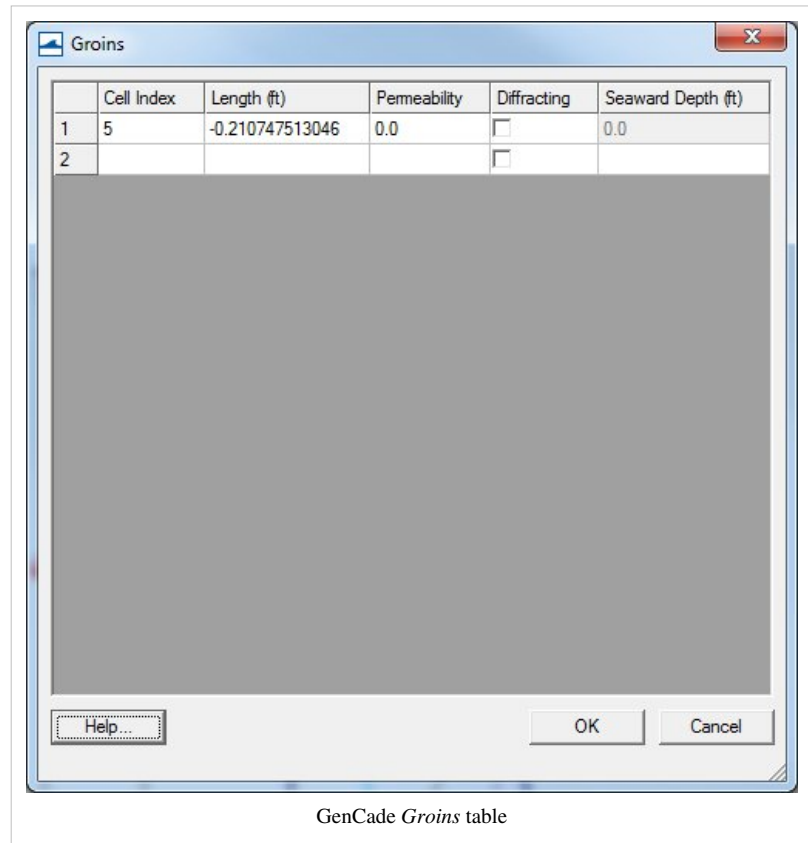
Cancel

GenCade *Seawalls* table

Groins

Parameters for characterizing groins include length, permeability, diffraction, and depth. To define a groin or jetty, proceed with the following steps:

1. Select the **Create Jetty/Groin** tool and click to specify the jetty or groin location. The groin or jetty will be created perpendicular to the grid.
2. Select the menu option *GenCade* | **Edit Groins**. The *GenCade Groins* dialog will appear allowing the entry of groin and jetty parameters. At this point, the user should verify that the parameters are correct. Further information concerning the selection of groin and jetty parameters is provided in the GenCade model documentation.



Inlets

To create an inlet, create a feature arc, right-click on it, select **Attributes** and change the attribute to *Inlet*. Inlets have a name, a left and right bypass coefficient, cell positions, shoal volumes, jetties, and dredging options.

Cell Positions

Cell beginning and ending positions can specified for the inlet and its left and right bypass. These can be set by creating attachment arcs by the inlet before mapping to a 1D grid. If there are two inlets with one attachment arc in between, the attachment arc will be associated to the inlet closest to the arc.

Shoal Volumes

The initial and equilibrium shoal volumes can be specified for the ebb, flood, bypass (left and right) and attachment (left and right) shoals.

Jetties

The jetties to the left and right of the inlet can be given a length, permeability, and a seaward depth. Also, whether the jetty exists and is diffracting can also be specified.

Dredging Events

Dredging events can have a beginning and ending date, which shoal is to be mined, and the amount to be removed from the specified shoal between the beginning and ending date.

Inlets (Reservoir Model and Jetties)

	Name of Inlet	Cell	(yd³)	Control	Manage	Y Left (ft)	Y Right (ft)
1	D	Position...	Volume...	Jetties...	Dredging...	1.0	0.05
2		Position...	Volume...	Jetties...	Dredging...		

Help...

OK

Cancel

GenCade Inlets table

Related Topics

- [GenCade](#)

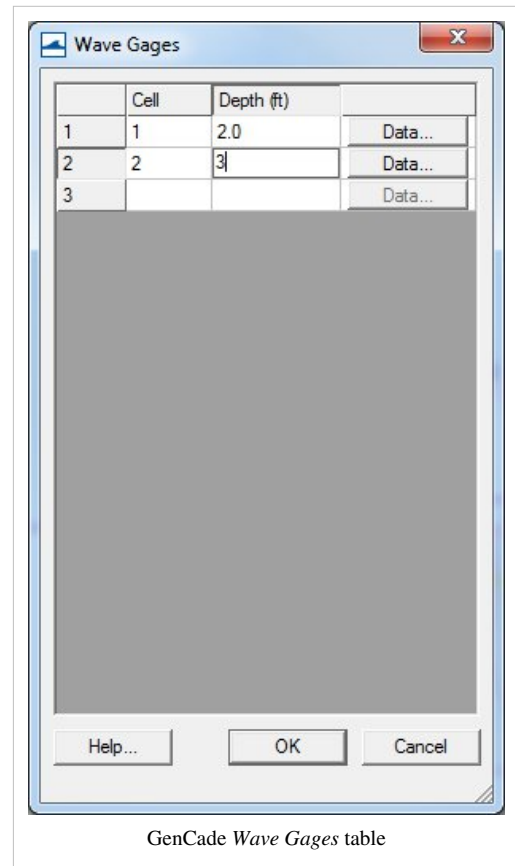
Wave Gages

Wave gages provide boundary condition information to a GenCade simulation. The wave height, period and direction influence the transport of material on the shoreline and the shoreline morphology. The GenCade model allows the user to define multiple wave gages, providing for spatially varied driving force.

The wave gage is created either as an *Attribute* on a feature point in the GenCade coverage, which will be mapped to a specific cell in the grid, or it may be specified directly on the grid. For each wave gage the user also specifies a depth and wave events. Each wave event has a date and time, a H0, a period, and a direction. The model computes the wave conditions for each cell in the grid by interpolating the data from the neighboring wave gages.

Wave gages specified in a GenCade coverage will be displayed at the feature point in the coverage, and will appear when the 1D grid (not grid frame) is created on the associated cell in the grid.

If the user changes attributes of a wave gage on a feature point, these changes do not impact the simulation unless the attributes are mapped to the grid again (Map → 1D grid). Wave gages can be edited using the **Edit Wave Data..** option from the *GenCade* menu after the grid is created and still take effect.



Wave Events

Wave events can be edited by clicking on the **Data...** button when editing the wave gage. This can be done for either a gage on a 1D grid, or a gage at a feature point.

The direction of a wave, for a given wave event, for a given wave gage may be specified in a meteorological, oceanographic, or Cartesian convention. Also, the wave direction can be specified in a shore normal convention as long as a 1D grid or 1D grid frame is present in the coverage which contains the wave gage. The default convention for wave gages is meteorological.

Wave Data for GenCade

Wave conditions, including significant wave height, peak period, and direction, drive the transport forces that impact coastal morphology. GenCade requires wave data for at least one location in the domain, and supports options for the user to specify wave data at multiple locations in the domain. Each wave gage includes:

- A location – (X,Y) coordinate as a feature point and Cell ID as an input to the model.
- A depth – the water depth at the gage.
- A time series of wave data events – The user obtains these events either from a buoy or from a local wave transformation model.

Issuing the command to edit wave data invokes the *Wave Gages* dialog. This dialog lists the gauge cell IDs and the depths. It also includes a button for each gage to view/edit the wave data at that gage.

Clicking on the **Data ...** button for any of the gages brings up the *Wave Events* dialog. This dialog displays the time, height, period and direction (in shore normal coordinates) for the gage. The data is in a spread sheet, so it may be copied and pasted from other spread sheet data. There is also an **Import** button that can be used bring up the import wizard to bring in wave events data stored in a text file. The data is saved and passed to the model in the wave file.

Related Topics

- GenCade

5.9. STWAVE - Steady State Spectral Wave

STWAVE

STWAVE	
Model Info	
Model type	Model for nearshore wind-wave growth and propagation.
Developer	Jane Smith ^[1]
Web site	STWAVE web site ^[2]
Tutorials	General Section • Data Visualization • Observation Models Section • STWAVE Several sets of sample problems and case studies are available. These include: • Model Validation cases from the STWAVE website ^[2]

STWAVE is a steady-state, finite difference, spectral model based on the wave action balance equation. STWAVE is written by the U.S. Army Corps of Engineers Waterways Experiment Station (USACE-WES).

Functionality

STWAVE simulates depth-induced wave refraction and shoaling, current-induced refraction and shoaling, depth- and steepness-induced wave breaking, diffraction, wave growth because of wind input, and wave-wave interaction and white capping that redistribute and dissipate energy in a growing wave field. The purpose of STWAVE is to provide an easy-to-apply, flexible, and robust model for nearshore wind-wave growth and propagation. Recent upgrades to the model include wind, surge and friction fields (spatially varied). Also, wind and surge fields can be temporally varied. The method of analysis used by the STWAVE code along with the file formats and input parameters are described in the STWAVE documentation. SMS supports both pre- and post-processing for STWAVE.

The new full-plane version of STWAVE is not a replacement for the half-plane version, but a supplement. The half-plane version will always have an advantage of substantially lower memory requirements (~ two orders of magnitude) and faster execution. The half-plane limitation is generally appropriate for nearshore coastal applications, with the exception of enclosed or semi-enclosed bays, estuaries, and lakes where seas and swells may oppose each other or there is no clear “offshore” direction. The full-plane version allows wave input on all boundaries and wave generation from all directions.

Saving STWAVE

When you do a *File* | **Save As...** the following files get saved in the *.sms

- *.mat referenced to new save location
- stw_grds.h5 referenced to new save location
- spec_grds.h5 referenced to new save location
- *.grd referenced to new save location
- *.ctl referenced to new save location
- *.sol referenced to a folder in the new save location

Using the Model / Practical Notes

- A grid for use with STWAVE is created and edited in SMS using the Map Module.
- The modeling parameters required by STWAVE are generated and applied to the mesh using commands grouped in the STWAVE menu.
- Post processing of solution data generated by STWAVE is done using the generic visualization tools of SMS.
- Wind can be entered in the STWAVE model control as either a constant value or by specifying an existing Cartesian Grid dataset.
- STWAVE requires metric units. All data in SMS needs to be in metric units before running STWAVE.
- Water depths are defined as positive numbers and land elevations are negative numbers.

Graphical Interface

The STWAVE Graphical Interface contains tools to create and edit a STWAVE simulation. The simulation consists of a geometric definition of the model domain (the grid) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the Cartesian Grid Module and setting the current model to STWAVE. If a grid has already been created for a STWAVE simulation or an existing simulation read, the grid object will exist in the Project Explorer and selecting that object will make the Cartesian grid module active and set the model to STWAVE. See Creating 2D Cartesian Grids for more information.

The interface consists of the Cartesian grid menus and tools augmented by the STWAVE Menu. See STWAVE Graphical Interface for more information.

External Links

- Aug 2007 ERDC/CHL CHETN-I-76 Modeling Nearshore Waves for Hurricane Katrina [3]
- Aug 2007 ERDC/CHL CHETN-I-75 Full-Plane STWAVE with Bottom Friction: II. Model Overview [4]
- Sep 2006 9th International Workshop On Wave Hindcasting and Forecasting Jane McKee Smith Modeling Nearshore Waves For Hurricane Katrina [5]
- Mar 2006 ERDC/CHL CHETN-I-71 Full Plane STWAVE: SMS Graphical Interface [6]
- Dec 2003 ERDC/CHL CHETN-IV-60 SMS Steering Module for Coupling Waves and Currents, 2: M2D (now know as CMS-Flow) and STWAVE [9]
- Jun 2002 ERDC/CHL CHETN-I-66 Grid Nesting with STWAVE [7]
- Jun 2002 ERDC/CHL CHETN-IV-41 SMS Steering Module for Coupling Waves and Currents, 1: ADCIRC and STWAVE [8]

Please see this forum post ^[9] for an explanation of ADCIRC and STWAVE steering

- Sep 2001 ERDC/CHL CHETN-I-64 Modeling Nearshor Wave Transformation with STWAVE [7]

Grid Nesting

Grid nesting refers to the ability to take output at specified locations from one grid to use for boundary conditions on another. We refer to the source grid as the parent grid and the grid using the boundary conditions as the child grid. This approach is often used in order to cover a large area with a coarse grid with a more refined grid in a specific area of interest.

Within SMS you can setup nesting for STWAVE using a STWAVE parent grid or a WAM parent grid. The functionalities are quite similar with some minor changes specific to each type of setup.

Nesting STWAVE → STWAVE

You can do nesting with full or half plane grids but the parent and child grids must be consistent (you can't nest a half-plane STWAVE grid within a full-plane grid). You can specify the nesting options by clicking on the options button in the *STWAVE Model Control* dialog.

Requirements

The following conditions must be met to create nested grids:

1. The cartesian grids being used must be composed of square cells
2. The offshore boundary of the child grid should be entirely contained in the parent grid.

Options

- **Select Nesting Grid** – This command indicates that the current grid should obtain input spectral values from a parent grid. By selecting this option the current grid is nested in the grid selected in the combo box below.
- **Type of Boundary Interpolation** – Choose to use either linear boundary interpolation or morphic boundary interpolation. The morphic interpolation method was developed to preserve the shape of the directional distribution. It is appropriate for climatic wave transformation studies where a parametric spectral shape is applied based on wave parameters.

When doing STWAVE → STWAVE nesting, some of the parameters in the child grid must match those in the parent grid. These include whether or not the model is using time steps, whether the model is using a reference time, whether the model is full or half plane, and the identifiers for each of STWAVE snaps (time steps or cases). These options will be disabled in SMS once you have chosen to do nesting.

Nesting WAM → STWAVE

Even though the output spectra from WAM is always full-plane, a full or half plane STWAVE grid may be nested within a WAM grid. You can set the nesting options by clicking on the options button in the model control dialog. Since WAM models are always run at specific date/times, child STWAVE models always use time steps with reference times.

Options

- **WAM Simulation** – The WAM simulation which will provide the input spectra.
 - **WAM Grid** – The WAM grid within the simulation to extract spectra.
 - **Start time** – The first time to run STWAVE. This time must be an output time specified in the WAM grid options.
 - **End time** – The last time to run STWAVE. This time must be an output time specified in the WAM grid options.
-

- **Interval** – The snap interval must be set so each snap will hit a WAM output time. For example if WAM data was being output every hour, the interval may be 1 hour, 2 hours, or 3 hours but not 1.5 hours.
- **Type of Boundary Interpolation** – Choose to use either linear boundary interpolation or morphic boundary interpolation. The morphic interpolation method was developed to preserve the shape of the directional distribution. It is appropriate for climatic wave transformation studies where a parametric spectral shape is applied based on wave parameters.

The start time, end time, and interval will determine the number of snaps used by STWAVE and the times associated with each.

Spectral Sites

Regardless of the type of nesting that is going to take place, spectra need to be provided to STWAVE along the boundaries with input spectra (see Spectral Events Dialog). Both WAM and STWAVE support the ability to output computed spectra at specified locations. These locations can be specified manually using cell attributes but preferably are automatically generated. To automatically generate these output locations, click on the spectral sites button in the model control dialog (the nesting options should have already be set). Be sure that if you are using a full-plane model, the sides which will have spectra input have already been identified. The dialog that comes up will allow you to choose between different options to control the number of nesting cells along the offshore boundary. The more nesting cells you have the better you can capture differences along the boundary. However, more nesting cells require more disk space and time to read and write the files.

STWAVE Graphical Interface

The STWAVE graphical interface includes tools to assist with creating, editing, and debugging a STWAVE model. The STWAVE interface exists in the Cartesian Grid Module.

Model Control

The *STWAVE Model Control* dialog is used to setup the options that apply to the simulation as a whole. These options include time controls, run types, output options, global parameters, print options and other global settings.

Running the Model

The STWAVE files are written automatically with the SMS project file or can be saved separately using the *File | Save STWAVE* or *File | Save As* menu commands. See STWAVE Files for more information on the files used for the STWAVE run.

STWAVE can be launched from SMS using the *STWAVE | Run STWAVE* menu command. A check of some of the common problems called the Model Checker is done each time the model is launched, or by selecting the *STWAVE | Model Check* menu command.

STWAVE Menu

See STWAVE Menu for more information.

Visualizing Results

The solution files for STWAVE are XMDF files. These files can be opened in the normal SMS interface. To view the observation or nesting spectral output simply open the files (outside of the *Spectral Energy* dialog), open the *Spectral Energy* dialog, and select the spectral grid. This will open the spectral grid for viewing in the *Spectral Energy* dialog. Make sure to set the original grid back before leaving the dialog to ensure the model does not change.

Related Topics

- Create a 2D Cartesian grid

STWAVE Iterations Control Dialog

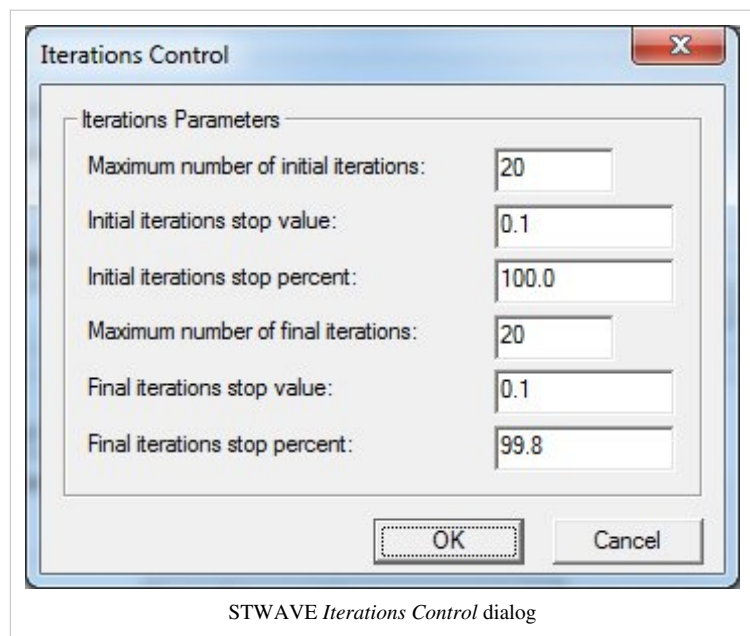
The STWAVE *Iterations Control* dialog is reached through the *STWAVE Model Control* dialog. Here the number, stop value, and stop percent of iterations can be set. The dialog contains the following options:

Iterations Parameters

- Maximum number of initial iterations
- Initial iteration stop value
- Initial iteration stop percent
- Maximum number of final iterations
- Final iteration stop value
- Final iteration stop percent

Related Topics

- STWAVE Model Control



STWAVE *Iterations Control* dialog

STWAVE Menu

The items in the *STWAVE* menu in the Cartesian Grid Module are described below:

STWAVE Menu Commands

The *STWAVE* menu has the following commands:

- **Assign Cell Attributes** – opens the *STWAVE Cell Attributes* dialog.
- **Model Control** – opens the *STWAVE Model Control* dialog.
- **Run STWAVE** – launches the finite element analysis engine.

Obsolete Menu Commands

The following commands are no longer available in current versions of SMS.

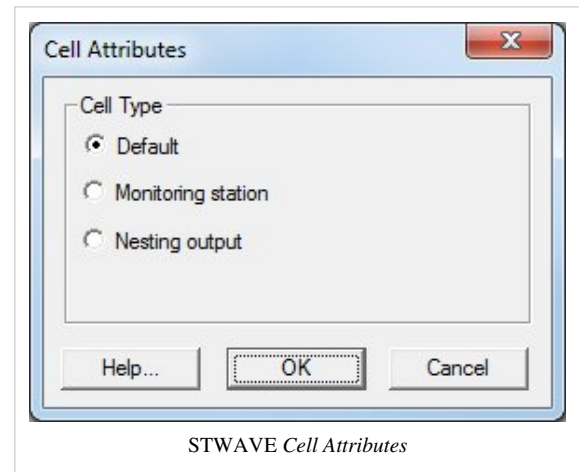
- **Spectral Energy** – opens the *Spectral Energy* dialog.

Assign Cell Attributes Command

The *STWAVE Cell Attributes* dialog is used to assign a cell type to selected Cells. It is opened using the menu command *STWAVE | Assign Cell Attributes*. The following cell types can be assigned:

Default

Cells are no longer assigned as Land or Ocean types but are assigned as Default where their elevation values are used to simulate whether flow exists or not. If no elevation data is available for part of the grid area, it is important to specify an extrapolation value of a negative number. Extrapolation values can be specified in the *Interpolation* dialog while doing **Map → 2D Grid**.



STWAVE Cell Attributes

Monitoring Station

Spectral output is generated for cells designated as monitoring station cells.

Nesting output

Spectral output is generated for nesting output cells to be used as input for a nested child grid.

Displaying Cell Attributes

The symbols and colors used to identify the attributes assigned to cells can be changed in the *STWAVE Display Options*.

Run STWAVE Command

After a simulation has been read or saved, SMS can launch the finite element analysis engine. To run STWAVE:

1. Select *STWAVE | Run STWAVE*. A dialog appears showing the engine that will execute.

2. If the executable program is the wrong version, or if you are given the message that it was not found, then click the file browser icon and choose the correct version of the program that should run.
3. Click the **OK** button or press the *ENTER* key.

A window will appear which displays various information as STWAVE runs. You should not need to type anything in this window. If the window prompts you for filenames, you need an updated version of the STWAVE model executable. When STWAVE has finished, you will be prompted to press the *ENTER* key. If the window goes away before doing this, STWAVE encountered a problem and crashed.

Related Topics

- STWAVE

STWAVE Model Control

The **Model Control...** command in the *STWAVE* Menu opens the *STWAVE Model Control* dialog. This dialog is divided into sections for different types of parameters which are used by the model as it runs. These include:

Grid Definition

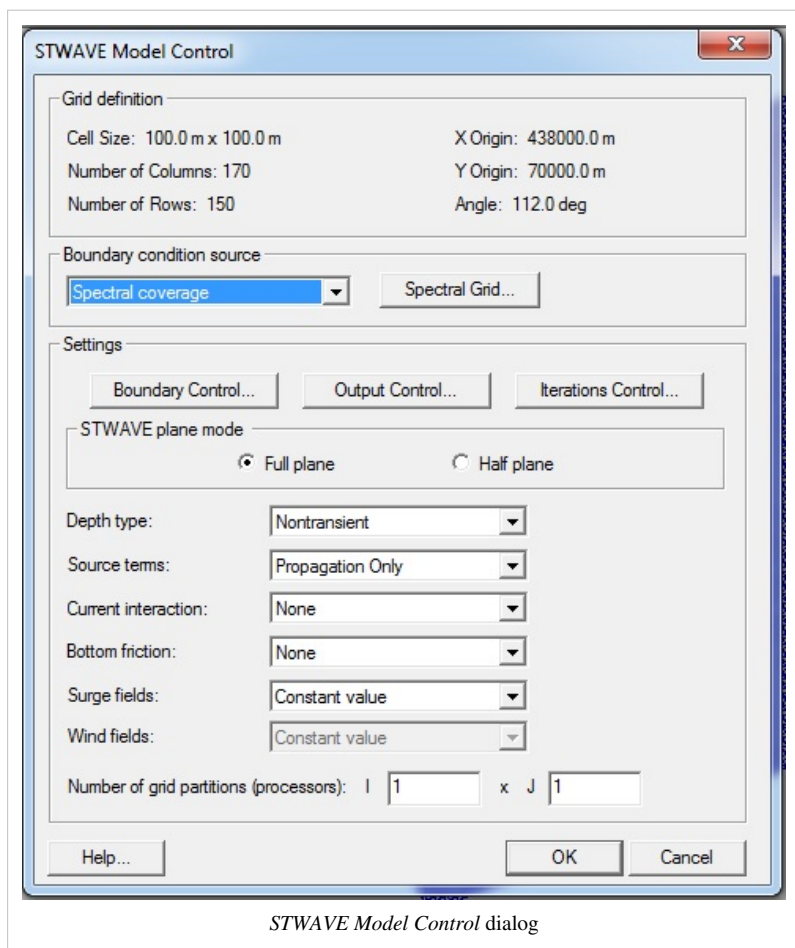
The grid definition section of the model control reports defining characteristics of the grid including:

- Cell size
- Number of rows and columns
- X, Y origin
- Angle of rotation

Boundary Condition Source

The boundary condition for STWAVE consists of one or more energy spectra entering on one or more open edges of the grid. Traditionally, the user specifies a single spectra that is assigned to all of the cells on the offshore edge. In this situation the user specifies one spectra for each wave case or time snap being simulated. Alternately, the input spectra can be interpolated to the cells on the offshore edge(s) from a parent STWAVE or WAM grid.

- **Source combo box** – This combo box lists the types of spectral sources. The user chooses from "Spectral Coverage", "STWAVE run", and "WAM run". If the option is "Spectral Coverage", the user must provide the spectra in a spectral coverage as well as a definition for the spectral grid to be used when exporting STWAVE simulation files. The grid definition can be specified by selecting the **Spectral Grid...** button. Both of the other



STWAVE Model Control dialog

options indicating that spectra will be coming from a parent grid and the associated options and spectral sites buttons become enabled. If an option is selected that is not valid (i.e. choosing an "STWAVE Run" when only a single grid exists in the project so not parent grid exists), SMS generates a message indicating the option is invalid and resets the source.

- **Options...** – This button brings up a dialog which allows the user to specify information for using output from a STWAVE or WAM parent grid as spectral input for this grid. When a parent STWAVE grid is desired, the user specifies which grid is the parent and what type of spectral interpolation should be used.
- **Spectral sites ...** – This button invokes a dialog that creates nesting sites in the parent grid. Spectral output sites for nesting may also be specified explicitly by selecting cells in the parent grid and assigning attributes for those cells. See grid nesting for more information.

Settings

- **Boundary Control** – Opens the *Spectral Events* dialog used to specify the STWAVE Boundary Conditions.
- **Output Control** – Opens the *STWAVE Output Control* dialog.
- **Iterations Control** – Opens the *STWAVE Iterations Control* dialog.
- **Full Plane** – Consult the STWAVE model documentation for more information.
- **Half Plane** – Consult the STWAVE model documentation for more information.
- **Depth Type** –
- **Source Terms** – Specifies whether STWAVE should generate waves using wind and the input spectrum or whether we are using the wave spectrum.
- **Current Interaction** – Specifies whether currents will be used as input to the model.
- **Bottom Friction** – Specifies which friction type is to be used, if any.
- **Surge Fields** – Specifies whether surge fields are being used, or a constant tidal offset.
- **Wind Fields** – Specifies whether wind fields (spatially varied wind) are being used, or a constant wind magnitude and direction.
- **Number of Iterations** – Number of iterations to run. (Full Plane Only).
- **Select buttons** – The select buttons by next to the current, bottom friction, surge, and wind options are used to specify a dataset on the grid to use as the desired dataset. These buttons open up the *Select Dataset* dialog. These datasets must be created prior to entering the model control dialog.

Using Datasets as Input

Datasets may be used as input to STWAVE to represent spatially varied currents, bottom friction (either manning's or JONSWAP), surge, or wind. Bottom friction cannot vary from snap to snap. The other dataset types can have different values for each snap.

Datasets are chosen within the model control by clicking on the **Choose** button to the right of the dataset choice. The dataset values for each timestep will be the values for the time that the time step occurs. The time steps for the dataset chosen do not need to match the time steps specified for the simulation. The dataset values passed to STWAVE can be interpolated from a dataset with values at different time steps than is being used by the model.

If the simulation is not using time steps, the datasets are chosen in the *Spectral Events* dialog. In this situation, a steady state dataset is chosen for each snap.

Related Topics

STWAVE Menu

STWAVE Output Control Dialog

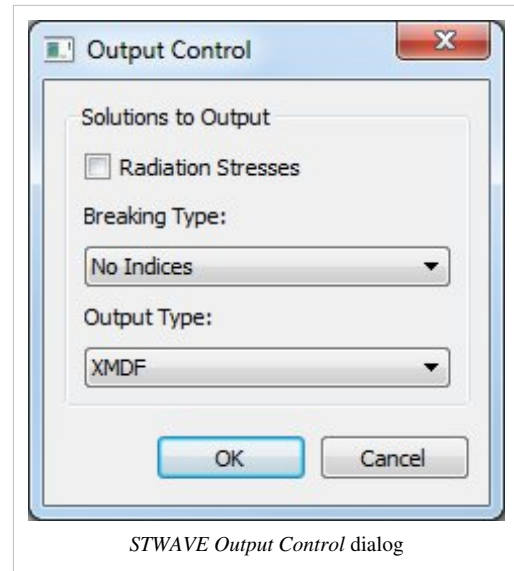
The *STWAVE Output Control* dialog is used to specify the solutions files STWAVE should output.

The following datasets are always written:

- **Height**
- **Period**
- **Direction**
- **Wave Vector**(magnitude of height with wave direction)
- **1/fma** – Period that represents the inverse of the spectral peak frequency redefined for local wind growth cases

The options the user can set include:

- *Radiation stresses* – If this is on, radiation stresses will be calculated and output into datasets
- *Breaking* – This will output a dataset representing the wave breaking. For full-plane, this can be no indices (off) or write indices which gives a value of 1 where breaking occurs and 0.0 otherwise. Half-plane has the full-plane options and has an additional option to calculate energy dissipation. This will give a dataset of energy dissipation.
- *Output type* – SMS supports the ability to write ASCII files, XMDF files or both. ASCII files refer to STWAVE version 6 global dataset files and XMDF files are datasets written using the XMDF library built upon HDF5. XMDF is the recommended file format because they are much faster to read/write. XMDF is only available on windows platforms.



Related Topics

- STWAVE Model Control

5.10. WAM - Wave Prediction Model

WAM

The global ocean WAve prediction Model called WAM is a third generation wave model. WAM predicts directional spectra as well as wave properties such as significant wave height, mean wave direction and frequency, swell wave height and mean direction, and wind stress fields corrected by including the wave induced stress and the drag coefficient at each grid point at chosen output times.

The model is continually updated to incorporate the latest results of research. The verification has been carried out in three areas National Oceanic and Atmospheric Administration (NOAA) moored buoys are available on the global Telecommunications System (GTS). It is hoped that the buoys chosen will allow the identification of both successes and failures in WAM model physics and will minimize shortcomings due to sub-grid scale effect.

Features

The present version of WAM makes the following assumptions:

- Time dependent wave action balance equation.
- Wave growth based on sea surface roughness and wind characteristics.
- Nonlinear wave and wave interaction by Discrete Interaction Approximation (DIA).
- Free form of spectral shape.
- High dissipation rate to short waves.

Graphical Interface

The WAM Graphical Interface contains tools to create and edit a WAM simulation. The simulation consists of a geometric definition of the model domain (the grid) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the Cartesian Grid Module and setting the current model to WAM. If a grid has already been created for a WAM simulation or an existing simulation read, the grid object will exist in the *Project Explorer* and selecting that object will make the Cartesian Grid Module active and set the model to WAM. See *Creating 2D Cartesian Grids* for more information.

The interface consists of the Cartesian Grid Module menus and tools augmented by the *WAM Menu*. See *WAM Graphical Interface* for more information.

External Links

- Influence of Marsh Restoration and Degradation on Storm Surge and Waves ^[1]
- Dynamics and modelling of ocean waves By G. J. Komen, L. Cavaleri, M. Donelan, S. Hasselmann, P. A. E. M. Janssen

WAM Map to Raster Utility

Map to Raster

The *Map to Raster* utility has been updated to allow for conversions between big endian to little endian. Most machines are little endian and therefore users should just use the utility like they did before. However, if the InOut/Map files were created from a machine that is big endian the user should convert to little endian. To convert: copy the InOut/Map files to a little endian machine. Then from the command line run `map_to_rast.exe` passing "big_endian" as an argument. It will look something like this: `c:\>map_to_rast.exe big_endian`. Other command line arguments can also be used outlined below:

- native

Specifies that unformatted data should not be converted.

- big_endian

Specifies that the format will be big endian for integer data and big endian IEEE floating-point for real and complex data.

- cray

Specifies that the format will be big endian for integer data and CRAY* floating-point for real and complex data.

- fdx (Linux, Mac OS X)

Specifies that the format will be little endian for integer data, and VAX processor floating-point format F_floating, D_floating, and X_floating for real and complex data.

- fgx (Linux, Mac OS X)

Specifies that the format will be little endian for integer data, and VAX processor floating-point format F_floating, G_floating, and X_floating for real and complex data.

- ibm

Specifies that the format will be big endian for integer data and IBM* System\370 floating-point format for real and complex data.

- little_endian

Specifies that the format will be little endian for integer data and little endian IEEE floating-point for real and complex data.

- vaxd

Specifies that the format will be little endian for integer data, and VAX* processor floating-point format F_floating, D_floating, and H_floating for real and complex data.

- vaxg

Specifies that the format will be little endian for integer data, and VAX processor floating-point format F_floating, G_floating, and H_floating for real and complex data.

Related Topics

- [WAM](#)

5.10.a. WAM Graphical Interface

WAM Graphical Interface

Overview

The WAM interface supports the wave model WAM.

WAM models are built by creating WAM grids, WAM simulations (right-click in blank area of project explorer), and dragging the WAM grids you want to use into each WAM simulation. There are options specific to each grid in a simulation as well as the simulation as a whole (model control).

WAM has the ability to use nested grids to improve resolution in an area of interest.

WAM Menu

The *WAM* menu has the following options:

- **Assign Cell Attributes** – this command will bring up the *Cell Attributes* dialog.
- **Model Check** – runs the WAM model check process.
- **Model Control** – brings up the WAM Simulation Model Control.

Grid Options

Once the WAM Grid has been created, different options have to be provided. In the Grid options dialog, there are three tabs.

- The *General* Tab
- The *Output* Tab
- The *Spatial Input* Tab

See WAM Grid Options for more information.

WAM Simulation

A new WAM simulation is created by right-clicking in the *Project Explorer* and selecting **WAM** from the *New Simulation* menu. The WAM grid is then dragged under the new simulation to create a link. If there are multiple grids in the same project (for example, nested grid and parent grid), all grids should be linked to the simulation.

Multiple simulations can also be created.

Model Control

Once a simulation is created and the grid(s) is linked, the grid options can be edited in the Simulation Model Control which can be accessed by right-clicking on the simulation. Once all information are entered, WAM files can be exported, the project can be saved and WAM can be launched.

See WAM Simulation Model Control for more information.

Model Wrapper

Once all of the simulation information is ready, the WAM files can be exported by right clicking on the simulation. After the WAM files are exported, then the model wrapper can be launched by right-clicking on the simulation and choosing *Launch WAM*. Both steps can be done with the *Save Project, Export, and Launch WAM* option, also under the right-click menu.

SMS will export all WAM files to a folder with the same name as the simulation under the WAM folder in the same directory as the saved SMS project. All wind dataset files will be in a folder called *WindInput* under the simulation folder. All WAM files, except the wind dataset, for each grid in the simulation will be in a folder with the same name as the grid under the simulation folder.

After all the files for a WAM simulation have been exported, the WAM executables can be launched. There are three WAM executables which are, *preproc*, *chief* and *map_to_rast*. These must be executed in order for a given grid. *Preproc* creates the *Grid_Info* and *Preproc_prot* files. *Chief* is the main process of WAM, and it creates the boundary value output files, the integrated parameter files, the restart files, the spectral output files and the *WAM_Prot* file. *Map_to_rast* creates the HDF5 file of the datasets that SMS can read.

SMS will automatically run all grids in a simulation, even nested grids. The model wrapper will run a process for a coarse grids before running the same process for one of the grids nested inside of it.

Nested Grids

- Used to make a finer grid of a particular area that will use input information from the coarse grid.
- To create a nested grid, make a grid frame around the area of the coarse grid to which you wish to make a nested grid. Then, before the nested grid is mapped, check the *Fine Grid* checkbox and select the coarse grid from the *Coarse Grid* combobox. This will change the values for the newly mapped grid, such as the origin and increment, to correspond to cells in the coarse grid.
- The dataset for the wind of the coarse grid may be used for the fine grid. Other datasets, such as for ice and currents, for the fine grid will need to be made in the same manner as the coarse grid, if they are to be used.
- The nested(fine) grid will have to be added to the simulation as well as the parent(coarse) grid.

WAM Wet and Dry Cells

Wet/dry status for WAM cells is determined solely upon the *z* values (represented as a depth). If the depth is positive, the cell is wet. If the depth is negative, the cell is dry.

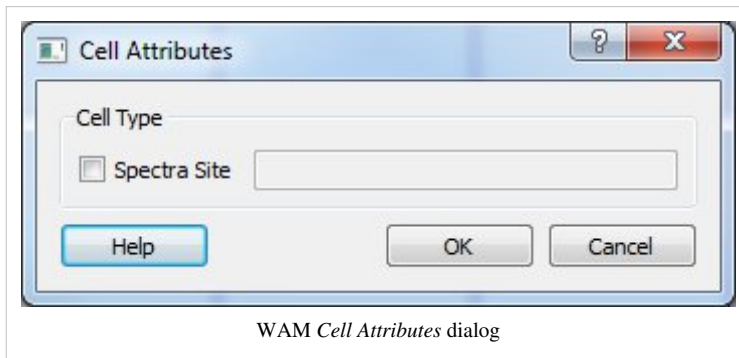
Only wet cells may be used as spectra sites.

To display wet/dry cells: Select the *WAM grid Display*→*Display Options* then turn on *land and ocean* cell. For best results also turn off *contours*.

The following operations may change the wet/dry status of a WAM cell since they are changing the depth values.

1. Select 1 or more WAM cells and then change the *Z* value found at the top.
2. Select and right click a WAM cell. Choose **Interpolate Bathymetry...** Enter a "single value" on the left side and press **Ok**
3. When converting a WAM map coverage to a WAM grid **Map**→**2DGrid** whatever value is in the *Depth Options'* section will be converted.

WAM Cell Attributes



Related Topics

- WAM

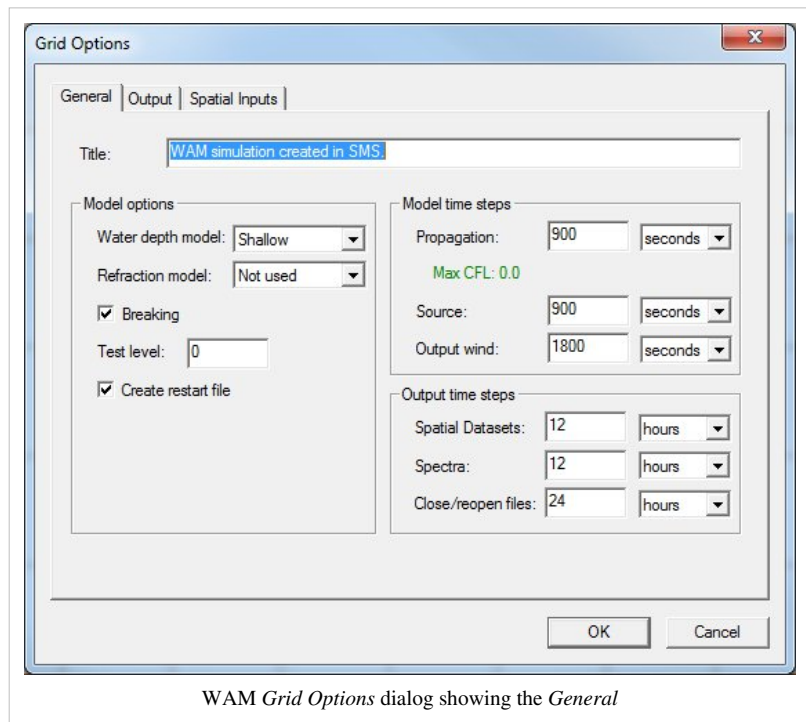
WAM Grid Options

The *Grid Options* dialog is accessed by clicking on **Model Control** command in the *WAM* menu. The *Grid Options* dialog contains three tabs which are as follows:

General Tab

Model options

- Water depth model – Choosing a shallow depth model specifies the use of the full dispersion equation. Dispersion equation is $w^2 = gk \tanh(kh)$ where w is radial frequency, g is acceleration due to gravity, k is the wave number, and h is the water depth. Computes all phase and group celerity where there is depth dependence.
- Refraction model – If running in deep water, specify "Not used." It is not recommended to use if grid resolution is greater than 3-min (0.05 deg) on a spherical grid.
- Breaking – It is recommended that breaking be used when the water depth model is "Shallow." This will implement limited breaking within the WAM model.



Model time steps

- Propagation time step – The propagation time step is controlled by the CFL condition. The CFL condition requires that the numerical solution cannot move faster than the group speed of the waves. It is recommended that the other model time steps and output time steps be multiples of the propagation time step.
- The CFL criteria must be less than 1.0 for WAM to run successfully. The estimated CFL value is shown below the time step. The value shows in green if the settings meet the criteria or red if not. The CFL computation is dependent upon whether deep or shallow water depth model is being used as well as the refraction model. Currently the CFL estimation in SMS does not account for currents but this is used within WAM. Therefore, the CFL estimate may look okay in SMS but fail when running WAM. If this occurs further reduce the propagation time step or change some of the other options.
- Source time step – This is the time step for the source term integration time step. Increasing this value can greatly decrease run times. However, it should be used with caution. It is recommended that the source term time step be no greater than 10 times the propagation time step.

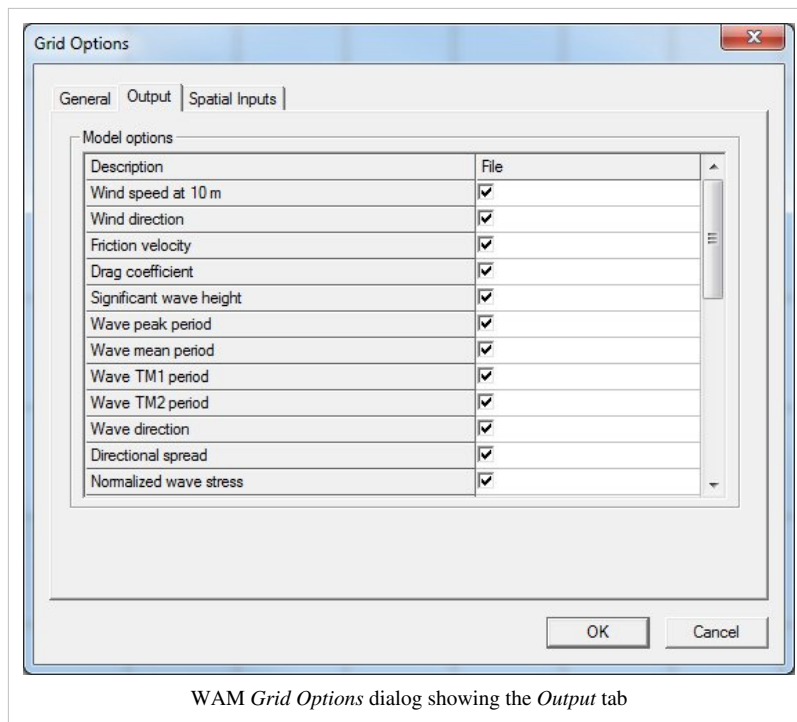
Output time steps

- Spatial datasets – This is the frequency of output that will be generated that can be loaded into SMS as datasets. The WAM file it creates is the "integrated parameter map file."
- Spectra – This is the frequency of output for spectra data at defined observation cells.
- Close/Reopen – When writing files on a computer, often data being written is stored temporarily in RAM before being written to disk. If a file is in this state and WAM crashes, the final data may not be complete. Closing and reopening the output files forces the file system to complete writing the data to file.

Output Tab

The *Output* tab consists of different Model options.

- Sea waves: waves generated by the wind blowing at the time, and in the recent past, in the area of observation.
 - Swell waves: waves which have travelled into the area of observation after having been generated by previous winds in other areas. These waves may travel thousands of kilometres from their origin before dying away. There may be swell present even if the wind is calm and there are no 'sea' waves.
-



WAM Grid Options dialog showing the Output tab

Spatial Inputs Tab

WAM uses spatial input for ice cover, currents, and wind. The *Spatial Inputs* section of the *WAM Model Control* is used to manage these datasets.

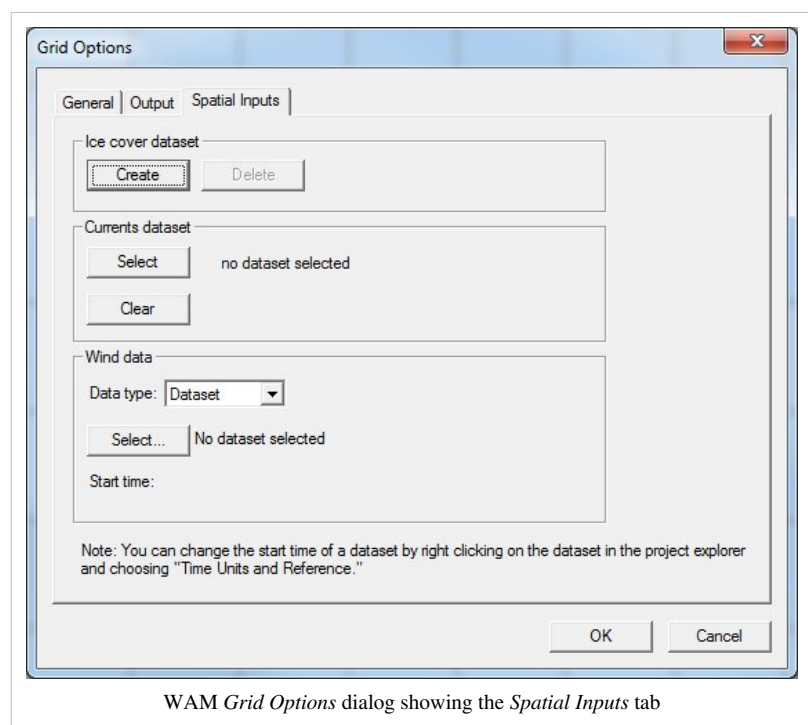
Ice Cover

Dataset

The ice cover dataset is an editable scalar dataset. After creating the dataset using the *Editable Datasets* dialog accessed from the Model Control, the dataset will be displayed in the project explorer in a folder named *Editable Datasets*. Only one ice cover dataset may exist at a time.

File

Rather than using a dataset you can specify the file name for ice.



WAM Grid Options dialog showing the Spatial Inputs tab

Currents

Dataset

The currents dataset is specified to be any vector dataset on the current grid. The dataset and time step must both be selected. The selection may be removed by clicking on the **Clear** button. This deselects the dataset as the specified currents dataset, but it does not remove the dataset from the SMS project.

File

Rather than using a dataset you can specify the file name for currents.

Wind

Dataset

The winds dataset can be set to any vector dataset that belongs to the current grid or another grid whose angle is zero and whose extents cover the current grid. Nested grids may use the wind dataset of the parent grid. The time steps in the dataset used must match those used with the WAM simulation.

File

PBL can be used to create an input wind file for WAM by selecting "WAM input" in the PBL Model Control. The fort.12 file created by PBL may be selected as the wind data. Note: This option is only useful if you have access to the "Planetary Boundary Layer" model which isn't available through SMS.

Related Topics

- WAM

WAM Simulation Model Control

The *WAM Simulation Model Control* can be accessed by right-clicking on the *Simulation* and selecting **Model Control** from the menu.

General Tab

Starting Frequency Band

This is a very important parameter in WAM. Based on historical testing of the source terms (see Komen et al., 1994) there was a prescribed frequency range to tune the Discrete Interaction Approximation and the other source terms defined in the energy balance equation. This work used a starting frequency band of 0.04177248 Hz. Also the definition of the frequency band range is dictated by:

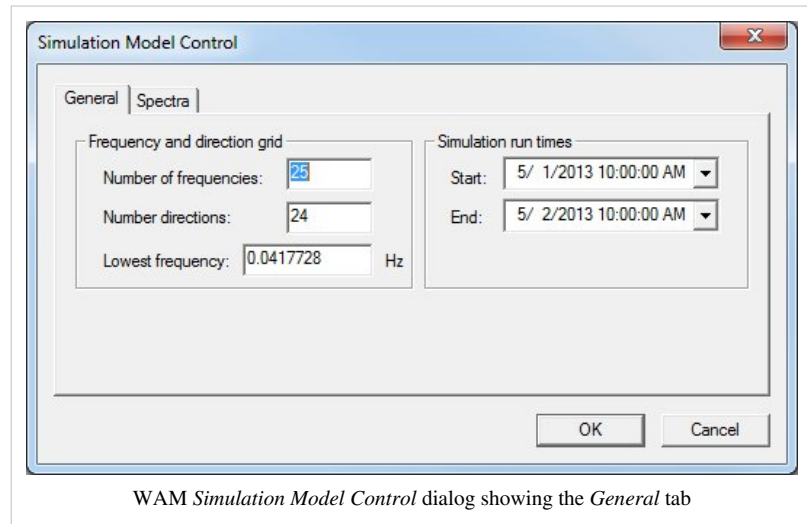
$$f(n+1) = 1.1 * f(n)$$

where

$$n = 1, NFRE - 1$$

(*NFRE* is number of frequency bands)

For many application this starting frequency may be too high for the case of a large-oceanic scale domain like the Pacific Ocean where there is the potential for energy carrying frequencies lower than the designated threshold (i.e. 25-sec peak spectral wave periods, where wave period is the inverse of the frequency). Also, for small-scale fully enclosed bodies of water (i.e. the Great Lakes, reservoirs etc.) the range of wave frequencies is much higher, where the lowest frequency may be on the order of 0.0833-Hz (or 12-sec). The initial frequency value can be changed, upward or downward, as long as the range of frequency values can be defined from the 0.0417728-Hz standard. For the example of the Pacific Ocean implementation, a starting value of 0.03138428-Hz is used, where $f(4) = 0.0417728$ -Hz as defined by the above equation. Setting a higher starting point for a Great Lakes simulation and knowledge of the wave climate an initial value could be 0.050545-Hz. Standard practices use conservatism allowing for energy to be transferred. Also, it is also advantageous to set the first frequency band high enough to minimize the computational time, where the propagation of wave energy is dictated by the group speed of the first frequency band. The lower the frequency, the shorter the stable propagation time step. The number of direction bands is nominally set to 24 or a 15-deg directional resolution of the wave spectra. Selection of a higher directional resolution (or increasing the number of directional bins) will linearly increase computational time because the directional loop is the outer-most loop in all of WAM's calculations.



WAM Simulation Model Control dialog showing the General tab

Simulation run times

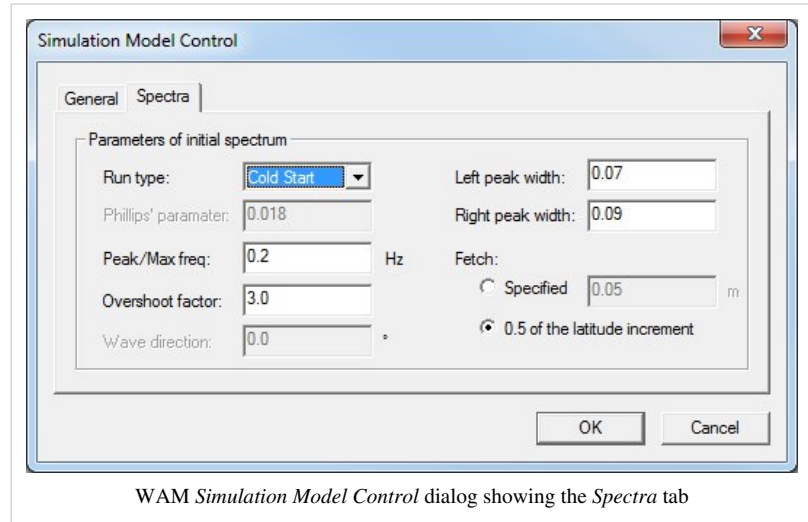
The starting and ending time of the simulation must be entered in this section. The run times must be less or equal to the time available for wind data.

Spectra Tab

The parameters in the *Spectra* tab with the exception of Fetch are Jonswap parameters.

Spectra Sites

Since the spectra data of every cell of the grid at each timestep would take an unreasonable amount of disk space, WAM writes spectra data at user specified cells. These cells are referred to as spectra sites and can be specified within the SMS interface. The spectra output can also be visualized on these cells.



WAM Simulation Model Control dialog showing the *Spectra* tab

Spectra sites can be made at any ocean cell on a Cartesian grid. To make a spectra site, select a cell on the desired Cartesian grid and change the cell attribute to Spectra Site and give it a name. If more than one cell is selected, then the name the spectra sites will receive is the name given concatenated with a number.

After WAM has run, and the spectra files have been read back into SMS, the spectral energy graph of the spectra sites may be viewed by right-clicking on the selected cell where the site is and selecting **Spectral Energy**. The spectra files may be read back into SMS by selecting the grid, and choosing **Read Spectral Files** from the *WAM* menu. The spectra files should be within the grid's folder within the simulation folder.

Related Topics

- WAM

WAM Spectra from STWAVE Grids

Spectra Sites

Since the spectra data of every cell of the grid at each timestep would take an unreasonable amount of disk space, WAM writes spectra data at user specified cells. These cells are referred to as spectra sites and can be specified within the SMS interface. The spectra output can also be visualized on these cells.

Spectra sites can be made at any ocean cell on a Cartesian grid. To make a spectra site, select a cell on the desired Cartesian grid and change the cell attribute to Spectra Site and give it a name. If more than one cell is selected, then the name the spectra sites will receive is the name given concatenated with a number.

After WAM has run, and the spectra files have been read back into SMS, the spectral energy graph of the spectra sites may be viewed by right-clicking on the selected cell where the site is and selecting **Spectral Energy**. The spectra files may be read back into SMS by selecting the grid, and choosing **Read Spectral Files** from the **WAM** menu. The spectra files should be within the grid's folder within the simulation folder.

WAM Spectra From STWAVE Grids

WAM can be used to provide boundary condition information to a spectral wave model such as STWAVE. WAM→STWAVE has a defined and support mechanism for converting WAM spectral output into STWAVE input.

The first part of this process is to define the locations where WAM will generate output for the STWAVE BC. Two or more WAM cells may be used along a STWAVE boundary. The STWAVE grids must exist and the boundaries types specified (which side(s) to apply spectra.

To create spectral sites along the STWAVE boundaries:

1. Verify/Modify the boundary types of a STWAVE Grid.
 - a. Select a STWAVE grid.
 - b. Select menu item **STWAVE** → **Model Control** then press the **Boundary Condition...** button.
 - c. Select the appropriate boundary type for each edge.
 - d. Repeat for each STWAVE grid.
1. Select and right-click on a WAM grid from the tree item on the left of SMS.
2. Select **Create spectral sites...** to bring up the *WAM spectral spectral sites from STWAVE* dialog.
3. Select the grids you would like to generate sites for.
4. Select a spacing option on the right hand side and the number of points/sites.
5. Press **Ok**.

Each STWAVE boundary cell will have a candidate WAM corner (where WAM spectra are output). A candidate corner is the nearest WAM corner to that STWAVE boundary cell. Multiple STWAVE cells may have the same candidate WAM corner. Duplicate WAM corners are automatically removed.

Spacing can be done by either spacing along candidate corners or by specifying an exact number of sites/corners along each boundary.

If spacing by candidate point(s) is chosen:

- Spacing by every 1 candidate points will keep all candidate corners and make them spectra sites.
- Spacing by every 2 candidate points will keep every other corner and make those spectra sites.

Regardless of the option selected at least two corners will be used on each STAVE boundary.

Related Topics

- WAM

6. Riverine and Estuarine Models

6.1. ADH - Adaptive Hydraulics Modeling

ADH

"ADH is a state-of-the-art ADaptive Hydraulics Modeling system developed by the Coastal and Hydraulics Laboratory (CHL), ERDC, USACE ^[1], and is capable of handling both saturated and unsaturated groundwater, overland flow, three-dimensional Navier-Stokes flow, and two- or three-dimensional shallow water problems." [2]

Information about the ADH model can be found at the ADaptive Hydraulics Modeling HomePage ^[2].

Graphical Interface

Dialogs

SMS provides the capability to view and edit model parameters via dialog boxes specific to the ADH interface. The following are descriptions of important windows accessible from the *ADH* menu and right-click menus.

ADH Menu

- Assign BC
- Assign Bed Layers
- Assign Consolidations
- Material Properties
- Hot Start Initial Conditions
- Time Series Data
- Model Check
- Model Control
- Run Model

Files

An ADH simulation is defined using three main files: mesh geometry (*.3dm), boundary conditions (*.bc), and hot start conditions (*.hot). ADH expects a 3-dimensional mesh geometry since it can solve 3D models, but SMS will only interface the 2D capabilities and will therefore only read and write 2D entities in the *.3dm file. A generic SMS mesh geometry file (*.2dm) can be used by ADH if the extension is changed to *.3dm. To facilitate handling the ADH files, SMS creates a simulation file (*.sim) which lists the individual files. Opening the *.sim will load the ADH files in SMS.

Each file is a card based text file that can be viewed and edited by a text editor (i.e. Notepad).

- Mesh geometry file (*.2dm/*.3dm)
 - Boundary condition file (*.bc)
 - Hot start file (*.hot)
 - Boat definition file (*.bt)
-

Using the Model / Practical Notes

- Sediment Transport and Bed Layers
- Time Series
- Wind Stations

ADH String Structures

ADH uses string structures to assign properties to entities. The following table describes the relation between the string structures and SMS structures.

ADH to SMS (Reading)

ADH String Structure		SMS Structure
NDS with DB card	and only 1 node ID	Node with boundary condition (BC)
	and 2 or more node IDs	Nodestring with BC
EGS with NB card (and possibly FLX card)		Nodestring with BC (and flux output)
MDS with FLX card		Nodestring with flux output
MTS with MP cards		Material with properties

SMS to ADH (Writing)

SMS Structure			ADH String Structure
Node with BC (Dirichlet)			NDS (only 1 node ID) with DB card
Nodestring	with Natural BC	with flux output	EGS with NB and FLX cards
			EGS with NB card
	with Dirichlet BC	with flux output	NDS (2 or more node IDs) with DB card, and MDS with FLX card
			NDS (2 or more node IDs) with DB card
	with flux output		MDS with FLX card
Material with properties			MTS with MP cards

Case Studies / Sample Problems

The following ADH simulation files have been provided by CHL ^[1] to test ADH and the SMS interface capabilities. Additional test cases will be posted here as the interface can successfully read, write, and run the ADH model from within SMS.

SMS 10.1 Development version using ADH rev #3669 dated August 2007:

- **Bayou Sorrel** ^[3] - Uses the **NB OVL** and **NB OTW** boundary condition cards. Includes two materials on a complex mesh.
- **Pool5** ^[4] - Uses the **NB OVL** and **NB OTW** boundary condition cards. Includes two inflows with one outflow and four materials on a complex mesh.
- **nb dis** ^[5] - Tests the **NB DIS** boundary card on a straight flume.
- **op bt** ^[6] - Tests the **OP BT** card which includes the vessel movement library. This test case contains a boat definition file (*.bt).
- **db lde** ^[7] - Tests the **DB LDE** pressure lid card on a straight flume.
- **db ldh** ^[8] - Tests the **DB LDH** pressure lid card on a straight flume.

- **db lid** ^[9] - Tests the **DB LID** pressure lid card on a straight flume.

Related Topics

- SMS Models

ADH Bed Layers Assignment

To open this window, global bed layers must be specified in *Model Control* on the *Global Material Properties* tab and at least one node must be selected. After a selection is made, select the **ADH | Assign Bed Layers...** menu item or use the right-click menu.

Dialog Description

The following controls specify the bed layer specifications to be assigned to the currently selected nodes to overwrite the global specifications. Overwriting is specified on a layer by layer basis at the selected nodes. This window is expandable by dragging the window edges to increase visibility of the spreadsheet data.

Bed Layers (untitled group box)

Spreadsheet contains the defined bed layers as columns and bed layer specifications as rows. The bed layers begin with Layer 1 and proceeds by ascending order by ID to the right. The specification rows are as follows:

- **Edit differing specifications** – Contains Edit check boxes and only appears when multiple nodes are selected and the bed layers specified at the nodes conflict. A check box is only placed in the columns of layers with conflicts (denoted by asterisks appended to the layer ID). The entire layer column will be disabled unless Edit is checked. If it is checked when accepting changes (exiting the window by clicking OK) the specifications for the bed layer will be applied to all selected nodes. Bed Layers that are not edited (remain unchecked) will not be changed.
 - **Overwrite global specification** – Contains Overwrite check boxes for each bed layer to specify whether the layer will be overwritten. If checked, the parameter cells of the column are enabled for editing, else the global specification values of the bed layer are displayed as read-only. The check box is disabled if a layer conflict exists and Edit is unchecked.
 - **Sediment distribution** – Title cell with each defined sediment listed below in its own row. The row contains Normalize buttons for each bed layer. Clicking on the button will normalize the distribution of each sediment such that the total distribution equals 100%. The button is disabled if a layer conflict exists and Edit is unchecked and read-only or enabled if Overwrite is unchecked or checked, respectively.
 - **Sediment (name)** – Consists of this sediment's distribution (as a percentage) for each layer. A sediment row is provided for all defined sediments. The value must be a positive real number and correlates to the MP SBN card. The cell is disabled if a layer conflict exists and Edit is unchecked and read-only or enabled if Overwrite is unchecked or checked, respectively.
 - **Total** – Displays the current total specified distribution of all sediments for each layer. The row updates automatically as specifications change and cannot be edited. Each layer's total sediment distribution must equal 100%. The cell is disabled if a layer conflict exists and Edit is unchecked.
 - **Thickness** – Consists of the each bed layer's thickness measured in meters. The value must be a positive real number and correlates to the MP SBN card. The cell is disabled if a layer conflict exists and Edit is unchecked and read-only or enabled if Overwrite is unchecked or checked, respectively.
 - **Bulk Density** – Consists of each bed layer's bulk density measured in kilograms per cubic meter. The value must be a positive non-zero real number and correlates to the MP CBN card. This row only appears if a cohesive sediment (clay or silt) is defined. The cell is disabled if a layer conflict exists and Edit is unchecked and read-only
-

or enabled if Overwrite is unchecked or checked, respectively.

- Erosion crit. shear – Consists of each bed layer's critical shear stress for erosion measured in newtons per square meter. The value must be a positive non-zero real number and correlates to the MP CBN card. This row only appears if a cohesive sediment (clay or silt) is defined. The cell is disabled if a layer conflict exists and Edit is unchecked and read-only or enabled if Overwrite is unchecked or checked, respectively.
- Erosion rate constant – Consists of each bed layer's unitless erosion rate constant. The value must be a positive non-zero real number and correlates to the MP CBN card. This row only appears if a cohesive sediment (clay or silt) is defined. The cell is disabled if a layer conflict exists and Edit is unchecked and read-only or enabled if Overwrite is unchecked or checked, respectively.
- Erosion rate exponent – Consists of each bed layer's unitless erosion rate exponent. The value must be a positive non-zero real number and correlates to the MP CBN card. This row only appears if a cohesive sediment (clay or silt) is defined. The cell is disabled if a layer conflict exists and Edit is unchecked and read-only or enabled if Overwrite is unchecked or checked, respectively.

Layer 1 is the deepest (or bottom) layer; the highest numbered layer is exposed to the stream flow. The layer IDs cannot be edited and bed layering order is fixed to ascending layer ID order (See Sediment Transport and Bed Layers).

Asterisks (*) denote bed layers with specifications that differ between the selected nodes. This note only appears when multiple nodes are selected and the bed layers specified at the nodes conflict. Asterisks are appended to the layer ID of differing layers in the title row across the top of the spreadsheet.

Related Topics

ADH

ADH Boat Definition File Cards

The following lists ADH *.bt cards that are recognized by SMS. For a description of card specifics, please use the ADH Quick Reference ^[1].

Boat Properties

	Read	Edit	Write	Notes
BLEN	Allowed	Allowed	Allowed	
BOAT	Allowed	Allowed	Allowed	
BWID	Allowed	Allowed	Allowed	
CBOW	Allowed	Allowed	Allowed	
CSTR	Allowed	Allowed	Allowed	
DRFT	Allowed	Allowed	Allowed	
ENDD	Allowed	Allowed	Allowed	
FDEF	Allowed	Allowed	Allowed	
PBOW	Allowed	Allowed	Allowed	
PROP	Allowed	Allowed	Allowed	
PSTR	Allowed	Allowed	Allowed	
SDEF	Allowed	Allowed	Allowed	

Related Topics

- ADH

ADH Extract WSE

SMS now has the ability to extract water surface elevation to a nodestring boundary condition from an existing dataset. This is done by right-clicking on the nodestring and selecting **Boundary Condition → Extract Water Surface Elevation...** and selecting the dataset to extract from. SMS will break up the existing nodestring into segments and extract the water surface elevation at the midpoint of each segment. The nodestrings are then assigned the Tailwater (outflow) boundary condition and the extracted data is set as the time series. This data can be edited in the *ADH Boundary Condition Assignment* dialog.

Related Topics

- ADH

ADH Hot Start File

ADH Hot Start (*.hot) File

The hot start file is mandatory and contains initial conditions for each node of a mesh. ADH allows for the specification of depth and velocity hydrodynamics, and sediment concentration and displacement datasets. The model requires at least depth initial conditions for hydrodynamics, and non-specified conditions will be defaulted to values of zeros (0.0).

The *.hot file format is actually the same as the ASCII dataset Files (*.dat) but necessitates the special extension since it should only contain following specifically named datasets:

Hot Start Datasets

Name	Initial Condition
"ioh" or "IOH"	Depth
"iov" or "IOV"	Velocity
"ioc" or "IOC"	Sediment concentration (SMS currently does not support this)
"iod" or "IOD"	Sediment displacement (SMS currently does not support this)
Name	Initial Condition
"ioh" or "IOH"	Depth
"iov" or "IOV"	Velocity

SMS will recognize and read in a hot start file if the dataset definition matches the current mesh definition (number of nodes and elements) and the ADH is the active model (other models also use a *.hot extension). Valid datasets with the above names are added to a locked "ADH Hot Start" folder in the Project Explorer. The folder and datasets can not be deleted or renamed from the tree. See Hot Start Initial Conditions for interface information.

Related Topics

- ADH

ADH Hot Start Initial Conditions

This window can be accessed by the **ADH | Hot Start Initial Conditions** menu item. ADH requires that at least an initial depth dataset is defined to start a simulation. ADH can not start with a completely dry domain.

The initial condition datasets will be generated and appear in the "ADH Hot Start" folder of the *Project Explorer*. This folder and the datasets are locked for editing except from this dialog.

Dialog Description

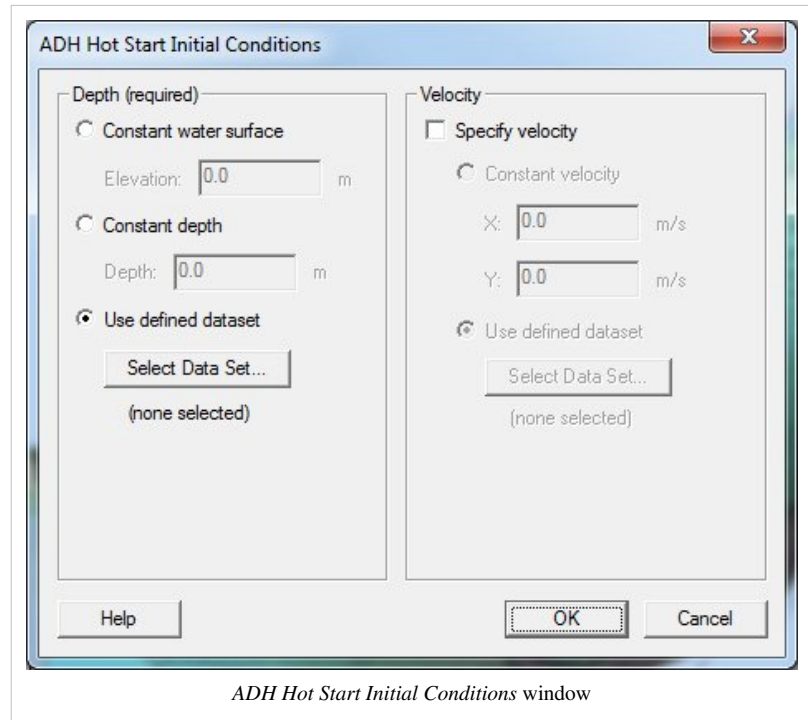
Depth

Constant water surface / Constant depth / Use defined dataset radio group

specifies the method for defining the initial depth dataset. The *Constant water surface* control group includes the *Elevation* real number edit field which only allows elevations greater than the maximum mesh elevation. This produces a fully flooded domain with a flat horizontal water surface. The *Constant depth* control group similarly includes the *Depth* positive real number edit field. The depth value is applied to every node to create a water surface which is offset from the mesh geometry. The *Use defined dataset* control group allows for the selection of an existing scalar dataset located in the Project Explorer via *Select Dataset...* button. The name and time step information (if transient) of the selected dataset will appear below this button.

Velocity

Specify velocity check box specifies whether an initial velocity dataset will be included and if checked, enables the *Constant velocity / Use defined dataset* method specification radio group. The *Constant velocity* control group includes *X* and *Y* velocity component real number edit fields. The composite velocity vector will be applied to every node in the domain. The *Use defined dataset* control group allows for the selection of an existing vector dataset located in the Project Explorer via *Select Dataset...* button. The name and time step information (if transient) of the selected dataset will appear below this button.



ADH Hot Start Initial Conditions window

Related Topics

- ADH

ADH Material Properties

To open this window, select the *ADH | Material Properties...* menu item.

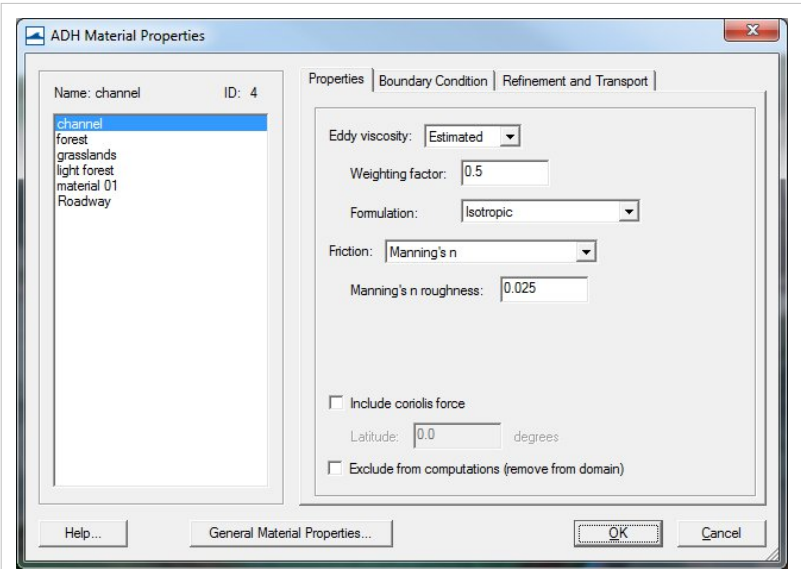
Dialog Description

Materials (Untitled group box)

Name text displays the name of the currently selected material in the material list.

ID text displays the numeric ID of the currently selected material in the material list.

List box contains the existing globally defined materials.



ADH Material Properties dialog showing the *Properties* tab

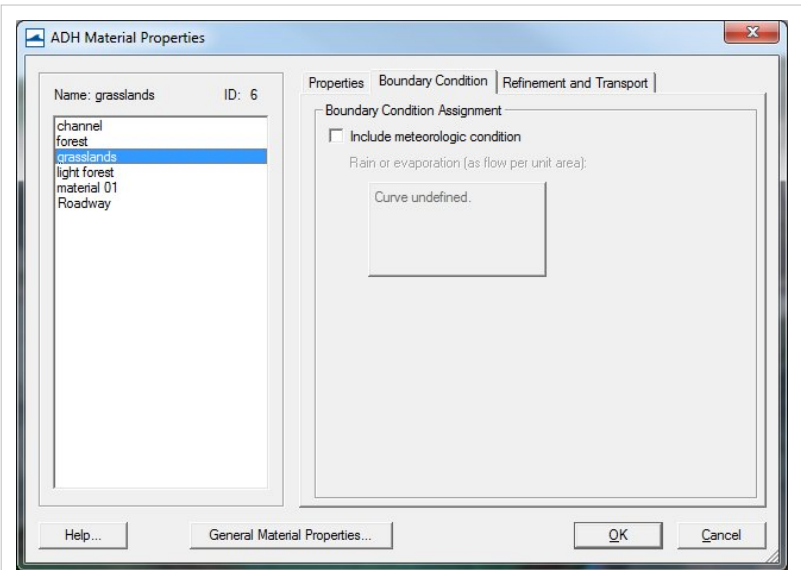
ADH Material Properties (Tab control)

This tab control contains the following three tabs:

Properties

Manning's $n_{roughness}$ real number edit field limited to a value of 0.0 to 0.1. This correlates to a **FR MNG** card applied to a material.

Kinematic eddy viscosity / Estimated eddy viscosity radio group specifies which method to use and correlates to the **MP EVS** and **MP EEV** cards, respectively. The *Kinematic eddy viscosity* control group includes the eddy viscosity tensor edit fields of *EVxx*, *EVyy*, and *EVxy*. The *EVxy* is specified once, and copied to the second term. All edit fields allow positive real number input. The

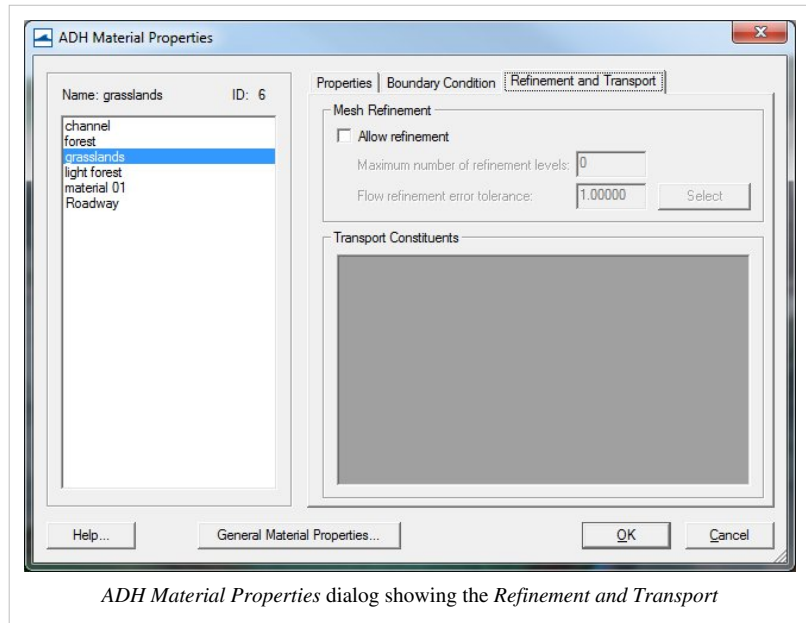


ADH Material Properties dialog showing the *Boundary Conditions* tab

Estimated eddy viscosity includes the *Weighting factor* real number field. This factor is used with other model conditions to calculate the eddy viscosity during the simulation.

Include coriolis force check box and the associated *Latitude* edit field will include the **MP COR** when checked. The edit field allows real number input between -90.0 and 90.0.

Exclude from computations (remove from domain) check box correlates to a **OFF** card applied to a material.



ADH Material Properties dialog showing the Refinement and Transport

Meshing and Boundary Conditions

Mesh Refinement

Maximum number of refinement levels positive integer edit field correlates to the **MP ML** card.

Error tolerance for refinement positive real number edit field correlates to the **MP SRT** card.

Boundary Condition Assignment

Apply rain or evaporation check box with the associated **Flow (per unit area) curve** button, *Series time units* combo box, and *Specify curve fir tolerance* check box and positive integer edit field defines the rain/evaporation series. Clicking on the curve button will open the XY Series Editor. This control group correlates to the **NB OVL** boundary condition card with a **XY1** series definition card.

Transport Constituents

List box contains the existing transport constituents.

Molecular diffusion rate edit field displays the parameter value for the currently selected transport constituent. This data pertains to the **MP DF** card.

Error tolerance for refinement edit field displays the parameter value for the currently selected transport constituent. This data pertains to the **MP TRT** card.

Miscellaneous (Outside of any group)

General Material Properties... button opens the *Materials Data* window. Any changes made to global materials will be reflected in the materials list box.

Related Topics

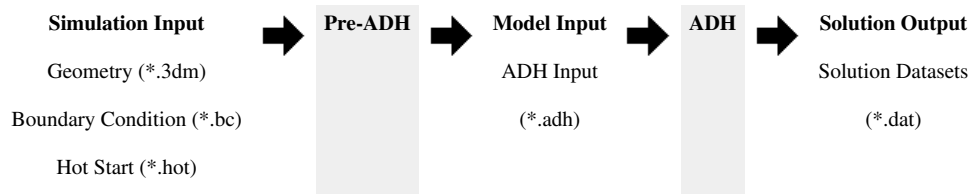
- ADH

ADH Run Model

To access the ADH model wrapper, select the *ADH* | **Run Model** menu item.

If the simulation has been changed and not yet saved, you will be prompted to do so before running the model. You will also be prompted to locate the Pre-ADH and ADH executables prior to running each if SMS does not know the location or find the executable. Model executable locations can be managed in on the *File Locations* tab of the *Preferences* dialog, accessed from the *Edit* | **Preferences...** menu item.

Pre-ADH takes the simulation input files (*.3dm, *.bc, and *.hot), performs some checks and creates the *.adh model input file. ADH reads the model input file, performs calculations and writes solution files (*.dat).



If Pre-ADH is not successful (the *.adh file is not written) then SMS will not allow ADH to be executed. Upon completion of ADH, the solution files can be automatically loaded into SMS or opened manually.

Dialog Description

Progress

Status line is blank for Pre-ADH and provides time step computation information for ADH.

Progress bar and text shows the percentage of completion.

Run Time displays the time since starting the executable. This is updated when the progress is updated.

Output

Pre-ADH Output or *ADH Output* field displays the last 1000 lines of commentary given by the running executable.

Output Text To File allows all commentary provided by the executable to be saved to a text file (*.txt). This button is enabled upon completion of the executable.

Load Solution check box is available only for ADH and is enabled upon successful completion of ADH. Check this option to automatically load the solution datasets into SMS.

Abort / Run ADH / Exit button controls the progress of the model wrapper.

Related Topics

ADH

Sediment Library Control

The *ADH Sediment Library Control* contains the graphical prompts for specifying model options. The dialog is accessed through the *ADH | Sediment Library Control* menu item

Dialog Description

Consolidation

See *ADH Model Control Consolidation* for more information.

Transport Constituents

See *ADH Model Control Transport Constituents* for more information.

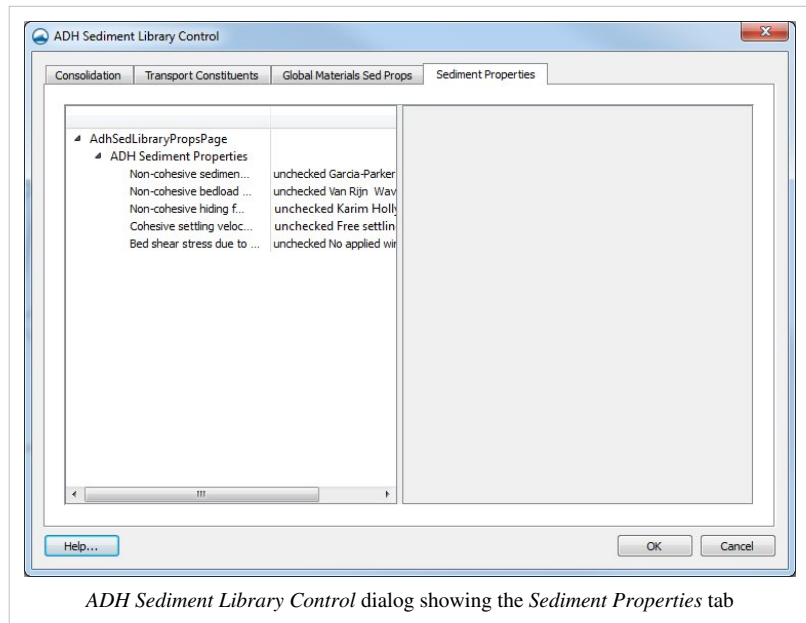
Global Materials Sed Props

The *Global Materials Sediment Properties* tab lets the user set the number of of bed layers. See *ADH Sediment Transport and Bed Layers* for more information.

Sediment Properties

This tab has five options.

- **Non-cohesive sediment entrainment.** Values of Option include: Garcia-Parker, Wright-Parker or Van Rijn. When this option is selected it will write out the SP NSE card.
- **Non-cohesive bedload entrainment.** Values of Option include: Van Rijn, Meyer Peter Mueller or Meyer Peter Mueller w/Wong Parker Correction. When this option is selected it will write out the SP NBE card.
- **Non-cohesive hiding factor.** Values of Option include: Karim Holly Yang or Egiazaroff. When this option is selected it will write out the SP HID card.
- **Cohesive settling velocity.** Values of Option include: Free settling or Hwang and Mehta. When this option is selected it will write out the SP CSV card.
- **Bed shear stress due to wind wave.** Values of Option include: No applied wind-wave stress, Grand and Madsen, or Teeter. When this option is selected it will write out the SP WWS card.



ADH Sediment Library Control dialog showing the *Sediment Properties* tab

CS2SHORE Sediment

This tab has ten options.

- Median sediment grain size with card SP C2SD50
- Breaking efficiency with card SP C2SEB
- Bottom dissipation efficiency with card SP C2SEF
- Bed porosity with card SP C2SBP
- Sediment fall velocity with card SP C2SWF
- Sediment specific gravity with card SP C2SSG
- Suspended load parameter with card SP C2SSL
- Wave-related bed load parameter with card SP C2SBLW
- Current-related bed load parameter with card SP C2SBLC
- Wave friction factor with card SP C2SFW

Related Topics

- ADH
- SMS:ADH Model Control

ADH Sediment Transport and Bed Layers

Sediments properties defined on the **CN CON**, **CN SND**, and **CN CLA** cards describe the properties of the sediment particle when it is first deposited.

Bed layers specified for a simulation are previously deposited layers defined by the **MP SBA**, **MP SBM**, and **MP SBN** cards. While running ADH, the previously deposited layers are aggraded and degraded and the solution datasets provide the resulting bed profile conditions during the simulation. For sediment transport with cohesive sediments, bed layers also have cohesive layer attributes defined by the **MP CBA**, **MP CBM**, and **MP CBN** cards to account for settlement and compaction of the previously deposited layer.

ADH provides the **MP SBM** and **MP CBM** cards to apply a bed layer specifications to nodes (which overwrites the global specifications) by specifying a material, however conflicts can arise since material type is an attribute of an element and not a node. As depicted in the image, a node can define the boundary between multiple materials and if each material has an associated bed layer specification, the central node can have three different specifications. ADH will assign material bed layer specifications by applying them to nodes and overwriting previous specifications in the order the cards are read from the *.bc file. Therefore, if **MP SBM** (or **MP CBM**) cards for material 1, material 2, and material 3 are read in that order, the central node (in the picture) will finish with the specification for material 3. SMS will read the *.bc file by the same process, however, bed layer specifications by material type will not be allowed in the interface (in Material Properties) and not written to file since materials are not currently prioritizable. Bed layers can be specified to nodes using a material type by selecting the Edit | Select by Material Type... menu item with the Select Node tool active, but the material type is only used to filter the selection and is not associated to the actual node or bed layer specification. SMS will only write **MP SBA**, **MP CBA**, **MP SBN**, and **MP CBN** cards to file.

Bed layers are in continuous ascending ordered by ID starting with 1 as the deepest specified layer in the profile and the highest numbered layer at the top, exposed to the stream flow. ADH automatically adds a solid boundary, consisting completely of an extra immobile (non-erodable) sediment, below Layer 1 which is not presented in the interface but will be included in solution files.

ADH Bed Layering Order

Stream Flow
Layer X
...
Layer 3
Layer 2
Layer 1
Solid Boundary

Bed layers specifications can be edited using the following:

- **ADH | Model Control...** menu item opens the Model Control window where global bed layers are specified on the *Global Material Properties* tab.
- **ADH | Assign Bed Layers...** menu item opens the *Bed Layers Assignment* window for specifying bed layers at the nodal level. Available only with a current node selection.
- **Select Node** tool **Assign Bed Layers...** right-click menu item opens the *Bed Layers Assignment* window for specifying bed layers at the nodal level. Available only with a current node selection.
- **Select Node** tool **Delete Bed Layers** right-click menu item deletes the nodal specification which overwrites the global. Available only when the current node selection includes at least one nodal bed layer specification.
- **Select Node** tool **Find Equivalent Bed Layers...** right-click menu item selects all nodes of the mesh that are equivalent to a bed layer specification of the current node selection.

Related Topics

- ADH

ADH Time Series

ADH includes several model specific time series (curve) groups to organize model curves by type. The curves of these groups can be viewed and edited in the Time Series editor and include time series attributes specific to ADH. The following table describes the groups:

ADH Curve Groups

Name	Description	Specification Use	Restrictions
Depth	Time offset with depth measurement (L)	Flow or pressure boundary condition (DB OVH or DB LDH card)	
Discharge	Time offset with flow rate (L ³ /T)	Flow boundary condition (NB DIS card)	
Draft	Time offset with draft measurement (L)	Pressure boundary condition (DB LID card)	
Elevation	Time offset with elevation value (L)	Pressure boundary condition (DB LDE card)	
Flow (per unit length)	Time offset with unit flow rate (L ² /T)	Flow boundary condition (NB OVL card in conjunction with an ADH edge string)	
Meteorologic	Time offset with unit flow rate (L/T)	Flow boundary condition (NB OVL card in conjunction with an ADH material string)	
Time Step Size	Time offset with time step size (T)	Simulation condition (TC IDT card)	Unavailable in steady state
Transport	Time offset with concentration measurement (PP)	Transport boundary condition (NB TRN or DB TRN card)	Unavailable in steady state
Velocity	Time offset with velocity components (L/T) (three columns)	Flow boundary condition (DB OVH or DB OVL card)	
Water Surface Elevation	Time offset with elevation value (L)	Flow boundary condition (NB OTW card)	
	(Units: T = time, L = length, PP = unitless parts per)		

Curves with similar value types (i.e. pressure boundary conditions) have been separated into individual groups to avoid the unintentional application.

Specifying and Selecting Time Series Data

In the ADH menu, selecting the Time Series Data... option will open the Time Series editor with the above mentioned available curve groups. This is provided so that the user can quickly view or specify all simulation time series data from one window. However, specifying a time series data in the editor accessed from this menu item does not associate the time series with a boundary or simulation condition. Various windows of the ADH interface (i.e. Boundary Condition tab of the Material Properties window) include a curve button which opens the Time Series editor with only the condition's curve group available. Association of the selected curve with a condition occurs when the editor's OK button is selected. These buttons will display the name and plot of the selected time series, unless in a spreadsheet (i.e. Transport Constituents tab of the Boundary Condition Assignment window).

Steady State Simulation

If setting up a steady state simulation, the Time Series editor is still used to specify and associate time series. There is no restriction on the time series data in the interface (the number of rows is not limited), however SMS will write a steady state series consisting only of the first condition value(s) listed. The written series is created using the simulation start and end times (zero offset and the simulation length from the Time tab in Model Control, respectively) with the same condition value(s). To avoid confusion, it is suggested that only one row should be specified in the editor (with a zero time offset) for all steady state series. SMS will validate that a time series selected to be associated with a boundary condition is appropriate steady state data.

Boundary Condition File

When saving the ADH Boundary condition file (*.bc), only the time series that are actually used by the simulation (associated with conditions) are written out. These are converted to the units expected by ADH (i.e. mph to ft/s) as necessary and written as XY1 cards with XYT cards if curve fit tolerance is specified (apart of time series attributes). Unused curves can be saved by exporting them from the Time Series editor. A written series that is not defined for the full simulation duration is automatically appended with the simulation end time and the last specified condition value(s). A time series in the file can appear different than as specified in the Time Series editor because of unit conversions and validation.

After reading a XY1 card from a file, SMS will assign the data to a curve group based on the type of card(s) that references it in the file. If multiple types of cards reference the same time series, then the data will be assigned to multiple curve group as separate and unique time series.

Related Topics

- ADH

ADH Time Series Attributes

This window is accessed from the *Time Series* window by clicking on the **Attributes...** button. The current curve group must be an ADH model specific curve group (i.e. ADH Discharge) to open this attributes window. The attributes are only associated with the selected curve.

A curve fit tolerance (XYT card) can be specified with each curve to be used when ADH interpolates values from a curve (while time adapting).

Dialog Description

Specify curve fit tolerance check box and the associated positive integer edit field specify whether a tolerance is included and its value. The edit field is enabled only if the check box is checked.

Related Topics

- ADH

ADH Velocity Series Editor

The *Velocity Series Editor* is used to define velocity versus time for ADH Dirichlet flow velocity boundary conditions. A velocity must be defined by specifying its x and y components. The components are used to calculate the velocity magnitude and direction.

Dialog Controls

Data spreadsheet contains three columns for time and velocity components. Define the series by entering values into each cell. When the empty row (last row) is edited, a new empty row will be appended. The empty row does not have any value and is not part of the series definition. An undefined series consists of only the empty row. Every cell of the series (the empty row is excluded) must contain a value and cannot be "blanked out" (left undefined). Copy and paste are enabled for the cells, and the spreadsheet cells include a right-click menu.

Insert New Row Above tool and spreadsheet right-click menu option will insert a row above the spreadsheet selection. The new row's values will be interpolated from the rows directly above and below, unless the new row is inserted above the spreadsheet's first row, in which case the values will be copied from the row directly below. This is only available if there is a spreadsheet selection that is not limited to just the empty row.

Delete Row(s) tool and spreadsheet right-click menu option will delete the row(s) included in the spreadsheet selection. The entire row(s) will be removed from the spreadsheet and all rows below the deletion will be shifted up. Since every cell of the series must contain a value, to remove a specific value, that value must be edited or the entire row (all three values) must be removed.

Curve Name text edit field specifies the name that will be associated with the curve defined in the spreadsheet. The ADH Dirichlet flow velocity series are written to file as two separate curves (velocity x component versus time and velocity y component versus time, using the same time values) so this name will be applied to and written out with both series definitions.

Related Topics

ADH

ADH Vessel Coverage

The ADH Vessel Coverage is used to add vessels to the simulation and give them paths to follow. Generally, one coverage represents one vessel. You can drag one or more vessel coverages into the ADH Mesh to add those coverages to the simulation (by creating links). This allows you to have more vessels defined than you are using in your simulation, and you can swap them in and out of the simulation to run different tests with different vessels.

Boat Path

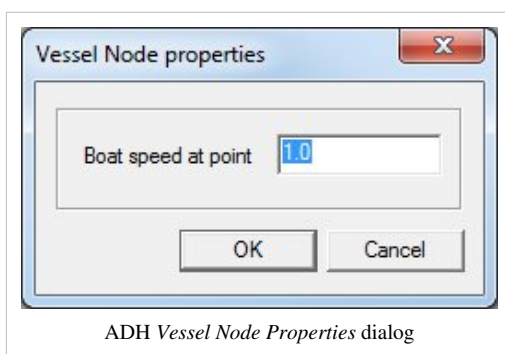
Each vessel coverage has at least one arc to define the boat path. The arc determines the vessel's starting position and starting speed, and where it will go from there. Additional segments can be added to the arc by adding vertices. Vertices add destinations but do not affect speed. To change the speed of the boat you can convert a vertice into a node, and enter the node's attributes.

In general, there will always be exactly one path per vessel coverage. The path can be made up of multiple arcs, but the arcs should not split into multiple paths. The one exception to the rule of "one path per coverage" is when you have multiple vessels that are identical except for their speeds and paths. In this case you can have multiple separate paths in the same coverage, and each will create its own separate copy of the boat defined in the coverage properties. These paths can cross each other as long as they do not connect to each other (at a node). Another option is to simply duplicate the coverage after the boat properties are defined. This allows you to add and remove the boats from the simulation separately, and the boats can have vertices and nodes in the same place (same x and y coord) along their path without conflicting. The boat path writes the FDEF and SDEF cards to the boat file.

Dialogs

Node Attributes

Nodes on arcs change the speed of the boat or, in the case of the first node of the path, define the boat's starting speed. To change the speed of the boat at a node, select it and right-click, then select **Node Attributes...**. Alternately, you can double-click the node.



Boat Parameters

The boat's parameters are defined in the coverage properties. Right-click on the coverage and choose **Properties....** This dialog lets you set the boat's size and shape, and define propellers if desired. If the OP BTS card is included, each vessel in the simulation will need to have propellers defined. Without the OP BTS card, propellers supposedly do nothing.

Each field corresponds directly to a card in the boat file.

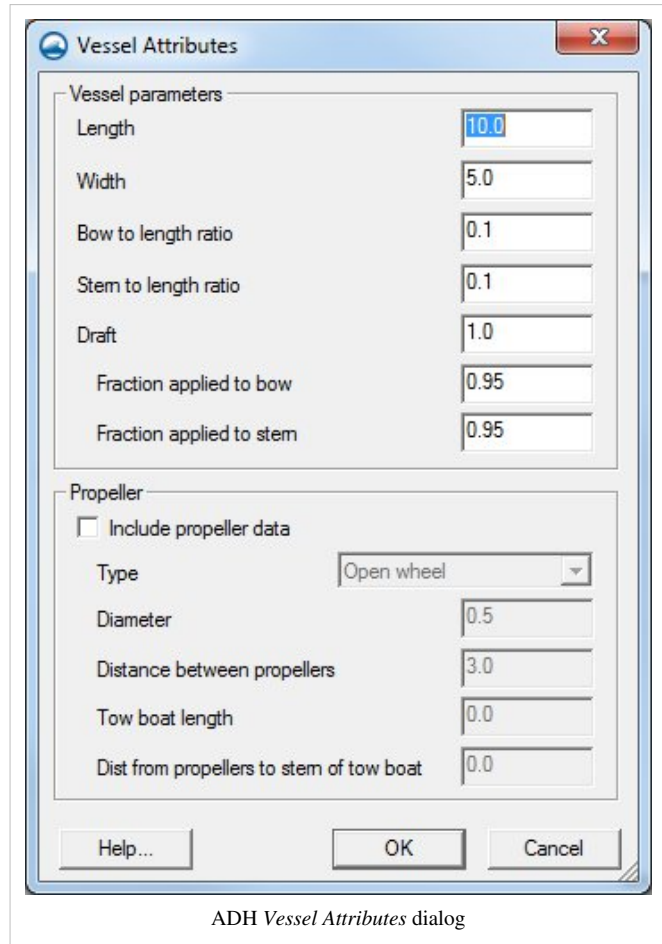
- Length (BLEN)
- Width (BWID)
- Bow to Length Ratio (PBOW)
- Stern to Length Ratio (PSTR)
- Draft (DRFT)
- Fraction of Draft Applied to PBOW (CBOW)
- Fraction of Draft Applied to PSTR (CSTR)

Propeller (PROP card):

- Propeller Type: Open wheel or Kort nozzle
- Propeller Diameter
- Distance between propellers
- Tow boat length: This length provides an offset distance of the propeller induced shear stresses from the vessel. Set to 0 if there is no tow boat.
- Distance from prop to tow boat stern

Related Topics

- ADH



ADH Vessel Attributes dialog

ADH Wind Stations

Wind station data can be included in an ADH simulation to apply wind stress on the water surface. This data is in addition to and should not affect the mesh geometry, so stations are specified on a feature coverage in the Map module. The Spatial Data coverage is used because it allows the specification of generic time series data at locations represented by feature points. ADH wind station data is specified in SMS as wind speed data (NOTE: ADH currently expects wind stress data, but is planned to accept wind speed data.)

Creating Wind Stations

To create a wind station for ADH, first create a new Spatial Data coverage and give it a descriptive name. Using the **Create Feature Point** tool, add a point at the location of the wind station. Right-click on the feature point with the **Select Feature Point or Node** tool, point at the *Add* option, and select *Time Series*. This will open the *Time Series* editor. Select a velocity curve group (either "Velocity – Components" or "Velocity – Mag. & Dir.") and click on the **New...** button. Specify a name, curve data, and reference time. When finished, clicking the **OK** button adds the selected curve to the selected feature point's data. Right-clicking on the feature point again, this menu will display the added time series and provide options to view, edit, or delete this data.

After wind stations have been created on a coverage, the ADH simulation must be told to include this information on the *Model Parameters* page of *Model Control*. Since a specific coverage is chosen, the user can create multiple coverages and quickly switch wind conditions as necessary. The velocity time series data of a coverage is not specific to ADH, and consequently do not include ADH time series attributes which contains curve fit tolerance. Therefore, a general curve fit tolerance can be specified in *Model Parameters* to be attributed to every velocity time series in the selected coverage.

Steady State Simulation

If setting up a steady state ADH simulation, a spatial data coverage is still used to specify wind stations and their time series. However, SMS will write a steady state velocity series consisting only of the first condition values listed regardless of the reference time. The written series is created using the simulation start and end times (zero offset and the simulation length from the *Time* tab in *Model Control*, respectively) with the same condition values. To avoid confusion, it is suggested that only one row should be specified in the editor (zero offset value with any reference time) for all steady state velocity series. The user is responsible for specifying appropriate steady state data.

Boundary Condition File

When saving the ADH Boundary condition file (*.bc), SMS will check to whether wind station data is included and, if specified, that the selected coverage is still valid (exists). SMS will then proceed to iterate through every feature point in the selected coverage, formatting series data and writing **XYC** and **XY2** cards (with **XYT** cards if curve fit tolerance is specified). Since the general velocity types (Velocity – Components and Velocity – Mag. & Dir. curve groups) include reference time, part of the series formatting includes extracting data and re-referencing it to the simulation start time. If the data does not cover the simulation start or end time, it will be appended with the time and the first or last specified condition values, respectively. The extracted series will start at the simulation start time and include all remaining data. Formatting also includes converting velocity data to component form (magnitude and direction to x,y components) and units expected by ADH (i.e. mph to ft/s) as necessary. A velocity time series in the file can appear different than as specified in the Time Series editor because of form and unit conversions, and validation. Any feature point without velocity data will be ignored along with all other feature objects the coverage may contain. If a feature point contains multiple velocity time series, then only the data of first time series found will

be written. Coverages not included in the simulation can be saved separately if desired.

As SMS reads an ADH boundary condition file, wind station data will be added to a new spatial data coverage with the default name of "ADH Wind Station". This coverage will be automatically selected in Model Parameters. If the file contains any curve fit tolerances for wind station velocity time series, the minimum tolerance (the values may be different) will be used for the entire coverage.

Related Topics

ADH

6.1.a. ADH Boundary Condition

ADH Boundary Condition

To open this window, at least one node or nodestring must be selected. After a selection is made, select the *ADH | Assign BC...* menu item or use the right-click menu.

ADH boundary conditions consist of two specification types (Dirichlet and Natural) and three condition types (flow, pressure, and transport). Dirichlet data is applied on the domain to individual nodes or to groups of nodes (defined with nodestrings). Natural data (flux) is applied through edges of the domain defined by nodestrings and includes a friction specification. A node or nodestring can be assigned only one condition option from each of the three condition types (two flow conditions cannot be assigned to a single node).

Boundary Conditions

Flow Condition Options	Specification Type	Information
None	---	No condition specified
Flow (per unit length)	Natural	Specified by a Flow (per unit length) time series; NB OVL card with a friction card
Sidewall (no through flow)	Natural	Creates a vertical wall with the specified shear stress; just a friction card
Subcritical inflow	Dirichlet	Specified by a Velocity time series; DB OVL card
Supercritical inflow	Dirichlet	Specified by a Depth time series and a Velocity time series; DB OVH card
Tailwater (outflow)	Natural	Specified by a Water Surface Elevation time series; NB OTW card with a friction card
Total discharge	Natural	Specified by a Discharge time series; NB DIS card with a friction card

Pressure Condition Options	Specification Type	Information
None	---	No condition specified
Lid (depth underneath)	Dirichlet	Specified by a Depth time series; i.e. a structure (ex. culvert, sluice gate, etc.) defined using bed elevation as datum; DB LDH card
Lid (draft)	Dirichlet	Specified by a Draft time series; i.e. a floating object (ex. boat, dock, etc.) with draft; DB LID card
Lid (elevation)	Dirichlet	Specified an Elevation time series; i.e. a structure (ex. culvert, sluice gate, etc.) defined using the elevation datum; DB LDE card

Transport Condition Options	Specification Type	Information
None	---	No condition specified
Dirichlet	Dirichlet	Specified by a Transport time series; DB TRN card
Equilibrium	Dirichlet	Specified by initial concentration (in ppm); This condition can only be assigned to sediments and specified in simulations without cohesive sediments; EQ TRN card
Natural	Natural	Specified a Transport time series; NB TRN card

Weir Options	Specification Type	Information
None	---	No condition specified
Weir		Nodestrings are specified as upstream, upstream edge, downstream or downstream edge. See the WER and WRS card

Natural Flow Condition Friction Options

Friction Options	Information
Manning's n	Specified by a roughness value; FR MNG card
Equivalent roughness height	Specified by a roughness height (in ft or m); FR ERH card
Submerged aquatic vegetation	Specified by an undeflected stem height (in ft or m); FR SAV card
Un-submerged rigid vegetation	Specified by a roughness height (in ft or m), averaged stem diameter (in ft or m), and average stem density (in stems/ft ² or stems/m ²); FR URV card

All mesh boundary edges that do not have a boundary condition will be treated as a frictionless sidewall with no through flow.

All boundary conditions listed above can only be assigned to the mesh boundary, except the pressure conditions which can also be assigned to the mesh interior.

When changing a condition type on any tab, the existing condition information is cleared from the window such that the information must be respecified upon returning to that type.

Dialog Description

The following controls specify the boundary condition to be assigned to the currently selected entities. This window is expandable by dragging the window edges to increase visibility of the spreadsheet data on the *Transport* tab.

Boundary Condition Type (Tab Control)

This tab control can contain the following three tabs:

Flow

This tab is only available if all of the currently selected nodes or nodestrings are on the mesh boundary.

Boundary type combo box contains the conditions available for flow (listed above) for the selected entities.

Series data curve button selects the time series to be associated with the selected boundary type. The text label above this curve button will change with the boundary type to denote the type of input. Clicking on the curve button will

open the Time Series editor (limited to the curve group of the boundary type). A second curve button will appear for boundary types that require two conditions with time series data.

Friction combo box specifies the type of bed shear stress to be applied to the boundary edge and is available only for Natural boundary types. The friction options are:

- *Manning's n* provides the *Manning's n* roughness real number edit field limited to a value of 0.0 to 0.3.
- *Equivalent roughness height* provides the *Roughness height* positive real number edit field.
- *Submerged aquatic vegetation* provides the *Undelected stem height* positive real number edit field.
- *Un-submerged rigid vegetation* provides the *Roughness height*, *Average stem diameter* and *Average stem density* positive real number edit fields.

Pressure

This tab is always available.

Boundary type combo box contains the conditions available for pressure (listed above) for the selected entities.

Series data curve button selects the time series to be associated with the selected boundary type. The text label above this curve button will change with the boundary type to denote the type of input. Clicking on the curve button will open the Time Series editor (limited to the curve group of the boundary type).

Transport Constituents

This tab is only available if all of the currently selected nodes or nodestrings are on the mesh boundary and constituents have been included in Model Control Transport Constituents.

Boundary type combo box contains the conditions available for transport (listed above) for the selected entities.

Spreadsheet contains the following information specific to each transport type:

- *None* clears the display of any information in the spreadsheet.
- *Dirichlet* provides each defined constituent (first column) with a *Include* check box (second column), *Specify Series...* curve button (third column), and a series name cell (fourth column). To include the constituent in the specification of the transport condition, check the *Include* control and specify a concentration series by clicking on the *Specify Series...* button which will open the Time Series editor (limited to the Transport curve group). When a series has been defined, the name will be listed in the series name cell. A series can be selected by multiple constituents. The concentration series button and name cells are only enabled if the constituent is included in the transport source.
- *Equilibrium* provides each defined sediment (first column) with a *Include* check box (second column) and a positive real number *Initial concentration* cell (third column). To include the sediment in the specification of the transport condition, check the *Include* control and specify a concentration (in parts per million). If the sediment is not to be present at the beginning of the simulation, specify a concentration of zero. The *Initial concentration* cell is only enabled if the sediment is included in the transport source.
- *Natural* provides the same as *Dirichlet*.

Output (untitled group box)

Output calculated flow across this entity check box is available only for nodestrings and correlates to the **FLX** card.

Related Topics

ADH

ADH Boundary Condition File Cards

The following lists ADH *.bc cards that are (or will be) recognized by SMS and their capability (reading, editing values and writing) currently available within the interface. For a description of card specifics, please use the ADH Quick Reference ^[1]. SMS will ignore any card that is not listed below when reading a *.bc file.

Constituent Properties

See Model Control Transport Constituents for interface information.					
		Read	Edit	Write	Notes
CN	CLA	X	X	X	SMS writes a constituent name character string in quotes preceding the regular card fields.
CN	CON	X	X	X	SMS writes a constituent name character string in quotes preceding the regular card fields.
CN	SAL				
CN	SLT	X	X	X	SMS writes a constituent name character string in quotes preceding the regular card fields.
CN	SND	X	X	X	SMS writes a constituent name character string in quotes preceding the regular card fields.
CN	TMP				
CN	VOR	X	X	X	

Friction Controls

See Boundary Condition Assignment for interface information.					
		Read	Edit	Write	Notes
FR	ERH				
FR	MNG	X	X	X	A natural boundary condition without a specified friction card will have a roughness value of zero.
FR	SAV				
FR	URV				

Iteration Parameters

See Model Control Model Parameters for interface information.					
		Read	Edit	Write	Notes
IP	FLI	X	X	X	This card and the IP MIT card are mutually exclusive.
IP	FNI	X	X	X	This card and the IP NIT card are mutually exclusive.
IP	ITL	X	X	X	
IP	MIT	X	X	X	This card and the IP FLI card are mutually exclusive.
IP	NIT	X	X	X	This card and the IP FNI card are mutually exclusive.
IP	NTL	X	X	X	
IP	RTL				
IP	SST				

Material Properties

See Model Control Global Material Parameters and Material Properties for interface information.					
		<u>Read</u>	<u>Edit</u>	<u>Write</u>	<u>Notes</u>
MP	CBA				
MP	COR	X	X	X	Material specific.
MP	DF				Material and transport constituent specific.
MP	DTL	X	X	X	Global property.
MP	EEV	X	X	X	Material specific.
MP	EVS	X	X	X	Material specific.
MP	G	X	X	X	Global property.
MP	ML	X	X	X	Material specific.
MP	MU	X	X	X	Global property.
MP	MUC	X	X	X	The ADH units system flag currently keys off of the current horizontal coordinate system.
MP	NBL				
MP	RHO	X	X	X	Global property.
MP	SBA				
MP	SBM				
MP	SBN				
MP	SRT	X	X	X	Material specific.
MP	TRT				Material and transport constituent specific.

Operation Parameters

See Model Control Model Parameters for interface information.					
		<u>Read</u>	<u>Edit</u>	<u>Write</u>	<u>Notes</u>
OP	BLK	X	X	X	
OP	BT	X	X	X	
OP	BTS				
OP	INC	X	X	X	
OP	PRE	X	X	X	
OP	SW2	X	---	X	Always written by SMS (not editable), denotes a 2D Shallow Water ADH model
OP	TPG				
OP	TRN				
OP	TEM				

Output Controls

See Model Control Output for interface information.				
	<u>Read</u>	<u>Edit</u>	<u>Write</u>	<u>Notes</u>
END	X	---	X	Always written by SMS (not editable), denotes the end of the file.
FLX	X	X	X	See Boundary Condition Assignment for additional interface information.
OC	X	X	X	

Solution Controls

See Boundary Condition Assignment for nodal and nodestring boundary condition interface information.					
		<u>Read</u>	<u>Edit</u>	<u>Write</u>	<u>Notes</u>
DB	LDE	X	X	X	
DB	LDH	X	X	X	
DB	LID	X	X	X	
DB	OVH	X	X	X	
DB	OVL	X	X	X	
DB	TRN				
EQ	TRN				
NB	DIS	X	X	X	
NB	OTW	X	X	X	
NB	OVL	X	X	X	See the Boundary Condition Assignment section of Material Properties for additional interface information.
NB	TRN				
OB	OF	X	---	---	This card is obsolete as the functionality is now handled internally in ADH.
OFF		X	X	X	This card can only be applied to a material string (MTS). See the Properties section of Material Properties for interface information.

String Structures

See Practical Notes for string structure information.				
	<u>Read</u>	<u>Edit</u>	<u>Write</u>	<u>Notes</u>
EGS	X	X	X	Use the Select Nodestring tool, and assign boundary conditions.
FCS				
MDS	X	X	X	Use the Select Nodestring tool, and assign boundary conditions for flow output.
MTS	X	X	X	Add and delete materials in the Materials Data window. Specify material properties, and using the Select Element tool, assign material type from the Elements Assign Material Type... menu command.
NDS	X	X	X	Use the Select Node or Select Nodestring tool, and assign boundary conditions.

Time Controls

See Model Control Time for interface information.					
		<u>Read</u>	<u>Edit</u>	<u>Write</u>	<u>Notes</u>
TC	IAC	X	X	X	
TC	IDT	X	X	X	
TC	NDP	X	X	X	
TC	SDI	X	X	X	
TC	STD	X	X	X	
TC	STH				
TC	TF	X	X	X	
TC	T0	X	X	X	SMS writes a "global time" character string in quotes preceding the regular card fields. The date/time information is used to relate the simulation with other data loaded in SMS concurrently.

Time Series

	<u>Read</u>	<u>Edit</u>	<u>Write</u>	<u>Notes</u>
XY1	X	X	X	Define series (curves) in various places. An output times series using the auto-build specifier can be read in and created, however, the auto-build format is not currently used when writing out. SMS writes a series name character string in quotes preceding the regular card fields.
XYC				
XYT	X	X	X	Available where series (curves) are defined as "curve fit tolerance".

Related Topics

ADH

6.1.b. ADH Model Control

ADH Model Control

The *ADH Model Control* dialog contains the graphical prompts for specifying model options. The dialog is accessed through the *ADH | Model Control...* menu item.

Dialog Description

Model control contains the following five pages:

- Model Parameters
- Iterations
- Time
- Output
- Global Material Properties
- Transport Constituents
- Consolidation
- Advanced (Cards)
- Solver

ADH Model Control Output

This page window is accessed from the Model Control window by clicking on the *Output* tab.

ADH will write solution data at startup and at each time step specified in a **XY1** series definition and referred to by the **OC** card. Flux data will be included with the solution data if output flow strings are specified (see assigning boundary conditions).

ADH Model Control Advanced (Cards)

Model options not available in the interface can be specified in the ADH model control under the advanced tab. Each row in the spreadsheet will be saved as a different line in the *.bc file. The cards can be inserted and deleted using the insert and delete rows.

- The lines will be written out exactly as typed.
 - The lines will be written out in the order in which they appear in the dialog.
 - Any lines encountered in a *.bc file that is not recognized will appear in this list.
 - Lines beginning with a "!" character are treated as comment lines and will be skipped over during a read.
 - Comment lines are automatically generated and will not be preserved.
 - All of these advanced cards will be grouped together at the bottom of the file regardless of where they appeared during reading.
-

Dialog Description

The following controls specify how often solution data will be written by ADH and the number of flux boundaries present.

Output Times

List box contains the times to be contained in the output series. Multiple selection is enabled for deleting times.

Add by specifying a range / Add individual output time radio group specifies time information to be added. The *Add by specifying a range* control group includes *Start* at positive integer/real number field and units combo box, *End* at positive integer/real number field, and *Increment* positive integer/real number field and units combo box. The *End* at time units are the same as specified for the *Start* at time. The *Add individual output time* control includes a positive integer/real number field and units combo box. Each edit field allows for positive integer seconds or positive real number minutes, hours, etc., with six decimal places. All times are rounded to the nearest whole second (a decimal minute is rounded to the nearest decimal minute with a whole second equivalent) before being added.

Add button includes the time information from the above radio group in the times list box. Duplicate times will be automatically removed from the list.

Delete button removes the selected time(s) from the list box.

Delete All buttons clears the entire times list without regard to the current selection.

View output times in combo box specifies the time unit the list box will display the output times in.

Specify curve fit tolerance check box and positive integer edit field specifies the series tolerance when checked.

Flow

Number of nodestrings specified to output flow calculated across themselves displays the total number of ADH flux boundaries included in the simulation.

Related Topics

ADH

ADH Model Control Model Parameters

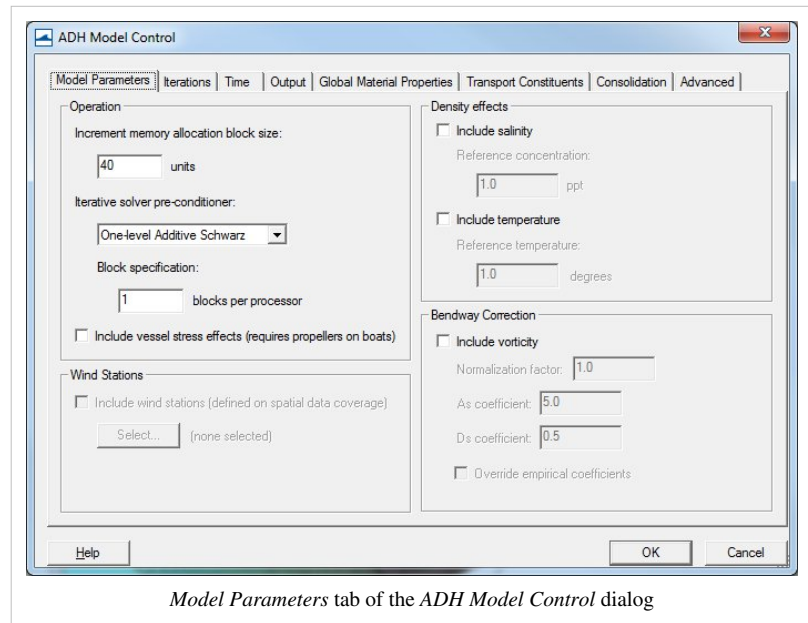
This window is part of the *ADH model control* dialog (*ADHModel Control...* command). It is accessed from the *Model Control* window by clicking on the *Model Parameters* tab.

Model Parameters

The model parameters window is divided into four sections including:

- Operation
- Wind Stations
- Density effects
- Bendway correction

Each of these sections are described below.



Operation

The operation section contains controls to specify general operation parameters for ADH. SMS automatically writes the OP SW2 card to all ADH boundary condition files because SMS interfaces with the 2D shallow water problems simulated by ADH. The other operational parameters specified in this section include:

Memory allocation block size (OP INC card)

The *Incremental memory allocation* block size edit field allows the user to specify a positive integer defining the size of the incremental memory allocations. The default value is 40 units and no explanation has been provided for when or what to change this to. We recommend most users leave the default value.

Iterative solver pre-conditioner (OP PRE and BLK cards)

The *Iterative solver pre-conditioner* parameters apply to parallel processing applications. If you are running ADH in single processor mode, this card has no impact.

If you are running ADH in parallel mode, the edit field defines how many blocks per processor are to be used in the preconditioning which subdivides the problem for each processor. No guidance has been provided by the AdH development team about when to use different pre-conditioning schemes. We recommend the defaults of "One-level Additive Schwarz" and "1 block per processor".

Vessel stress effects (OP BT card)

This toggle instructs SMS to write the OP BTS card which includes vessel stress effects check in the simulation when checked. This calculates and outputs bed shear stresses due to vessels in dyn/cms. This currently requires use of metric units. If you enable this option, make sure you have vessel coverages in your simulation and that each boat has propellers defined.

Wind Stations (XYC cards)

Wind can be included in a number of ways in an ADH simulation. These include on a material by material basis described in a different section. The *Include wind stations (defined on spatial data coverage)* check box lets the user specify that wind shear stress should be included in the simulation, computed from a set of spatially defined wind stations. The Include check box is only available if at least one spatial data coverage exists. In the spatial data coverage the user should define a set of wind gages (defined as feature points) with time series of wind data associated.

If more than one spatial data coverage is defined, the associated **Select...** button undims and the user can select a single set of wind data for the current simulation. Clicking on the **Select...** button opens the *Select Coverage* window which lists all available spatial data coverages.

SMS will write an XYC card for each wind station. This card defines the curve containing the wind data and the location of the station.

Specify curve fit tolerance check box and the associated positive integer edit field specify whether a tolerance for all wind time series is included and its value. These controls are enabled only if a coverage is selected (the tolerance check box must also be checked to enable the edit field).

For more information see Wind Stations.

Density effects

The ADH model includes the ability to simulate the effects of salinity and temperature on flow. This section includes two toggles for the user to activate these options.

Salinity (CN SAL card)

The *Include salinity* check box with the associated Reference concentration allow the user to turn on the simulation of salinity and define a positive non-zero real number for the background or reference concentration. The concentration is specified in parts per thousand.

Temperature (CN TMP card)

The *Include temperature* check box with the associated *Reference temperature* allow the user to turn on the inclusion of temperature effects in the simulation of specify a positive non-zero real number which correlates to the CN TMP card reference temperature. The temperature is measured in degrees.

Bendway Correction (CN VOR card)

The SW2 option in ADH simulates two-dimensional flow. However, some three dimensional effects can be approximated. This includes a method for correcting for the effects of vorticity around bends. Selecting the *Include vorticity* check box will cause SMS to add this card and activate this feature. It also adds the OP TRN card since vorticity is simulated in ADH as a transport constituent.

When the feature is active, The user must also specify three coefficients. These include the As and Ds empirical coefficients determined by integrating against measured values (defaults of 5.0 and 0.5) and a normalization factor.

The *Override empirical coefficients* toggle activates the edit fields for the other coefficients. If the option is unchecked, the coefficients will be reset to the default values. Zero values for both coefficients will assume the default values. Currently we do not recommended overriding the empirical coefficients.

Related Topics

- ADH Model Control

ADH Model Control Iterations

This window is part of the *ADH model control* dialog (*ADHModel Control...* command). It is accessed from the *Model Control* window by clicking on the *Iterations* tab.

Iterations

The iterations tab allows the user to specify the level of precision for the conservation of mass and momentum as ADH performs its calculations.

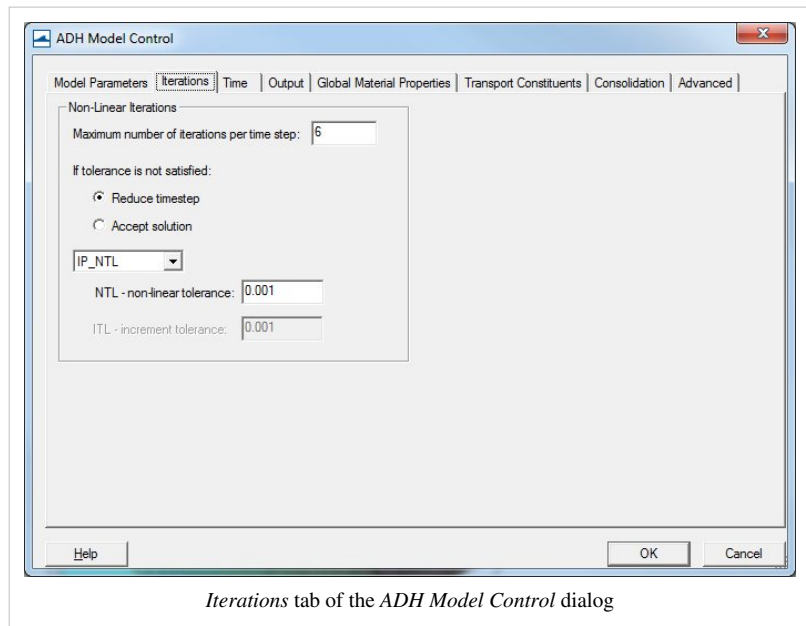
ADH solves conservation statements concerning water volume, momentum, and constituent mass. ADH is written in conservative form and thus can be

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.

ADH computes the water levels and velocities at each computation point in the domain for each time step in the simulation in an iterative process. At the end of each iteration, calculations are made to determine how much the solution is changing and what the errors in conservation are. If the errors are small, and the solution is not changing, the process is said to be converged and calculations can proceed to the next time step. If the solution is changing and/or the errors in conservation are large, ADH can attempt another iteration.

There are several iteration parameter controls in ADH. These are described in detail in the ADH users manual ^[1]. The Solver options tab controls the linear iterations. This dialog controls the non-linear iteration parameters.

At the end of each iteration ADH computes the conservation properties for the current approximations of water depth and velocity. If the errors are less than specified tolerance, the solver moves on to the next time step. If the errors are greater than the specified tolerance, but still reasonable, and the maximum number of iterations specified on the IP NIT card has not been reached, ADH will attempt another iteration. If the errors have become unreasonable (divergence), ADH will exit the loop with a failure notification. The controls in this dialog allow the user to define how ADH should determine convergence and how to proceed when convergence is not reached.



Maximum number of iterations per time step (IP NIT and IP FNI cards)

In this edit field the user specifies the maximum number of iterations that ADH will try for a single time step. By examining the output for successive iterations, a user can determine if the process is converging in general (trending towards zero), or oscillating. If the trend is towards convergence, this maximum number of iterations can be increased. As the value increases, ADH can spend more time processing individual time steps which may decrease the overall efficiency of the solver. For this reason, a user should not simply increase this parameter without verifying that it will be beneficial.

SMS writes either the IP NIT or IP FNI card based on the setting of the If tolerance is not satisfied radio group.

- IP NIT card is written if the Reduce timestep option is selected. (Recommended)
- IP FNI card is written if the Accept solution option is selected.

The IP FNI card instructs ADH to continue with the solution as if it had converged. This is not recommended.

When the Reduce timestep option is selected and the solution has diverged or the maximum number of iterations have been attempted, ADH will reduce the time step size to ¼ of the previous and continue the calculations.

When the Accept solution option is selected, and the maximum number of iterations have been attempted, ADH proceeds as if convergence has been reached to the next time step. (This option is not recommended.)

NIT/ITL/Both combo box

NIT (IP NIT card)

If the NIT option is selected, ADH will check for non-linear convergence on the "maximum residual norm". An initial estimate for an appropriate tolerance value for a problem can be computed as:

$$\frac{(total\_area * 10^{-6})}{\#\_elements}$$
 This is the maximum allowance, so you will likely want to set the tolerance higher than this estimate so your simulation will progress.

ITL (IP ITL card)

If the ITL option is selected, ADH will check for non-linear convergence on the "maximum increment norm" or the change in the solution (velocity, depth and concentration).

Both

If the Both option is selected, SMS will write both cards to the BC file and ADH will check for convergence on both terms. Both must be satisfied for the solution to proceed.

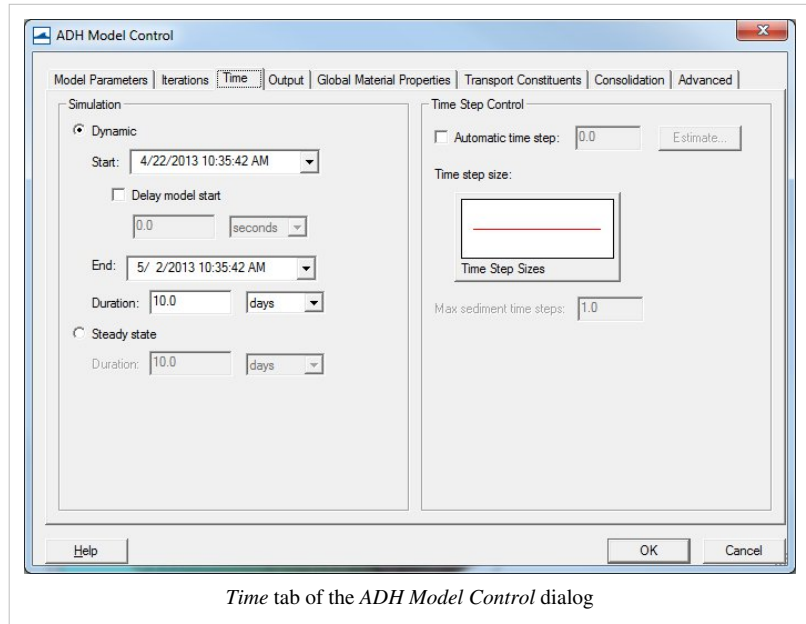
Related Topics

ADH

ADH Model Control Time

This page window is accessed from the *Model Control* window by clicking on the *Time* tab.

ADH is set up as a transient conditions model, however, a steady state simulation can be invoked which utilizes an iterative process to converge to a solution. A steady state solution is accepted if the end time of the model is reached or the iteration tolerance requirement (see Model Parameters) is met. Users are required to ensure that all series data are acceptable (constant linear series) if the steady state method is used. ADH also includes a quasi-unsteady method which strings multiple steady state simulation together to form a step function style hydrograph.



Time tab of the ADH Model Control dialog

Due to the nature of a steady state simulation, constituent transport and bed layers cannot be included in the simulation. If any constituent (and bed layer) is specified in a dynamic simulation and then the simulation is changed to steady state, SMS will clear the constituent data (and bed layer specification). A message will warn the user of this effect when the steady state simulation type is selected and constituent data (and a bed layer) exists. Another message will state that SMS has cleared the data (if steady state is still selected) when switching to another tab or exiting Model Control by selecting OK.

Since series data specified for a dynamic simulation is usually inappropriate for a steady state simulation, a change in simulation type from dynamic to steady state will force SMS to clear all previously specified series data when changes are accepted in Model Control. This data is usually associated with the boundary conditions of materials, nodes, and nodestrings. A message will warn the user of this effect when the simulation type is changed and series data exists. Another message will state that SMS has cleared the data (if the simulation type remains changed) upon accepting the changes (exiting Model Control).

Time

The following controls specify the timing parameters for the ADH model run.

Simulation

Dynamic / Steady state / Quasi-unsteady radio group specifies the simulation type.

- *Dynamic* option provides the following:
 - *Start* date and time field and the *Delay model start* check box with positive real number and units combo box correlates to the **TC T0** card.
 - *End* date and time field (with the defined start) specifies the simulation run length on the **TC TF** card.
 - *Duration* positive real number edit field and units combo box also specifies the simulation run length. This field and the End field will update as the other is changed.
- *Steady state* option provides the following:

- *Duration* positive real number edit field and units combo box to specify the simulation length.
- *Quasi-unsteady* option is currently unavailable.

Apply adaptive time control check box includes the **TC IAC** card or **TC NDP** card when checked or unchecked, respectively.

Time Step Control

Time step size curve button with the associated *Series time units* combo box, and *Specify curve fit tolerance* check box and positive integer edit field defines the data to be associated with the **TC IDT** card and its **XY1** series definition card. Clicking on the curve button will open the XY Series Editor. This series must be defined for the entire model run length, so include or exceed the time *Model run time* listed in the *Simulation* group. This control group is available only when the *Dynamic* simulation type is selected.

Initial time step positive real number edit field and units combo box specifies the time step value of the **TC STD** card. This control group is available only when the *Steady state* simulation type is selected.

Specify extra sediment transport time steps check box and positive integer edit field while include the **TC SDI** card when checked. If sediment transport is included in the simulation and this control is not checked, then the model will calculate sediment according to the time increment specified by the *Time step size* series.

Dynamic simulation type option, in the *Simulation* group, provides the following:

- *Time step size* curve button selects the time series data to be associated with the **TC IDT** card. Clicking on the curve button will open the Time Series editor (limited to the Time Step Size curve group). The selected series should be defined for the entire model run length, so include or exceed the time *Model run time* listed in the *Simulation* group.
- *SNumber* of sediment transport time steps per hydrodynamic time step positive non-zero integer edit field correlates to the **TC SDI** card. This field is only enabled if sediments have been specified on the Transport Constituents tab of Model Control. This applies only to sediments and not any other constituents. If sediment transport is not included in the simulation, then the **TC SDI** card is not included in the file.
- *Max sediment time steps* value is associated with the **TC SDI** card. This allows users to specify smaller sediment transport time steps that what is used for the hydrodynamic time step. Sediment constituents must exist otherwise this option will be grayed out and the card will not be written.
- *Automatic time step* option, provides the following:
- Specifies the time step data for the **TC ATF** card. Pressing the 'calculate' button calculates the time step based on the formula: $= x / \sqrt{-g * h}$

Where:

x = mesh element length (currently the distance between the 1st and 2nd node in the 1st element)

g = gravitational acceleration

h = min water depth

If an initial dataset doesn't exist, the automatic time step option is disabled and the **TC ATF** card is not written.

Steady State simulation type option, in the *Simulation* group, provides the following:

- *Initial time step* positive real number edit field and units combo box specifies the time step value of the **TC STD** card. This control group is available only when the *Steady state* simulation type is selected.

Related Topics

- ADH Model Control

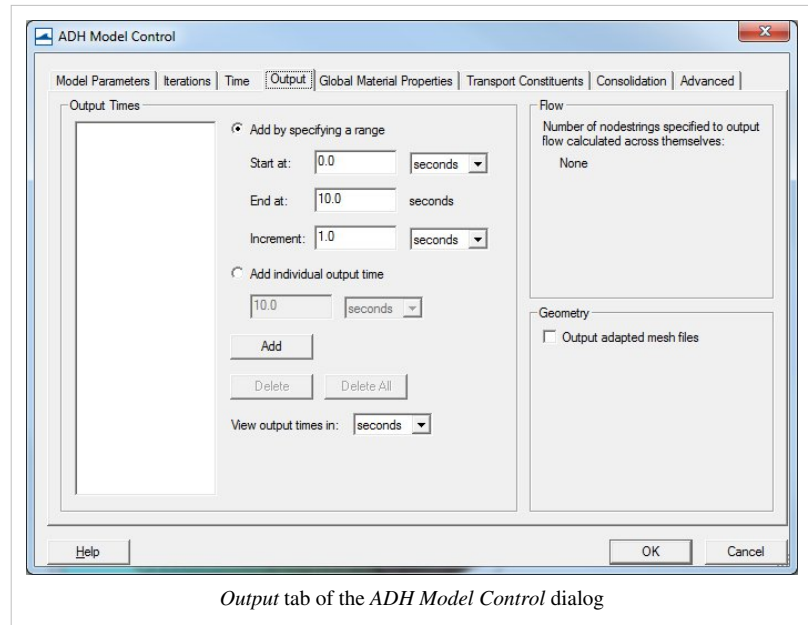
ADH Model Control Output

This window is part of the *ADH model control* dialog (*ADHModel Control...* command). It is accessed from the *Model Control* window by clicking on the *Output* tab

ADH will write solution data at startup and at each time step specified in a **XY1** series definition and referred to by the **OC** card. Flux data will be included with the solution data if output flow strings are specified (see assigning boundary conditions).

Output Control

The following controls specify how often solution data will be written by ADH and the number of flux boundaries present.



Output tab of the ADH Model Control dialog

Output Times

List box contains the times to be contained in the output series. Multiple selection is enabled for deleting times.

Add by specifying a range / Add individual output time radio group specifies time information to be added. The *Add by specifying a range* control group includes *Start at* positive integer/real number field and units combo box, *End at* positive integer/real number field, and *Increment* positive integer/real number field and units combo box. The *End at* time units are the same as specified for the *Start at* time. The *Add individual output time* control includes a positive integer/real number field and units combo box. Each edit field allows for positive integer seconds or positive real number minutes, hours, etc with six decimal places. All times are rounded to the nearest whole second (a decimal minute is rounded to the nearest decimal minute with a whole second equivalent) before being added.

Add button includes the time information from the above radio group in the times list box. Duplicate times will be automatically removed from the list.

Delete button removes the selected time(s) from the list box.

Delete All buttons clears the entire times list without regard to the current selection.

View output times in combo box specifies the time unit the list box will display the output times in.

Specify curve fit tolerance check box and positive integer edit field specifies the series tolerance when checked.

Flow

Number of nodestrings specified to output flow calculated across themselves displays the total number of ADH flux boundaries included in the simulation.

Geometry

Output adapted mesh files check box specifies whether geometry files will be saved as ADH adapts the mesh. The geometry will be outputted at the same interval specified in the list box for the solution data. This correlates to the **PC ADP** card.

Related Topics

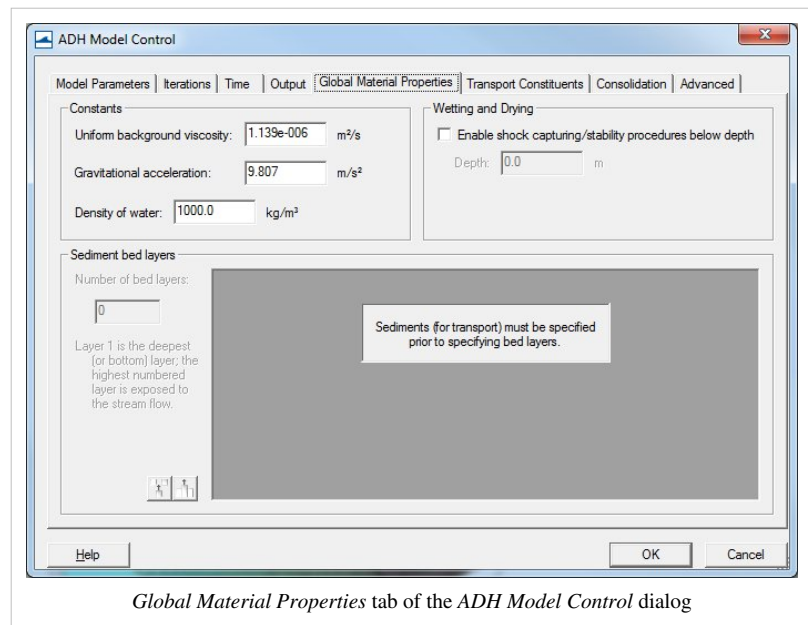
- ADH
- ADH Model Control

ADH Model Control Global Material Properties

This window is part of the *ADH model control* dialog (*ADH\Model Control...* command). It is accessed from the *Model Control* window by clicking on the *Global Material Properties* tab.

ADH contains parameters that apply to all materials and parameters that are specific to (and must be defined for) each material. The parameters found within this tab are of the former type.

The bed layers specified here are applied to all mesh nodes and can only be overwritten at the nodal level in the *Bed Layers Assignment* window. When bed layers are created or deleted here, nodal bed layers are defaulted or deleted, respectively.



Global Material Properties tab of the ADH Model Control dialog

Global Material Properties

Parameters (Untitled group box)

Enable wetting/drying limits check box and the associated *Dry depth* and *Wet depth* positive real number edit fields correlate to the optional **MP DTL** card. If the water depth of a node is below the *Dry depth*, then the node is completely dry and is not included in the ADH shallow water equations. If the water depth is above the wet depth, then the node is included in the equations. A water depth which falls within these two depths will include the node in the equation with a restricting factor.

Constant

Uniform background viscosity positive real number edit field correlates to the **MP MU** card.

Gravitation acceleration positive real number edit field correlates to the **MP G** card.

Density positive real number edit field correlates to the **MP RHO** card.

Wetting and Drying

Enable shock capturing/stability procedures below depth is check box and the associated wetting and drying process. When this toggle is selected, ADH performs extra calculations for all cells with depth values less than the specified minimum to stabilize the process. As the specified depth value increases, extra calculations also increase and model performance decreases. When this option is selected, the *Depth* edit field is enabled by SMS. In this field the users specifies a positive real number correlating to the optional **MP DTL** card.

Sediment Bed layers

Number of bed layers positive integer edit field correlates to the **MP NBL** card. The value specified is the number of bed layer column provided in the spreadsheet. If the value is increased, the new bed layer columns are appended; if decreased, the existing bed layer columns are removed. The effected bed layers are always the highest numbered layers (at the end of the spreadsheet). This is enabled only if sediments have been defined on the Transport Constituents tab.

Layer 1 is the deepest (or bottom) layer; the highest numbered layer is exposed to the stream flow. The layer IDs cannot be edited and bed layering order is fixed to ascending layer ID order (See Sediment Transport and Bed Layers).

Edit spreadsheet column toolbar is enabled when a cell selection exists in the spreadsheet, and provides:

- Insert New Column Before tool inserts a single new bed layer column before (to the left of) the current selection.
- Delete Column(s) tool removes all of the bed layer columns of the currently selected cells.

Spreadsheet contains the defined bed layers as columns and bed layer specifications as rows. The bed layers begin with Layer 1 and proceeds by ascending order by ID to the right. This spreadsheet will be empty if no sediments have been defined on the Transport Constituents tab or zero bed layers are specified. The specification rows are as follows:

- Sediment distribution – Title cell with each defined sediment listed below in its own row. The row contains Normalize buttons for each bed layer. Clicking on the button will normalize the distribution of each sediment such that the total distribution equals 100%.
- Sediment (name) – Consists of this sediment's distribution (as a percentage) for each layer. A sediment row is provided for all defined sediments. The value must be a positive real number and correlates to the **MP SBA** card.
- Total – Displays the current total specified distribution of all sediments for each layer. The row updates automatically as specifications change and cannot be edited. Each layer's total sediment distribution must equal 100%.
- Thickness – Consists of each bed layer's thickness measured in meters. The value must be a positive real number and correlates to the **MP SBA** card.
- Bulk Density – Consists of each bed layer's bulk density measured in kilograms per cubic meter. This row only appears if a cohesive sediment (clay or silt) is defined. The value must be a positive non-zero real number and correlates to the **MP CBA** card.
- Erosion crit. shear – Consists of each bed layer's critical shear stress for erosion measured in newtons per square meter. This row only appears if a cohesive sediment (clay or silt) is defined. The value must be a positive non-zero real number and correlates to the **MP CBA** card.

- Erosion rate constant – Consists of each bed layer's unitless erosion rate constant. This row only appears if a cohesive sediment (clay or silt) is defined. The value must be a positive non-zero real number and correlates to the **MP CBA** card.
- Erosion rate exponent – Consists of each bed layer's unitless erosion rate exponent. This row only appears if a cohesive sediment (clay or silt) is defined. The value must be a positive non-zero real number and correlates to the **MP CBA** card.

Related Topics

- ADH Model Control

ADH Model Control Transport Constituents

This window is part of the *ADH model control* dialog (*ADHModel Control...* command). It is accessed from the *Model Control* window by clicking on the *Transport Constituents* tab.

ADH transport constituents consist of two types: regular and sediment. Transport is only available for a normal (dynamic) simulation (specified on the *Time* tab of *Model Control*).

Constituents

Transport Constituents

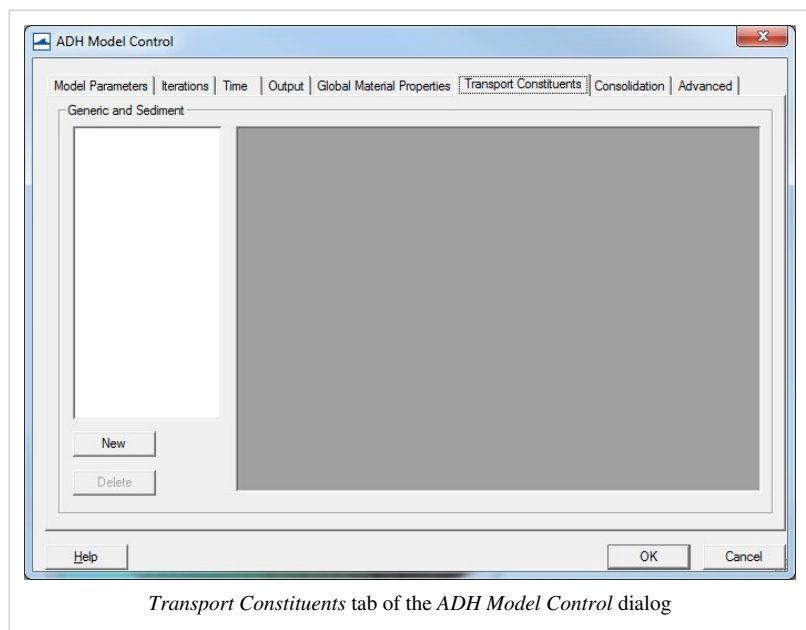
- General – **CN CON**
- Salinity – **CN SAL**
- Temperature – **CN TMP**
- Vorticity – **CN VOR**

Sediment Transport Constituents

- Sand or gravel – **CN SND**
- Clay or silt – **CN CLA**

Non-sediment Transport Constituents

- Salinity – **CN SAL** (This is a currently unavailable option)
- Temperature – **CN TMP** (This is a currently unavailable option)
- Vorticity – **CN VOR**



Transport Constituents tab of the *ADH Model Control* dialog

Dialog Description

Sediment

List box contains type and name of sediments to be included in the model simulation.

New button inserts a defaulted general sediment into the sediment list.

Delete button removes the currently selected sediment from the list.

Spreadsheet contains the parameters and values specific to the currently selected sediment in the list. Available parameters will be updated when the sediment type is changed. The sediment list box will update when the sediment name or type change.

Bendway Correction

Include vorticity check box and the associated *Normalization factor*, *As coefficient*, and *Ds coefficient* positive real number edit fields correlates to the **CN VOR** card.

Generic and Sediment

List box contains type and name of user-defined constituents to be included in the model simulation.

New button inserts a defaulted generic constituent into the user-defined constituent list.

Delete button removes the currently selected constituent from the list.

Spreadsheet contains the parameters and values specific to the currently selected constituent in the list. Constituent parameters are:

- Constituent type – Options are: Generic, Sand or gravel sediment, and Clay or silt sediment
- Name – Within SMS, the constituent will be referenced by this name. Names must be unique.
- Characteristic concentration – The characteristic concentration of the constituent is measured in micromass per unit mass or parts per million.
- Grain diameter (sand/gravel and clay/silt types) – Measured in millimeters (this is converted to meters for ADH).
- Specific gravity (sand/gravel and clay/silt types) – Value must be greater than or equal to 1.0.
- Grain porosity (sand/gravel type only)
- Bulk density (clay/silt type only) – Measured in kilograms per cubic meters.
- Erosion critical shear (clay/silt type only) – Critical shear stress for erosion is used in the erosion flux equation. Measured in Newtons per square meters.
- Erosion rate constant (clay/silt type only) – Used in the erosion flux equation.
- Deposition critical shear (clay/silt type only) – Critical shear stress for deposition is measured in Newtons per square meters.
- Settling velocity (clay/silt type only) – Measured in meters per second.

The constituent list box will update when the constituent name or type changes. If the type of a constituent is changed, the name and characteristic concentration are retained, but all other values are cleared. All value fields are restricted to positive non-zero real numbers unless stated otherwise.

Related Topics

ADH Model Control

ADH Model Control Consolidation

This window is part of the *ADH model control* dialog (*ADHModel Control...* command). It is accessed from the *Model Control* window by clicking on the *Consolidation* tab.

Consolidation

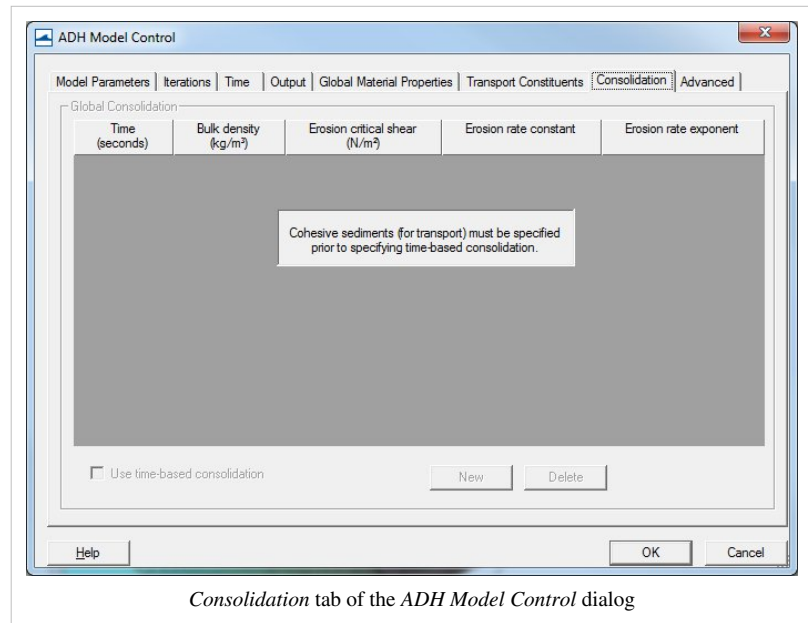
This window consists of a table defining the properties of cohesive sediments included in an ADH simulation.

The sediment classes must be specified before entering these time-based parameters for each sediment. The check box labeled *Use time-based consolidation* at the bottom of the page tells ADH that consolidation should be computed.

The spreadsheet contains the parameters and values specific to the currently selected constituent in the list. Constituent parameters are:

- *Time* (sec) – Measured in seconds.
- *Bulk density* (clay/silt type only) – Measured in kilograms per cubic meters.
- *Erosion crit. shear* (clay/silt type only) – Critical shear stress for erosion is used in the erosion flux equation. Measured in Newtons per square meters.
- *Erosion rate constant* (clay/silt type only) – Used in the erosion flux equation.
- *Erosion rate exponent* (clay/silt type only) – Used in the erosion flux equation.

All value fields are restricted to positive non-zero real numbers unless stated otherwise.



Related Topics

- ADH
- Sediment Library Control

ADH Model Control Advanced

This window is part of the *ADH model control* dialog (*ADH* | **Model Control...** command). It is accessed from the *Model Control* window by clicking on the *Advanced* tab.

Advanced Cards

SMS includes support for many of the features in ADH, however, this list of features is a dynamic set. The user base of ADH includes a development team that is constantly experimenting with new options.

This window allows SMS to maintain the options that are not otherwise supported by the interface. For

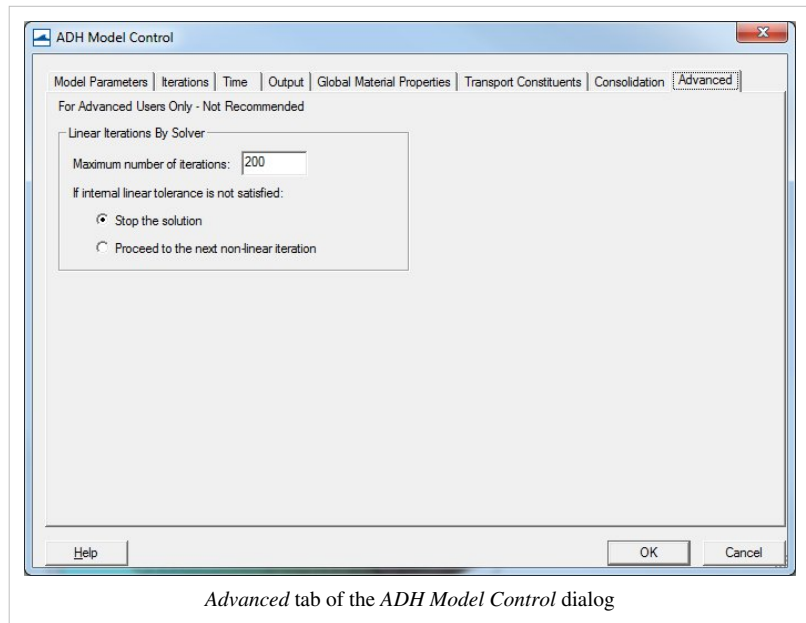
example, when SMS reads the BC file for an ADH simulation, and encounters a card that it does not recognize, that card is stored, verbatim, to a list of "Advanced Cards". That list is displayed in this window.

Users may enter cards in this dialog to experiment with other new features in the model. As those features are added to the list of supported features, they would automatically be moved to their own location in the model control the next time SMS reads the BC file.

Unsupported or advanced cards are written at the end of the BC file, just before the END card. This relies on the attribute of ADH that does not require any order dependence in the BC file.

Related Topics

- ADH
- ADH Model Control



ADH Model Control Solver

This window is part of the *ADH model control* dialog (*ADH | Model Control...* command). It is accessed from the *Model Control* window by clicking on the *Solver* tab.

(As noted at the top of this dialog, this is an advanced feature and we recommend that only users working with the ADH development team use these options.)

Linear Iterations By Solver

This window includes the options to control linear iterations in ADH. The controls are similar to the controls for non-linear iterations included on the *Iterations* tab.

The distinction between linear and non-linear iterations are described in some detail in the ADH users manual. ^[1]

Maximum number of iterations (IP MIT and IP FLI cards)

In this edit field the user specifies the maximum number of linear iterations that ADH will try for each non-linear iteration.

SMS writes either the IP MIT or IP FLI card based on the setting of the *If internal linear tolerance is not satisfied* radio group.

- IP MIT card is written if the *Stop the solution* option is selected. (Recommended)
- IP FLI card is written if the *Proceed to the next non-linear iteration* option is selected.

The IP FLI card instructs ADH to continue with the solution as if it had converged. This is not recommended.

When the *Stop the solution* option is selected and the solution has not converged at the maximum number of iterations have been attempted, ADH will treat the solution as diverged and the non-linear iterations will be aborted.

Related Topics

ADH

6.2. Finite Element Surface Water Modeling System (FESWMS)

FESWMS

FESWMS	
Model Info	
Model type	Two-dimensional finite element surface water computer program that can compute the direction of flow and water surface elevation in a horizontal plane. Also has the ability to model hydraulic structures commonly used by hydraulic engineers.
Developer	David C. Froehlich, Ph.D., P.E.
Web site	FHWA web site ^[1]
Tutorials	General Section • Mesh Editing • Observation • Overview • Sensitivity Models Section • FESWMS • FESWMS Steering • FESWMS Weirs

The Finite Element Surface Water Modeling System (FESWMS) consists of multiple modules used to simulate surface-water flow in a two-dimensional horizontal plane. The SMS includes an interface for the FST2DH (Flow and Sediment Transport) module of FESWMS. FESWMS is sponsored by the Federal Highway Administration. David C. Froehlich, Ph.D., P.E. originally developed FESWMS for the United States Department of Transportation Federal Highway Administration (FHWA) ^[2] and the United States Geological Survey (USGS) ^[3]. The FHWA has continued to maintain and sponsor development of subsequent versions, which continue to incorporate features specifically designed for modeling highway structures in complex hydraulic environments.

Functionality

FST2DH is the FESWMS program that performs the two-dimensional hydraulic computations in surface-water bodies. FST2DH can perform either steady state or dynamic flow modeling and provides analysis of highway crossings and structures, including bridges, culverts, weirs, roadway embankments, and drop-inlet spillways. FST2DH simulates the movement of water and non-cohesive sediments in rivers, estuaries, and coastal waters by applying the finite element method to solve steady-state or time-dependent systems of equations that describe two-dimensional depth-averaged surface-water flow and sediment transport. ^[1]

Using the Model / Practical Notes

- FESWMS Spindown Steering (Incremental Loading)
- The representation of the wind direction in the FESWMS Manual is incorrect. The corrected image is shown in the *FESWMS Model Control* dialog of the SMS interface.
- FESWMS known model issues
- Support Forum FESWMS with Sediment Transport Tips User Post ^[5]
- Transform your mesh so it is close to the origin (0,0). Large x, y coordinates can be problematic for FESWMS.
- Verify mesh does not have incorrect elevation values (often due to extrapolation). Try looking at the mesh in a rotated view.

General Steps to build a FESWMS Model

Various sequences of steps can be used to create and edit a two-dimensional model in SMS. A suggested methodology is provided below.

1. Import background data consisting of:
 1. Elevation data covering the area to be modeled. See Scatter Module
 2. Open and register background image(s) if desired. See Images
2. Create a FESWMS coverage and define the Conceptual Model that will be used to generate the finite element mesh. See Map Module
 1. Define the physical boundaries of the model
 2. Build feature polygons within the model domain. See Build Polygons in the Feature Objects menu.
 3. Assign mesh types to the feature polygons and adjust the density and spacing of vertices (i.e., redistribute vertices) as necessary to achieve appropriate element shapes and sizes
3. Create an area properties coverage and assign material types. This is an optional step. The FESWMS coverage can also be used to define material regions. See Map Module
4. Generate a finite element mesh from the Conceptual Model. See Map Module
5. Check the final quality of the finite element mesh and edit the mesh as needed. See 2D Mesh Module
 1. Designate element types (i.e., linear or quadratic; triangular or quadrilateral)
 2. Review the mesh quality
6. Specify FESWMS element attributes (i.e., material properties) such as Manning's or Chezy roughness factors and eddy viscosity to the material types. See 2D Mesh Module
7. Create nodestrings and assign model boundary conditions not included in the conceptual model. See 2D Mesh Module
8. Renumber the finite element mesh.
9. Specify initial conditions and model control parameters. See FESWMS Model Control
10. Run the model.
11. View model results. If necessary, troubleshoot error/warning messages generated during the model run and revise the model accordingly.

Graphical Interface

SMS provides a graphical interface that is designed to allow users to visualize the projects they are creating, easily modify project parameters, and view the solutions produced by the FESWMS model. See FESWMS Graphical Interface for more information.

The FESWMS Graphical Interface contains tools to create and edit an FESWMS simulation. The simulation consists of a geometric definition of the model domain (the mesh) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the 2D Mesh Module and setting the current model to FESWMS. If a mesh has already been created for a FESWMS simulation or an existing simulation read, the mesh object will exist in the Project Explorer and selecting that object will make the 2D Mesh module active and set the model to FESWMS. See the Mesh Module documentation for guidance on building and editing meshes as well as visualizing mesh results.

The interface consists of the 2D Mesh Module Menus and tools augmented by the FESWMS Menu. See FESWMS Graphical Interface for more information.

Saving FESWMS

When you do a *File* | **Save As...** the following files get saved in the *.sms.

- *.fpr referenced to new save location
- *.mat referenced to new save location
- *.flo referenced to original location unless reran then moves to saved location.

External Links

- Sept. 2002 (Revised October 2003) – User's Manual for FESWMS FST2DH – Publication No. FHWA-RD-03-053 [6]
- U.S. Department of Transportation Federal Highway Administration – Hydraulics Engineering Software Page [7]
- Ipson, Mark K. (2006). Analysis of the Sediment Transport Capabilities of FESWMS FST2DH. Thesis, Brigham Young University. [8]

Related Topics

- FESWMS Files
- FESWMS Graphical Interface
- FESWMS Hydraulic Structures
- FESWMS Model Control Dialog
- FESWMS Spindown
- FESWMS Sediment Transport
- FESWMS Known Model Issues

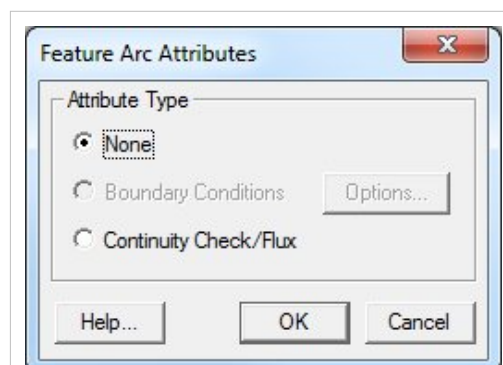
References

- [1] Sept. 2002 - User's Manual for FESWMS Flo2DH - Publication No. FHWA-RD-03-053

FESWMS Arc Attributes Dialog

The *FESWMS Feature Arc Attributes* dialog is used to set the attributes for feature arcs. Attributes that can be specified for each feature arc include:

- Arc Type
 - None
 - Boundary Conditions – Options button opens the FESWMS Nodestring Boundary Conditions dialog
 - Continuity Check/Flux



FESWMS Feature Arc Attributes dialog

Related Topics

- Feature Objects Menu

FESWMS BC Nodestrings

Generic 2D Mesh Boundary conditions are generally defined on feature arcs in the conceptual model or nodestrings on the 2D mesh. Boundary conditions constrain the water surface elevation and/or flow at the model boundary.

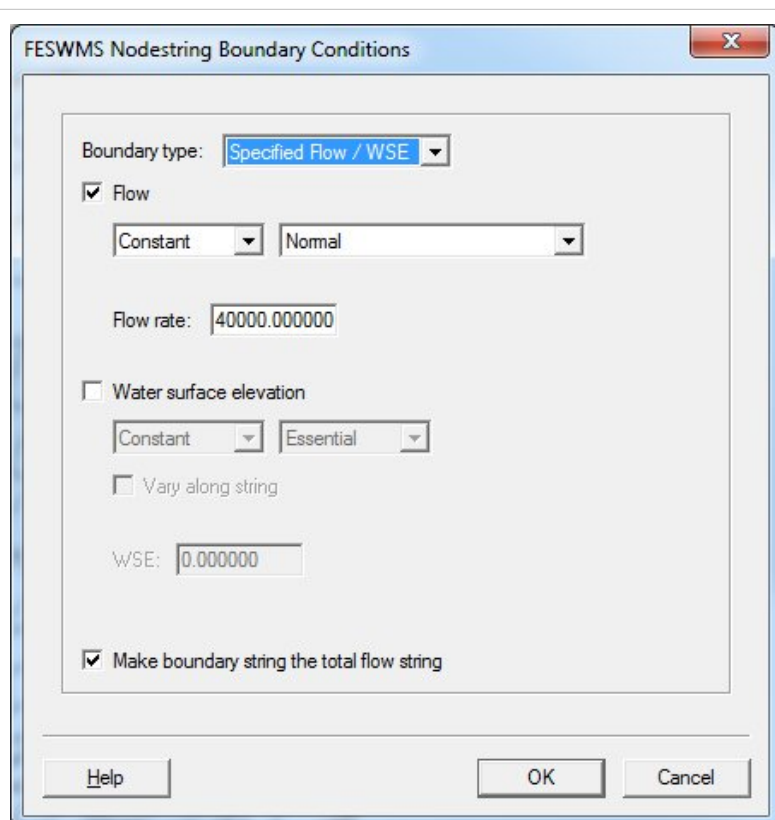
General Boundary Options

Constant vs Dynamic

FESWMS boundary conditions can be constant or dynamic. If a constant boundary condition is used with a dynamic simulation the boundary condition does not change during the simulation. It is not possible to have a dynamic boundary condition with a steady state simulation.

Total Flow String

The nodestring that is the total node flow string identifies the flowrate that will be considered 100% of the flow in the FESWMS printed output file (*.prt). Only one nodestring may be the total flow nodestring and it will be the last one identified as the total flow nodestring.



FESWMS Nodestring Boundary Conditions dialog

Boundary Condition Types

Specified Flow / WSE

This type of boundary constrains the flowrate and/or water surface elevation to fixed values. Both the flowrate and water surface elevation should only be specified for supercritical inflow boundary conditions.

A flow boundary condition can be a normal boundary or a weakly reflecting boundary. A weakly reflecting boundary can be used to allow tidal variation to affect the boundary. A weakly reflecting boundary simulates a fictitious river without frictional resistance upstream of the boundary.

Generally a fixed water surface elevation boundary is used for the downstream boundary condition of the model. The water surface elevation can vary from one end of the boundary to the end by using the vary along string option.

Supercritical Outflow

For supercritical boundaries, both the flowrate and water surface elevation is defined on the inflow boundary. Even though no data is associated with supercritical outflow boundaries, these boundaries must be identified so FESWMS will allow flow through the boundary.

Computed WSE

This is used to provide a rating curve (water surface vs flow) relationship to compute the downstream boundary condition. FESWMS can compute the rating curve using the friction slope option. Strangely enough FESWMS requires the user to provide flow information to use this option.

Storm/Tide WSE

This is used to tell FESWMS that the boundary water surface elevation will be determined from the storm associated with the simulation.

Related Topics

- FESWMS Arc Attributes Dialog

FESWMS Executable Known Issues

There are several different builds of the FESWMS executable (fst2dh.exe). Unfortunately there are some known issues and likely some unknown issues as well. This page is intended to help users know which executable to use for which type of project they are working on.

FESWMS development is sponsored by the Federal Highway Administration. Aquaveo does not have access to the FESWMS source code and cannot make changes to fix the issues found in the FESWMS numerical model. Aquaveo can fix problems in the SMS interface, used for pre and post processing of FESWMS files.

This list may not contain all know issues, so please feel free to add to it.

Sediment Transport

Aquaveo employees have successfully run test cases using the following options:

- Equilibrium inflow
- Clear water inflow

We have been unsuccessful in our attempts to create test cases using the other options ^[1].

Version 3.3.2

This is the version of fst2dh.exe that is currently distributed with SMS 10.0

- Does not work with Piers.

Version 3.3.2 can be downloaded at <ftp://pubftp.aquaveo.com/download/fst2dh332.exe> ^[2].

Version 3.3.3

- Fixed the issue with Piers but does not work with transient models. It is unknown what is wrong specifically other than models that run using 3.3.2 do not always work with 3.3.3. The problem seems to be that the model doesn't work with transient models.

Version 3.3.3 can be downloaded at <ftp://pubftp.aquaveo.com/download/fst2dh333.exe> ^[3].

Related Topics

- FESWMS

References

- [1] Ipson, Mark K. (2006). Analysis of the Sediment Transport Capabilities of FESWMS FST2DH. Thesis, Brigham Young University. (<http://contentdm.lib.byu.edu/ETD/image/etd1542.pdf>)

FESWMS Files

Input and Output files for FESWMS.

Input Files

- FST2DH project file (*.fpr)
- Control data (*.dat)
- Mesh data (*.msh – *.net for SMS)
- Flow input data (*.flo)
- Sediment input data (*.sed – *.sdi for SMS)
- Boundary condition data (*.bcs)
- Wind data (*.wnd)
- Wave data (*.wve)
- Time-dependent data (*.tim)

Output Files

- Report data (*.rpt – *.prt for SMS)
- Flow output data (*.flo)
- Sediment output data (*.sed)
- Restart-recovery data (*.rsr)
- Upper coefficient matrix (*.upp)
- Lower coefficient matrix (*.low)
- Scalar output data (*.scl)
- Vector output data (*.vec)
- Profile output data (*.pro)
- Run status data (*.sta)

Related Topics

- FESWMS
 - FESWMS Hydraulic Structures
 - FESWMS Model Control Dialog
 - FESWMS Spindown
-

FESWMS Graphical Interface

The FESWMS Graphical Interface includes tools to assist with creating, editing and debugging a FESWMS model. The FESWMS interface exists in the Mesh Module.

Model Control

The *FESWMS Model Control* dialog is used to setup the options that apply to the simulation as a whole. These options include time controls (steady state/dynamic), run types, output options, global parameters, print options and other global settings.

Boundary Conditions

All numeric models require boundary condition data. Boundary conditions in FESWMS include flows in/out of the model domain or known water surface elevations. In FESWMS boundary conditions are generally defined on nodestrings but may also be defined on nodes. The default boundary condition is a closed boundary (no flow). See FESWMS BC Nodestrings and FESWMS Point Attributes Dialog for more information.

Material Properties

Each element is assigned a material type. Material properties describe the hydraulic characteristics of each material type.

Hydraulic Structures

FESWMS was designed for use around highways and includes support for several different types of structures including weirs, culverts, drop inlets, gates and piers. See FESWMS Hydraulic Structures for more information.

Running the Model

The FESWMS Files are written automatically with the SMS project file or can be saved separately using the *File | Save FESWMS* or *File | Save As* menu commands. See FESWMS Files for more information on the files used for the FESWMS run.

FESWMS can be launched from SMS using the *FESWMS | Run FSTD2H* menu command. A check of some of the common problems called the Model Checker is done each time the model is launched, or by selecting the *FESWMS | Model Check* menu command.

FESWMS Menu

See FESWMS Menu for more information.

Related Topics

- FESWMS
- FESWMS Files
- FESWMS Hydraulic Structures
- FESWMS Model Control Dialog
- FESWMS Spindown
- Total Flow Nodestring

FESWMS Hydraulic Structures

Overview

FESWMS has the capability to model hydraulic structures including: bridges, roadway embankments, culverts, weirs, and drop-inlet spillways. In the finite element network, bridges and roadway embankments are represented with a collection of two-dimensional elements, which overlay the plan view of these structures. However, since culverts, weirs, and drop-inlet spillways are difficult to characterize with elements, these structures are modeled with either one or two node points, with these nodes representing points of inflow and outflow. An exception to this modeling technique occurs when a culvert spans a large channel or is large in comparison to the size of the defined floodplain elements; in this instance, the user should consider modeling the culvert with two-dimensional elements.

Culvert

Small culverts are modeled with inlet and outlet node points in FESWMS unless they are located on the model boundary, in which case they are represented with one node point. Parameters for characterizing small culverts include barrel geometry (shape, length, span, slope, and number of barrels), inlet configuration, barrel roughness, and tailwater depth. FESWMS employs the FHWA ^[2] culvert routing routines as presented in *Hydraulic Design Series 5 (HDS 5)*, *Hydraulic Design of Highway Culverts* [1] [2] and discharge through a culvert is determined by the energy heads at the nodes where the culvert is placed. To define a culvert with two mesh nodes, proceed with the following steps:

1. Select the culvert's inlet and outlet node by choosing the **Select Mesh Node** tool  and clicking on the nodes of interest while pressing the *Shift* key.
2. Select the menu option **FESWMS | Culvert**. The *FESWMS Culvert* dialog will appear allowing the entry of culvert parameters. Note that when the Flap-gate option is selected, the model only permits flow to travel in the downstream direction. At this point, the user should verify that the nodes identified as "upstream" and "downstream" in the FESWMS Culvert dialog represent the culvert inlet and outlet, respectively.
3. Culvert material, shape, and inlet type may be designated by selecting the appropriate drop-down menus and clicking on the desired values. Additionally, the *FESWMS Culvert* dialog offers help in determining the culvert entrance loss coefficient (k_e) and the Mannings roughness coefficient of the culvert barrel (n_c); simply select the **Help** button next to these two parameters for information. To enter inlet control flow coefficients other than the default values provided (not recommended for novice users), click on the Override defaults box next to the appropriate parameters and enter a new value. Further information concerning the selection of culvert parameters is provided in HDS 5.



A small culvert under a pedestrian walkway.

FESWMS Culvert

Name: culvert

Comment:

Type and Specifications

Type code: 1011

Material: Concrete

Shape: Circular

Inlet: Headwall, square edge

Number of barrels: 1

Entrance loss coefficient (ke): 0.000000 Help

Manning roughness coefficient (nc): 0.000000 Help

Rise (height): 0.000000

Span (width): 0.000000

Barrel length (Lc): 252.020694

Invert elevation at upstream node: 0.000000

Invert elevation at downstream node: 0.000000

Tailwater elevation for outlet control: 0.000000

Minimum head difference: 0.000000

Riprap basin depth: 0.000000

Flow Direction

Upstream node ID: 2160

Downstream node ID: 2202

Switch nodes

☐ Flap-gate

Inlet Control Flow Coefficients

	Default	Override value
K:	0.009800	☐ 0.000000
M:	2.000000	☐ 0.000000
c:	0.039800	☐ 0.000000
Y:	0.670000	☐ 0.000000
alpha:	1.040000	☐ 0.000000

Help... OK Delete Culvert Cancel

FESWMS Culvert dialog

Weir

FESWMS weirs can be used to model flows over topping bridge decks, embankments, guide banks (spur dikes), and other structures. As with culverts, weirs located in the interior of the finite element network are described by two node points, one on the upstream side and one on the downstream side of the weir. Weir flow is typically modeled by dividing the bridge deck, embankment, or other structure into a series of weir segments. Each segment is described by the appropriate number of nodes (either one or two), a discharge coefficient, submergence criteria, and the length and crest elevation of the segment (all necessary as input to the broad-crested weir-flow equation). To define a weir segment, perform the following steps:

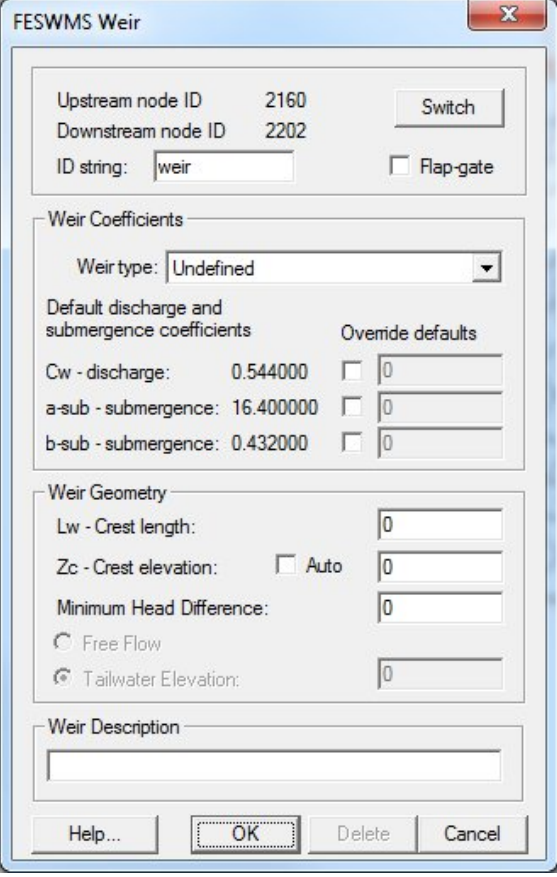


The weir at Coburg Lake in Victoria (Australia) after heavy rainfall.

1. Select one or two nodes (depending on if the weir is on the boundary or in the interior of the model) by clicking on the Select Mesh Node tool  and choosing the node or nodes of interest in defining the weir. If defining a "two-node" weir, hold the *Shift* key to select both the upstream and downstream node.
2. Select the menu option **FESWMS | Weir** to open the *FESWMS Weir* dialog.
3. Adjust the default parameters and enter the user-defined values as necessary. The "paved roadway" option is typically selected from the Weir type drop-down menu for bridge decks. As with culverts, when the Flap-gate option is selected, the model only permits flow to travel in the downstream direction. Also, the Switch button can be used to change the flow direction if the nodes identified as "upstream" and "downstream" do not correspond to the actual flow direction over the modeled weir.

When defining a series of weir segments to model a bridge deck or other structure, one can create nodestrings on the upstream and downstream sides of the structure. After creating the nodestrings, select each nodestring using the **Select Nodestring** tool  in turn while holding the *Shift* key, click on the menu option **FESWMS | Weir**, and assign the appropriate crest elevation and other parameters. Further information on weir flow and the broad-crested weir

equation can be found in the FESWMS Users Manual [3] and FHWA publication Hydraulic Engineering Circular (HEC) 22, Urban Drainage Design Manual [4] [5].



The image shows the 'FESWMS Weir' dialog box. It has a title bar with a close button. The main area is divided into several sections: 'Upstream node ID' (2160) and 'Downstream node ID' (2202) with a 'Switch' button; 'ID string' (weir) and a 'Flap-gate' checkbox; 'Weir Coefficients' with a 'Weir type' dropdown (Undefined) and a table for coefficients; 'Weir Geometry' with fields for crest length, elevation, and head difference; and 'Weir Description' with a text area. At the bottom are 'Help...', 'OK', 'Delete', and 'Cancel' buttons.

Default discharge and submergence coefficients		Override defaults
Cw - discharge:	0.544000	☐ 0
a-sub - submergence:	16.400000	☐ 0
b-sub - submergence:	0.432000	☐ 0

Weir Geometry	
Lw - Crest length:	0
Zc - Crest elevation:	☐ Auto 0
Minimum Head Difference:	0
☐ Free Flow	
☒ Tailwater Elevation:	0

Buttons: Help..., OK, Delete, Cancel

FESWMS Weir dialog

Drop-Inlet Spillways

Drop-inlet spillways are sometimes used as flow conveyance structures in reservoirs, and the discharge through this type of spillway is limited by one of three flow conditions (depending on the existing water surface at the inlet and the configuration of the structure):

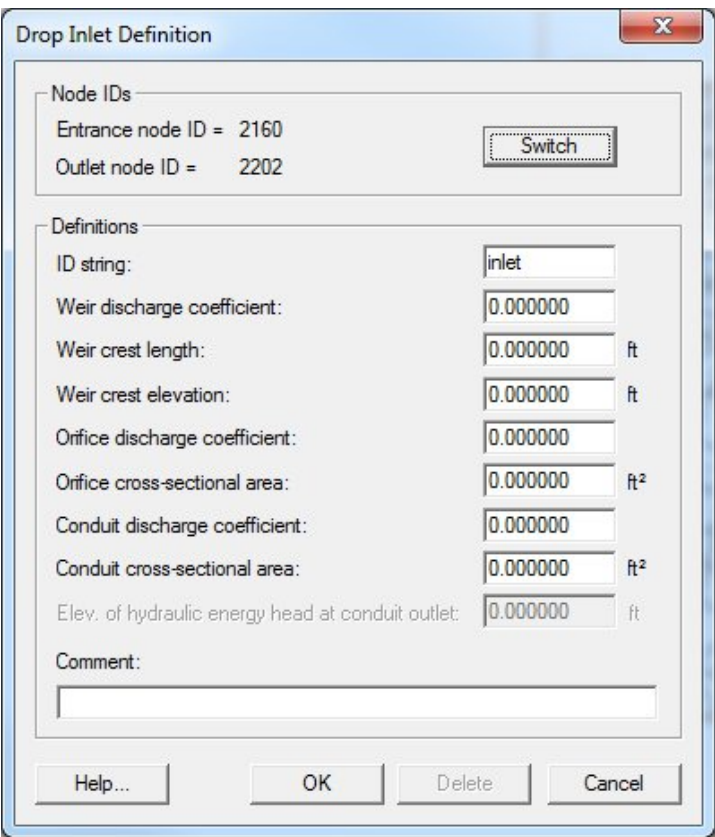
1. Weir flow
2. Orifice flow
3. Full pipe flow

Flow through a drop-inlet spillway in the interior of a finite element network is modeled by designating an inflow (entrance) node, an out-flow (exit) node, and a set of parameters that describe the structure. If the structure is located on one of the model boundaries, only an entrance node is specified (as flow is assumed to exit the model domain).

To specify a drop-inlet spillway in the FESWMS model:

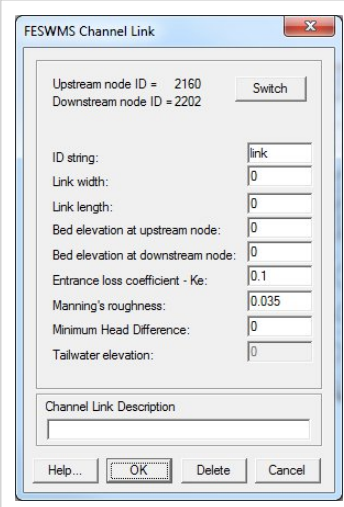
1. Select one or two nodes (depending on if the drop-inlet spillway is on the boundary or in the interior of the model) by clicking on the **Select Mesh Node** tool  and selecting the nodes of interest.
2. Select the menu option **FESWMS | Drop Inlet**. The *Drop Inlet Definition* dialog will open, allowing the entry of the inlet characteristics including geometry and discharge coefficients.

Help with selection of weir and orifice parameters is provided in the FHWA publication Hydraulic Engineering Circular (HEC) 22, Urban Drainage Design Manual [4] [5].



FESWMS Drop Inlet Definition dialog

Channel Link



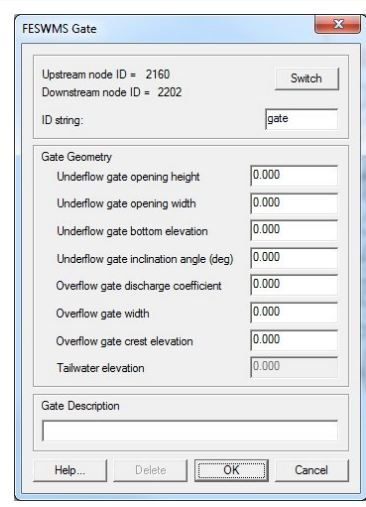
The FESWMS Channel Link dialog box contains the following fields and controls:

- Upstream node ID = 2160
- Downstream node ID = 2202
- Switch button
- ID string: link
- Link width: 0
- Link length: 0
- Bed elevation at upstream node: 0
- Bed elevation at downstream node: 0
- Entrance loss coefficient - K_e : 0.1
- Manning's roughness: 0.035
- Minimum Head Difference: 0
- Tailwater elevation: 0
- Channel Link Description text area
- Buttons: Help..., OK, Delete, Cancel

FESWMS Channel Link dialog

- Switch
- ID String
- Link width
- Link length
- Bed elevation at upstream node
- Bed elevation at downstream node
- Entrance loss coefficient - K_e
- Manning's roughness
- Minimum Head Difference
- Tailwater elevation

Gate



The FESWMS Gate dialog box contains the following fields and controls:

- Upstream node ID = 2160
- Downstream node ID = 2202
- Switch button
- ID string: gate
- Gate Geometry section:
 - Underflow gate opening height: 0.000
 - Underflow gate opening width: 0.000
 - Underflow gate bottom elevation: 0.000
 - Underflow gate inclination angle (deg): 0.000
 - Overflow gate discharge coefficient: 0.000
 - Overflow gate width: 0.000
 - Overflow gate crest elevation: 0.000
 - Tailwater elevation: 0.000
- Gate Description text area
- Buttons: Help..., Delete, OK, Cancel

FESWMS Gate dialog

- Switch
- ID string
- Gate Geometry
 - Underflow gate opening height
 - Underflow gate opening width
 - Underflow gate bottom elevation
 - Underflow gate inclination angle (deg)
 - Overflow gate discharge coefficient
 - Overflow gate crest elevation
 - Overflow gate crest elevation
- Tailwater elevation

Bridges & Roadway Embankments

Unlike culverts, weirs, and drop-inlet spillways, bridges and roadways will usually be modeled with a collection of two-dimensional elements. Bridge modeling considerations include:

1. Deck roughness
2. Weir flow over decking
3. Pressure flow through the bridge opening
4. Flow around bridge piers
5. Embankment elevations

Once bridge elements are created (in the Map Module during network construction) such that they conform to the two-dimensional plan view of the bridge deck, further treatment of the bridge and associated embankments continues as described below:

Deck Roughness

To define the roughness of the bridge deck elements in the Mesh Module, proceed as follows:

1. Click on the **Select Element** tool  and highlight the elements representing the bridge deck while holding the *Shift* key.
2. Select the menu option **FESWMS | Material Properties** and click on the *Roughness Parameters* tab of the *FESWMS Material Properties* dialog.
3. To define a material property group for the bridge elements (if not previously defined during network construction), select the **General material properties** button, click on the *New* button in the *Materials Data* window, and enter the property name for the bridge elements. Once finished, click **OK**.
4. In the *FESWMS Material Properties* dialog, highlight the name of the bridge elements in the left-hand window and enter the Manning's n in the *Deck Roughness* edit field. Click **OK**.

A similar procedure can be used to define roughness values for the roadway, including embankments.

Weir Flow Over Decking

Weir flow over decking can be modeled by segmenting the bridge and approaches and establishing a weir for each segment as described previously.

Pressure Flow Through The Bridge Opening

In order for pressure flow to be calculated, the property group representing the bridge deck needs to be coded for potential pressure flow and a bridge deck ceiling elevation (also known as the low chord or low steel elevation) needs to be specified for each element node in the deck as follows:

1. To indicate potential pressure flow, first select **FESWMS | Material Properties** and under the *Roughness Parameters* tab highlight the bridge properties group and select the *Potential Pressure* flow box.
2. Next, code in the ceiling elevation for each element node in the deck by using the **Select Mesh Node** tool  and highlighting the element nodes corresponding to the bridge deck by holding the *Shift* key and clicking on each node to be assigned the ceiling elevation. An alternative means of selecting all bridge nodes is to choose the **Select Mesh Node** tool , click on the menu option **Edit | Select by Material Type**, choose the *Bridge* material type, and click on *Select*. Once the appropriate bridge deck nodes have been highlighted, select **FESWMS | Local Parameters** and click on the **Ceiling Options** button. In the *Specify ceiling for selected items* box, enter the ceiling or low chord elevation of the selected nodes and click **OK**.

Bridge and Embankment Elevation Adjustments

To adjust bridge and embankment elevations:

1. Focus in on the area surrounding the embankment or bridge nodes of interest using the **Zoom** tool.
2. Select the corner nodes of the roadway element using the **Select Mesh Node** tool .
3. Adjust the elevations of the nodes to the desired values by overwriting the z-coordinate value in the Z node coordinate display edit field in the SMS window. Note that adjusting the corner nodes automatically adjusts the center and midside nodes.
4. Repeat these steps until all bridge and roadway elements have been set to the desired elevations.

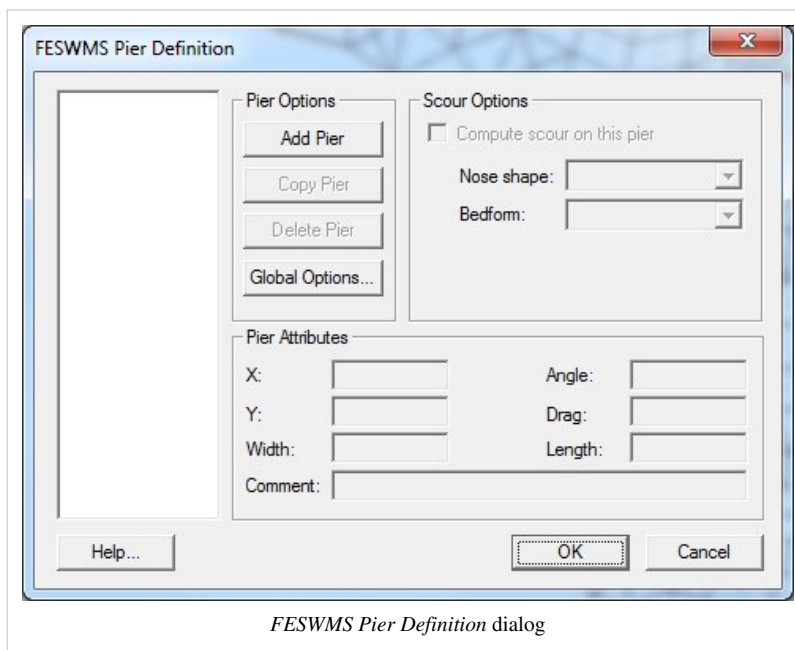
Piers

Piers may be modeled in SMS by either of two methods:

1. Creating elements of the proper geometric sizes and elevations in the network to mimic piers
2. Utilizing the tools provided in SMS including the *FESWMS Pier Definition* dialog

To create a pier in the Mesh Module using the *FESWMS Pier Definition* dialog:

1. Select the **Create Pier** tool and click on the approximate position of the pier in the finite element mesh.
2. Choose the **Select Pier** tool, select the pier of interest, and choose the menu option *FESWMS | Pier*. The *FESWMS Pier Definition* dialog will open. Pier parameters used by FESWMS describe the pier location, geometry, and flow resistance. The pier angle is the orientation of the long axis of the pier measure counterclockwise from the x-axis.
3. Adjust the pier parameters, including coordinate location, as necessary by filling in the various edit fields under *Pier Attributes*. Note that pier names can be changed by clicking on the name displayed in the *FESWMS Pier Definition* dialog.
4. Various parameters for pier scour computations can also be specified by selecting the *Global Options* button. For more detailed information on pier parameters and scour calculations, refer to FHWA publication Hydraulic Engineering Circular (HEC) 18, Evaluating Scour at Bridges [6] [7] [8] [9].



Modeling Tips

- When dividing a weir into segments on an element basis, it is recommended that you assign 2/3 of the length to the midside node of an element, and 1/6 of the length to each of the adjacent vertex nodes.
- More than two rows of elements may be required to accurately model pressure flow through a river crossing.
- A smaller, more highly refined finite element mesh should be used at and around piers, abutments, and other areas of curvilinear/complex flow patterns to improve the accuracy of model results.

External Links

- Hydraulic Design Series 5 (HDS 5), Hydraulic Design of Highway Culverts [1] [2]
- Hydraulic Engineering Circular (HEC) 18, Evaluating Scour at Bridges [6] [7] [8] [9].
- User's Manual for FESWMS Flo2DH [3]
- Urban Drainage Design Manual Second Edition [4] [5]

Related Topics

- FESWMS
- FESWMS Files
- FESWMS Graphical Interface
- FESWMS Model Control Dialog
- FESWMS Spindown

FESWMS Material Properties

FESWMS material properties include roughness, turbulence, and wind/wave parameters. The *Materials Properties* dialog is accessed through the *FESWMS* menu.

Roughness Parameters

The roughness helps determine the energy losses as water flows over elements. Each material includes roughness information.

- Manning n values (n_1 , n_2 , depth1, depth2)

The primary roughness property is the manning n value associated with the element. The manning n value can vary with depth by specifying n values at two depths. The n_1 value is used below depth1. The n_2 value is used above depth2. Between depth1 and depth2 the n value is linearly interpolated.

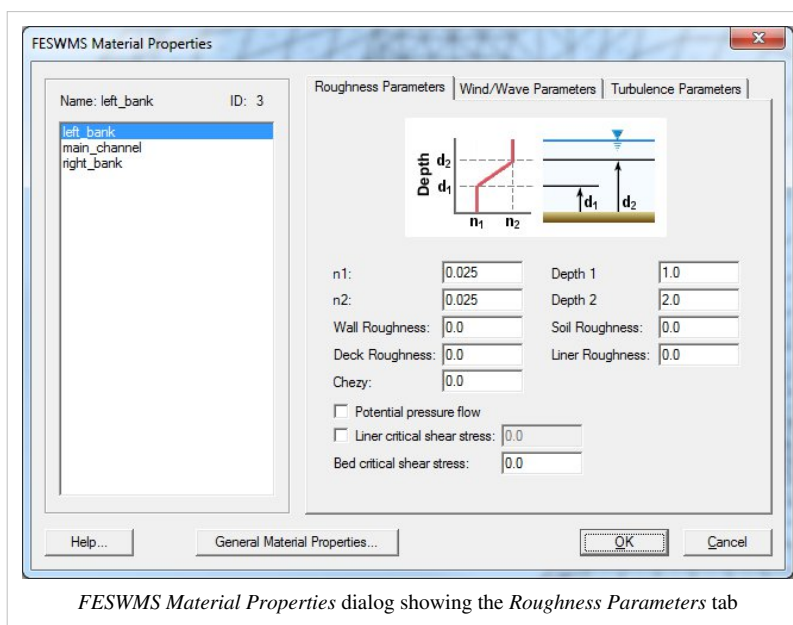
- Wall roughness

The wall roughness is used on the edge of the model domain. Wall roughness is ignored unless the model is using semi-slip boundaries.

- Soil Liners

FESWMS can be used with materials representing a soil liner. To use a liner, turn on the linear critical shear stress and set the value. When using a liner n_1 , n_2 , depth1, and depth2 are ignored.

- Pressure flow



FESWMS Material Properties dialog showing the Roughness Parameters tab

It is required to toggle on "potential pressure flow" with all materials that are assigned to elements with a ceiling elevation. If pressure flow is enabled, the deck roughness is the manning value for the bridge deck. Otherwise deck roughness is ignored.

- Chezy

Chezy values are an alternative to using Manning n values for roughness. You must turn on chezy in the model parameters to use this value.

- Bed critical shear stress

The bed critical shear stress is used to compute clear water scour.

Turbulence Parameters

Turbulence parameters are used to control the energy lost in turbulence.

- V_o

Base eddy viscosity in $\text{Length}^2/\text{second}$

- $Cu1$, $Cu2$

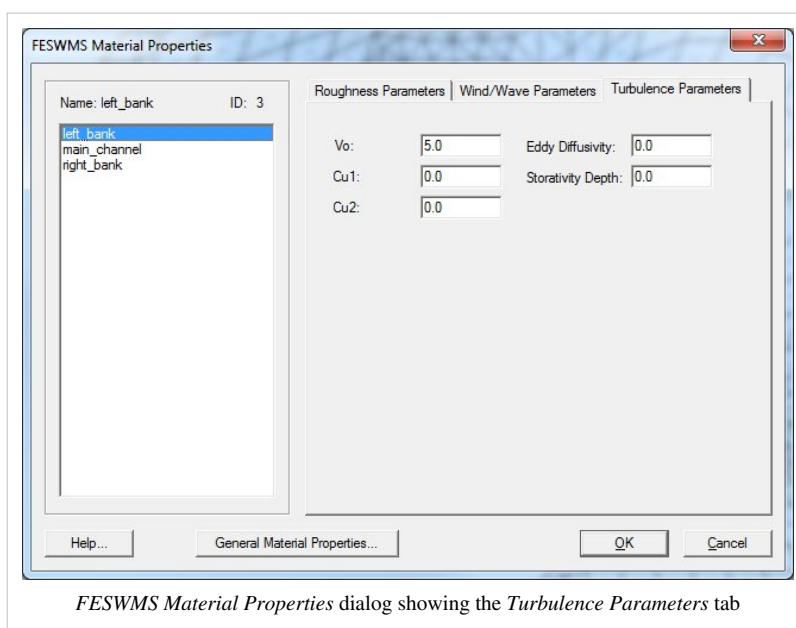
Turbulence model coefficients used to modify base eddy viscosity

- Eddy diffusivity

Used to modify base eddy viscosity and is related to the amount of curves in the channel

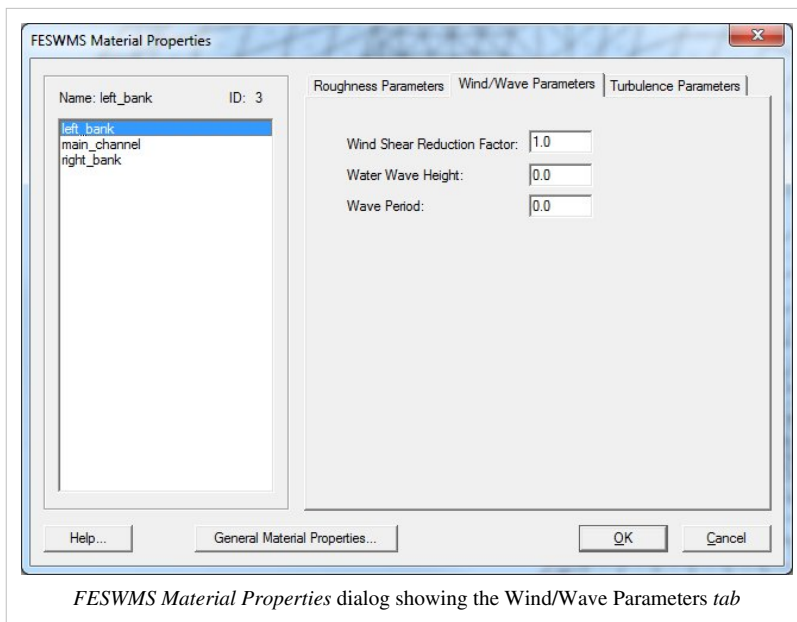
- Storativity depth

The global storativity depth (specified in the model parameters) can be overridden on a material level by entering a non-zero value.



Wind/Wave Parameters

- Wind Shear Reduction Factor
- Water Wave Height
- Wave Period



Related Topics

- FESWMS Menu

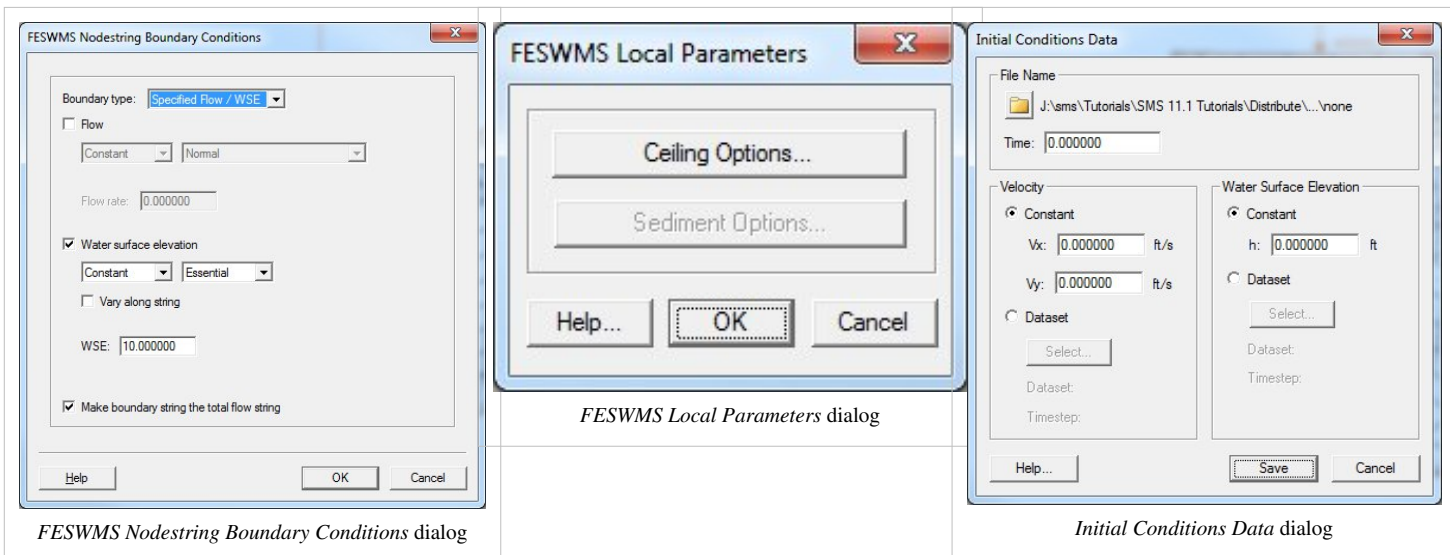
FESWMS Menu

The following menu commands are available in the FESWMS Menu:

Boundary Conditions

- Assign BC (Boundary Condition)
- Local Parameters
- Initial Conditions

Boundary Conditions Dialogs



Hydraulic Structures

- Weir
- Culvert
- Drop Inlet
- Channel Link
- Gate
- Pier

Model Parameters

- FLUX String
- Material Properties
- Model Check
- Model Control
- Run FST2DH

Related Topics

- FESWMS

FESWMS Model Control Dialog

The *FESWMS model control* dialog is used to set general simulation options. This document highlights the more commonly used options. Refer to the FESWMS manual[3] for a more detailed description of how these parameters affect the model results.

General Tab

The controls in the *General* tab include model description, FESWMS version, run controls, input options, and output options.

- Network stamp and BC descriptor

These are comments that are added at the top of the FESWMS input files. The network file (*.net) includes only the network stamp but the bc file includes both the network stamp and the bc descriptor.

- Slip conditions

This controls how boundaries are handled by FESWMS. Slip boundaries allows water to

move freely along the boundary. No slip fixes the velocity at fixed boundaries to 0.0. Semi-slip allows water to move along boundaries but reduces the velocities.

- Run Type

A simulation can compute hydrodynamics, sediment, or both hydrodynamics and sediment (semi-coupled). It is not recommended to use the sediment only option. Use the semi-coupled option instead.

- Solution Type

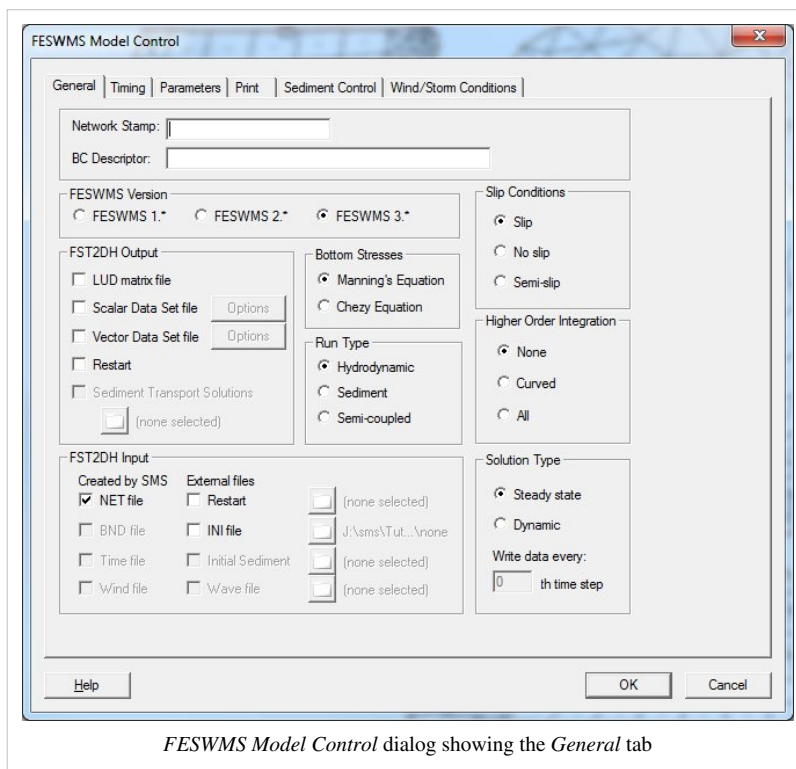
This controls whether the simulation will be treated as steady state or dynamic (time dependent). A steady state run requires boundary conditions that do not change and computes a solution that does not change through time.

- Input Options

This controls the files that will be used in FESWMS.

An INI (initial conditions) file is used to provide an initial solution for FESWMS. The initial conditions file format is identical to the FESWMS solution file format and a FESWMS solution may be used for an initial conditions file. Initial conditions files can also be created from inside of SMS. INI files are used as hot starts in the spin down process.

- Output Options



The scalar dataset file and vector dataset files are additional output files written in the EMRL dataset format. Regardless of whether this file is used or not, the datasets water surface elevation, depth, and velocities are written to the *.flo solution file. The dataset files will have the same base name as the other project files and have the extensions *.scl (scalar) and *.vec (vector). The data written to the dataset file can be specified by clicking on the options button. Available scalar datasets include: Depth-averaged velocity magnitude, Unit flow rate magnitude, Froude number, Mechanical energy head elevation, Bed shear stress magnitude, Vorticity magnitude, Wind Speed, Depth of general scour, Scoured bed elevation, Sediment concentration, Sediment volumetric transport rate, Sediment volumetric transport capacity, and Sediment transport rate deficit. Available vector datasets include: Depth-averaged flow velocities, Unit flow rate, Bed shear stress, Tropical cyclone wind velocities, Sediment volumetric transport rates, and Sediment volumetric transport capacities.

Timing Tab

- Relaxation factor

Affects how fast and reliable FESWMS will find a solution. Reducing this value may help FESWMS achieve a solution but may increase runtime.

- Iterations

This is the maximum number of iterations to perform. There will be fewer iterations if the model reaches the convergence parameters before reaching this number of iterations (see parameters tab for convergence parameters).

- Starting time

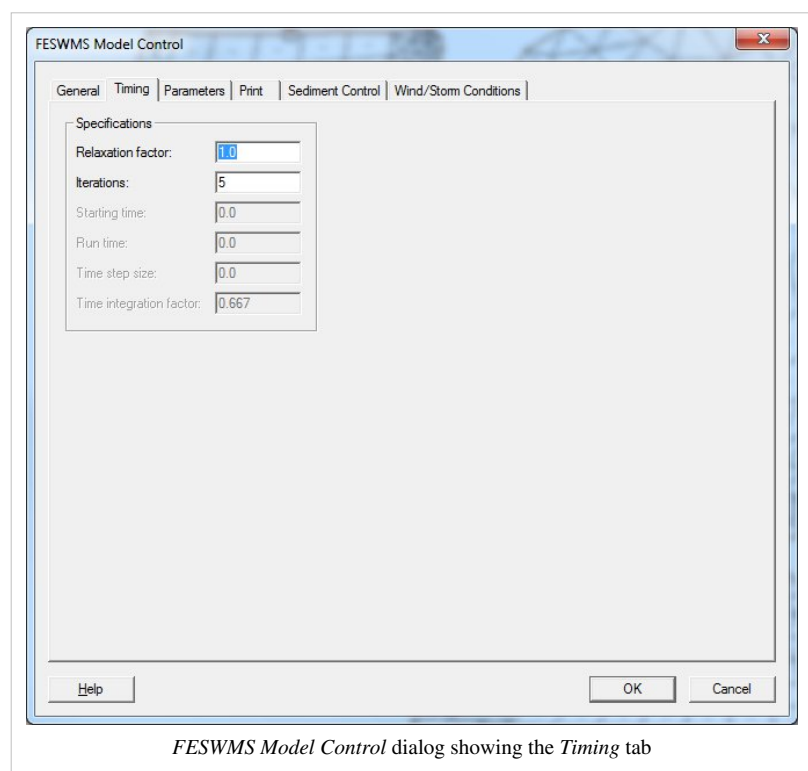
The time to start a dynamic simulation in hours.

- Run time

The length of the simulation in hours.

- Time step size

The length of time for each time step in hours. FESWMS is an implicit model so the time step size is not dependent upon the courant number.



FESWMS Model Control dialog showing the Timing tab

Parameters Tab

- General Parameters

- Water surface elevation

This is the initial water surface elevation for a cold start simulation. This number should always be larger than the elevation of the highest node in the mesh.

- Unit flow convergence

The maximum change of unit flow between iterations that will be considered a acceptable for a converged solution.

- Water depth convergence

The maximum change of water depth between iterations that will be considered acceptable for a converged solution.

- Storativity depth

Storativity allows an element to remain active as long as the water surface elevation is above the highest node value minus the storativity depth. Specifying a non-zero storativity value generally increases the stability of the model.

- Element wetting/drying

Unless this is checked, FESWMS will not allow elements to wet and dry during a simulation.

- Depth tolerance for drying

Indicates the required depth to re-wet a dry node. If this value is zero, any depth above the node will re-wet a dry element. Using a value greater than zero helps prevent wet/dry excessive wet/dry oscillations that can lead to instabilities.

- Reference Point Location

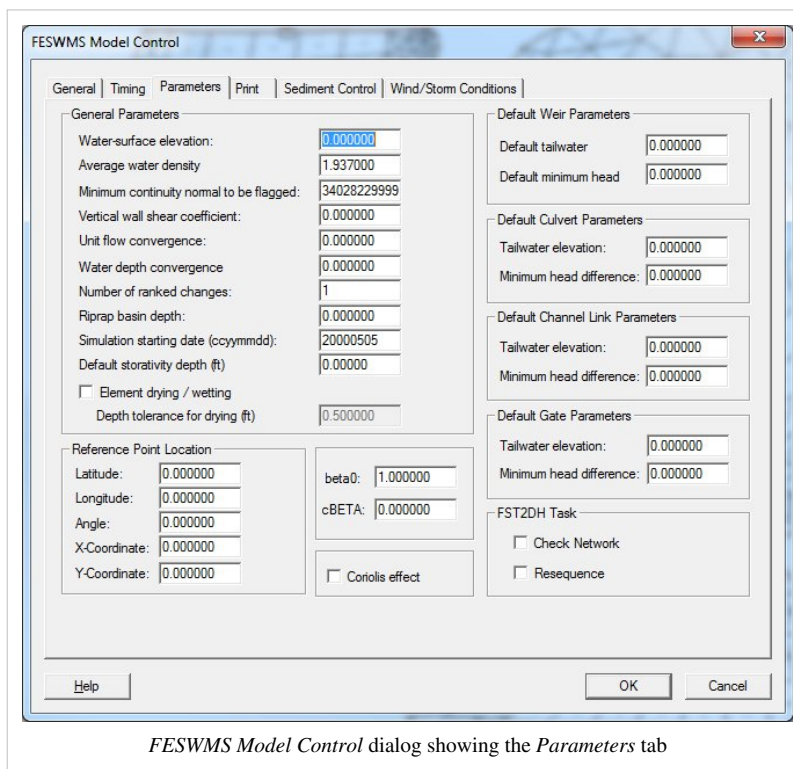
- Latitude
- Longitude
- Angle
- X-Coordinate
- Y-Coordinate

- Default Weir Parameters

- Default tailwater
- Default minimum head

- Default Culvert Parameters

- Tailwater elevation
- Minimum head difference



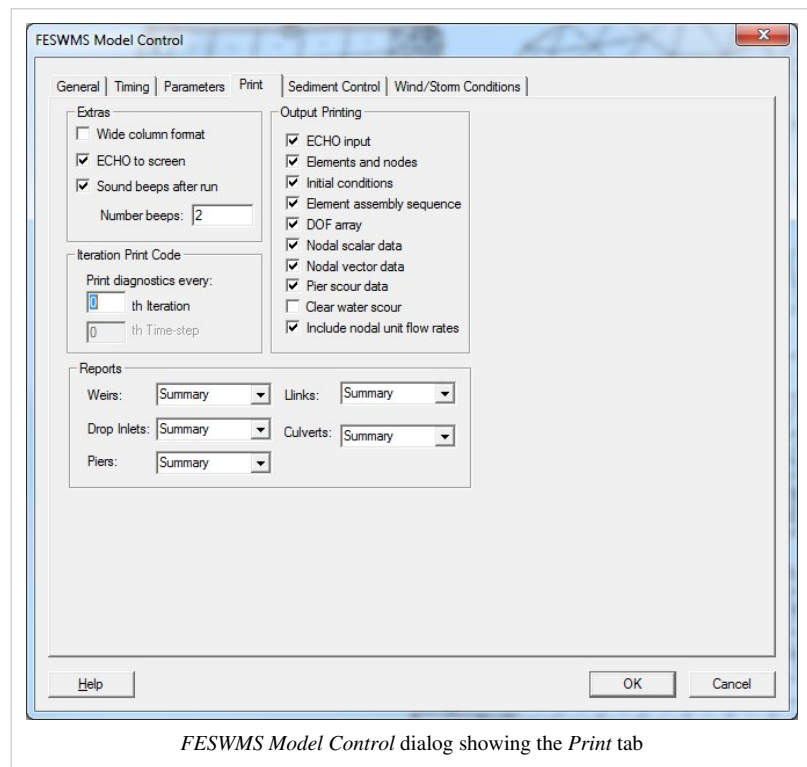
FESWMS Model Control dialog showing the Parameters tab

- Default Channel Link Parameters
 - Tailwater elevation
 - Minimum head difference
- Default Gate Parameters
 - Tailwater elevation
 - Minimum head difference
- FST2DH Task
 - Check Network
 - Resequencing

Print Tab

This tab controls the information that is written to the printed output file (*.prt).

- Extras
 - Wide column format
 - ECHO to screen
 - Sound beeps after run
 - Number beeps
- Output Print
 - ECHO input
 - Elements and nodes
 - Initial conditions
 - Element assembly sequence
 - DOF array
 - Nodal scalar data
 - Nodal vector data
 - Pier scour data
 - Clear water scour
 - Include nodal unit flow rates
- Iteration Print Code
 - Print diagnostics every:
- Reports
 - Weirs
 - Drop Inlets
 - Piers
 - Links
 - Culverts



FESWMS Model Control dialog showing the Print tab

Sediment Transport

This tab defines the parameters used for sediment transport including bed configuration. See FESWMS Sediment Control for more information.

Wind/Storm Conditions Tab

- Condition

Wind conditions include none, a directional wind, or a storm (cyclone or hurricane).

- Wind Parameters

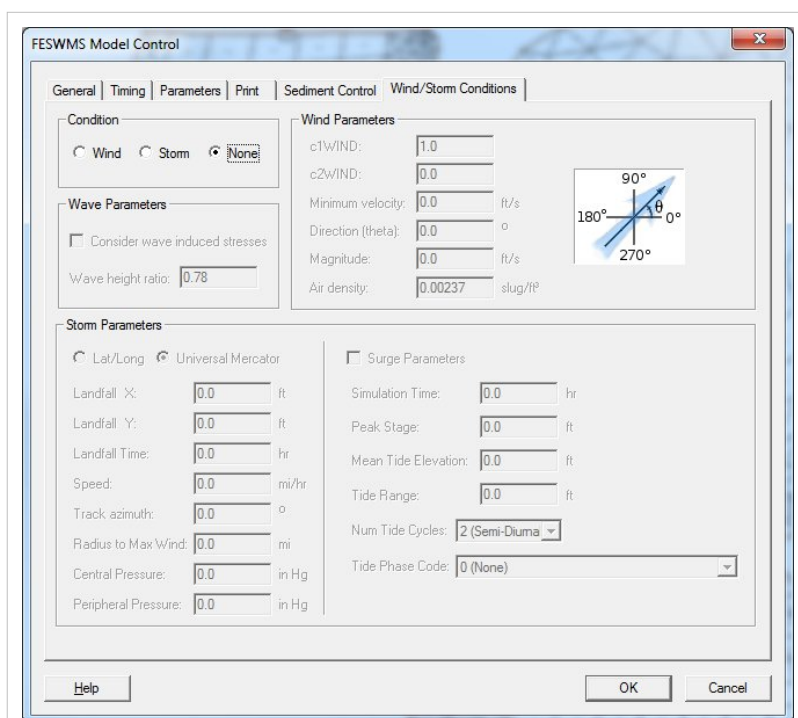
These parameters are used to define the parameters for a directional wind (the same direction magnitude for the entire model domain).

- Storm Parameters

These parameters are used to define a hurricane or cyclone moving across the model domain. The storm parameters such as direction and central pressure are constant throughout the simulation. The effect of a storm surge combined with the wind fields can be defined using the surge parameters toggle. See the FESWMS manual for a description of the input parameters.

- Wave Parameters

FESWMS has the ability to consider wave induced stresses. The wave height ratio is the breaking water wave height to depth ratio. Waves will break when the significant wave height equals the water depth times this ratio.



FESWMS Model Control showing the Wind/Storm Conditions tab

Related Topics

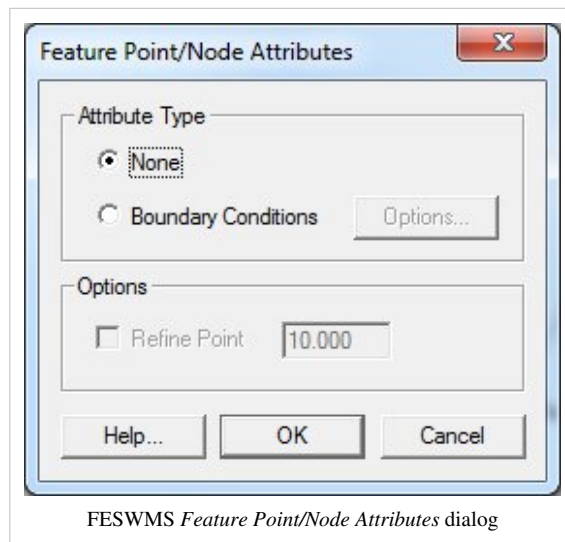
- FESWMS
- FESWMS Files
- FESWMS Graphical Interface
- FESWMS Hydraulic Structures
- FESWMS Spindown

FESWMS Point Attributes Dialog

The FESWMS *Feature Point/Node Attributes* dialog is used to set the attributes for a feature point / refine point represented by a feature point in a 2D Mesh model coverage. Attributes that can be specified for each feature point / refine point include:

Attribute Type Frame

- None
- Boundary Conditions – Options button opens the *FESWMS Nodal Boundary Conditions* dialog.
 - FESWMS BC Nodes – FESWMS allows specifying boundary conditions on a nodal basis although it is generally preferable to use boundary conditions on nodestrings. Please refer to the FESWMS manual for information on the nodal boundary condition options.



Options Frame

- Refine point (checked = on)
 - Element size – Specify the nodal spacing, or element edge length in the vicinity of the refine point. Refine points are only used if the mesh is generated using the Paving or Scalar Paving Density mesh generation methods.

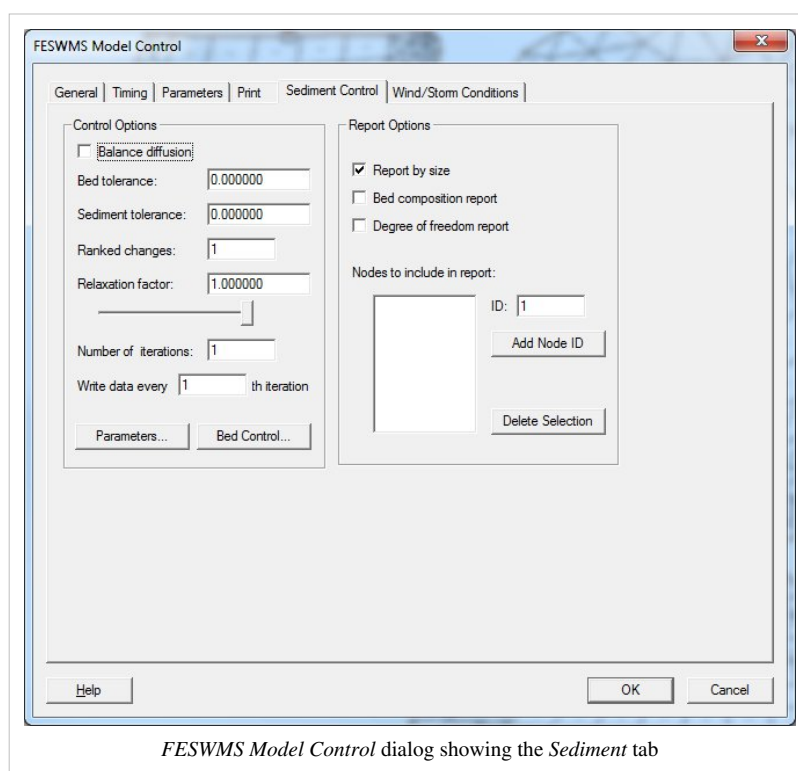
Related Topics

- Feature Objects Menu
- Mesh Generation

FESWMS Sediment Control

Overview

The FESWMS engine was modified in 2005 to include a capacity for sediment transport. In fact at this time the official name of the system changed from Flo2DH to FST2DH. The "ST" stands for "Sediment Transport". That functionality is supported in the SMS interface for the model. Shortly after the capability was added to the numerical engine, a study was performed at Brigham Young University to verify the capabilities and provide sample applications. It was discovered that the FESWMS engine does indeed provide reasonable estimates of sediment transport, including scour and deposition patterns for some applications of non-cohesive sediments. However, the study also revealed that several of the advertised capabilities do not function, and the model is prone to numerical instabilities. We advise caution and patience for modelers attempting to use this capability of the FESWMS engine.



FESWMS Model Control dialog showing the Sediment tab

Before simulating sediment transport, set up a non-transport project. Then add the sediment transport parameters and attributes. This is done in three main steps:

1. Set the model parameters
2. Define the bed conditions and sediment grain sizes
3. Define the sediment boundary conditions

Model Parameters

A sediment transport simulation requires several settings in the *General* tab of the *FESWMS Model Control* be set. These include the "Run Type", the "FST2DH Output" and the "Solution Type". Set the "Run Type" to "Sediment" or "Semi-coupled". The "Sediment" option uses an existing hydrodynamic solution. The "Semi-coupled" option instructs the engine to switch back and forth between hydrodynamic calculations and sediment calculations at each time step. Tests have shown that for stability, the "Semi-coupled" option should be used. The "Solution Type" should be set to "Dynamic". The "Sediment Transport Solutions" toggle should also be turned on.

The *Sediment Transport* tab of the *FESWMS Model Control* provides access to the other parameters related to sediment transport calculations in FESWMS. The "Report Options" section allows specification of what information will be saved to the report (*.prt) file by FESWMS. The "Control Options" section of the sediment control allows for specification of the numeric properties such as convergence criteria and maximum iterations the engine should use while solving the sediment transport quantities. This section also includes two buttons to access the sediment parameters and the bed definition options. The parameters include a choice of eight transport formulae. Each formula

requires associated variables to be specified.

Bed Conditions

FST2DH performs sediment calculations using a three layer system. The first layer is the "Active bed-layer". This is the layer where deposition and scour can occur. The scour at a single time step is limited to the thickness of the active layer. The second layer consists of material deposited on top of the native bed. If core samples have been taken and show a variation in particle distributions, a depositional-layer may be defined. The third layer defines the native bed. Total scour is limited to the depth or thickness of this layer.

The "Bed Control" button in the sediment control options allows the user to access the interface to define the global bed. This include the thickness of the three computation layers and the distribution of particle sizes in each of those layers. FST2DH supports up to 8 particle sizes. The user defines the characteristic particle size for each gradation and the percent that gradation makes of each of the three layers. If a single size gradation is specified, the three percentages would all be 100%.

The global bed definitions apply to the entire domain unless a local specification is made. This is done by selecting the nodes in the area the local definition changes and define the "Local Paramters" in the FESWMS menu.

Limitation

Tests have shown that numerical sensitivity increases as the number of particle sizes increases. Use as few as you can to represent the situation. Sensitivity also increases with long duration runs (over 1 week).

Sediment Boundary Conditions

Sediment boundary conditions must be applied at all open boundaries just as hydraulic conditions must be defined at those locations. In the case of sediment, the user specifies the amount of sediment entering the domain over that open boundary using one of the following options:

- Don't specify inflow sediment. This means clear water is entering the domain.
- Specify volume of material per time as an essential or natural boundary condition for each particle size.
- Specify concentration (ppm) as an essential or natural condition for each particle size.
- Allow FST2DH to compute the inflow sediments assuming equilibrium conditions.

Limitation

Tests have shown that only the clear water and equilibrium condition boundary conditions produce reasonable results.

Related Topics

- FESWMS

FESWMS Spindown

For cold start simulations, the initial velocities are zero and the water surface elevation is constant. This is often referred to as the "bathtub condition." Often FESWMS will not directly converge using these initial conditions.

Incremental Loading Strategy

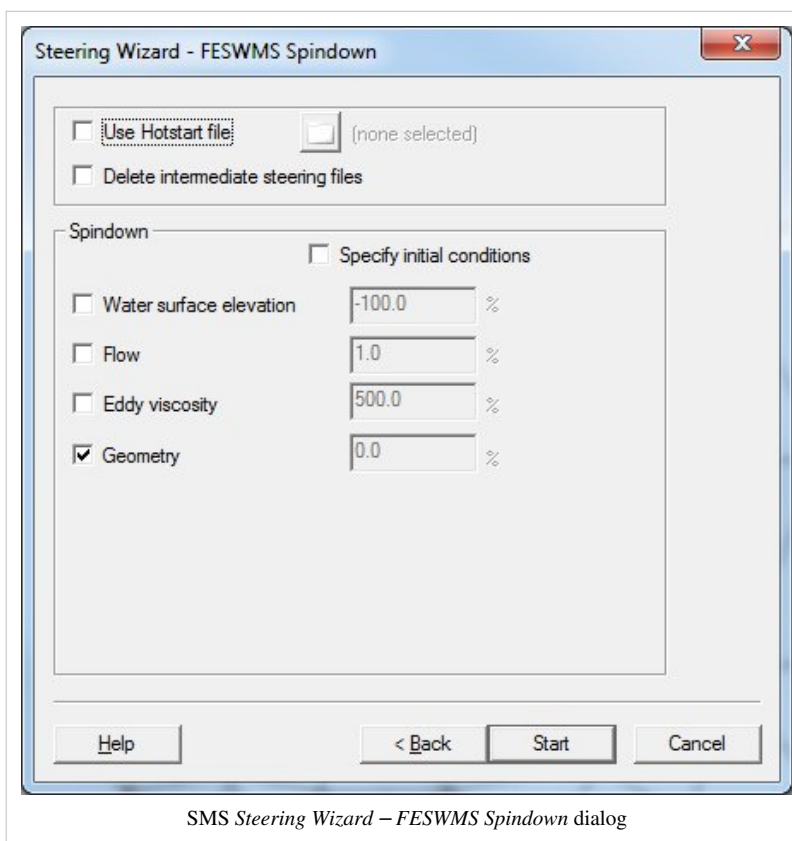
The flow equations are nonlinear and thus require an iterative solution, starting from some initial guessed value. Convergence of the iterative solution is not guaranteed. Since the desired boundary conditions may be vastly different from a cold start condition, it may be impossible to get convergence starting from this bathtub type condition. However, a solution can be obtained using a series of

"Runs" that generate solutions progressively closer to the desired answer. Intermediate boundary conditions that are closer to the final desired boundary conditions are specified to generate a set of flow conditions. These conditions do not represent the final desired flow conditions, but are closer to the final desired flow conditions than the original cold start, and can be used as initial conditions for a subsequent run. Starting the model from a previous solution is called "Hot Starting". In the incremental loading strategy, "loads" consisting of applied flow rates and water surface elevations along the boundary increment from a cold start condition to the final condition. By choosing a suitably small increment in the boundary conditions, convergence can be attained.

FESWMS Spindown refers to the process of using the Steering Module to automate the process of incremental loading. The Steering Module can vary the following:

- Boundary Conditions
 - Water surface elevation
 - Flow rate
- Model Parameters
 - Eddy viscosity
- Finite Element Network
 - Geometry (nodal elevations)

This replaces the need to perform incremental loading by hand in FESWMS.



SMS Steering Wizard – FESWMS Spindown dialog

Related Topics

- [FESWMS](#)
 - [FESWMS Files](#)
 - [FESWMS Graphical Interface](#)
 - [FESWMS Hydraulic Structures](#)
 - [FESWMS Model Control Dialog](#)
 - [Steering](#)
-

6.3. HYDRO AS-2D

HYDRO AS-2D

HYDRO AS-2D	
Model Info	
Model type	2D current, pollutant, and sediment transport simulation.
Developer	Dr.- Ing. Marinko NUJIC
Web site	HYDRO AS-2D web site ^[1]
Tutorials	General Section • Data Visualization • Mesh Editing • Observation

HYDRO AS-2D performs 2D modeling of bodies of water. The procedure integrated in HYDRO AS-2D is based on the numerical solution of the 2D current equations with Finite-volume-Discretization. In addition to 2D current simulation, HYDRO AS-2D can also simulate pollutant and sediment transport.

Functionality

Features

- Solves hundreds of thousands of elements very quickly
- Calculation accuracy guaranteed by extensive lab tests and real-world applications
- High stability, robustness and exactness for a wide spectrum of hydrodynamic conditions
- Volumetric accuracy of the tidal wave propagation on complicated topography
- Solutions showing flow speed, direction, and flood depth
- The bed changes as a result of sediment erosion and sediment deposition are modeled
- Fully coupled transport and hydrodynamic solver ensures current changes due to sediment scour/deposition are accurately simulated
- Simulate transport of up to five grain dimensions at the same time

Applications

- River flooding analysis
- Sediment erosion/depositions studies in river channels and floodplains
- Tidal wave propagation
- Pollutant dispersion in a waterway

Graphical Interface

HYDRO AS-2D uses the Generic Model Graphical Interface.

External Links

- PowerPoint presentation of a Dam Break Study ^[2]
- Hydrotec HYDRO AS-2D website (in German) ^[3]
- Hydrotec HYDRO AS-2D website (in German) ^[4]
- HYDRO AS-2D model developer website (in German) ^[1]

Related Topics

- Generic Model Interface

6.4. RIVERFLO-2D

RIVERFLO-2D

RiverFLO-2D	
Model Info	
Model type	2D current and sediment transport simulation.
Developer	Dr. Reinaldo Garcia
Web site	hydronia.net ^[1]
Tutorials	General Section • Data Visualization • Mesh Editing • Observation

RiverFLO-2D v3 is a hydrodynamic, mobile bed model for rivers and estuaries that uses a stable and powerful finite element method to compute high resolution flood hydraulics, including supercritical and subcritical flows over dry or wet river beds. Use of flexible triangular mesh allows resolving the flow field, sediment transport and bed elevation changes around key features in complex river environments. RiverFLO-2D has been applied on numerous river projects worldwide including large rivers in the United States, South America, Europe and Asia.

Functionality

Features

- Flexible triangular finite element meshes
- Mesh refinement to resolve complex river features
- Supercritical and subcritical flow
- Steady and time-dependent flow
- Mobile-bed sediment transport
- Numerically stable explicit solution scheme
- Dry-wet bed capability not requiring initially wet elements
- Parallelized code for multiple-core processor computers
- Documented input ASCII, free format files
- Comprehensive ASCII output files
- Graphical output in a variety of formats

Using the Model / Practical Notes

The RiverFLO-2D installation package includes several executable files. To connect SMS to the numeric engine:

1. Select the **Preferences** command from the *Edit* menu
2. Click on the *File Locations* tab in the *Preferences* dialog
3. In the *Model Executables* area, scroll down to the *Generic* line and click on the button on the right side. A dialog to select the executable will appear.
4. Browse to the area where SMS is installed (the default is in "Program Files\SMS 11.1 ##-bit"). From there browse into the "models\RiverFLO-2D" folder.
5. Select the executable file named "RiverFLO-2Dm3.exe". This is the engine itself that SMS will launch to analyze a simulation.
6. Click the **Open** button to select the file and the **OK** button to close the *preferences* dialog.

Graphical Interface

RiverFLO-2D uses the Generic Model Graphical Interface which allows you to construct the grid, assign model parameters and boundary conditions based on a template provided by the RiverFLO-2D developers, and view the output of an analysis.

Modeling hints for RiverFLO-2D

- In the model parameters, RiverFLO-2D supports a *Variable Time step* option. When this is on, the user must specify a multiplier. Our experience has shown that you may need to try various values for this multiplier in order to get proper mass conservation.
- The global parameter group *Graphic Output Options* causes the engine to launch a separate window during analysis to provide graphical feedback of its progress.

Reading Legacy Projects into SMS

An existing RiverFLO-2D project consists of the following files:

- *.dat – basic input data
- *.fed – geometry
- *.ifl – boundary conditions in open countours
- *.hvt – water surface elevation vs. time
- *.qvt – flow rate vs. time
- *.plt – plot options
- *.sed – sediment data
- *.tba – list of boundary nodes

Aquaveo does not currently have a utility to convert these files directly to SMS format. In order to work with data from this project in the SMS environment follow these steps:

Mesh Data

- Open the *.fed file in a file editing program such as Notepad ++. Copy and paste the data into Microsoft Excel.
- In Excel, click on the *data* menu and select **Text to Columns**. Then select **Fixed Width**. This will divide the data into individual columns. Click **next** twice, then **finish**.
- We are only concerned with the data in the first three columns so delete the data in the other columns.
- We will delete one more sets of data now. Scroll down and you will notice that the data changes. This is because it switches from mesh data to element data. Delete the element data so you only have the mesh data.

- Do a file 'Save as' and save the file as a comma delimited (*.csv) file. Click **Ok**, then click **No**. The file is now ready to be read into SMS.
- When reading the file into SMS, use the Import Wizard. In the *Import Wizard*, make sure the file is set to *Delimited* and click **next**. Change the *SMS data type* to *mesh*, then click **finish**.
- Once read into SMS, delete the undesired elements.

Nodestring Boundary Conditions

- Open the *.hvt file in a file editor and copy and paste the data into Excel. Click on the *data* menu and select **Text to Columns**. Then select *Fixed Width*. Click **next** twice, then **Finish**.
- Repeat the above step for the *.qvt file. The data in these files are now ready to be copied and pasted into SMS and assigned as boundary conditions in their respective nodestrings.

Control Data

The *.dat file does not need to be converted

- Open the *.dat file into the Argus One RiverFLO-2D engine. Note the data that appears.
- In SMS, click on the *RiverFLO-2D* menu and click *Global Parameters*. Click on the *Control Data* tab and record the data from the *.dat file in the RiverFLO-2D engine.

Remaining Files

These files are also read into the Argus One RiverFLO-2D engine, and the data from there can be manually recorded in SMS under their respective tabs in *RiverFLO-2D* | **Global parameters**.

Case Studies / Sample Problems

Hoh River Washington

This tutorial is distributed with RiverFLO-2D and includes a typical riverine application and floodplain.

Related Topics

- Generic Model Interface

6.5. SRH-2D

SRH-2D

SRH-2D	
Model Info	
Model type	Two-dimensional (2D) hydraulic, sediment, temperature, and vegetation model for river systems
Developer	Yong Lai Bureau of Reclamation
Web site	SRH-2D web site ^[1]
Tutorials	General Section • Data Visualization • Mesh Editing • Observation Models Section • SRH-2D (Pending)

About SRH-2D

SRH-2D, Sedimentation and River Hydraulics – Two-Dimensional model, is a two-dimensional (2D) hydraulic, sediment, temperature, and vegetation model for river systems under development at the Bureau of Reclamation. Different versions of SRH-2D contain different modules, as listed below (only version 2 is released and the rest will be released in the future):

- SRH-2D version 2: Modeling of flow hydraulics for stream/river systems
- SRH-2D version 3: Mobile bed sediment transport module added to v.2 (**Not yet released**)
- SRH-2D version 4: Temperature and vegetation module added to version v.2 (**Not yet released**)
- For watershed runoff modeling, SRH-W v.1.1 should be used.

About SRH-2D Version 2

SRH-2D version 2 solves the 2D dynamic wave equations, i.e., the depth-averaged St. Venant equations. Its modeling capability is comparable to some existing 2D models but SRH-2D claims a few boasting features. First, SRH-2D uses a flexible mesh that may contain arbitrarily shaped cells. In practice, the hybrid mesh of quadrilateral and triangular cells is recommended though purely quadrilateral or triangular elements may be used. A hybrid mesh may achieve the best compromise between solution accuracy and computing demand. Second, SRH-2D adopts very robust and stable numerical schemes with a seamless wetting-drying algorithm. The resultant outcome is that few tuning parameters are needed to obtain the final solution. SRH-2D was evolved from SRH-W which had the additional capability of watershed runoff modeling. Many features are improved from SRH-W.

Major Features of SRH-2D v. 2

Major SRH-2D capabilities are listed below

- 2D depth-averaged dynamic wave equations (the standard St. Venant equations) are solved with the finite-volume numerical method
- Steady state (with constant discharge) or unsteady flows (with flow hydrograph) may be simulated
- An implicit scheme is used for time integration to achieve solution robustness and efficiency
- An unstructured arbitrarily-shaped mesh is used which includes the structured quadrilateral mesh, the purely triangular mesh, or a combination of the two. Cartesian or raster mesh may also be used. In most applications, a combination of quadrilateral and triangular meshes is the best in terms of efficiency and accuracy
- All flow regimes, i.e., subcritical, transcritical, and supercritical flows, may be simulated simultaneously without the need for special treatments
- Robust and seamless wetting-drying algorithm; an
- Solved variables include water surface elevation, water depth, and depth averaged velocity. Output variables include the above, plus Froude number, bed shear stress, critical sediment diameter, and sediment transport capacity.

SRH-2D is a 2D model, and it is particularly useful for problems where 2D effects are important. Examples include flows with in-stream structures, through bends, with perched rivers, with side channel and agricultural returns, and with braided channel systems. A 2D model may also be needed if one is interested in local flow velocities, eddy patterns, flow recirculation, lateral velocity variation, and flow over banks and levees.

Features

- SRH-2D solves the 2D depth-averaged form of the diffusive wave or the dynamic wave equations. The dynamic wave equations are the standard St. Venant depth-averaged shallow water equations
- Both the diffusive wave and dynamic wave solvers use the implicit scheme to achieve solution robustness and efficiency
- Both steady or unsteady flows may be simulated
- All flow regimes, i.e., subcritical, transcritical, and supercritical flows, may be simulated simultaneously without the need of a special treatment
- Solution domain may include a combination of main channels, side channels, floodplains, and overland
- Solved variables include water surface elevation, water depth, and depth averaged velocity. Output information includes above variables, plus flow inundation, Froude number, and bed shear stress

SRH-2D is a 2D model and it is particularly useful for problems where 2D effects are important. Examples include flows with in-stream structures, through bends, with perched rivers, with multiple channel systems, and with complex floodplains. A 2D model may also be needed if one is interested in local flow velocities, eddy patterns and flow recirculation, lateral variations, flow spills over banks and levees, and flow diversion and bifurcation.

The Bureau of Reclamation does not provide technical support for SRH-2D.

Graphical Interface

SRH-2D uses a custom interface to specify boundary conditions, model parameters, model control and material parameters. The interface includes the following:

- SRH Menu
- SRH Model Control
- SRH Material Properties – SRH is able to work with a number of material zones. Materials may be created from a Materials Coverage, an SRH coverage, or directly from the mesh. Materials are added by selecting *Edit | Materials Data* when the coverage or mesh is activated.
- SRH Monitor Points Coverage
- SRH file I/O – When a user wishes to execute an SRH-2D model, he/she should export the model native files using the **Export SRH-2D Files** or **Save/Export/Launch** commands in the *SRH-2D* menu. The native files include:
 - SRHHYDRO – Contains key information about the simulation while acting as a directory to other files for SRH to use.
 - SRHGEOM – Tells SRH where each element is located and the characteristics of that element.
 - SRHMAT – Gives each element a material type.
 - SRHMONITORPTS – Tells SRH that there are monitor points to watch and where those points are located.

In the past, this model has been utilized through the Generic Model Graphical Interface. The SRH-2D version 2.0 Distribution ^[2] included SRH2D template files for both SMS 8.0 and SMS 10.0. These are no longer needed with the custom interface.

Steps to Create an SRH-2D Model

To create an SRH model, the following general steps should be followed:

1. Gather data pertinent to the project and location. This should include bathymetry data, roughness data (Manning's *n* value), coordinate system corresponding to the data, and flow data.
2. Specify a coordinate system. This is done in the *Display Projections* dialog accessed through the *Display* menu.
3. Add bathymetry data. This may come as survey data, Lidar data, or Raster DEM data to name a few.
4. Check the triangulation or raw data display. It is important to make sure that SMS is reading the data the same way that it was measured. Turning on contours will allow the user to view what SMS sees and make adjustments as needed. Contours may be turned on using the **Display Options** command. Optionally, the user may use the tools available in SMS for refinement of the data.
5. Create coverages. A simple SRH project would likely include a SRH-2D main coverage which would hold data about mesh type and bathymetry data, a materials coverage, which would map material types to region, and a monitor points coverage which will hold data relating to specific locations where site specific data will be gathered. Coverages may be created by right-clicking on the map data folder from the data tree and selecting **New Coverage**. The coverage is assigned a type upon creation. For an SRH model, select the SRH coverages as they relate to the data that will correspond to that type.
6. Outline the workspace with arcs. Here the user is defining regions of the model location that will have unique features. For example, locations of more water interaction will need more detail which equates to more nodes; locations with different roughness values will need to be separated for material type assignments. Create polygons for areas of similar characteristics. Keep in mind that SMS has a variety of tools available to adjust the arcs to meet the modeling needs of the project.
7. Build polygons.
8. Assign attributes to the polygons. Direct SMS to what materials and what mesh type is to be built over that polygon. Specify how SMS should assign elevation data by selecting the bathymetry source

9. Assign attributes to the arcs by giving the arcs boundary conditions. For interior arcs, the only option is a monitor line. For exterior arcs the user may choose from a variety of inlet conditions, whether subcritical or supercritical, as well as outlet or water surface elevation options. For no flow boundaries, the option of a wall or symmetry is available.
10. Prepare to build the mesh. Review the information given to SMS to ensure that the field data matches what is represented virtually for the region. After a review of the inputs for the model, the mesh is ready to be built. The mesh is built by converting the Map coverage to a 2D Mesh.
11. Build the mesh. If all of the attributes for the arcs and polygons were assigned correctly, the mesh will represent the same data over the elements and nodestrings. If the data is incomplete or changes need to be made, they will need to be done on the elements (for material properties) or the nodestrings (for boundary conditions). Note, that if necessary, it is possible to delete the mesh and adjust values over the polygons and arcs before regenerating the mesh.
12. Run the model. First, be sure that the project is saved. Next, from the *SRH-2D* menu, select **Export SRH**. Finally, from the *SRH-2D* menu, select **Launch SRH**.
13. Analysis of results and post processing.

External Links

- Bureau of Reclamation SRH-2D Website ^[1]

External Links – SRH-2D Version 2.0

- SRH-2D version 2.0 Theory and User Manual ^[3]
- SRH-2D version 2.0 Distribution ^[2] – Includes Software, Manual, Template file for integration into SMS interface, and Tutorials

Papers / Presentations

- SRH-2D Theory Paper ^[4]
- SRH-2D Training Presentation ^[5]
- 2005 US-China Workshop Paper ^[6]^[7]
- 2006 FISC Paper on Savage Rapids Dam Removal Project - "Comparison of Numerical Hydraulic Models Applied To The Removal of Savage Rapids Dam Near Grants Pass, Oregon" ^[8]
- FISC 2006 Short Course Presentation ^[9]
- Using Bathymetric LiDAR and a 2-D Hydraulic Model to Identify Aquatic River Habitat ^[10]
- List of journal articles using SMS by Prof. Greg Pasternack, UC Davis ^[11]

Project Reports

- Bountry J.A. and Lai, Y.G. (2006). "Numerical modeling of flow hydraulics in support of the Savage Rapids Dam removal." ^[12]
- Lai, Y.G., Holburn, E.R., and Bauer, T.R. (2006). "Analysis of sediment transport following removal of the Sandy River Delta Dam." ^[13]
- Lai, Y.G. and Bountry, J.A. (2006). "Numerical hydraulic modeling and assessment in support of Elwha Surface Diversion Project." ^[14]
- Lai, Y.G. and Bountry, J.A. (2007). "Numerical modeling study of levee setback alternatives for lower Dungeness River, Washington" ^[15]

In the News

- Wired.com article "Computer Modeling Smooths a Dam Hard Job" ^[16]
- Photos related to Wired.com article "Computer Modeling Smooths a Dam Hard Job" ^[17]

External Links – SRH-W

SRH-W Version 1.1

- SRH-W version 1.1 User Manual ^[18]
- SRH-W version 1.1 Distribution Package ^[19] – Includes Software, Manual, and Tutorials

Papers / Presentations

- 2006 FISC Watershed Modeling Paper - Watershed Simulation with an Enhanced Distributed Model ^[20]^[21]

Related Topics

- Generic Model Interface
-

6.6. Steering

Steering

The steering tool has been added to facilitate the process of launching models multiple times. To launch the steering tool, choose the menu command *Data | Steering Module*.

The steering tool can be used for single model (spin down) simulations of RMA2 and FESWMS.

The steering tool can also be used to facilitate the transfer of data from wave models to circulation models and back. Currently, CMS-Flow / CMS-Wave links is supported.

Related Topics

- SMS Models
- RMA2 Spindown
- FESWMS Spindown
- CMS-Flow / CMS-Wave Steering

NOTE: CMS-Flow (formerly known as M2D) and CMS-Wave (formerly known as WABED) are components of the Coastal Modeling System (CMS).



External Links

- Jun 2002 ERDC/CHL CHETN-IV-41 SMS Steering Module for Coupling Waves and Currents, 1: ADCIRC and STWAVE [8]
 - Please see this forum post ^[9] for an explanation of ADCIRC and STWAVE steering
- Jun 2002 ERDC/CHL CHETN-IV-42 Coupling of Regional and Local Circulation Models ADCIRC and M2D (now know as CMS-Flow) [1]
- Dec 2003 ERDC/CHL CHETN-IV-60 SMS Steering Module for Coupling Waves and Currents, 2: M2D (now know as CMS-Flow) and STWAVE [9]

6.7. TABS-MD (Multi-Dimensional) Numerical Modeling System - RMA2/RMA4

TABS

The TABS-MD (Multi-Dimensional) Numerical Modeling System was one of the first widely used collection of programs designed for studying multi-dimensional hydrodynamics in rivers, reservoirs, bays and estuaries. The hydrodynamic engine for the system is the RMA2 engine. RMA2 and RMA4 were written by Resource Management Associates, Lafayette, California, and modified by WES. SED2D was written jointly by Resource Management Associates and WES.

Models

GFGEN

The pre-processor for the TABS software programs. This utility converts ASCII geometry into binary format and does data checking along the way. SMS will launch this utility as needed before running other components of the system. See GFGEN for more information.

RMA2

A one-dimensional/two-dimensional numerical model for depth-averaged flow and water levels. RMA2 is a two dimensional depth averaged finite element hydrodynamic numerical model. It computes water surface elevations and horizontal velocity components for subcritical, free-surface flow in two dimensional flow fields. RMA2 computes a finite element solution of the Reynolds form of the Navier-Stokes equations for turbulent flows. Friction is calculated with the Manning's or Chezy equation, and eddy viscosity coefficients are used to define turbulence characteristics. Both steady and unsteady state (dynamic) problems can be analyzed.

It should be noted that the commercially available version of the model does not include all functionality included in the ERDC documentation.

See RMA2 for more information.

RMA4

A one-dimensional/two-dimensional numerical model for depth-averaged transport. This program uses a provided hydrodynamic solution (either node by node or in RMA2 format) to compute transport of a constituent in solution. It is assumed that the depth concentration distribution is uniform. While model documentation claims up to six constituents can be considered simultaneously, practical application has shown the only application of multiple constituents to be DO/BOD. Either conservative or non-conservative diffusion is computed. See RMA4 for more information.

SED2D

Formerly STUDH, a two-dimensional numerical model for depth-averaged transport of cohesive or a representative grain size of noncohesive sediments and their deposition, erosion, and formation of bed deposits. The interface for this model has been removed from version SMS 10.0. Boundary condition files can still be constructed/edited in text editors and SMS can still read solution data.

RMA10

A one-dimensional/two-dimensional/three-dimensional hydrodynamic numerical model. Not available for public use at this time.

Using the Model / Practical Notes

- The TABS models are built to expire after a set date. You can download the latest version from the Software Updates section of the Aquaveo website ^[1].

Related Links

- SMS Models page

External Links

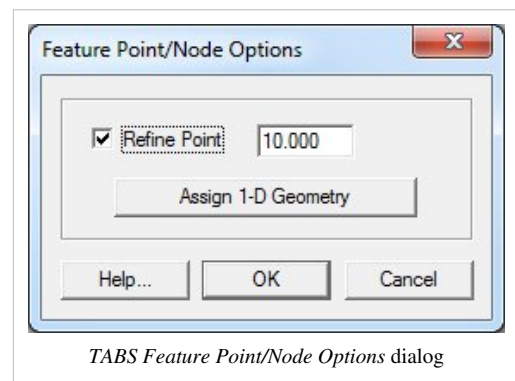
- CHL TABS Numerical Modeling website [2]
- CHL RMA2 Frequently Asked Questions [3]

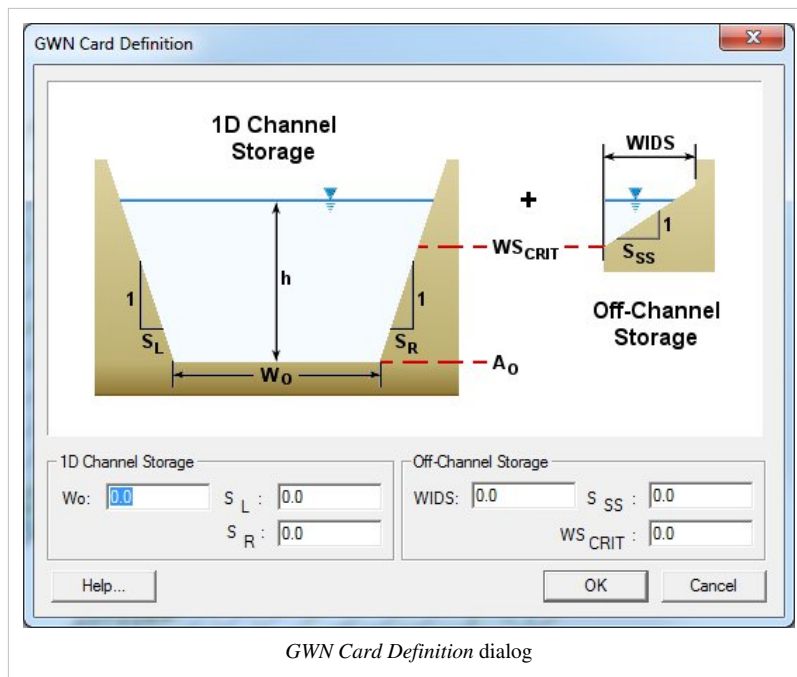
TABS Attribute Dialog

TABS Point Attributes Dialog

The *TABS Feature Point/Node Options* dialog is used to set the attributes for a feature point / refine point represented by a feature point in a 2D Mesh model coverage. Attributes that can be specified for each feature point / refine point include:

- Refine point (checked = on)
 - Element Size – Specify the nodal spacing, or element edge length in the vicinity of the refine point. Refine points are only used if the mesh is generated using the Paving or Scalar Paving Density mesh generation methods.
- Assign 1-D Geometry – Opens the *GWN Card Definition* dialog





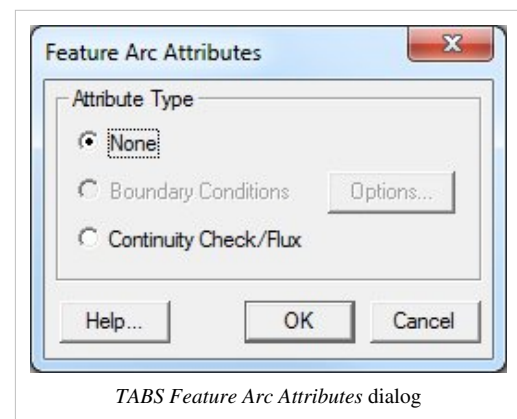
TABS Arc Attributes Dialog

The *TABS Feature Arc Attributes* dialog is used to set the attributes for feature arcs. Attributes that can be specified for each feature arc include:

- Arc Type
 - **None**
 - **Boundary Conditions** – **Options** button opens the RMA2 Nodestring Boundary Conditions dialog
 - **Continuity Check/Flux**

Related Topics

- Feature Objects Menu
- Mesh Generation



Total Flow Nodestring

WARNING: Article could not be rendered - ouputting plain text.

Potential causes of the problem are: (a) a bug in the pdf-writer software (b) problematic Mediawiki markup (c) table is too wide

Two hydrodynamic models supported by SMS, RMA2 and FST2DH, support a concept called the Total Flow Nodestring. These models print out a table of continuity or flux checks across specified nodestrings. Such checks are used to determine if there is mass loss through the model, to determine the amount of flow in various sections of a braided stream, or other similar purpose. In order to participate in the continuity table, the nodestring must either be assigned as a boundary condition or assigned as a flux (FST2DH) or geometric continuity (GC) (RMA2) string. The nodestring assigned as total flow will always be reported first in the table, and will always have 100% of the flowrate across it. The flux across all other cross sections will be reported in the continuity table as a percentage of this flowrate, whether it be more, less, or equal to 100%. Note that nodestrings should all be created from right to left, while looking downstream. When a nodestring is selected, arrows appear on its ends which show the assumed downstream direction. The figures below show advisable locations of the total flow nodestring. Related TopicsMesh Module

6.7.a. GFGEN

GFGEN

GFGEN (Geometry File **GE**Nerator) converts ASCII geometry into binary format and does data checking along the way.

Using the Model / Practical Notes

- The TABS models are built to expire after a set date. You can download the latest version from the Software Updates section of the Aquaveo website ^[1].
- GFGEN Executable Known Issues

External Links

- Users Guide to GFGEN - Version 4.35 [1]

Related Topics

- SMS Models Page
 - TABS Models
 - RMA2
 - RMA4
-

GFGEN Executable Known Issues

Unfortunately there are some known issues and likely some unknown issues as well related to the GFGEN executable. This page is intended to help make users aware of known issues and how to work around them.

GFGEN (TABS) is developed by Resource Management Associates, Lafayette, California, and modified by WES^[1]. Aquaveo does not have access to the GFGEN source code and cannot make changes to fix the issues found in the GFGEN numerical model. Aquaveo can fix problems in the SMS interface, used for pre and post processing of GFGEN files.

This list may not contain all know issues, so please feel free to add to it.

Node Location Truncation

GFGEN does not correctly calculate the midside node location if the x or y coordinate of a node are large. You may receive an error similar to the following:

```
* Node      50 Violates MIDDLE THIRD RULE *****
The node lies      0.0831 of the way from node      1 to node      49

=== Problem inside COEFS routine. Do 180.
Element= 535 Gauss pt= 1 out of NGP= 16
Divide by DETJ= -0.0229
J11-J22-J12-J21= 1.1526 0.38006 0.74030 0.62257
stop in coefs
```

The current work around is to translate the mesh so the x and y coordinates are smaller values.

See the following forum posts:

- RMA2 – same mesh, same BC, different results?, Results differ depending on nodal coordinates^[2]
- Problem inside COEFS routine^[3]

Element Limit

GFGEN can only handle up to 100,000 elements.

Related Topics

- GFGEN

6.7.b. RMA2 - Resource Management Associates

RMA2

RMA2	
Model Info	
Model type	Two-dimensional depth averaged finite element hydrodynamic numerical model. Computes water surface elevations and horizontal velocity components for subcritical, free-surface flow in two dimensional flow fields.
Developer	Resource Management Associates United States Army Corps of Engineers (USACE)
Web site	RMA2 web site ^[1]
Tutorials	General Section • Mesh Editing • Observation • Overview • Sensitivity Models Section • RMA2 • RMA2 Steering

RMA2 is a 1D/2D hydrodynamic model using the finite element method. RMA2 was written by Ian King and is maintained by the Army Corp of Engineers Engineering Resource Development Center (ERDC) ^[2]. RMA2 has been applied to multi-dimensional problems since the mid 1970s. As such, it was one of the first widely used multi-dimensional hydrodynamics engine applied to riverine and estuarine applications. The current interface to RMA2 in the Surface-water Modeling System originated as a program named FastTABS.

Functionality

RMA2 has these capabilities:

- Compute water levels and velocities for a 2D mesh structure
- 1D elements (trapezoidal shape)
- 1D flow control structures
- Allow elements to wet and dry
- 2D structures are permitted in version 4.50 or higher (US Army Corps of Engineers employees only)
- Account for effects of the earth's rotation ([SMS:RMA2_Material_Properties#Coriolis | Coriolis effect])
- Accepts a wide variety of boundary conditions including:
 - Discharge by node/element/ or line
 - Tidal radiation boundary conditions by line
 - Discharge as a function of elevation by line
 - Water surface elevation along a line
 - Apply wind stress either

- Uniformly over the model domain; constant or time-varying
- As a storm; front or tropical cyclonic event (time-varying)
- Read an STWAVE radiation stress file to incorporate wave induced currents
- Identify errors in the computational mesh specification
- Accept either English or standard SI units
- Restart (Hotstart) the simulation from a prior RMA2 run and continue
- Account for Marsh Porosity wetting and drying (wetlands)
- Employ either direct or automatic dynamic assignment of Manning's n-value by water depth
- Employ user selectable manual or automatically assigned turbulent exchange coefficients
- Compute flow across continuity check lines.
- Revisions within a time step (both coefficients and/or boundary conditions)

Using the Model / Practical Notes

- RMA2 Spindown Steering (Incremental Loading)
- RMA2 2D Control structures are only available to employees of the US Army Corps of Engineers.
- The TABS models are built to expire after a set date. You can download the latest version from the Software Updates section of the Aquaveo website ^[1].
- According to the RMA2 model documentation, "Because of lack of experience in using storms in a simulation, including storms in simulations remains experimental."
- Boundary condition time series curves are resampled by the model based on the time step. If you change the time step, you may want to re-enter the time series curve since SMS will automatically resample the curve and resampling the curve multiple times may result in a loss of important information.

Graphical Interface

SMS provides a graphical interface that is designed to allow users to visualize the projects they are creating, easily modify project parameters, and view the solutions produced by the RMA2 model. See RMA2 Graphical Interface for more information.

The RMA2 Graphical Interface contains tools to create and edit an RMA2 simulation. The simulation consists of a geometric definition of the model domain (the mesh) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the 2D Mesh Module and setting the current model to RMA2. If a mesh has already been created for a RMA2 simulation or an existing simulation read, the mesh object will exist in the Project Explorer and selecting that object will make the 2D Mesh module active and set the model to RMA2. See the Mesh Module documentation for guidance on building and editing meshes as well as visualizing mesh results.

The interface consists of the 2D Mesh Module Menus and tools augmented by the RMA2 Menu. See RMA2 Graphical Interface for more information.

Saving RMA2

When you do a *File | Save As...* the following files get saved in the *.sms

- *.mat referenced to new save location
- *.sim referenced to new save location

Related Topics

- SMS Models Page
- TABS Models
- RMA4
- Nodal Transition Dialog
- Roughness Options Dialog
- Rainfall Values Dialog

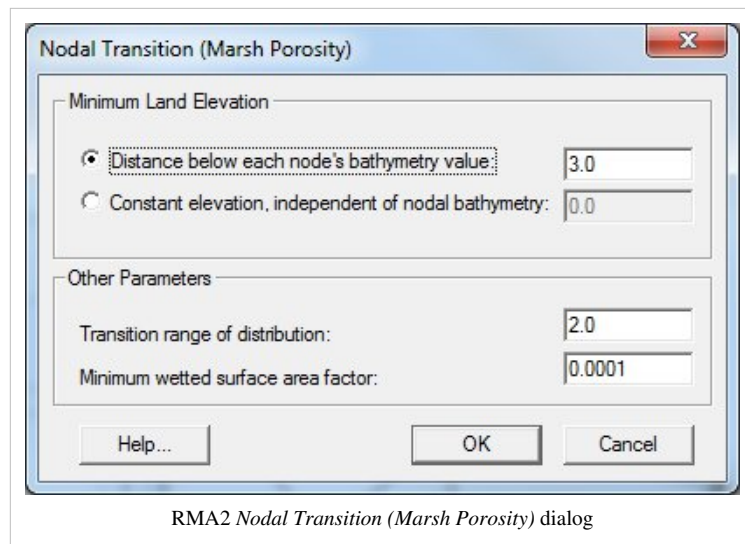
External Links

- CHL RMA2 Website [3]
- CHL RMA2 Frequently Asked Questions [3]
- Users Guide To RMA2 WES Version 4.5 [4]
- Users Guide to RMA2 Version 4.35 [5]
- MERGAVE – Utility for Merging RMA2 Solution Files [6]

Nodal Transition (Marsh Porosity) Dialog

The *RMA2 Nodal Transition (Marsh Porosity)* dialog is used to define the RD Card: Automatic Roughness Coefficient Assignment by Depth. See the section "Wetting and Drying", "Elemental Elimination", "Marsh Porosity", and DA and DM Card descriptions in the RMA2 model documentation for more information. The following parameters can be set in the *Nodal Transition (Marsh Porosity)* dialog:

- Minimum Land Elevation
 - Distance below each node's bathymetry value
 - Constant elevation, independent of nodal bathymetry
- Transition range of distribution
- Minimum wetted surface area factor



Related Topics

- Marsh Porosity
- RMA2 Model Control dialog

Rainfall Values Dialog

The *RMA2 Rainfall Values* dialog is used to assign rainfall or evaporation. Rainfall or evaporation is applied in inches/hour or cm/hour if using metric units. Positive values represent rainfall, while negative values represent evaporation. Rainfall and evaporation can occur at any time during the simulation. See the "Adding Rainfall And Evaporation" section and RA Card description in the RMA2 model documentation for more information.

Related Topics

- RMA2 Model Control dialog



RMA2 Rainfall Values dialog

RMA2 1D Control Structure

Overview

In RMA2, a one-dimensional Control Structure element is a single point which contains two nodes and has an IMAT value ≥ 904 . The order of the node numbering at a Control Structure element should be that the side with higher elevation comes first, then the side with the lower elevation. This is generally the “upstream” side of the structure followed by the “downstream” side. This order sets the “orientation” of the flow through the structure.

Since flow through and over some structures does not fit the 2D assumptions, some structures can be represented as 1D components in the 2D mesh. These include weirs, culverts, drop inlets and some gates.

There are strict rules that must be followed in order to get convergence with 1D elements in RMA2. These rules include:

- Transition elements must be connected to the free end of a 2D element. The element should not be connected to other 2D elements on the sides.
- The transition element must be perpendicular to the free end of the 2D element.
- The bottom elevation of the free end should be flat.
- Control structure elements cannot be constructed on transition elements.

Culvert

A culvert is a conduit used to enclose a flowing body of water. It may be used to allow water to pass underneath a road, railway, or embankment, for example. Culverts can be made of many different materials; steel, polyvinyl chloride (PVC) and concrete are the most common.

Culvert flow rate depends primarily on headwater depth. Other factors that affect flow rate are tailwater elevation and properties of the culvert such as shape, cross sectional area, inlet geometry, length, slope, and roughness.

Weir

A weir is a small overflow-type dam commonly used to raise the level of a river or stream. Weirs have traditionally been used to create mill ponds in such places. Water flows over the top of a weir, although some weirs have sluice gates which release water at a level below the top of the weir. The crest of an overflow spillway on a large dam is often called a weir.

Weirs also give hydrologists and engineers a simple method of measuring the rate of fluid flow in small to medium sized streams. Since the geometry of the top of the weir is known, and all water flows over the weir, the depth of water flowing over the weir will be an indication of the flow. There are different types of weir. It may be a simple metal plate with a V notch cut into it or it may be a concrete and steel structure across the bed of a river. A v-notch weir will give a more accurate indication of low flow rates.

Related Topics

- RMA2
-

RMA2 Boundary Conditions

Assigning Boundary Conditions

Boundary conditions are generally defined on nodestrings but may also be defined on nodes. The default boundary condition is a closed boundary (no flow).

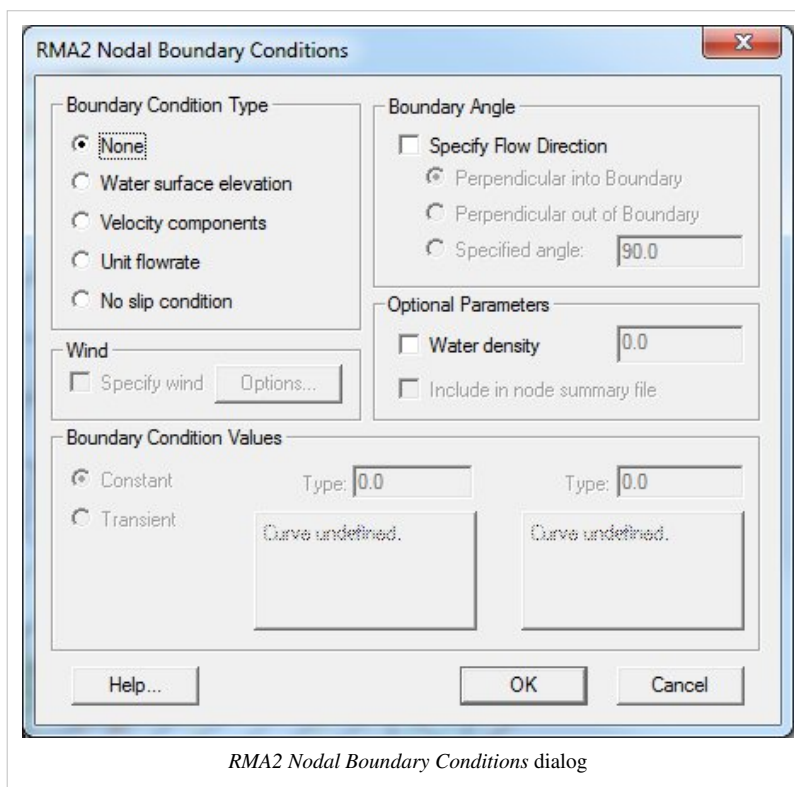
Nodal Boundary Conditions

Boundary Condition Type

- None

No Boundary Condition is assigned to this Nodestring
- Water Surface Elevation

A Water surface elevation BC is assigned to this Nodestring. This boundary condition is normally located at the tailwater location. However, RMA2 does sometime edit the water depth at the exit boundary to satisfy the finite element equations.
- Velocity Components
- Unit Flowrate
- No Slip Condition



RMA2 Nodal Boundary Conditions dialog

Boundary Angle

- Specify Flow Direction
 - Perpendicular into Boundary
 - Perpendicular out of Boundary
 - Specified angle

Optional Parameters

- Water Density
- Include in node summary file

Boundary Condition Value

- Constant
- Transient

Nodestring Boundary Conditions

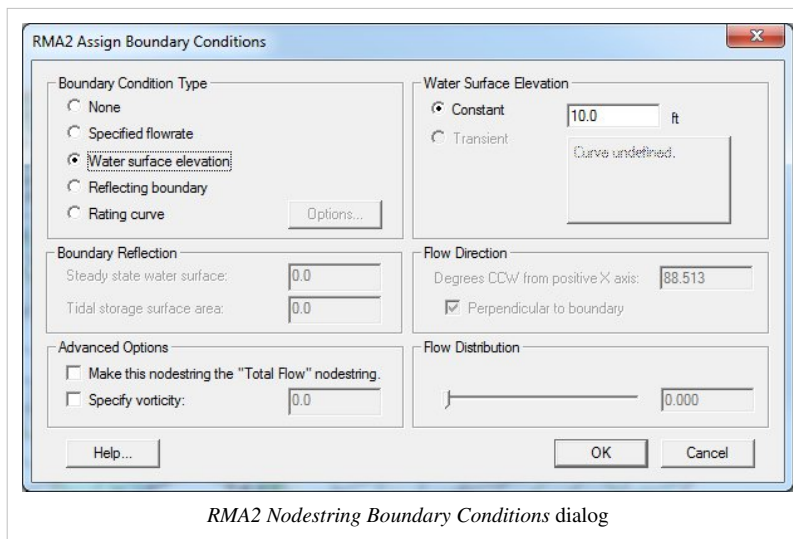
Boundary Condition Type

- None

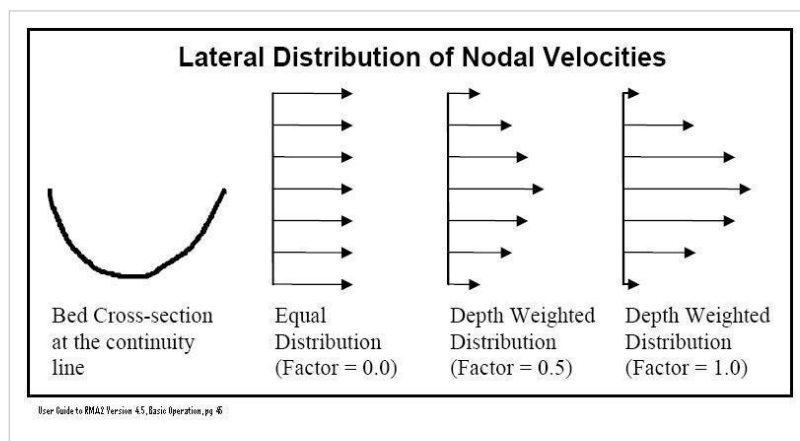
No Boundary Condition is assigned to this Nodestring

- Specified Flow Rate

A flow rate BC is assigned to this nodestring. It is the most popular way to provide a total flow across a particular continuity line. The total flow (cubic units per second), flow angle (degrees from positive x-axis) and distribution factor can be specified.



RMA2 Nodestring Boundary Conditions dialog



- Water Surface Elevation

A Water surface elevation BC is assigned to this Nodestring. This boundary condition is normally located at the tailwater location. (However RMA2 does sometime edit the water depth at the exit boundary to satisfy the finite element equations)

- Reflecting Boundary

Considered an advanced feature — for more information see the "Boundary Permeability (Reflection/Absorption)" section on page 99 of the Users Guide to RMA2 version 4.5

- Rating Curve

A Rating curve can also be defined by selecting the **Options...** button

Flowrate

- Constant
- Transient

Boundary Refelction

- Steady state water surface
- Tidal storage surface area

Flow Direction

- Degree CCW from positive X axis
- Perpendicular to boundary

Advanced Options

- Make this nodestring the "Total Flow" nodestring
- Specify vorticity

Related Topics

RMA2

RMA2 Files

Input and Output files for RMA2.

Input Files

- ASCII Geometry (*.geo) → GFGEN → Binary Geometry (*.bin) and printed output file (*.ot1)
- Run Control Input File (*.bc)
- Hotstart File (*.hot) – optional
- Alternate BC File (*.abc)
- Run File (*.run) – _gr.run (GFGEN) and _rm.run (RMA2)

Output Files

- Binary Solution (*.sol)
- Printed Output File (*.ot2)
- Node summary file (*.sum)
- Hotstart Output (*.hot) – optional

Related Topics

- RMA2
 - RMA2 Hydraulic Structures
 - RMA2 Model Control Dialog
 - RMA2 Spindown
-

RMA2 Graphical Interface

The RMA2 Graphical Interface includes tools to assist with creating, editing, and debugging a RMA2 model. The RMA2 interface exists in the Mesh Module. Users can visualize and create projects, easily modify project parameters, and view the solutions produced by the RMA2 model. Below are links describing the SMS interface that interacts with the RMA2 model.

Model Control

The *RMA2 Model Control* Dialog is used to setup the options that apply to the simulation as a whole. These options include time controls (steady state/dynamic), run types, output options, global parameters, print options and other global settings.

Boundary Conditions

All numeric models require boundary condition data. Boundary conditions in RMA2 include flows in/out of the model domain or known water surface elevations. In RMA2 boundary conditions are generally defined on nodestrings but may also be defined on nodes. The default boundary condition is a closed boundary (no flow). See Nodestring Boundary Conditions and Nodal Boundary Conditions for more information.

Material Properties

Each element is assigned a material type. Material properties describe the hydraulic characteristics of each material type.

RMA2 Hydraulic Structures

RMA2 includes support for several different types of structures including weirs, culverts, drop inlets, gates and piers. See RMA2 Hydraulic Structures for more information.

Running the Model

The RMA2 Files are written automatically with the SMS project file or can be saved separately using the *File | Save RMA2* or *File | Save As* menu commands. See RMA2 Files for more information on the files used for the RMA2 run.

RMA2 can be launched from SMS using the *RMA2 | Run FSTD2H* menu command. A check of some of the common problems called the *Model Checker* is done each time the model is launched, or by selecting the *RMA2 | Model Check* menu command.

RMA2 Menu

See RMA2 Menu for more information.

Related Topics

- SMS Models Page
- TABS Models
- RMA2
- RMA4
- Total Flow Nodestring

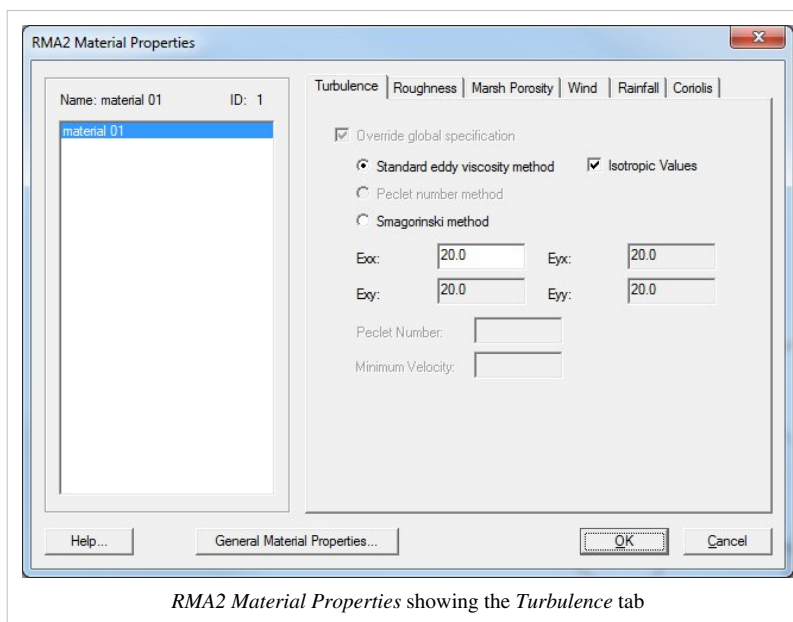
RMA2 Material Properties

For an in depth discussion of the RMA2 material properties, consult the RMA2 Users Guide. [4] [5]

Turbulence

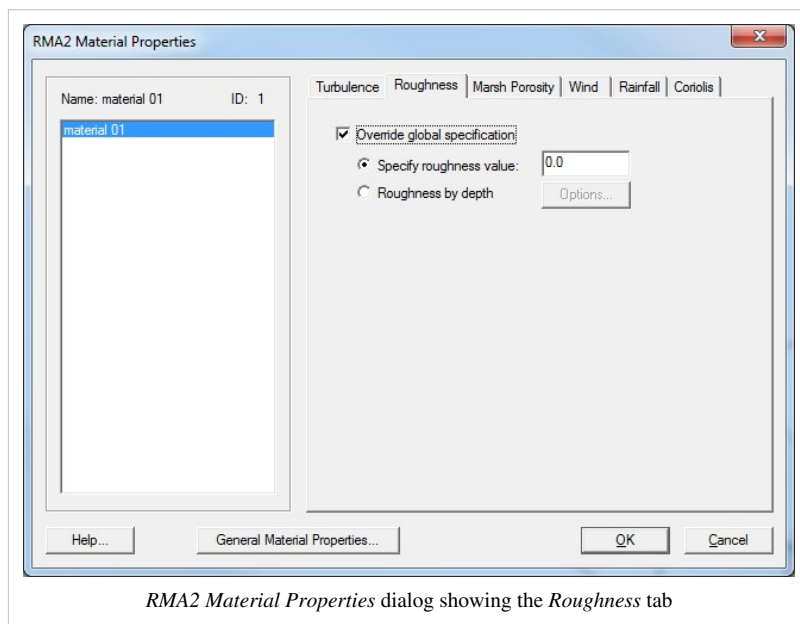
The global turbulence values (specified in the RMA2 model control) can be overridden for each material.

- Standard eddy viscosity method
 - E_{xx} – The x-momentum of turbulent exchange in the x-direction (lb-sec/ft² or Pascal-sec for SI-units)
 - E_{xy} – The x-momentum of turbulent exchange in the y-direction (lb-sec/ft² or Pascal-sec for SI-units)
 - E_{yx} – The y-momentum of turbulent exchange in the x-direction (lb-sec/ft² or Pascal-sec for SI-units)
 - E_{yy} – The y-momentum of turbulent exchange in the y-direction (lb-sec/ft² or Pascal-sec for SI-units)
 - Note: 1 lb-sec/ft² = 47.879 Pascal-sec
- Smagorinski method
 - TBFAC – Smagorinsky coefficient for turbulent exchange. A negative value applies the default coefficient (0.05).
 - TBFAC – Not applied in RMA2. Smagorinsky coefficient for diffusion (Vorticity). A negative value applies the default coefficient (0.05).
 - TBMINF – Smagorinsky minimum turbulent exchange factor. A negative value applies the default coefficient (1.0).
 - TBMINF – Not applied in RMA2. Smagorinsky minimum diffusion (Vorticity) factor. A negative value applies the default coefficient (1.0).



Roughness

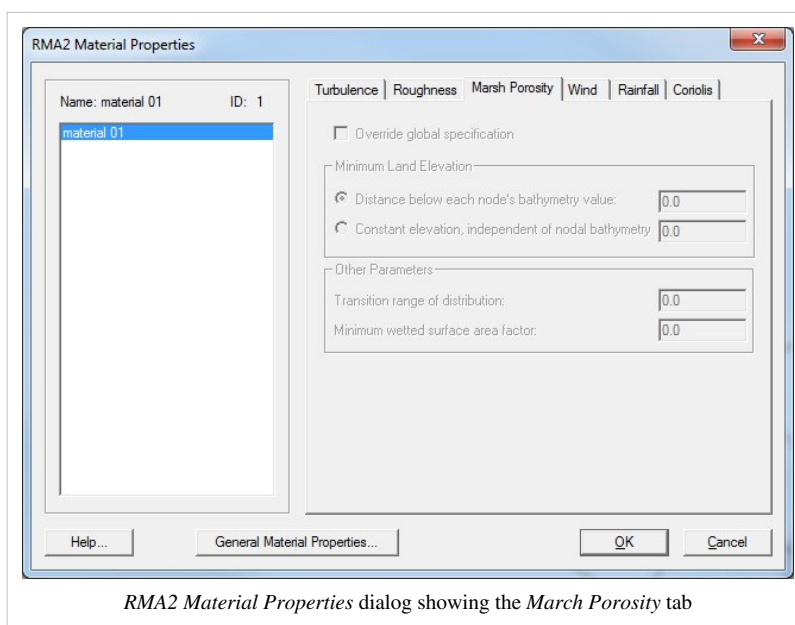
The global roughness value (specified in the *RMA2 model control*) can be overridden for each material. The material roughness can be specified as a single value, or depth dependent.



RMA2 Material Properties dialog showing the Roughness tab

Marsh porosity

The global marsh porosity values (specified in the *RMA2 model control*) can be overridden for each material. Since wetting and drying cause abrupt changes to the shape of the finite element mesh, numerical stability problems can result when wetting and drying occurs. In an attempt to address the fact that wetting and drying is a continuous process, while the mesh is a discrete data set, marsh porosity was implemented for RMA2. When marsh porosity is used, elements are considered "dry" only if ALL nodes on the element are dry.



RMA2 Material Properties dialog showing the Marsh Porosity tab

Abrupt changes in the marsh porosity parameters between adjacent node can lead to divergence. The results from a simulation using marsh porosity can be difficult to interpret due to the fact that partly dry marsh elements appear completely wet even though the water surface elevation is lower than part of the element.

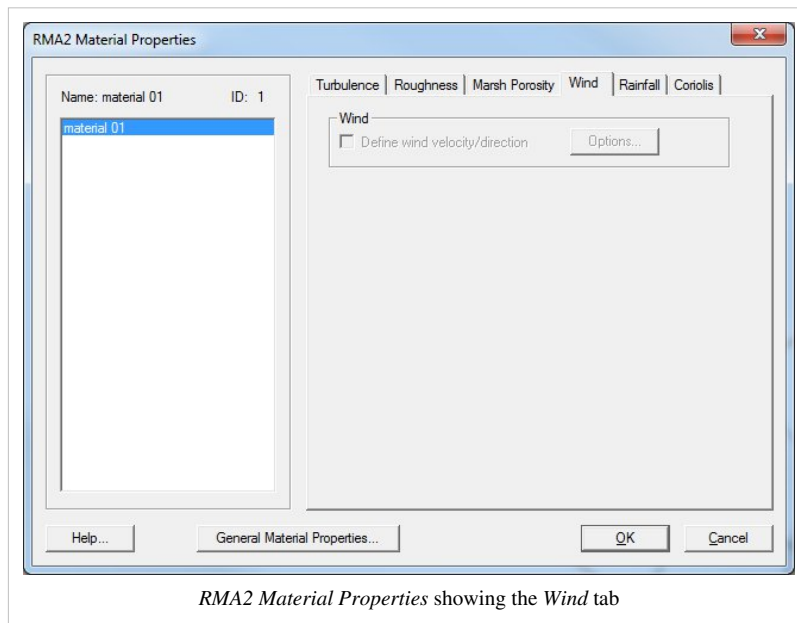
Marsh porosity is only available when running a dynamic RMA2 simulation.

Wind

According to the RMA2 model documentation, "Because of lack of experience in using storms in a simulation, this feature remains experimental."

Wind shear stress at the water surface is caused by friction between the moving air and water. The shear stress coefficient is a function of wind speed. The global wind values (specified in the RMA2 model control) can be overridden for each material.

Wind parameters are only available when running a dynamic RMA2 simulation.

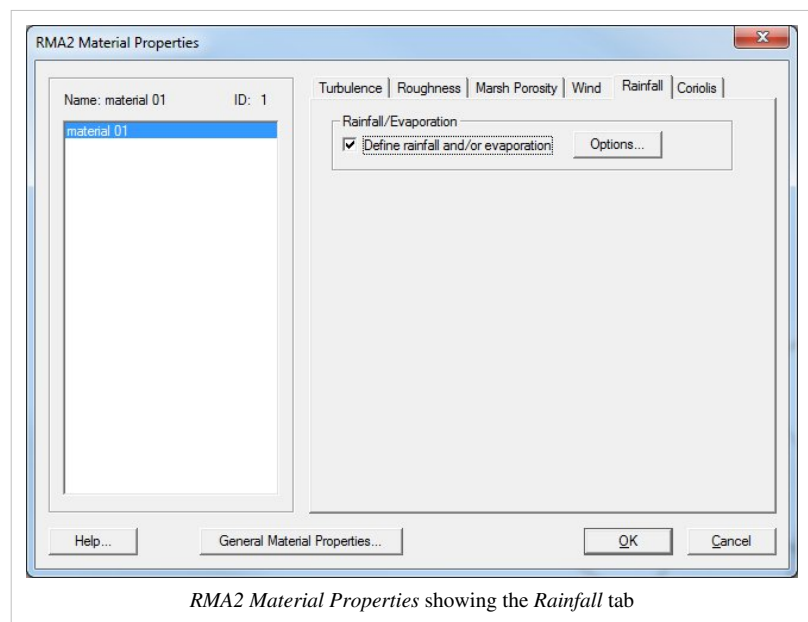


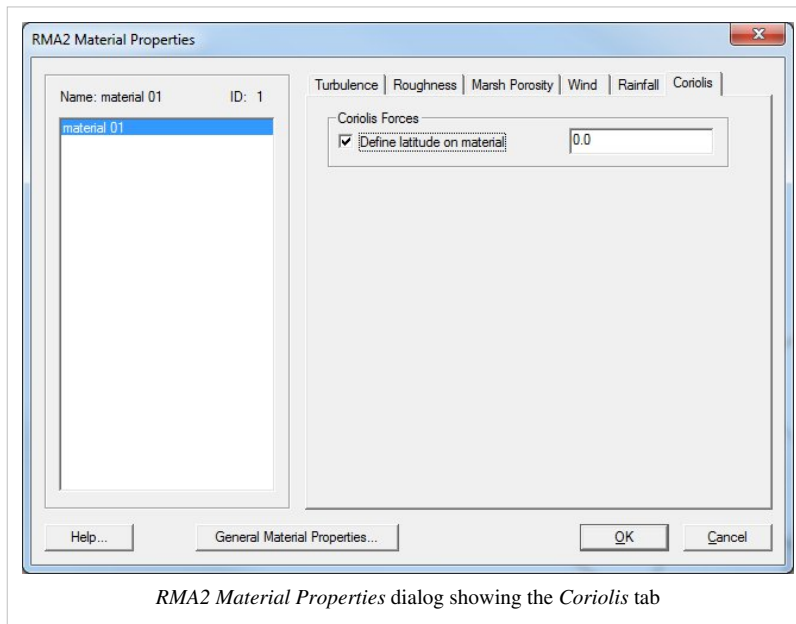
Rainfall

The global rainfall and evaporation values (specified in the *RMA2 model control*) can be overridden for each material.

Coriolis

The global coriolis^[1] values (specified in the *RMA2 model control*) can be overridden for each material.





Related Topics

RMA2

RMA2 Menu

The following menu commands are available in the RMA2 Menu:

Boundary Conditions

- Assign BC – Specify boundary conditions for the selected node(s) or nodestring(s)
- Delete BC – Deletes the boundary condition assigned to the selected node(s) or nodestring(s)
- Extract Nodal BC – Extracts nodal boundary conditions (dynamic simulations only)

Hydraulic Structures

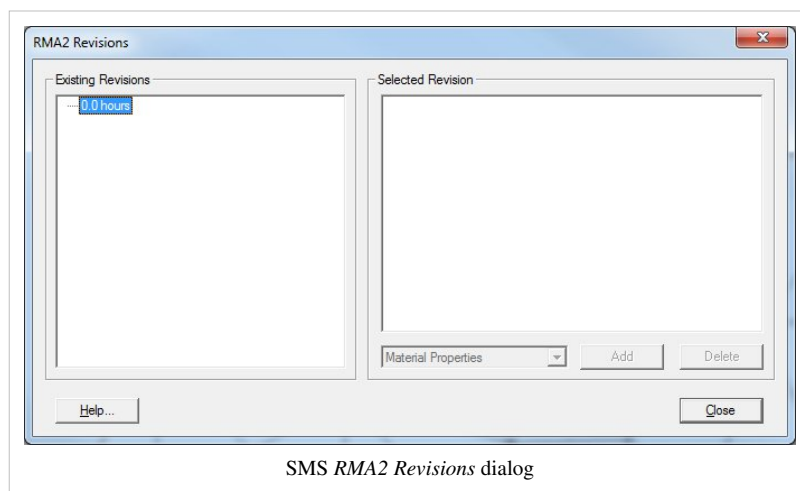
- 1D Node Geometry – Specify 1D node geometry (1D Control Structures).

Model Parameters

- Add GC Strings – Designates the selected nodestrings as Geometry Continuity Check Line nodestrings
 - Material Properties – Specify turbulence, roughness, marsh porosity, wind, rainfall, and coriolis properties for each material
 - Model Check – Checks the RMA2 simulation for common input errors
 - Model Control – Organize input files, specify model parameters, choose output options, etc.
 - Revisions – Create and organize revisions. See below.
 - Run RMA2 – Launches the RMA2 model
-

RMA2 Revisions

RMA2 allows the user to revise both model coefficients and/or boundary conditions for each time step. This is done through the *RMA2* menu selecting the **Revisions** command which will bring up the *RMA2 Revisions* dialog.



SMS RMA2 Revisions dialog

Related Topics

- RMA2

RMA2 Model Control Dialog

This article highlights frequently used RMA2 model control parameters. Please see the RMA2 model documentation for a description of items not discussed or for more information on specific items.

RMA2 model parameters are divided into the following groups in the RMA2 Model Control:

General

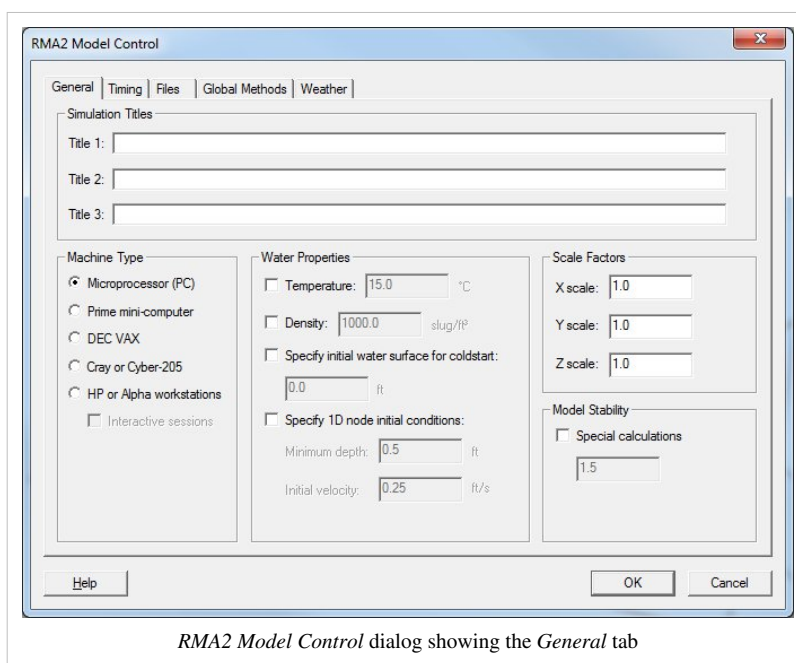
Simulation Titles / \$T1 Card

The RMA2 .geo file contains three comment lines, which can be set using the Title 1, Title 2, and Title 3 edit fields.

Machine Type / \$M Card

Set the machine type the RMA2 simulation will be run on. Options include:

- Microprocessor (PC) - Default option
- Prime mini-computer
- DEC VAX
- Cray or Cyber-205
- HP or Alpha workstations



RMA2 Model Control dialog showing the General tab

Water Properties

Set the properties of the water being simulated.

- Temperature / FT Card
- Density / FD Card
- Coldstart initial water surface / IC Card
- 1D node initial conditions
 - Minimum depth / IC Card
 - Initial velocity / IC Card

Scale Factors

- X scale / GS Card
- Y scale / GS Card
- Z scale / GS Card

Model Stability

- Special calculations

Timing

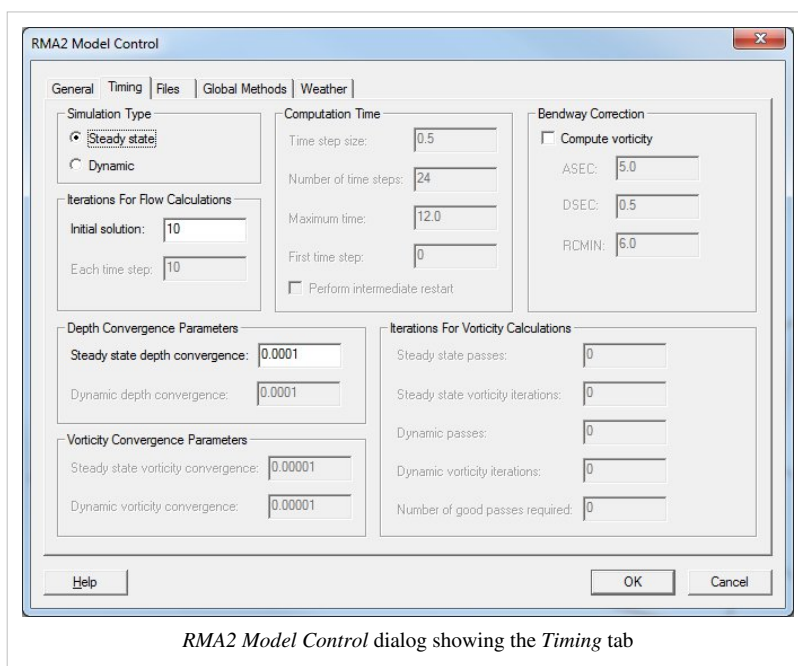
Simulation Type

- Steady state – Boundary conditions do not change with time. A "snapshot" in time.
- Dynamic – Boundary conditions change with time. Computation Time parameters will need to be specified.

Iterations For Flow Calculations

See the TI Card description in the RMA2 model documentation.

- Initial solution – Maximum number of iterations to perform for the initial solution.
- Each time step – Used for dynamic simulation - maximum number of iterations to perform after the first time step.



RMA2 Model Control dialog showing the Timing tab

Computation Time

See the TZ Card description in the RMA2 model documentation.

- Time step size – DELT Variable
- Number of time steps – NCYC Variable
- Maximum time – TMAX Variable
- First time step – NSTART Variable
- Perform intermediate restart – MBAND Variable

Bendway Correction

Used to supply values associated with the calculation of vorticity. See the VO Card description in the RMA2 model documentation.

- Compute vorticity – IVOR Variable
- ASEC – ASEC coefficient for the vorticity equation. Recommend: 5.0
- DSEC – DSEC coefficient for the vorticity equation. Recommend: 0.5
- RCMIN – Minimum radius of curvature that will be allowed. Recommend: 6 feet (or 2 meters).

See also BV card, TV card.

Depth Convergence Parameters

- Steady state depth convergence
- Dynamic depth convergence

Vorticity Convergence Parameters

See the VO Card description in the RMA2 model documentation.

- Steady state vorticity convergence
- Dynamic vorticity convergence

Iterations for Vorticity Calculations

The maximum number of iterations allowed for the vorticity calculation is set by tvariables NVITI or NVITN on the TV card. If the number of passes between the phases reaches NPASS1 or NPASS2, then the calculation has failed to converge. If the number of “good passes” between the two phases exceeds NGOODMAX, the calculation is finished and the code can proceed to the next time step. See the "Bendway Correction (Vorticity)" section and TV Card description in the RMA2 model documentation.

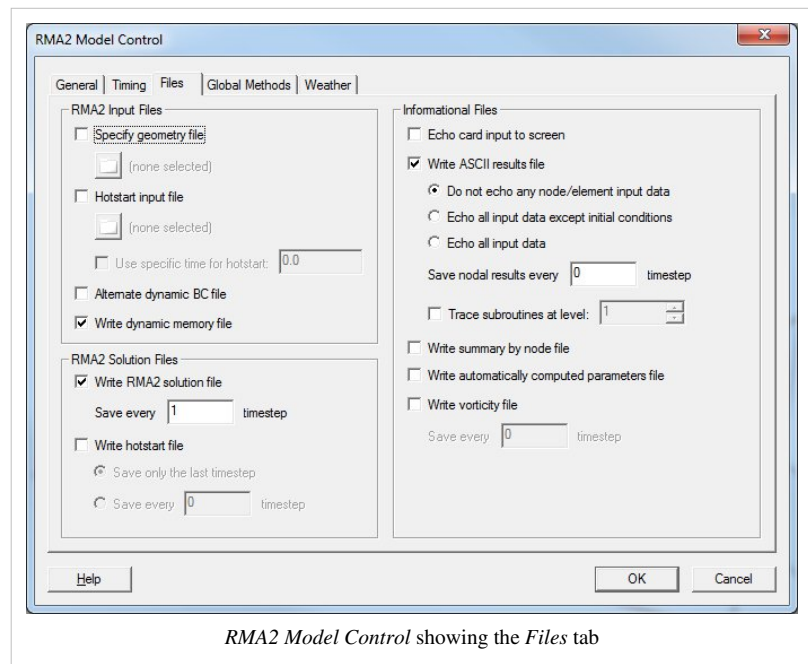
The following TV Card variables are used to control the number of iterations for vorticity calculations:

- Steady state passes – NPASS1 Variable
 - Steady state vorticity iterations – NVITI Variable
 - Dynamic passes – NPASS2 Variable
 - Dynamic vorticity iterations – NVITN Variable
 - Number of good passes required – NGOODMAX Variable
-

Files

RMA2 Input Files

- Specify geometry file – select an existing RMA2 ASCII Geometry (*.geo) file to use with the simulation
- Hotstart input file – specify an existing RMA2 Hot Start File (*.hot) to use with the simulation
 - Use specific time for hotstart
- Alternate dynamic BC file – See the "RMA2 Alternate Dynamic BC File Format" section of the RMA2 model documentation
- Write dynamic memory file – See the "Dynamic Memory Allocation" section of the RMA2 model documentation



RMA2 Model Control showing the Files tab

RMA2 Solution Files

- Write RMA2 solution file
 - Write frequency in time steps
- Write hot start file – Write an RMA2 Hot Start Output (*.hot) file. See RMA2 Spindown for a discussion of hot starting.
 - Save only the last time step – Write only the last successful RMA2 time step solution to the hot start file.
 - Write frequency in time steps – Write multiple time steps to the hot start file.

Informational Files

- Echo card input to screen
- Write ASCII results file
 - Do not echo any node/element input data
 - Echo all input data except initial conditions
 - Echo all input data
- Write frequency of nodal results in time steps
- Trace subroutines level
- Write summary by node file
- Write automatically computed parameters file
- Write vorticity file
 - Write frequency in time steps

Global Methods

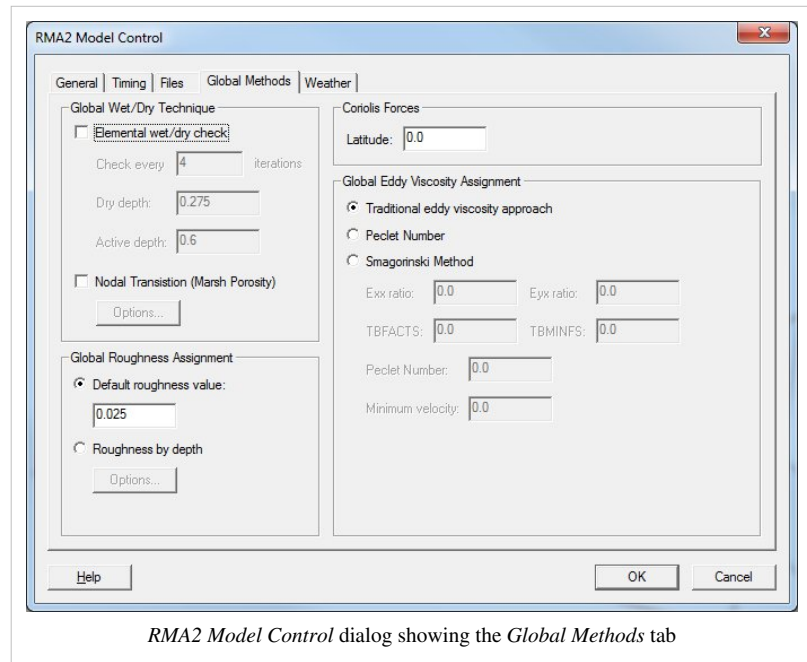
Global Wet/Dry Technique

See the Material Properties article for a description of the *Marsh Porosity* parameters.

- Elemental wet/dry check
 - Check frequency in timesteps
 - Dry depth
 - Active depth
- Nodal Transition (Marsh Porosity)
 - Options – Opens the *Nodal Transition (Marsh Porosity)* dialog

Global Roughness Assignment

- Default roughness value
- Roughness by depth
 - Options – Opens the *Roughness Options* dialog



RMA2 Model Control dialog showing the *Global Methods* tab

Coriolis Forces

- Latitude

Global Eddy Viscosity Assignment

See the Material Properties article for a description of the *Turbulence* parameters.

- Traditional eddy viscosity approach (default)
- Peclet Number
- Smagorinski Method
 - Exx ratio
 - Eyx ratio
 - TBFacts
 - TBMInfs
 - Peclet Number
 - Minimum velocity

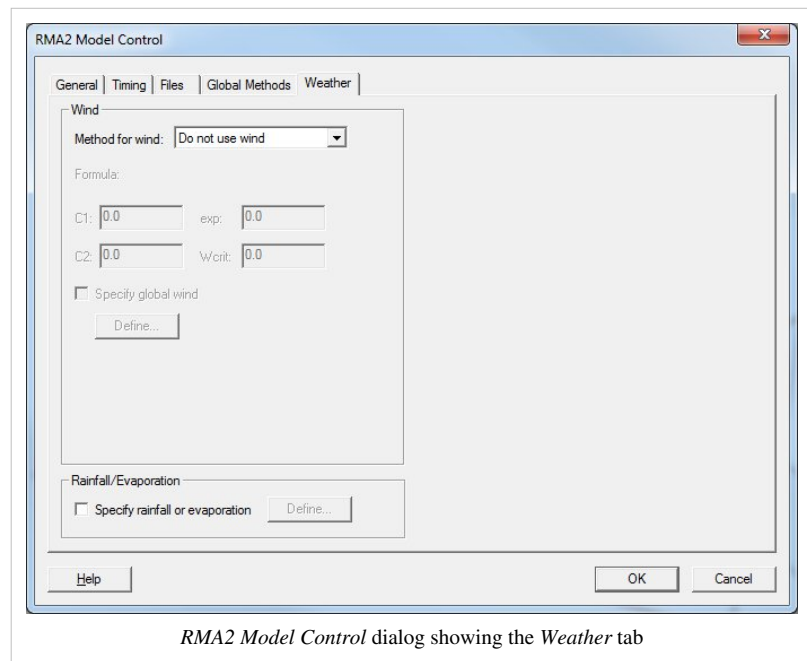
Weather

Wind

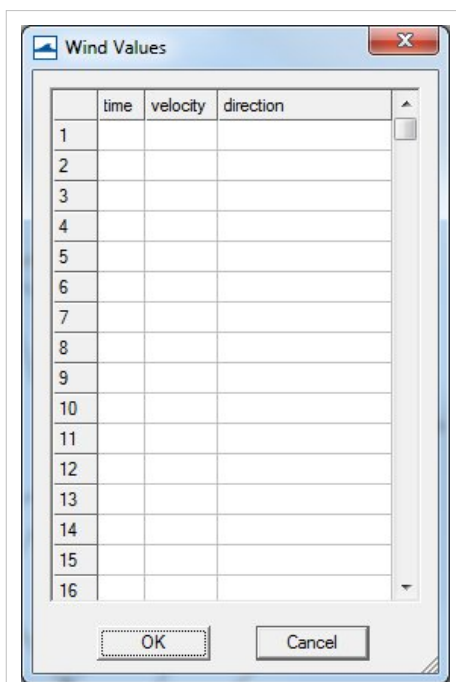
According to the RMA2 model documentation, "Because of lack of experience in using storms in a simulation, this feature remains experimental."

See the "Simulating With Storms" section and BWC Card description in the RMA2 model documentation for more information.

- Method for wind
 - Do not use wind
 - Original RMA2 Formula
 - Van Dorn Formula
 - Wu Formula
 - Safaie Formula
 - Ekma Formula
 - Generic Formula
- A
- exp
- C
- rho
- Specify global wind
 - Define – Opens the *Wind Values* dialog



RMA2 Model Control dialog showing the Weather tab



Wind Values dialog

Rainfall/Evaporation / RA Card

See the "Adding Rainfall And Evaporation" section and RA Card description in the RMA2 model documentation [1] for more information.

- Specify rainfall or evaporation
 - Define – Opens the *Rainfall Values* dialog

Related Topics

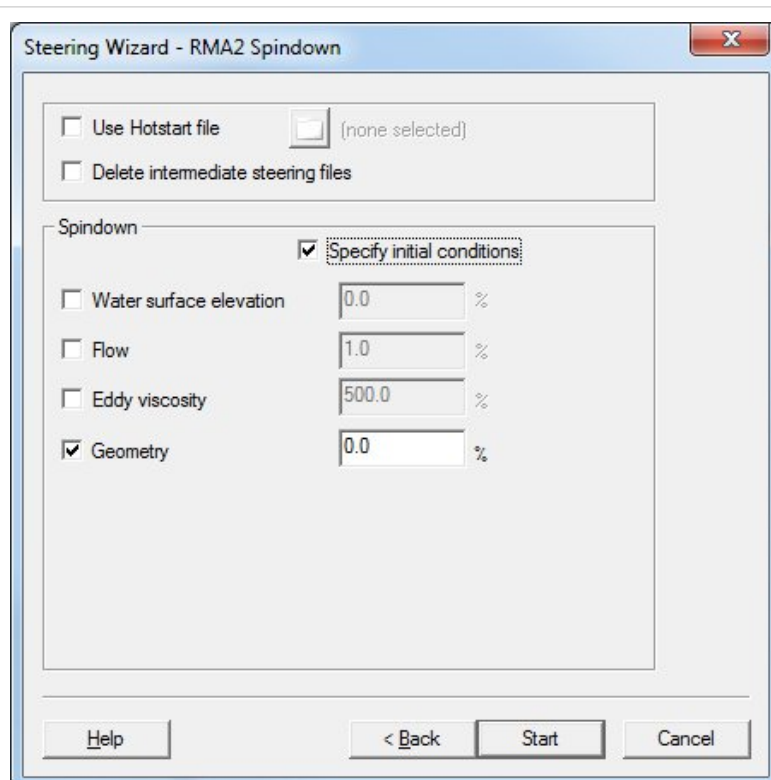
- RMA2

RMA2 Spindown

For cold start simulations, the initial velocities are zero and the water surface elevation is constant. This is often referred to as the "bathtub condition." Often RMA2 will not directly converge using these initial conditions.

Incremental Loading Strategy

The flow equations are nonlinear and thus require an iterative solution, starting from some initial guessed value. Convergence of the iterative solution is not guaranteed. Since the desired boundary conditions may be vastly different from a cold start condition, it may be impossible to get convergence starting from this bathtub type condition. However, a solution



SMS Steering Module – RMA2 Spindown dialog

can be obtained using a series of “Runs” that generate solutions progressively closer to the desired answer. Intermediate boundary conditions that are closer to the final desired boundary conditions are specified to generate a set of flow conditions. These conditions do not represent the final desired flow conditions, but are closer to the final desired flow conditions than the original cold start, and can be used as initial conditions for a subsequent run. Starting the model from a previous solution is called “Hot Starting”. In the incremental loading strategy, “loads” consisting of applied flow rates and water surface elevations along the boundary increment from a cold start condition to the final condition. By choosing a suitably small increment in the boundary conditions, convergence can be attained.

RMA2 Spindown refers to the process of using the Steering Module to automate the process of incremental loading. The Steering Module can vary the following:

- Boundary Conditions
 - Water surface elevation
 - Flow rate
- Model Parameters
 - Eddy viscosity
- Finite Element Network
 - Geometry (nodal elevations)

This replaces the use of "REV" cards in RMA2.

Related Topics

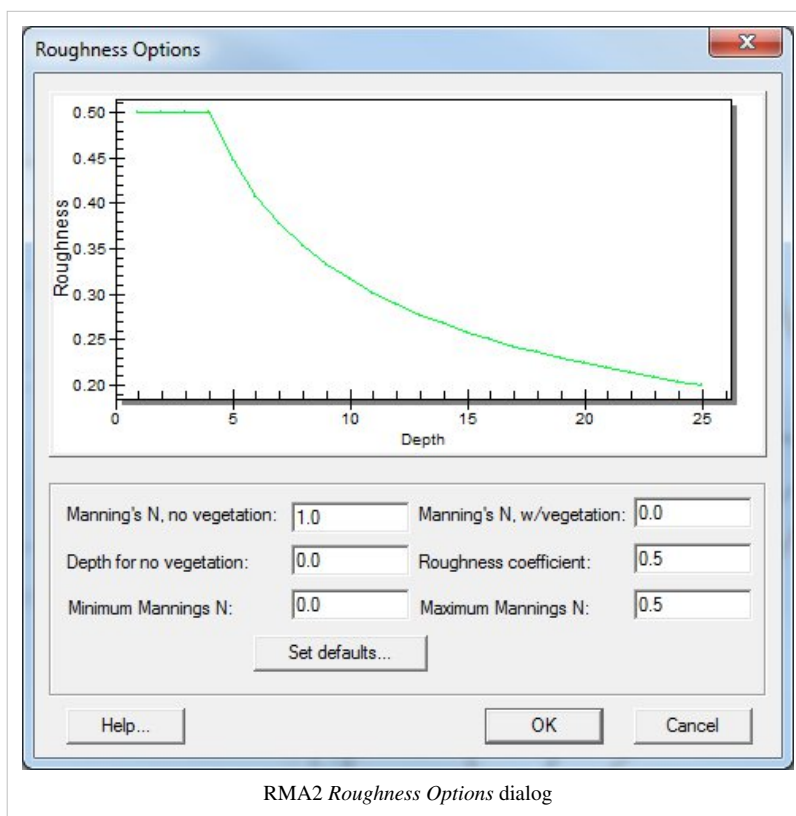
- RMA2
 - Steering
-

Roughness Options Dialog

The *RMA2 Roughness Options* dialog is used to define the RD Card: Automatic Roughness Coefficient Assignment by Depth. See the RD Card description in the RMA2 model documentation for more information.

The following parameters can be set in the *Roughness Options* dialog:

- Manning's N , no vegetation
- Depth for no vegetation
- Manning's N , with vegetation
- Roughness coefficient
- Minimum Manning's N
- Maximum Manning's N
- Set Defaults
 - Opens the *Roughness By Depth Defaults* dialog. Defaults from several Army Corps projects can be automatically entered into the *Roughness Options* dialog.



A graph of the Depth vs. Roughness is calculated using the given parameters and shown in the *Roughness Options* dialog.

Related Topics

- RMA2 Model Control dialog
- RMA2

RMA2 GCL Card

The GCL card is used to specify a line within the grid where the flow rate is of interest. GCL lines may be used to specify the location of boundary conditions.

Example GCL Card

The GCL Card permits the line # to be assigned, and the end of the line is marked with a negative number.

CO	Line#	Corner Node numbers... Negative marks end of line.						
GCL	1	10	11	12	13	14	15	
GCL		16	17	18	19	20	-1	
GCL	2	100	101	102	109	107	99	-1

Reltated Topics

- [RMA2 Menu](#)
- [RMA2](#)

6.7.c. RMA4 - Resource Management Associates

RMA4

RMA4	
Model Info	
Model type	Finite element water quality transport numerical model in which the depth concentration distribution is assumed uniform. It computes concentrations for up to 6 constituents, either conservative or non-conservative, within the computational mesh domain.
Developer	Resource Management Associates United States Army Corps of Engineers (USACE)
Web site	RMA4 web site ^[1]
Tutorials	General Section • Mesh Editing • Observation • Overview • Sensitivity Models Section • RMA4

RMA4 is a finite element water quality transport numerical model. RMA4 was designed to simulate the depth-average advection / diffusion process in an aquatic environment. The model can be used for the evaluation of any conservative substance that is either dissolved in the water or may be assumed to be neutrally buoyant within the water column. The model is also used for investigating the physical processes of migration and mixing of a soluble substance in reservoirs, rivers, bays, estuaries and coastal zones. The model is useful for evaluation of the basic processes or for defining the effectiveness of remedial measures. For most applications, the model utilizes the depth-averaged hydrodynamics from RMA2. The input hydrodynamics solution can be either a steady state or a dynamic simulation and is used to compute particle dispersions.

Functionality

RMA4 has been applied to:

- Define horizontal salinity distribution.
- Trace temperature effects from power plants
- Calculate residence times of harbors or basins
- Optimize the placement of outfalls
- Identify critical area for oil spills or other pollutants spread
- Monitor water quality criterion within game and fish habitats
- Determine the limits of salinity intrusion
- Perform flushing analysis
- Turbidity monitoring
- SMS does not support BOD/DO modeling with RMA4 because of issues with RMA4.

Using the Model / Practical Notes

- RMA4 includes support of the SI card which controls what units the engine will use. If SI is set to 0, the model runs with English units. If it is set to 1, the model runs in metric units. SMS saves the SI card based on the coordinate system (projection) specified by the user. However, it is recommended that metric units be used for RMA4. This applies to the hydrodynamics that are fed into RMA4 as well.
- RMA4 is more sensitive to mass gain/loss than RMA2 across large closed boundary angles. The suggested minimum boundary angle between any two adjacent elements is 10°. This becomes a very great concern, as this applies simply to the wet/dry interface and not to the outer mesh boundary.
- RMA4 should not be used to evaluate highly volatile materials such as gasoline, nor materials that do not mix with water, such as oil. Although RMA4 has been used to measure temperature effects from such locations as power plant discharges, this is not a recommended application. Although the documentation claims to be able to model up to 6 constituents, the SMS developers have never had success with more than a single constituent at a time. Multiple constituents require multiple boundary condition files, each with values for one of the constituents. Multiple constituents can be named and joined through the Data Calculator and exported to a single file for future use.
- Card order matters. The RMA4 run control and full print files are very helpful for determining what options are being used.
- The TABS models are built to expire after a set date. You can download the latest version from the Software Updates section of the Aquaveo website ^[1].

Graphical Interface

SMS provides a graphical interface that is designed to allow users to visualize the projects they are creating, easily modify project parameters, and view the solutions produced by the RMA4 model. See RMA4 Graphical Interface for more information.

The RMA4 Graphical Interface contains tools to create and edit an RMA4 simulation. The simulation consists of a geometric definition of the model domain (the mesh) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the 2D Mesh Module and setting the current model to RMA4. If a mesh has already been created for a RMA4 simulation or an existing simulation read, the mesh object will exist in the Project Explorer and selecting that object will make the 2D Mesh module active and set the model to RMA4. See the Mesh Module documentation for guidance on building and editing meshes as well as visualizing mesh results.

The interface consists of the 2D Mesh Module Menus and tools augmented by the RMA4 Menu. See RMA4 Graphical Interface for more information.

Saving RMA4

When you do a *File | Save As...* the following files get saved in the *.sms

- *.mat referenced to new save location
- *.mat referenced to new save location
- *.qsl referenced to new save location (RMA4 solution file)
- *.sol referenced to new save location and original location (RMA2 solution file)

Related Topics

- SMS Models Page
- TABS Models
- RMA2

External Links

- CHL RMA4 Frequently Asked Questions [2]

RMA4 Boundary Conditions

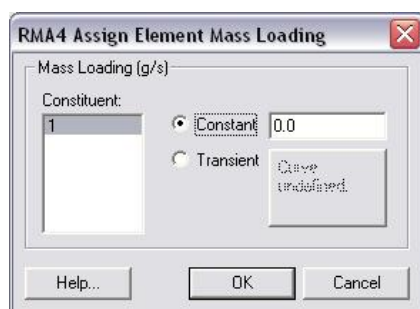
Assigning Boundary Conditions

Boundary conditions are generally defined on nodestrings but may also be defined on elements. The default RMA4 boundary condition is a closed boundary (no constituent). To assign boundary conditions, select the item (nodestring, element, etc.) to which the boundary condition should be applied and use the menu command *RMA4* | **Assign Boundary Condition** which will open a dialog allowing the specification of boundary condition options.

Element Loading

Starting in SMS 10.1, it is now possible to assign a mass loading to an element.

To do this, select the element then go to *RMA4* | **Assign BC**.



Deleting Boundary Conditions

Boundary conditions are removed using the menu command *RMA4 | Delete Boundary Condition*

Nodestring Boundary Conditions

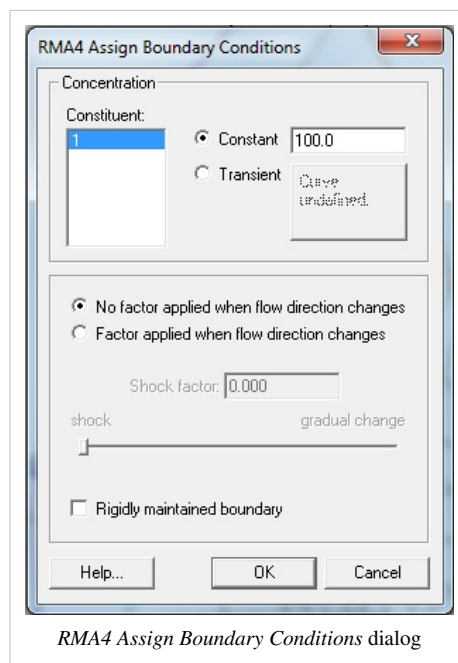
Nodestrings can be used as boundary conditions to define constituent concentrations being modeled. Constituent concentrations can be defined as constant or transient using the XY Series Editor.

RMA4 does not care about the units of the concentration because the output is relative to the initial number you specify. For example, if we specify a concentration of 1,000, the values in the solution will range from 0 to 1,000 as the plume spreads downstream. We can say that the concentration is ppm, ppt, or kg/kg; RMA4 treats all concentrations as relative values.

Since a concentration in water is rarely rigidly maintained, a shock factor may be applied to allow fluctuation of the concentration when the flow direction changes. If no shock factor is applied, no matter how much the flow pushes the concentration out of the model, the concentration at the boundary will not change. However, applying a shock factor is like creating a buffer zone outside the model where the constituent can go until the flow begins to carry it back into the model. This provides for a more realistic solution in some cases. Depending on the situation, a different shock factor may be applied from zero for no shock to 1.0 for a gradual change due to a change in flow direction.

By right-clicking on a selected nodestring and selecting *Assign BC|RMA4...* will bring up the *RMA4 Assign Boundary Conditions* dialog. The dialog has the following options:

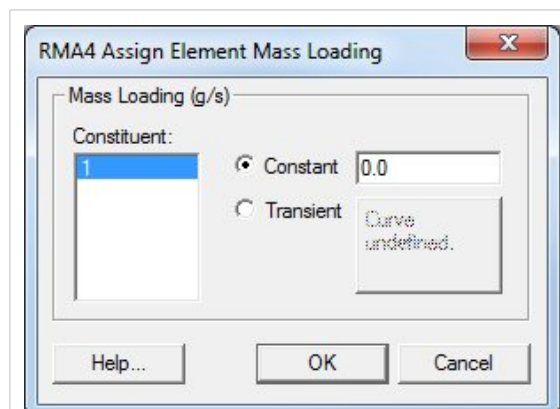
- Constituent
 - Constant
 - Transient
 - Curve undefined
- No factor applied when flow direction changes
- Factor applied when flow direction changes
- Rigidly maintained boundary



RMA4 Assign Boundary Conditions dialog

Element Boundary Conditions

A boundary mass loading can be added to an element in the mesh by selecting first the desired element and then the *RMA4* | **Assign BC** menu command. This opens the *RMA4 Assign Element Mass Loading* dialog where a constant or transient entry can be made in grams/second for each constituent defined in the *RMA4 model control* dialog. The time variant option uses a XY Series File, XYZ Format editor to enter a series of time step-mass loading pairs. The time steps should be consistent with those of the *RMA4* simulation and for each constituent. The mass loadings may be deleted using the *RMA4*|**Delete BC** menu command.



RMA4 Assign Element Mass Loading dialog

Card Type	RMA4 BLE Card: Boundary Loading (Mass)		
Description	Used to assign mass loading on the element boundary.		
Required	NO		
Format	BLE		
Sample	BLE 237 2 3		
Field	Variable	Value	Description
0	IC1	BLE	Card type identifier.
1	ISTART	+	Element id
2	PBCX(J,1)	+	Mass loading for constituent 1 at ISTART=J (g/s)
3	PBCX(J,2)	+	Mass loading for constituent 2 at ISTART=J (g/s)
4	PBCX(J,3)	+	Mass loading for constituent 3 at ISTART=J (g/s)
5	PBCX(J,4)	+	Mass loading for constituent 4 at ISTART=J (g/s)
6	PBCX(J,5)	+	Mass loading for constituent 5 at ISTART=J (g/s)
7	PBCX(J,6)	+	Mass loading for constituent 6 at ISTART=J (g/s)

Card Type	END
Description	Identifies the end of the time step
Required	NO

Related Topics

- RMA4
- RMA2 Boundary Conditions

RMA4 Files

Input and Output files for RMA4.

Input Files

- Hydrodynamic Solution (usually an RMA2 *.sol file with a path name less than 67 characters long)
- RMA4 Run Control (Boundary Condition) input (*.trn)
- GFGEN Output Geometry (*.bin)

Output Files

- Transport Solution (*.qsl)
- Full Print Listing File (*.ot3)

Related Topics

- RMA4

RMA4 Graphical Interface

The RMA4 Graphical Interface includes tools to assist with creating, editing, and debugging a RMA4 model. The RMA4 interface exists in the Mesh Module. Users can visualize and create projects, easily modify project parameters, and view the solutions produced by the RMA4 model.

The following links to describe the SMS interface that interacts with the RMA4 model:

- RMA4 Menu

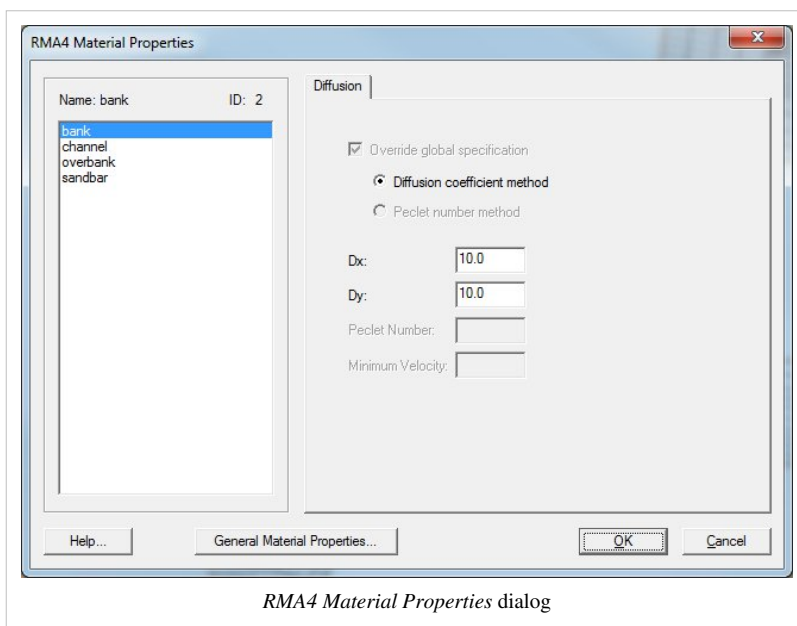
Related Topics

- TABS Models
 - GFGEN
 - RMA2
 - RMA4
-

RMA4 Material Properties

Diffusion

Because RMA4 does not have the ability to model turbulence, diffusion coefficients may be used to approximate turbulence. By assigning a diffusion coefficient in the x and y directions for each material, the flow over that material will be altered somewhat to provide an approximation of turbulent flow over that region. A value of -1.0 may be applied to allow normal flow over the material. Positive values provide turbulence. The higher the value, the greater the effect is.



Related Topics

- RMA4

RMA4 Menu

The following menu commands are available in the *RMA4* Menu:

Boundary Conditions

- **Assign BC** – Specify boundary conditions for the selected nodestring(s) or element(s)
- **Delete BC** – Deletes the boundary condition assigned to the selected nodestring(s) or element(s)

Model Parameters

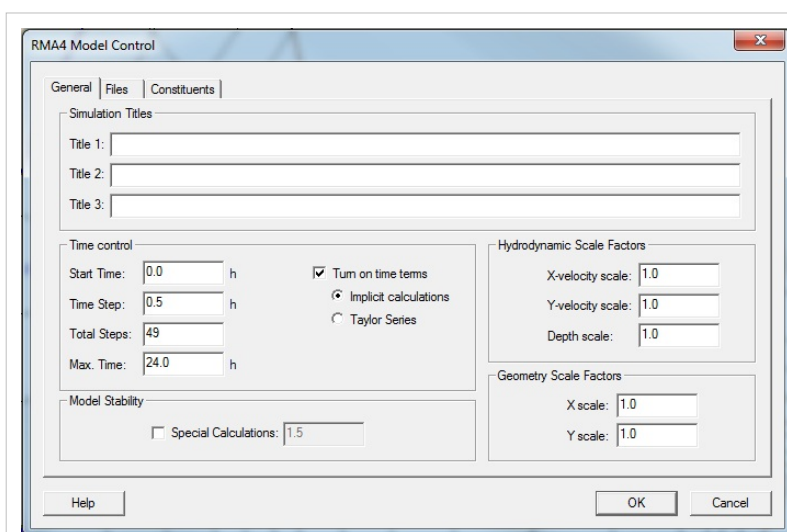
- **Delete Simulation** – Deletes the current RMA4 simulation. Sets all values in the Model Control to default values.
- **Material Properties** – Specify turbulence, roughness, marsh porosity, wind, rainfall, and coriolis properties for each material
- **Model Control** – Organize input files, specify model parameters, choose output options, etc.
- **Run RMA4** – Launches the *RMA4* model

Related Topics

- RMA4

RMA4 Model Control

The *Model Control Dialog* is used to define general, file, and constituents characteristics. The **General** tab includes options to set up the period of time the model will run, and hydrodynamic and geometry scale factors. The **File** tab includes the time step used from the RMA2 velocity file and the time subtracted from the RMA2 velocity file, *RMA4 Output files* options, and *Informational Files* to be produced when running the model. The **Constituents** tab includes diffusion coefficient for the constituent, constituent control, and mass check.



RMA4 Model Control dialog showing the *General* tab

General

Simulation Titles. The RMA4 *.geo file contains three comment lines, which can be set using the Title 1, Title 2, and Title 3 edit fields.

Time control

- Start time
- Time Step
- Total Steps
- Max. Time

Model Stability

- Special Calculations

Hydrodynamic Scale Factors

- X-Velocity scale
- Y-Velocity scale
- Depth scale

Geometry Scale Factors

- X scale
- Y scale

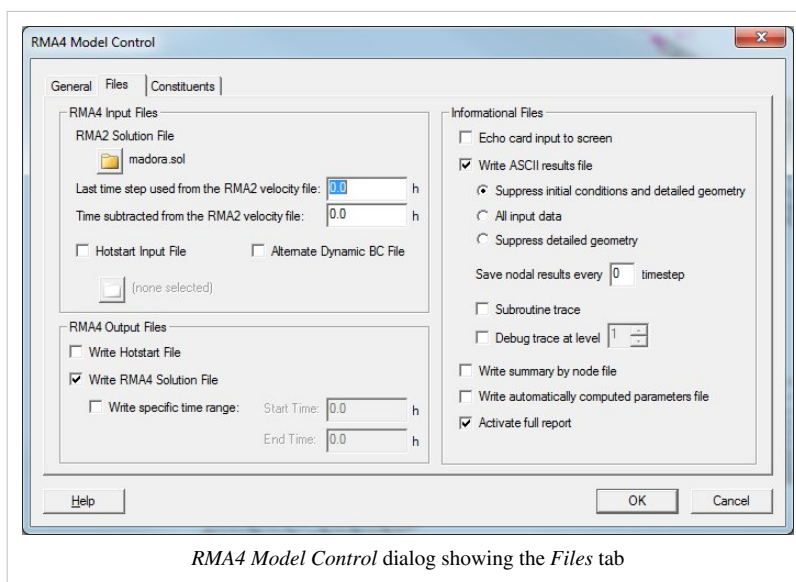
Files

RMA4 Input Files – Time steps from the RMA2 solution file that will be used for the RMA4 simulation.

- Last time step used from the RMA2 velocity file
- Time subtracted from the RMA2 velocity file
- Hotstart Input file
- Alternate Dynamic BC File

RMA4 Output files – Output files when running RMA4

- Write Hotstart File
- Write RMA4 Solution File
 - Write specific time range



RMA4 Model Control dialog showing the Files tab

Information Files – Files to create when running RMA4

- Echo card input to screen
- Write ASCII results file
 - Suppress initial conditions and detailed geometry
 - All input data
 - Suppress detailed geometry

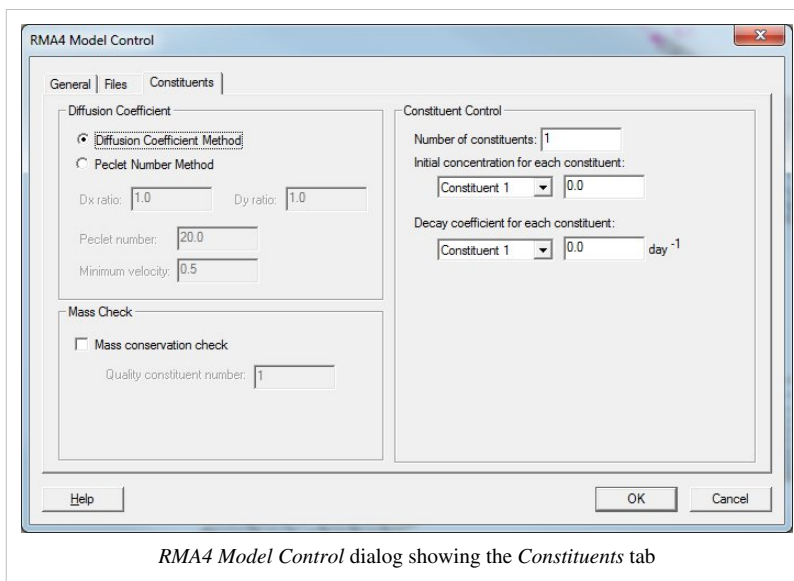
- Save nodal results every ___ time step
- Subroutine trace
- Debug trace at level
- Write summary by node file
- Write automatically computed parameters file
- Activate full report

Constituents

Diffusion Coefficient – Diffusion Coefficient Method and Peclet Number Method.

Constituent Control – Number of constituents, initial concentration for each of the constituents, and decay coefficient for each coefficient.

Mass Check – Mass conservation check.



RMA4 Model Control dialog showing the Constituents tab

Related Topics

- RMA4

6.8. TUFLOW

TUFLOW

TUFLOW	
Model Info	
Model type	One-dimensional (1D) and two-dimensional (2D) flood and tide simulation software. It simulates the complex hydrodynamics of floods and tides using the full 1D St Venant equations and the full 2D free-surface shallow water equations.
Developer	Bill Syme BMT WBM (Australia) ^[1]
Web site	TUFLOW web site ^[2]
Tutorials	Models Section • TUFLOW 2D • TUFLOW 1D

TUFLOW ENGINE

The TUFLOW computational engine computes 1D and 2D hydraulic solutions. The engine is very stable making it an excellent choice for models with lots of wetting and drying. TUFLOW includes 1D cross-sections, 1D pipe networks, 1D and 2D hydraulic structures, and allows rainfall directly on the model domain so it is useful in many different applications including coastal, riverine and urban settings.

For more information see the TUFLOW webpage www.tuflow.com ^[3].

SMS Interface

The TUFLOW Interface in SMS can be used to construct TUFLOW models and view and analyze the results.

Some of the features of the interface include:

- Create 2D domains from bathymetry data
- Extract 1D cross-section domains from TIN data
- Define 1D pipe domains including links to the surface
- Import data from ArcGIS or MapInfo GIS formats
- Define boundary conditions at 1D nodes, 2D lines, or 2D polygons
- Create 2D geometry modifications to model objects such as levees
- Set material properties such as Manning n values and hydrologic losses
- Define simulations from building blocks above
- Simulations share common data to prevent data duplication errors and make it easy to update project with new data

The interface consists of the Cartesian grid tools, the TUFLOW menu and the TUFLOW coverages.

Using the Model / Practical Notes

- For the past several years, TUFLOW required the horizontal and vertical coordinates used for input files to be in meters. Starting with the release of TUFLOW 2012, the numeric engine now supports an option for customary units. We recommend that you set the units to be the desired units at the beginning of a project. SMS will allow you to switch units, but since this feature is so new, the responsibility to verify model parameters and boundary conditions are in the correct units remains with the user. Geographic Coordinates cannot be used since it is a latitude/longitude system defined in decimal degrees.

TUFLOW Overview

TUFLOW

Overview

TUFLOW is a one-dimensional (1D) and two-dimensional (2D) flood and tide simulation software. It simulates the complex hydrodynamics of floods and tides using the full 1D St Venant equations and the full 2D free-surface shallow water equations.

TUFLOW Interface

The TUFLOW interface in SMS exists in several modules including: Cartesian Grid Module, Map Module, Scatter Module, and Mesh Module. Many of the TUFLOW building blocks are managed using the Project Explorer.

How do I use TUFLOW?

The SMS tutorials are a good place to start learning to use SMS and associated models. The following tutorials will help get you started using TUFLOW.

- Models | TUFLOW 2D
- Models | TUFLOW 1D
- General | Data Visualization
- General | Observations

General Steps to Build a TUFLOW Model

- Define the domain or domains
 - Setup boundary conditions
 - Create the simulation
 - Define Material Properties
 - Set Model Parameters
 - Run TUFLOW
 - Find problems and verify model setup using check files
 - View and analyze the results
 - Repeat above steps to generate new simulations
-

Optional Steps

- Define flow constrictions
- Define geometry modifications such as levees

TUFLOW Coverages

Much of the data used in TUFLOW is specified on a coverage level into one of the TUFLOW Coverages.

Numeric Engine Background

The TUFLOW computational engine computes 1D and 2D hydraulic solutions. The engine is very stable making it an excellent choice for models with lots of wetting and drying. TUFLOW includes 1D cross-sections, 1D pipe networks, 1D and 2D hydraulic structures, and allows rainfall directly on the model domain so it is useful in many different applications including coastal, riverine and urban settings.

For more information see the TUFLOW webpage www.tufLOW.com ^[2]

Saving TUFLOW

When you do a *File | Save As...* the following files get saved in the *.sms

- *.mat referenced to new save location
- _grds.h5 referenced to new save location
- *.mat_h5 referenced to new save location
- Geomcomps.h5 referenced to new save location
- scatter.h5 referenced to new save location
- tufLOW files don't get saved to new location unless exported again

Running TUFLOW

Before running TUFLOW, the TUFLOW files must be generated from SMS. These files are separate from the files SMS uses to store its TUFLOW information.

The options to export the TUFLOW files and run TUFLOW are in the right-click menu of the simulation.

- Export TUFLOW files – This will export the TUFLOW model files into the run directory (by default a directory named TUFLOW under the project directory).
- Launch TUFLOW – This will launch TUFLOW under the assumption that the files have already been exported as above.
- Save Project, Export TUFLOW files, and Launch TUFLOW – This option will save the current SMS project file, export the TUFLOW files, and launch TUFLOW on these files.

Viewing Results

TUFLOW results are written in SMS mesh format in the \TUFLOW\results folder. The mesh module tools and menus are used to visualize the solution.

The solution files all start with the name of the simulation. The extension identifies the type of data in the result file.

- *.2dm – two-dimensional output mesh in SMS format.
- *.dat – The dataset files associated with the mesh solution (d for depth, h for water surface, etc.)
- *.mat – The output materials from TUFLOW. Most of the element materials come from the *.tmf file. TUFLOW also uses the materials to identify boundary cells.
- *.ALL.sup – A file that opens the *.2dm file, all the *.dat files, and the *.mat file.
- *.hV.sup – A file that opens the *.2dm file, the water surface and velocity datasets, and the *.mat file.

Related Topics

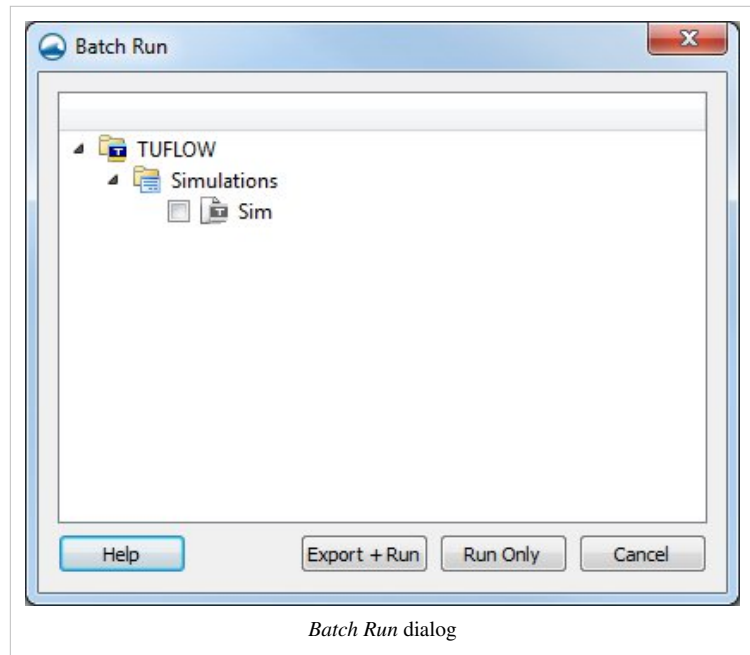
- Define TUFLOW domain
 - Mesh Module TUFLOW model
 - Q&A TUFLOW
 - Reading a TUFLOW simulation
 - TUFLOW 2D Flow constriction Shape Coverage
 - TUFLOW 2D Geometry Components
 - TUFLOW 2D Z Lines/Polygons (Simple) Coverage
 - TUFLOW 2D Z Lines (Advanced) Coverage
 - TUFLOW AD
 - TUFLOW Boundary Conditions
 - TUFLOW Combining 1D and 2D Domains
 - TUFLOW Inlet Database
 - TUFLOW Irregular Culverts
 - TUFLOW Linking 2D Domains
 - TUFLOW Manholes
 - TUFLOW Model Parameters
 - TUFLOW Simulation
 - TUFLOW Check Files
 - TUFLOW Grid Options
 - TUFLOW Material Properties
 - TUFLOW Reading Simulation
-

Batch Runs

The **Batch Run** command allows multiple simulations (each with their own defined scenarios/events to run) to run simultaneously. A batch run is initiated by right-clicking on the TUFLOW simulations folder (the folder that holds the individual simulations) and choosing **Batch Run**.

The *Batch Run command brings up a dialog to choose the simulations to run. After checking the simulations to run, you can choose whether or to Export and Run the simulations or 'Run only*. The run only option can be used if the simulation files have been exported previously and don't need to be changed.

After the simulations have been selected in the *Batch Run* dialog, the runs will be launched and can be monitored in the *Model wrapper* dialog. The top part of this dialog has progress bars showing the individual progress of each run. The screen output for each run can be viewed by clicking on the associated progress bar. The screen output is shown in the bottom part of the dialog.



Batch Run dialog

Related Topics

- TUFLOW

Building TUFLOW 2D Domains

TUFLOW 2D domains are cartesian (rectilinear) grids. 2D domains are created using a Cartesian Grid Frame inside a TUFLOW Grid Extents Coverage.

Grids store elevation values at each corner, midside, and center. There are several Grid Options that apply to a grid that can be specified by right-clicking on the grid and choosing *Properties*.

A 2D domain is represented as a 2D Geometry Component in the TUFLOW interface. A 2D Geometry component includes the grid and associated coverages.

Grid elevations can be modified using the select grid location tool by selecting and changing the z values at specific directions. A 2D TUFLOW Geometry Modification Coverage can also be used to force elevations using arcs or polygons.

Multiple grids can be used inside the same simulation. The grid used in each area of the domain is specified using a TUFLOW 2D/2D Link Coverage.

Command Objects

Command Objects provide the ability to provide additional commands for TUFLOW simulations using the TUFLOW command file syntax. The SMS interface provides some guidance for using these commands and allows users to link to objects created in SMS. Command objects can be used for:

- Create scenarios to do multiple runs using the same simulation.
- Share commands between multiple simulations or 2D Geometry components.
- Utilize commands not available in the SMS interface.
- Enter commands to read GIS data generated outside of SMS.

TUFLOW Command Object syntax is described in the TUFLOW Manual. Each line (except for blank/comment lines) has a command identifier. After the identifier most commands have are followed by a double equals “==” and a command value. In some cases, the command value may have multiple parts separated by blank spaces or “.” A few commands don’t have any arguments and do not have a “==” at all.

Types of Commands

There are several kinds of commands that can be utilized through command objects. The types of commands that can be issued used include: 2D Model control commands (*.tcf), 1D Model control commands (ecf), 2D Geometry commands (tgc), Boundary conditions commands (*.tbc), and commands for specific events (*.tef). The event commands are defined with each event when using composite events. The rest are managed in the project explorer as command objects.

The Appendix of the TUFLOW manual contains a complete list of the commands that are available, the purpose of each command, and its syntax. The identifiers above in parentheses (*.tcf, *.ecf, *.tgc, *.tbc, and *.tef) are the abbreviations used by TUFLOW for file naming and also used to find the appropriate section of the Appendix for a set of commands.

Creating Command Objects

Command objects are created using right-click commands in the project explorer. After creating a TUFLOW simulation, you should see a folder named “Command Objects” in the TUFLOW root folder. You can create command objects by right clicking on the “Command Objects” folder, choosing “Create Command Object”, and choosing the appropriate type. This will create a folder for command objects of this type along with a new command object. This approach can be repeated to create additional command objects of the same or different type.

Editing Command Object Contents

When command objects are first created, their contents are blank. You can edit the contents of a command object by double clicking on the object or right-clicking and choosing “edit.” This will bring up the command object editor dialog.

The command object editor dialog has two main parts. One part consists only of a text editor. The text in this editor is what is stored with the command object and will be used with the TUFLOW model. The commands for the object may be typed directly into the editor. Comments may be added and must start with the exclamation mark (!).

The other part of the dialog contains controls to assist the user in creating commands for the command object. There is a control showing the list of commands in a tree structure organized by the appendix where the command is found in the TUFLOW manual. This part of the dialog will also contain other controls based upon the command that is selected in tree control. For example, if the command requires a single floating point value a edit control will shown where the user can specify the value for the command.

Once the command creation controls have been specified as desired, the command is inserted into the command text using the insert button. Once this has been done, editing of the command must be done in the editor. You can't edit commands already created using the create command controls. You can delete an existing command and insert another if you wish to use the controls to setup the command text.

Using Command Objects in Simulations

With the exception of event command objects (discussed later), command objects are used by creating links to the objects from simulations and geometry components. Links are created by dragging the command object under the object you wish to add the link to. 2d model control command objects and 1d model control command objects are linked to simulations. 2D geometry commands and boundary condition commands are linked to 2D Geometry Components.

When a command object is linked to a simulation or geometry component, SMS writes a file with the exact text from the command object and links to the file from the TCF or TGC file as appropriate. Command objects are always used after any commands specified in the SMS interface and therefore will override duplicate specifications. Multiple command objects can be linked to the same object and will be used in order from top to bottom. Command objects can override settings defined in command objects used previously.

Event command objects are only used with composite events (** need to add link to this help topic). To set commands for a composite event, you click on the button “Additional commands” in the TUFLOW events dialog (**link to dialog help). This text will be stored with the event and written to a file with the extension “tef” which will be used when running TUFLOW. The event command objects do not show up in the project explorer.

Scenarios in Command Objects

TUFLOW command objects can define scenarios in order to setup multiple models that have shared data without the need for separate simulation objects. Anything not specified for one or more scenarios is used in all scenarios for the simulation. Scenarios could be used to model different geometries by including z-lines or tins for specific scenarios but not others.

Scenarios can only be defined using command objects. Scenarios are defined by using the following commands:

- If Scenario
- Else If Scenario
- Else
- End If

A scenario block must start with “If Scenario” and end with “End If.” The other commands are optional.

Define TUFLOW domain

TUFLOW supports 2D domains, 1D cross-section based domains, and 1D pipe network domains. Multiple domains of each type can be used within the same TUFLOW simulation. To use Multiple 2D domains in TUFLOW, you must purchase the add-on module.

Types of Domains

2D Domains

Two-dimensional domains are Cartesian grid (rectilinear) and include elevation data.

TUFLOW 2D domains are cartesian (rectilinear) grids. 2D domains are created using a Cartesian Grid Frame inside a TUFLOW Grid Extents Coverage.

Grids store elevation values at each corner, midside, and center. There are several Grid Options that apply to a grid that can be specified by right-clicking on the grid and choosing *Properties*.

A 2D domain is represented as a 2D Geometry Component in the TUFLOW interface. A 2D Geometry component includes the grid and associated coverages.

Grid elevations can be modified using the select grid location tool by selecting and changing the z values at specific directions. A TUFLOW 2D Z Lines/Polygons (Simple) coverage can also be used to force elevations using arcs or polygons.

Multiple grids can be used inside the same simulation. The grid used in each area of the domain is specified using a TUFLOW 2D/2D Link Coverage.

1D Cross-section Domains

Cross-section based domains include channel and cross-section definitions.

1D cross-section domains use both the TUFLOW network coverage and the TUFLOW cross-section coverage. The network coverage describes the channel attributes such as the length and the cross-sections are defined in the cross-section coverage.

1D Pipe Network Domains

Pipe network domains can be used to model such things as culverts and storm sewers.

1D Pipe networks are created using culvert channels inside a 1D Network Coverage. The shape of the pipe segments are specified using the attributes button in the channel attributes dialog.

Channel inlets, storage, and invert elevations can be defined on the arc nodes (endpoints). Starting in version 10.1, you can create your own inlet definitions and use them within your pipe network model. See `TUFLOW_Inlet_Database`

Combining Multiple Domains

1D to 2D Domains

A very powerful and useful feature in TUFLOW is the ability to combine a 1D cross-section domain and a 2D domain. This method can be used to embed a 2D domain between two 1D domains, embedding pipe networks, or to model the channel in 1D and floodplain in 2D. See [Combining 1D and 2D Domains](#).

2D Domain to 2D Domain

One of the disadvantages to TUFLOW is that it uses a fixed grid rather than a mesh which can transition very well from large to small cells. This can be overcome somewhat by using 1D and 2D domains. TUFLOW also supports using multiple 2D domains of different resolutions. The SMS interface makes it easy to set up see [Linking 2D Domains](#).

Reading a TUFLOW Simulation

Reading a TUFLOW simulation in SMS

TUFLOW simulation files can be opened into SMS by opening the simulation's control file (*.tcf). This can be done by dragging and dropping the file into the graphics window or by using File | Open. Multiple simulations can be opened together, and SMS will only create one copy of the data that is shared between the projects.

TCF File

The following commands are supported when reading a TCF file into SMS:

- BC CONTROL FILE – read the specified TUFLOW Boundary Condition file
 - BC DATABASE – set the active database
 - BC EVENT NAME – set the active event filename to be used
 - BC EVENT TEXT – set the text to be substituted by the BC EVENT NAME value
 - CELL SIDE WET/DRY DEPTH – set the depth for when cell sides wet and dry
 - CELL WET/DRY DEPTH – set the wet/dry depth for when a cell wets and dries
 - CHECK INSIDE GRID – set to ERROR, WARNING, or OFF
 - DISPLAY WATER LEVEL – display water level for cell at specified location
 - END 1D DOMAIN – end block of 1D commands
 - END 2D DOMAIN – end block of 2D domain definition
 - END TIME – set the simulation finish time in hours
 - ESTRY CONTROL FILE (AUTO) – set the ESTRY control filename
 - GEOMETRY CONTROL FILE – read the specified TUFLOW Geometry Control file
 - HX ZC CHECK – set whether or not to check the minimum ZC elevation is above the 1D bed level
 - INSTABILITY WATER LEVEL – set the water level used to detect instabilities
 - MAP OUTPUT DATA TYPES – define data types to be output in map format
 - MAP OUTPUT INTERVAL – set the output interval for map based output in seconds
 - MASS BALANCE OUTPUT – set whether or not to output mass balance data
 - MI PROJECTION – set the geographic projection
 - NUMBER ITERATIONS – specify the number of iterations per timestep
 - READ FILE – read the specified external file as an addition TUFLOW control file
 - READ GIS FC – read the specified flow constriction mif/mid files
 - READ GIS IWL – read the specified initial water level mif/mid files
 - READ MATERIALS FILE – read the specified TUFLOW materials file
 - READ MI PROJECTION - set the projection
 - READ RESTART FILE – set the model to use the specified restart file
 - SCREEN/LOG DISPLAY INTERVAL – set the frequency for display of output to the computer screen and log file
 - SET IWL – set a default initial water level for all cells
 - START 1D DOMAIN – start a block of 1D commands
 - START 2D DOMAIN – begin block of 2D domain definition
 - START MAP OUTPUT – set simulation time in hours when map output begins
 - START TIME – set the start time of the simulation in hours
 - STORE MAXIMUMS AND MINIMUMS – set whether or not to store the highest and lowest values
 - SX ZC CHECK – set whether or not to check if the minimum ZC elevation is below the connected or snapped 1D node bed level
-

- TIMESTEP – set the computation time step in seconds
- VISCOSITY COEFFICIENT – set the viscosity coefficient(s)
- VISCOSITY FORMULATIONS – set the viscosity formulation
- WRITE RESTART FILE AT TIME – set when to write the restart file in hours
- WRITE RESTART FILE INTERVAL – set the interval in hours between writing the restart file
- ZERO NEGATIVE DEPTHS IN SMS – set whether or not to zero depths if negative

The following commands are not supported and are written to an external file:

- ADJUST HEAD AT ESTRY INTERFACE
 - APPLY WAVE RADIATION STRESSES
 - APPLY WIND STRESSES
 - BC WET/DRY METHOD
 - BC ZERO FLOW
 - BED RESISTANCE CELL SIDES
 - BED RESISTANCE DEPTH INTERPOLATION
 - BED RESISTANCE VALUES
 - BOUNDARY CELL SELECTION
 - CALIBRATION POINTS MI FILE
 - CHANGE ZERO MATERIAL VALUES TO ONE
 - CHECK MI SAVE EXT
 - CSV TIME
 - DEFAULTS
 - DENSITY OF AIR
 - DENSITY OF WATER
 - DEPTH/RIPPLE HEIGHT FACTOR LIMIT
 - DISTRIBUTE HX FLOWS
 - DOUBLE PRECISION
 - EXCEL START DATE
 - FIRST SWEEP DIRECTION
 - FREE OVERFALL
 - FREE OVERFALL FACTOR
 - FROUDE CHECK
 - FROUDE DEPTH ADJUSTMENT
 - GLOBAL FC CH FACTOR
 - GLOBAL WEIR FACTOR
 - INPUT DRIVE
 - INSIDE REGION
 - LATITUDE
 - LINE CELL SELECTION
 - MAP CUTOFF DEPTH
 - MAXIMUM POINTS
 - MAXIMUM VERTICES
 - MESHPARTS
 - MODEL PRECISION
 - NULL CELL CHECKS
 - NUMBER 2D2D LINK ITERATIONS
 - OBLIQUE BOUNDARY ALIGNMENT
 - OBLIQUE BOUNDARY METHOD
-

- OUTPUT DRIVE
- READ GIS CYCLONE
- READ GIS HURRICANE
- READ GIS ISIS NETWORK
- READ GIS ISIS NODES
- READ GIS ISIS WLL
- READ GIS ISIS WLL POINTS
- READ GIS GLO
- READ GISLP
- READ GISPO
- READ GIS XP NETWORK
- READ GIS XP NODES
- READ GISXP WLL
- READ GIS XP WLL POINTS
- READ ROWCOL IWL
- RECALCULATE CHEZY INTERVAL
- SHALLOW DEPTH STABILITY FACTOR
- SHALLOW DEPTH WEIR FACTOR CUT OFF DEPTH
- SHALLOW DEPTH WEIR FACTOR MULTIPLIER
- SNAP TOLERANCE
- START TIME SERIES OUTPUT
- START WIND OUTPUT AT TIME
- SUPERCRITICAL
- SX HEAD ADJUSTMENT
- TIME SERIES OUTPUT INTERVAL
- TIMESTEP DURING WARMUP
- UK HAZARD DEBRIS FACTOR
- UZ HAZARD FORMULA
- UZ HAZARD LAND USE
- UNUSED HX AND SX CONNECTIONS
- VERBOSE
- VG Z ADJUSTMENT
- WARMUP TIME
- WATER LEVEL CHECKS
- WAVE PERIOD
- WETTING AND DRYING
- WIND OUTPUT INTERVAL
- WIND/WAVE SHALLOW DEPTHS
- WRITE EMPTY MI FILES
- WRITE PO ONLINE
- WRITE Z1D CHECK FILES

The following commands are ignored when read in by SMS because they are hardwired:

- CHECK MI SAVE DATE – hardwired to “OFF”
 - LOG FOLDER – hardwired to “.\log”
 - MI PROJECTION CHECK – hardwired to “WARNING”
 - OUTPUT FOLDER – hardwired to “..\results”
 - WRITE CHECK FILES – hardwired to “..\check\”
-

TGC File

The following commands are supported when reading a TGC file into SMS:

- ALLOW DANGLING Z LINES – allows dangling z lines
- CELL SIZE – sets the cell size
- GRID SIZE (N, M) – sets the number of I and J in the grid
- GRID SIZE (X, Y) – sets the grid size in x and y
- ORIENTATION -- sets the x,y location of a point along the x-axis
- ORIENTATION ANGLE – sets the angle of the grid
- ORIGIN – sets the location for the origin
- READ FILE – reads an external file
- READ GIS CODE – reads mif/mid files and stores the codes in polygons in a bc coverage
- READ GIS FC SHAPE – reads mif/mid files to create a flow constriction coverage
- READ GIS FLC – reads mif/mid files to create a 2D spatial coverage with form loss coefficient as the coverage property
- READ GIS IWL – reads mif/mid files to create a 2D spatial coverage with initial water level as the coverage property
- READ GIS LAYERED FC SHAPE – reads mif/mid files to create a flow constriction coverage
- READ GIS LOCATION – set the grid location
- READ GIS MAT – reads mif/mid files to create an area property coverage
- READ GIS VARIABLE Z SHAPE – reads mif/mid files to create a z-shape coverage
- READ GIS WRF – reads mif/mid files to create a 2D spatial coverage with weir factor as the coverage property
- READ GIS Z LINE – reads mif/mid files to create a z-lines coverage
- READ GIS Z SHAPE – reads mif/mid files to create a z-shape coverage
- READ GIS ZPTS – reads mif/mid files to create points in a z-lines coverage
- READ ROWCOL CODE – sets code values for grid cells
- READ ROWCOL MAT – sets material values for grid cells
- READ ROWCOL ZPTS – sets z values for grid cells
- READ TIN ZPTS – creates a scatter set from the points
- SET CODE – set the default code
- SET IWL – if opening with a simulation, sets the default initial water level, otherwise the card is written to an external file
- SET MAT – set the default material id
- SET ZPT – set the default z-value for each grid cell

The following commands are not supported and are written to an external file:

- READ TGA
- READ TGC
- EXTERNAL BNDY
- READ GIS CNM
- READ GIS FRIC
- READ ROWCOL GRID
- SET CNM
- SET FRIC
- SET CODE ZERO ABOVE ZC
- WRITE MI DOMAIN
- WRITE MI GRID
- CREATE TIN ZPTS

- DEFAULT LAND Z
- INTERPOLATE ZC
- INTERPOLATE ZHC
- INTERPOLATE ZUV
- INTERPOLATE ZUVC
- INTERPOLATE ZUVH
- PAUSE WHEN POLYLINE DOES NOT FIND ZPT
- TIN COINCIDENT POINT DISTANCE
- READ MI Z HX LINE
- READ MI Z HX LINE RIDGE or MAX or RAISE
- READ MI Z LINE RIDGE or MAX | GULLY or MIN | HX
- READ MI Z SHAPE RIDGE or MAX or RAISE | GULLY or MIN or LOWER

TBC File

The following commands are supported when reading a TBC file into SMS:

- BC DATABASE – set the database to be used
- BC EVENT NAME – set the name of the event file to be used
- BC EVENT TEXT – set the text to be replaced by the event name
- READ GIS BC – read the boundary condition MIF/MID files
- READ GIS RF – read the rainfall MIF/MID files
- READ GIS SA – read the source flow MIF/MID files

The following commands are not supported and are written to an external file:

- GLOBAL RAINFALL AREA FACTOR
- GLOBAL RAINFALL BC
- GLOBAL RAINFALL CONTINUIN LOSS
- GLOBAL RAINFALL INITIAL LOSS
- READ GIS SA ALL
- READ GIS SA PITS
- READ GIS RF
- READ ROWCOL RF
- UNUSED HX AND SX CONNECTIONS

ECF File

The following commands are supported when reading a ECF file into SMS:

- DEPTH LIMIT FACTOR – set the depth limit for detecting instabilities
- MANHOLE DEFAULT C EXIT COEFFICIENT – set the K coefficient for automatic circular manholes
- MANHOLE DEFAULT LOSS APPROACH – set the loss approach for automatic manholes
- MANHOLE DEFAULT R EXIT COEFFICIENT – set the K coefficient for automatic rectangular manholes
- MANHOLE DEFAULT SIDE CLEARANCE – set the side clearance for circular and rectangular manholes (automatic)
- MANHOLE DEFAULT TYPE – set the type (circular, rectangular, no chamber, automatic) for automatic manholes
- MANHOLE K MAXIMUM BEND/DROP – set the K coefficient for energy loss when using Engelhund approach with automatic manholes

- MANHOLE MINIMUM DIMENSION – set the minimum dimension for circular and rectangular automatic manholes
- MANHOLES AT ALL CULVERT JUNCTIONS – set whether or not to automatically create manholes at culvert junctions
- MINIMUM NA – set the minimum surface area in all NA tables
- OUTPUT INTERVAL – set the output interval for ESTRY output
- PIT INLET DATABASE – read the specified inlet database file
- READ GIS NETWORK – read the network mif/mid files
- READ GIS TABLE LINKS – read the table link mif/mid files
- READ GISI WLL – read water level lines mif/mid files
- READ GIS WLL POINTS – read elevations and points from water level lines from mif/mid files
- STORAGE ABOVE STRUCTURE OBVERT – define how surface area is to be contributed to the NA table
- TIMESTEP – set the computation time step in seconds
- READ FILE – read the external file as an ECF file
- SET IWL – set the initial water level

The following commands are not supported and are written to an external file:

- BG DATA
 - BRIDGE FLOW
 - CHECK MI SAVE DATE
 - CHECK MI SAVE EXT
 - CREATE NODES
 - CS DATA
 - CSV FORMAT
 - CSV TIME
 - CULVERT ADD DYNAMIC HEAD
 - CULVERT CRITICAL H/D
 - CULVERT FLOW
 - DEFAULTS
 - END TIME
 - FIXED FIELD FLAGS
 - FLOW CALCULATION
 - FROUDE CHECK
 - HEAD RATE CREEP FACTOR
 - HEAD RATE LIMIT
 - HEAD RATE LIMIT MINIMUM
 - INTERPOLATE CROSS-SECTIONS
 - INTERPOLATE CULVERT INVERTS
 - LOG FOLDER
 - M11 NETWORK
 - MI PROJECTION
 - MINIMUM CHANNEL STORAGE LENGTH
 - MINIMUM NA PIT
 - MOMENTUM EQUATION
 - NA DATA
 - ORDER OUTPUT
 - OUTPUT FOLDER
 - OUTPUT TIMES SAME AS 2D
-

- PIT CHANNEL OFFSET
- READ MATERIALS FILE
- READ GIS IWL
- RELATIVE RESISTANCE
- S CHANNEL APPROACH
- SNAP TOLERANCE
- START OUTPUT
- START TIME
- STRUCTURE LOSSES
- TAPER CLOSED NA TABLE
- TRIM XZ PROFILES
- VEL RATE CREEP FACTOR
- VEL RATE LIMIT
- VEL RATE LIMIT MINIMUM
- VG DATA
- WEIR FLOW
- WLL ADDITIONAL POINTS
- WLL ADJUST XS WIDTH
- WLL AUTOMATIC
- WLL NO WEIRS
- WLL VERTICAL OFFSET
- WLLP INTERPOLATE BED
- WRITE CHECK FILES
- WRITE CSV ONLINE
- WRITE EMPTY MI FILES
- XS DATABASE
- ZERO CULVERT COEFFICIENTS

The following commands are ignored when read in by SMS because they are hardwired:

- APPLY ALL INVERTS – hardwired to “ON”
 - CONVEYANCE CALCULATION – hardwired to “ALL PARALLEL”
 - FLOW AREA – hardwired to “TOTAL”
 - WLL APPROACH – hardwired to “Method B”
-

TUFLOW 2D Geometry Components

A 2D geometry component groups grids and data that apply to the grid. The other kinds of data in a geometry component include coverages and TIN (scatterset). A TIN is used to modify the Z values. The purpose of each coverage is dependent upon the coverage type. Some coverages will modify z values and others will control other attributes. There can be multiple objects of the same type in a geometry component. When this happens, the data is applied in the order of the objects in the component. This means that the objects below in the project explorer are the ones that will control. This applies to Z as well as other attributes. Links can be rearranged by dragging them up or down within the 2D Geometry domain.

Coverages in a geometry component

TUFLOW 2D Geometry components associate grids with TUFLOW coverages that apply specifically to the domain. For example a grid component may contain an area property coverage that modifies the materials in the grid. Grids and coverages are represented as TUFLOW Project Explorer Links inside of a 2D Geometry component. Only one grid or grid frame coverage may be linked in each geometry component.

The following coverages can be put inside of a 2D Geometry domain

- Area Property Coverage
- Boundary Condition Coverage
- 2D Z Lines/Polygons (Simple)
- 2D Spatial Attributes Coverage
- 2D Z Lines (Advanced)

TINs (scatterset) in a geometry component

- In addition, Scatter Data can also be put inside a 2D Geometry domain to modify Z values. Unlike the other components, it has additional TUFLOW options that can be set in its link's right-click menu. The TUFLOW Options that can be set for how the scatter data modifies Z values are All, Add, Min and Max.
- Clip Regions for Read Tin z pts

Considerations for modifying Z data

As mentioned previously, the order links are listed matters for coverages and scatter data. This can be particularly useful for Z data but you have make sure you are applying the attributes in the correct order.

Suppose you have a situation where you want to incorporate a 2.0 m high fenceline and also raise a portion of the domain to model a proposed fill for a development. The fenceline is defined a z-line and the raised domain is a TIN (you could use a 2D z lines/polygons (simple) if it is a simple fill scenario). If the fenceline is above the TIN in the project explorer, anywhere that they overlap the TIN would take precedence and the fenceline would get wiped out. If the fenceline is below the TIN, the TIN elevations will get applied and then the fenceline will raise the elevations 2.0 m above the new elevations. Depending upon the scenario you want to represent, either option may be the preferred mechanism.

External files

- An external file can be read in as part of a geometry component. As previously mentioned, the order of links is important, and the external file behaves the same way as the z-modifying data. An external file is added by right-clicking on the geometry component and selecting "Add External File...". After a file has been selected, a tree item with the filename will appear at the bottom of the geometry component tree item group. This external file item can be moved around to the desired location and the filename can be changed by right-clicking and selecting **Browse...**

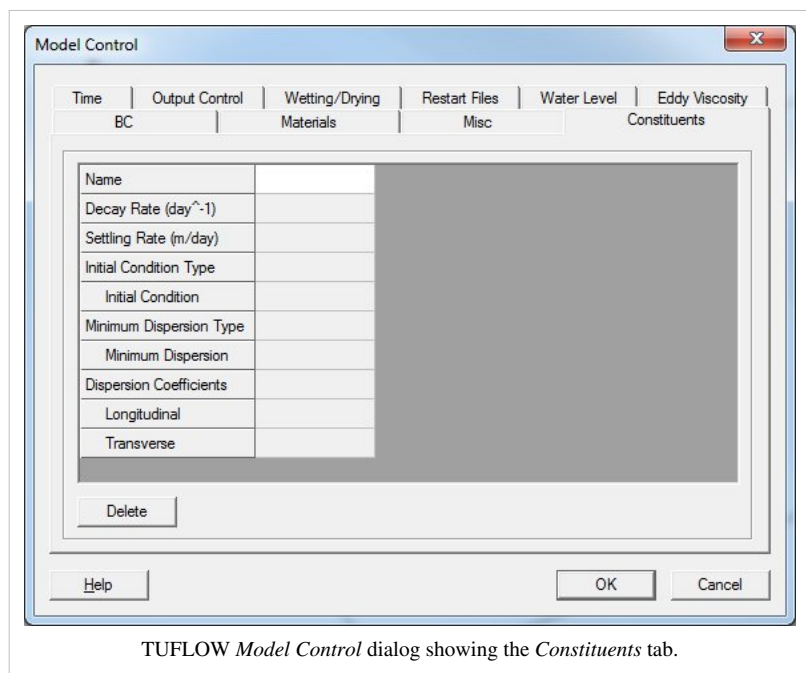
TUFLOW AD

SMS supports the TUFLOW Advection Dispersion (AD) module. This allows users to model constituents using TUFLOW.

Creating the Constituents

The constituent data is entered in the *Model Control* of the TUFLOW simulation. The *Constituents* tab allows the user to enter in the following parameters for each constituent:

- Name
- Decay rate (units of day^{-1})
- Settling rate (units of m/day)
- Longitudinal Dispersion Coefficient
- Transverse Dispersion Coefficient
- Initial Condition – Specified Value or Polygon by Polygon value
- Minimum Dispersion – Specified Value or Polygon by Polygon value



TUFLOW *Model Control* dialog showing the *Constituents* tab.

Boundary Conditions

For each boundary condition and constituent the user can provide a curve representing concentration at the boundary. This can be done by selecting the **Advection Dispersion BC...** button in the *properties* dialog for the arc or polygon.

External Link

- TUFLOW AD Documentation ^[1]

TUFLOW Boundary Conditions

TUFLOW Boundary conditions may be defined at nodes, arcs, or polygons in a Boundary Conditions Coverage.

Boundary Condition Locations

1D Boundary Conditions at Nodes

Boundary conditions defined at points are 1D boundary conditions and must be placed at the same location as a 1D boundary node. (See Snapping Feature Objects) The boundary condition information is specified in the *BC Node Attributes* dialog.

2D Boundary Conditions at Arcs

Arcs can be used to define 2D Boundary conditions applied to the 2D domain. The BC attributes are specified in the *BC Arc Attributes* dialog.

Node Attributes

Type

Inlet - Rectangular

☐ User Specified Id

rzi_node_0063

☐ Nodal Area Curve

Curve undefined.

Additional Nodal Area

0.0

☐ Ground Level

-99999.0

☐ Invert (Bottom)

-99999.0

Number of inlets

1

☐ Manhole

Manhole Attributes...

Shape

Width

1.0

Height

1.0

Contraction Coefficients

Width

0.6

Height

0.9

Losses

Entry

0.5

Exit

1.0

☐ Create Connection to 2D domain (SX)

☐ Force cell z to at or below node z (invert/ground level) (SXZ)

☐ Lower 2D cell elevation (SXL) 0.0

☒ Don't alter cell Z

Method to choose additional cells

Use neighboring cells of higher elevation + lower to invert (grade)

Method to determine number of connected 2D cells

Automatic plus

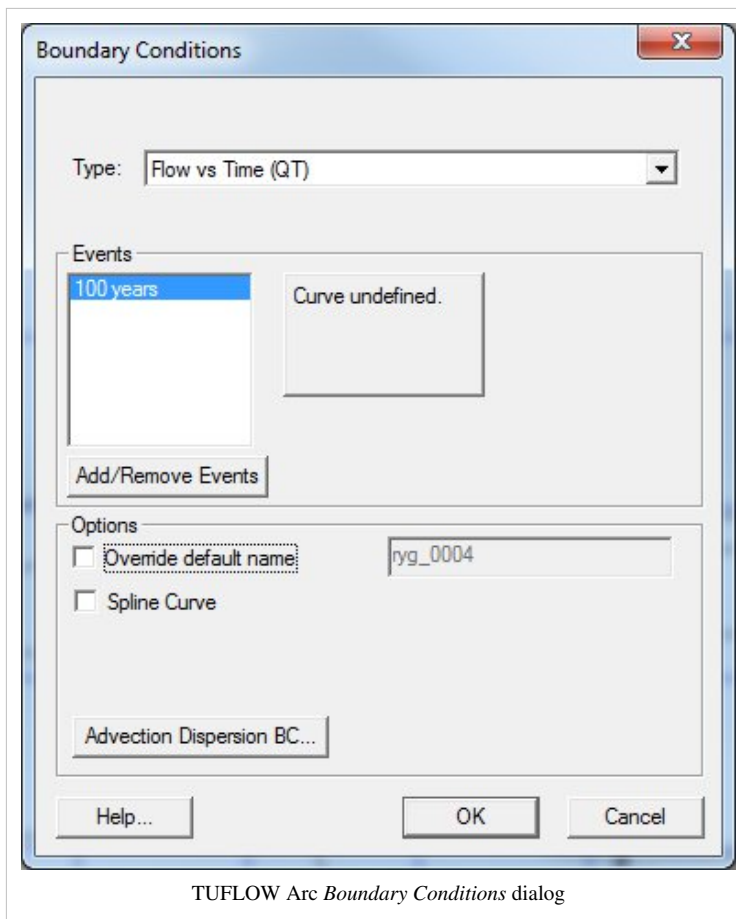
Number of Additional 0

Help...

OK

Cancel

TUFLOW Boundry Condition Nodal Attributes dialog



2D Boundary Conditions for Polygons

Polygons can be used to define rainfall applied to 2D domains or to specify active/inactive information for 2D domains. The *BC Polygon Attributes* dialog specifies the type of information stored with the polygons.

Convert GIS Rainfall Data to TUFLOW BC Polygons

GIS files containing TUFLOW rainfall data may be imported into SMS and a Boundary Condition coverage with the data assigned to polygons will be created. The data for each polygon should be contained in the following columns: 8 STARTYEAR – year of first data point (integer)

- STARTMONTH – month of first data point (integer)
- STARTDAY – day of first data point (integer)
- STARTHOUR – hour of first data point (integer)
- STARTMIN – minutes of first data point (integer)
- STARTSEC – seconds of first data point (integer)
- TIMESPAN – timespan between data point in minutes (integer)
- NUMRAINPTS – number of data points (integer)
- PT0 – data for the first point (double)
- PT1, PT2,... – data for each point

When converting GIS rainfall data to TUFLOW rainfall polygons, the user will be prompted to set a reference time. Each rainfall data point will be stored as total mm for each interval of time (hrs) from the reference time.

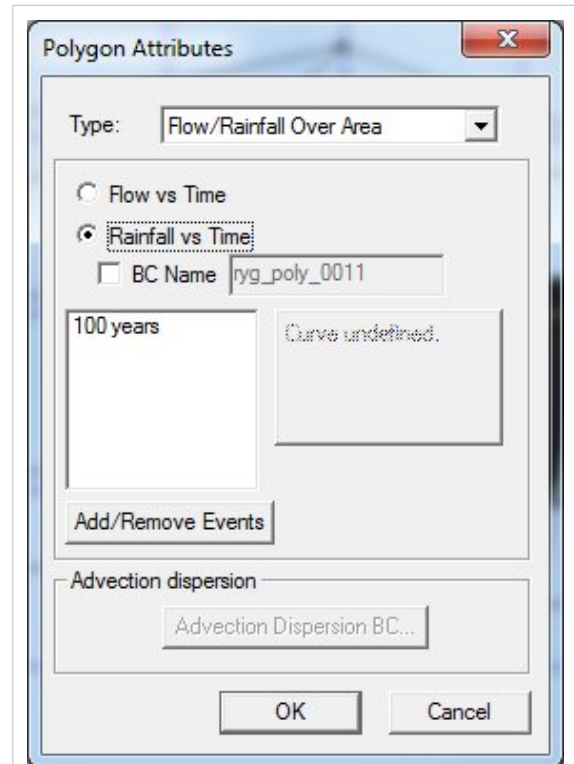
Events

Every boundary condition should be setup for each event that will be used with the boundary condition. The events can be defined by clicking on the **Add/Remove Events** button. The list of events is common to all the boundary conditions in the project.

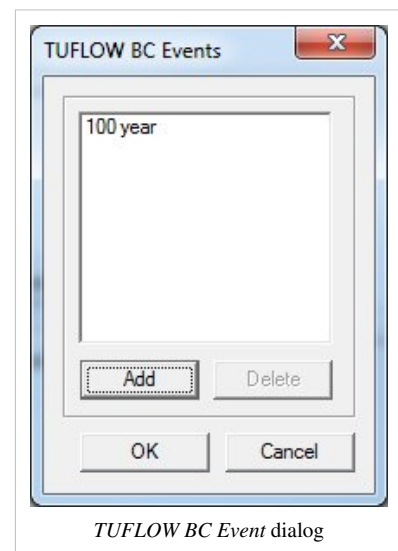
Options

Depending upon the type of boundary condition, there are several additional boundary condition options.

- Override Component Identifier – SMS creates a default unique boundary condition identifier based upon the coverage and id of the arc. You can override this value to more easily find boundary condition information in the TUFLOW check, and log files.
- Spline curve – If this option is on, TUFLOW will smooth the boundary condition curve using a spline algorithm. This should generally only be used for smooth, cyclic boundary conditions such as a tidal boundary.



TUFLOW Boundary Condition Polygon Attributes dialog



TUFLOW BC Event dialog

Boundary Condition Types

The types of boundary conditions available depend upon whether the boundary condition is at nodes, arcs, or polygons.

Boundary Conditions

Type	Available Locations	Description
Not used	Points, Arcs	No boundary condition exists at the location.
Flow vs Time	Points, Arcs	Sets the flowrate as a curve through time.
Flow vs Wse	Points	Sets the flowrate based upon a relationship with the water surface elevation.
Wse vs Time	Points, Arcs	Sets the water surface elevation as a curve through time.
Wse vs Flow	Points, Arcs	Defines a relationship to determine the water surface elevation from a flowrate. The relationship can be specified or TUFLOW can compute it using water surface slope.
1D Flow/2D Water Level Connection (HX)	Arcs	Defines the interface between a 1D and 2D domain. See Combining 1D and 2D Domains.
Non-directional Flow vs Time	Arcs	Defines a flowrate that enters along each cell of the boundary.
Non-directional Flow vs Wse	Arcs	Defines a flowrate that enters along each cell of the boundary based upon a relationship to the water surface elevation.
Flow Source from 1D model (SX)	Arcs	Defines an interface between a 1D and 2D domain which is often useful for culverts rather than using the 1D Flow/2D Water Level Connection.
Flow/Rainfall over Area	Polygons	Defines either an amount of flow entering the polygon (applies to all wet cells or lowest cell if none) or rainfall amounts (applies to all cells in the polygon).
Cell Codes	Polygons	Specifies whether cells are active or inactive and whether or not they will be included in the mesh output files.

Related Topics

- [TUFLOW](#)
- [TUFLOW Coverages](#)

TUFLOW Check Files

TUFLOW provides excellent feedback for finding and fixing model errors. This feedback exists in several files. All of these files are prefixed by the name of the simulation that you are running and the filename ending identifies the type of data contained in the file.

Log Files

In the \runs\log directory (within the TUFLOW files for your simulation), are the log files for individual run.

The most useful file for finding model problems is the _messages.mif file. This file contains spatially located messages including errors, warnings or checks (listed in descending order of severity). These can be read into the GIS module of SMS. This provides quick feedback of model errors and the exact locations that the errors occur. This is very useful for pinpointing model setup problems and should be the first step in fixing model issues.

The *.tlf file contains information (in ASCII format) about how the run proceeded. It includes information on the parameters and settings used, the files that were read and used, as well as a copy of the screen information that is written as TUFLOW runs. This file can be used to verify the model is setup and is running as expected.

Check Files

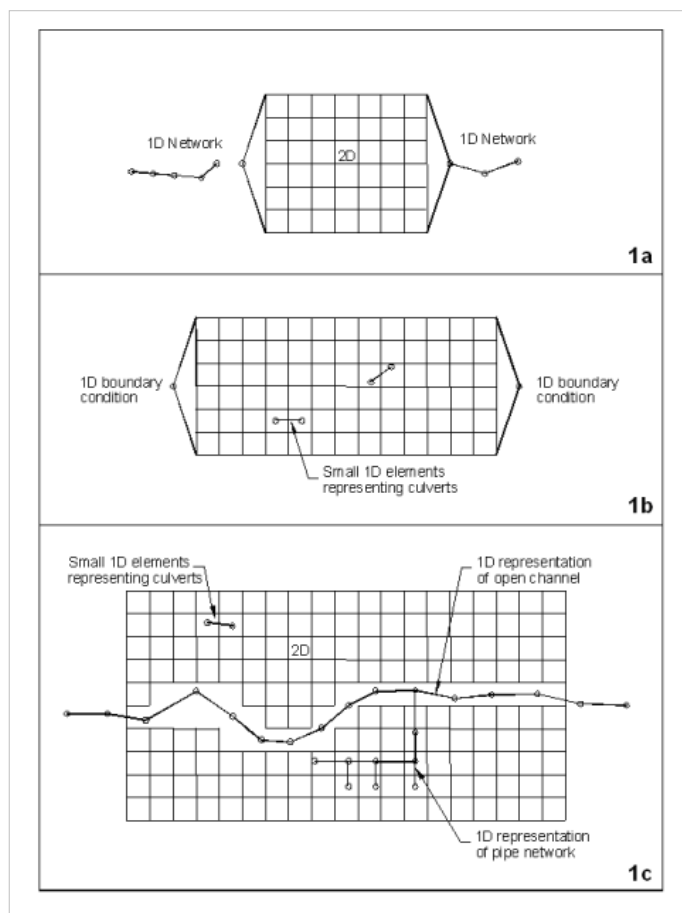
In addition to the log files mentioned previously, TUFLOW writes additional files that can be used to verify model setup. These files are in the directory labeled "\\check" and are only written if the 'Write Check Files' toggle is checked in the misc tab in the simulation model control dialog (this is the default). Many of the check files are in MapInfo MIF/MID files that can be read into the SMS GIS module where the info tool can be used to see the values at individual locations.

TUFLOW generates potentially lots and lots of check files. A description of the check files and what they contain can be found in the TUFLOW manual.

TUFLOW Combining 1D and 2D Domains

1D and 2D Domains can be linked in three different ways.

1. A 2D domain within a 1D domain (1a)
2. 1D elements representing culverts or pipe networks (1b or 1c)
3. 1D cross-section based domain for river channel and 2D domain for floodplain (1c)



Linking 1D Pipe network within a 2D network

The easiest way to link a 1D pipe network to a 2D network is to double click on one of the nodes in the network coverage and define an inlet (connection to the 2D domain). The inlet can be circular or rectangular and the shape and losses can be specified.

Linking 1D domain and 2D domains

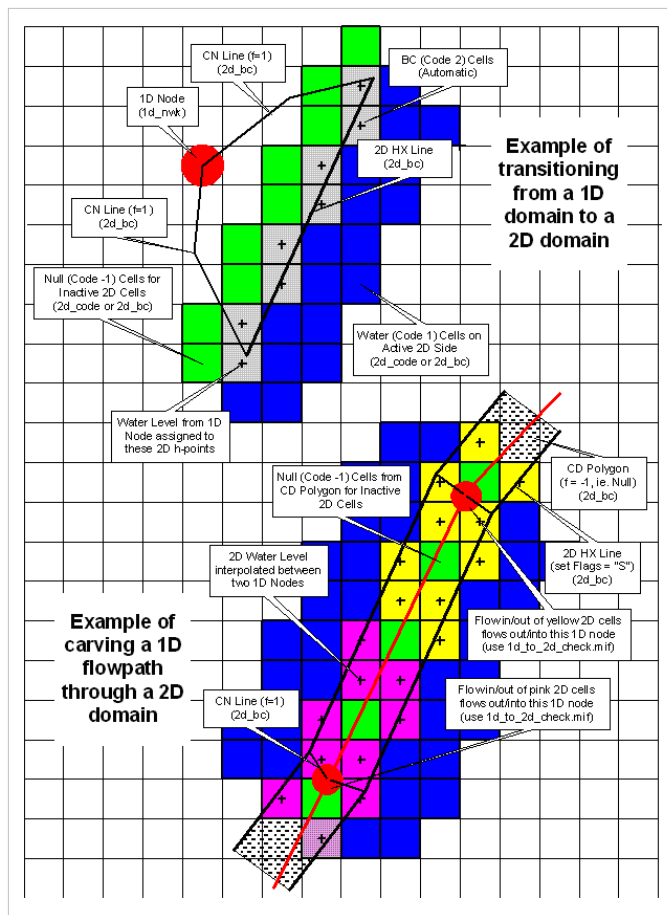
Except for pipe networks defined above, linking a 1D network to a 2D network requires several coverages.

- Network coverage – contains the 1D channels
- BC coverage – 1D Flow/2D Water Level Connection (HX) or Flow source from a 1D model (SX) arcs define the interface between the 1D and 2D domains.
- 1D/2D Connections – Arcs connect 1D nodes to ends of 1D Flow/2D Water Level arcs or Flow source arcs. When used with Flow source arcs only 1 connection arc is needed where the

1D Flow/2D Water Level arcs require 2 arcs (one to each end of the arc).

Arcs in the connections coverage must snap to the 1D network nodes and the endpoints of the 1D Flow/2D Water Level Connection arcs. The snapping option setup in the map tab of the preferences dialog should be used to ensure that the nodes are placed correctly.

The following image shows how 1D/2D linkages are setup both for both items 1 and 3 above.



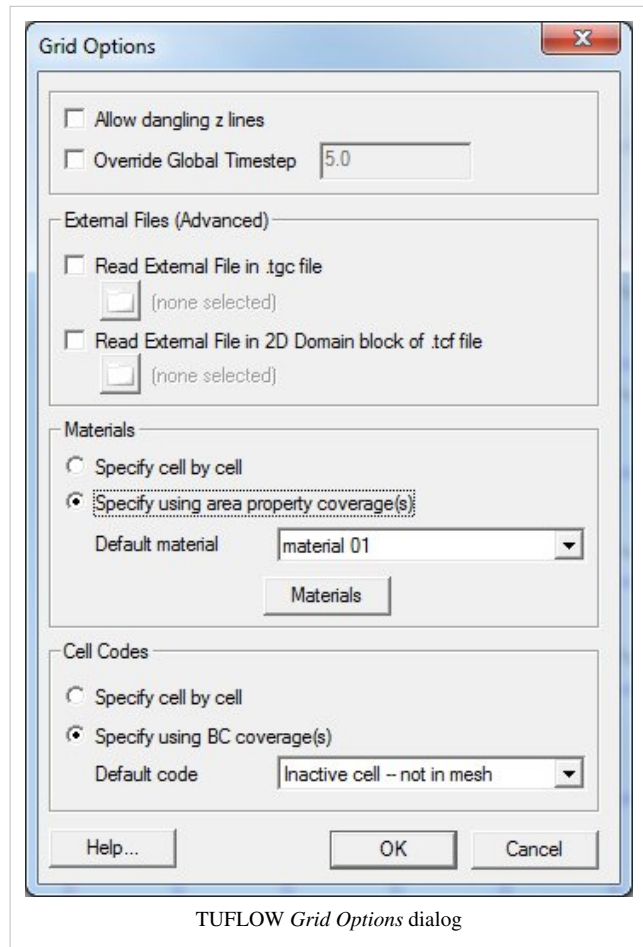
TUFLOW Grid Options

The TUFLOW *Grid Options* dialog (right-click on 2D geometry component in the project explorer and select **Options**) is used to specify how TUFLOW will handle the grid data. Some of the options deal with what types of data will be specified on the grid cells. This data can alternatively be specified using coverages and this has several advantages including size of the data, speed of execution, and more opportunities for reuse.

The general options include:

- Allowing dangling z lines – This turns on/off the TUFLOW command to check for dialing z lines (see the TUFLOW manual for description).
- Override Global Timestep
- External Files (Advanced)
 - Read External File in TGC file – This can be used to read an external file from the TUFLOW TGC file.
 - Read External File in 2D Domain block of TCF file.
- Materials
 - Specify cell by cell
 - Specify using area property coverage(s)
- Cell Codes
 - Specify cell by cell
 - Specify using BC coverage(s)
- Time step – Since the required time step depends heavily on the size of grid cells, it is often convenient for each grid to have a unique time step when using multiple grids in a simulation. These controls allow the user to override the global time step on a grid by grid basis.

The rest of the dialog is used to tell SMS what kind of data to write from the grid in SMS. The optional data types are materials and cell codes. Materials may be specified using an area property coverage and cell codes can be specified from polygons in a TUFLOW BC Coverage. If a grid (instead of a TUFLOW grid extents coverage) is being used in the geometry component, these properties can also be specified cell by cell. If using an area property coverage, a default material is specified for any areas not inside of a polygon.



Related Topics

- TUFLOW
- TUFLOW Menu

TUFLOW Inlet Database

TUFLOW supports several types of inlets that connect 1D pipe network domains to 2D overland domains. These include circular, rectangular, and weir. TUFLOW now also supports the ability to define depth vs flow curves to allow for more flexibility. The user defined inlet information is stored in an inlet database.

The inlet database stores information about the user defined inlet types. This information includes the flow area, width, and the depth vs flow curve. The flow area is used to compute the velocities in the inlet channel and the width is used to determine how many 2D cells to connect the inlet to. For more information about how TUFLOW uses the inlet database, see the TUFLOW documentation.

To define the Inlet Database

- Create a 1D network coverage and an arc.
- Double-click on the end node of an arc (which is where inlets are defined)
- Change the inlet type to "Inlet — From Database"
- Click on the button labeled "Inlet Database..."
- In the dialog that comes up, define the type (name), flow area, flow width, and curve for each of the inlets you wish to define.

To use an inlet from an Inlet Database

- Create a 1D network coverage and an arc.
- Double-click on the end node of an arc (which is where inlets are defined)
- Change the inlet type to "Inlet – From Database"
- Select the inlet type by name in the combo-box labeled "Inlet Database Name."

Importing/Exporting the Inlet Database

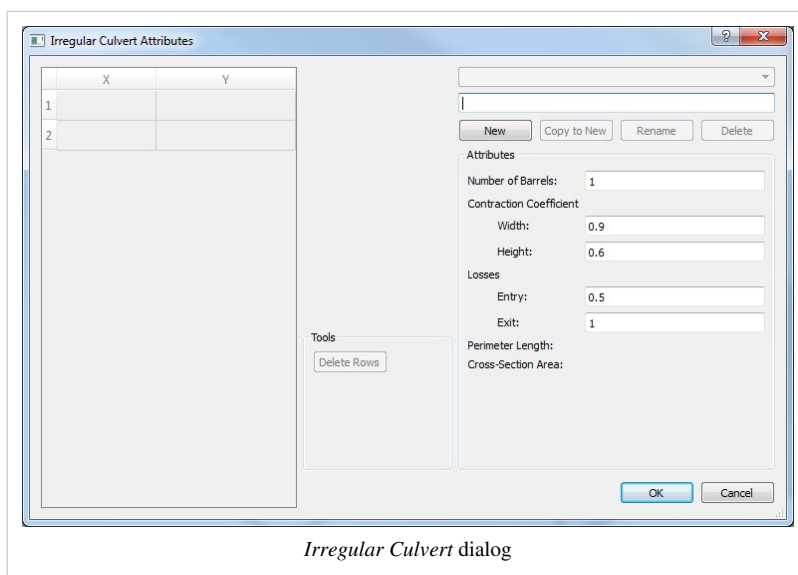
From the inlet database dialog, it is possible to export or import the database in to the format used by TUFLOW (two or more csv files). This format is very easy to create or use for graphs using a spreadsheet program. For more information on this format, see the TUFLOW documentation.

TUFLOW Irregular Culverts

SMS now supports irregular culverts for TUFLOW.

Creating the Irregular Culverts

In the network coverage, change the type of an arc to "Irregular Culvert" in the *Channel Attributes* dialog. By clicking on **Attributes** at the bottom of the *Channel Attributes* dialog, the user can specify what shape the culvert is. The drop-down specifies what polygon will be used for this arc. New polygons can be created and named. The table specifies the points of the polygon in clock-wise order. Please note that negative values cannot be entered for y-values. The number of barrels, contraction coefficients, and losses can be entered in this dialog, similar to other culverts. The perimeter length of the polygon, and its area, are given for convenience.



When the TUFLOW simulation is exported, an HW table will be generated for each unique irregular culvert shape in use.

TUFLOW Linking 2D Domains

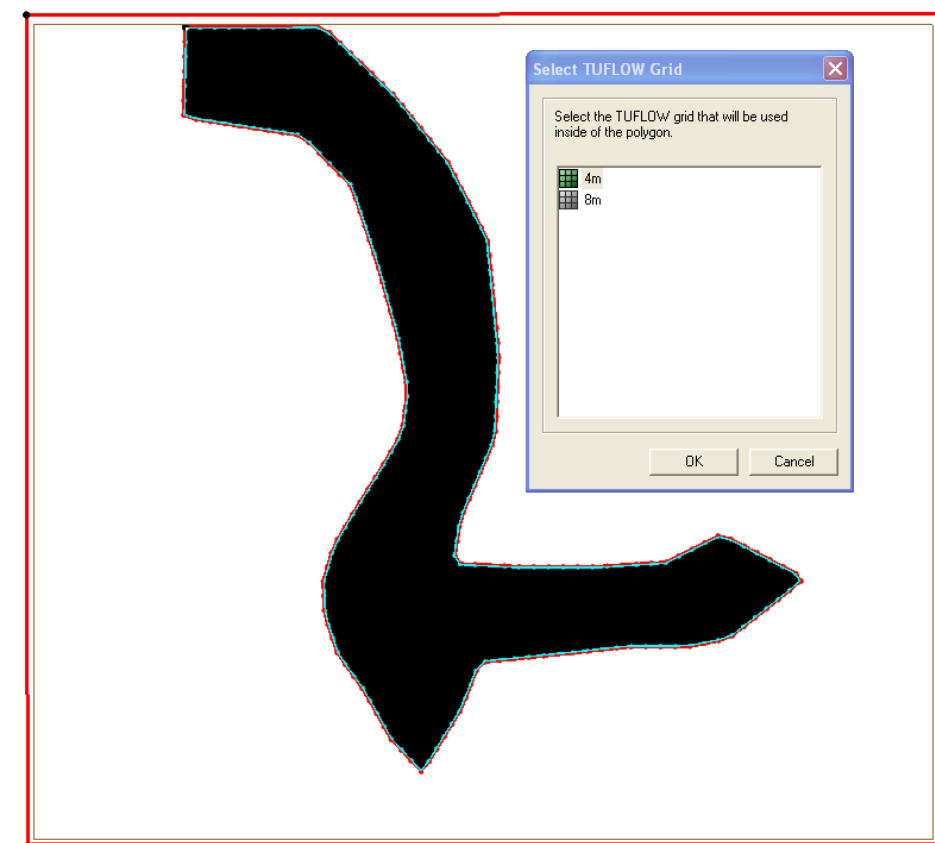
Any number of grids of varying sizes and/or orientations may be used in TUFLOW. Using multiple 2D domains requires that multiple 2D domains are licensed in TUFLOW.

To setup multiple 2D domains in SMS:

1. Create multiple TUFLOW grids and associated grid components.
2. Define a 2D/2D Linkage coverage and attributes (see below).
3. Add all the 2D domains being used and the 2D/2D Links Coverage to the simulation.

There are a few important items to keep in mind when setting up a 2D/2D Linkage coverage.

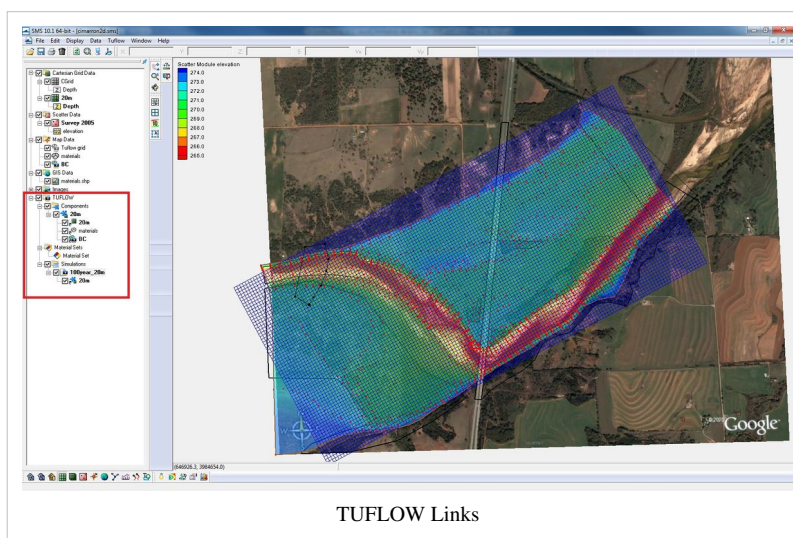
1. You need a polygon enclosing each domain you wish to use. In the case below, you would need a polygon that covers the area outside of the main channel polygon.
2. Each polygon needs to be linked to a specific 2D domain. This is done by double clicking on each polygon and specifying the appropriate 2D grid.
3. Set the vertex spacing on the arcs that join multiple 2D domains. The spacing along these arcs determines how flows move between the domains. The rule of thumb to start with is to have the vertex spacing equal to 1.5 times the larger cell size.



TUFLOW Links

A TUFLOW simulation is comprised of a grid, feature coverages, and model parameters. SMS allows for the creation of multiple simulations and each includes “links” to these items. Links are like shortcuts in windows and they call for the geometry component as well as each coverage used that is not part of the geometry component.

By using links the data is not duplicated; it just knows where to go to get the data that has to be used for each simulation. The use of links allows



TUFLOW Links

these items to be shared between multiple simulations. Multiple simulations can be created if some components need to be changed to create different scenarios and links would call for the same components used in previous simulations. Simulations also store the model parameters used by TUFLOW.

Links are located in the Project Explorer under TUFLOW Simulation once a simulation has been created.

TUFLOW Manholes

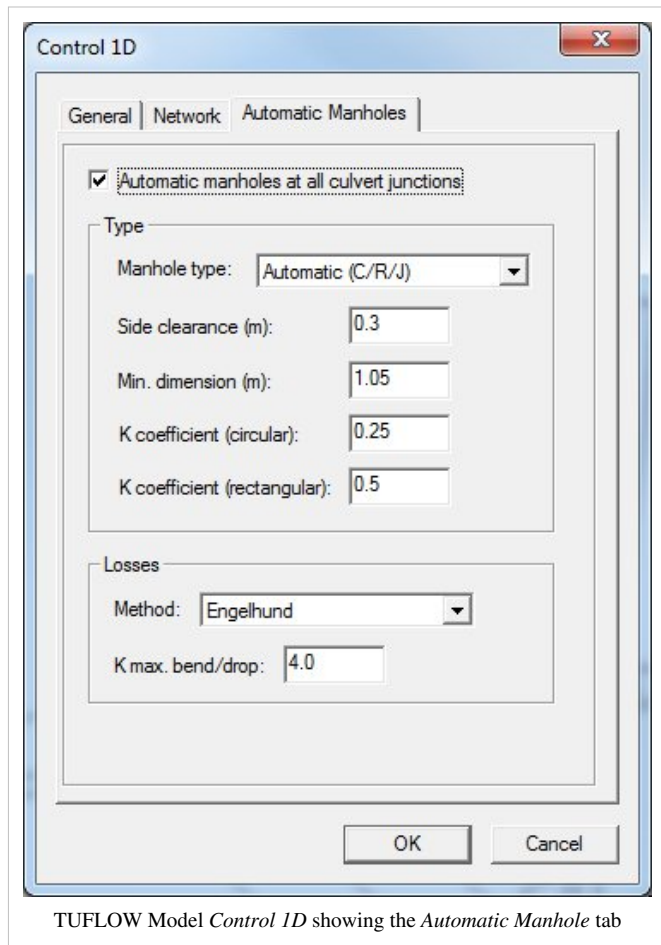
Automatic Manholes

Global manhole attributes can be set up for a TUFLOW simulation in the TUFLOW 1D model control. TUFLOW will then automatically create manholes when the following criteria are met:

- There is at least one incoming and one outgoing culvert – a culvert is a C, I or R channel.
- There are no open channels connected (ie. no bridges, weirs or any other channel that is not a culvert).
- Pit channels can be connected, but are not included in any of the calculations for determining manhole energy losses.

The following global attributes are set:

- **Automatic Manholes at All Culvert Junctions** – whether or not to have TUFLOW automatically create manholes.
- **Manhole Type**– circular, rectangular, junction, or automatically determined.
- **Side Clearance** – for circular manholes, the default side clearance from the side of the largest culvert to the side of the manhole chamber. For rectangular manholes, the side clearance from the side of the culvert to the side of the manhole chamber.
- **Minimum Dimension** – minimum diameter for circular manholes and minimum width and length for rectangular manholes.
- **C Exit Coefficient** – K coefficient for flow out of circular manholes.
- **R Coefficient** – K coefficient for flow out of rectangular manholes.
- **Loss Approach** – loss approach to be used (none, Engelhund, Fixed).
- **K Maximum Bend/Drop** – maximum K energy loss coefficient that can occur for the sum of the loss coefficients for bends and drops at a manhole when using the Engelhund approach.



TUFLOW Model Control 1D showing the Automatic Manhole tab

Manual Manholes

The global manhole parameters can be overridden by creating manholes in a TUFLOW 1D Network coverage. Toggling on *Manhole* and selecting the **Attributes...** button in the 1D Network node properties dialog allows the user to set the following parameters for manually created manholes:

- **ID** – set a name for the manhole
- **Type** – circular, junction, rectangular
- **Loss Approach** – loss approach to be used (none, Engelhund, Fixed)
- **Invert** – bottom or bed elevation of manhole
- **Flow Width** – The flow width in meters (this is the diameter for circular manholes)
- **Flow Length** – The flow length in meters (not used for circular manholes)
- **Additional Storage Surface Area** – usually set to zero
- **K Fixed** – Set the fixed component of the calculated manhole loss coefficient
- **Km** – exit coefficient used by Engelhund approach
- **K Bend Max** – Upper limit of the combination of K theta and K drop used by Engelhund approach

If a TUFLOW 1D Network coverage has manholes defined, the manholes will be written out as their own layer when exporting TUFLOW files.

(TUFLOW Manual 2010)

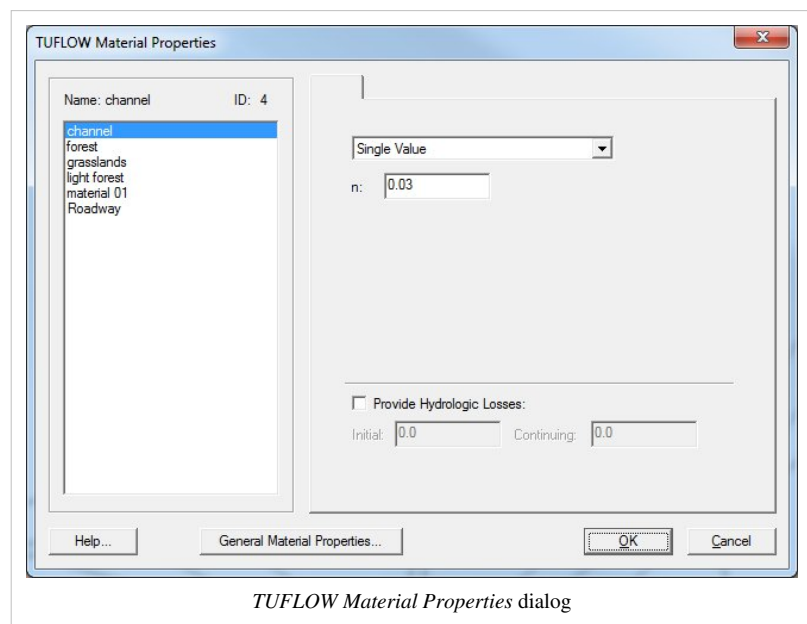
TUFLOW Material properties

Material properties include roughness and hydrologic losses (if used). TUFLOW uses a manning n value for roughness which can be specified as a single parameter or it can vary with depth. Hydrologic losses include an initial loss (in mm) and a continuing loss rate (in mm/hr). Hydrologic losses are generally only necessary when using direct rainfall on the model domain.

Material Sets

Multiple definitions of material properties can exist in a project. Each definition includes the material

properties for each material in SMS and is referred to as a "material set." A material set can represent roughness for different conditions such as winter/summer conditions or can be used in a sensitivity analysis. To build a material set, the TUFLOW solution must already be built. Right-click on the *project explorer* item labeled "Material Sets" and choose **New Material Set**.



Editing Material Properties

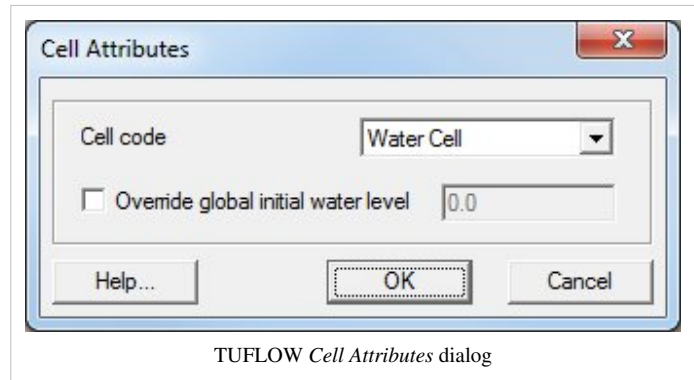
Material properties can be edited by right-clicking on the material set in the project explorer and choosing **Properties**. This will bring up the *TUFLOW Material Properties* dialog.

TUFLOW Menu

The following menu commands are available in the TUFLOW Menu:

- **Cell Attributes**

This command is enabled when one or more cells on a TUFLOW grid are selected. The attributes that can be set using this command are the cell code and the option to override the global initial water level (set in the 2D Model Control). The cell code (water cell, inactive cell – in mesh, inactive cell – not in mesh) for the selected cells will only be used if *Specify cell by cell* is selected in *Cell codes* section of the *TUFLOW Grid Options* dialog.



- **Assign Material Type**

This command is enabled when one or more cells on a TUFLOW grid are selected. The command will open the *Materials Data* dialog. The *Materials Data* dialog will allow the user to select a material to assign to a cell. The material set for the cell will only be used if *Specify cell by cell* is selected in the *Materials* section of the *TUFLOW Grid Options* dialog.

The following menu commands are available in the *TUFLOW Simulation* Menu:

- **Delete**

This command will delete the TUFLOW simulation.

- **Duplicate**

This command will create another copy of the simulation.

- **Rename**

This command will allow you to rename the simulation.

- **Export TUFLOW files**

SMS will save the TUFLOW input files in the TUFLOW directory for that simulation. If one does not already exist, the user will be prompted to select a location.

- **Launch TUFLOW**

SMS will launch TUFLOW using input files that have previously been exported.

- **Save Project, Export and Launch TUFLOW**

SMS will save the project, write out the TUFLOW input files, and launch TUFLOW.

- **2D Model Control**

This command will bring up the *2D Model Control* dialog where various model parameters are set. See *TUFLOW 2D Model Control* page.

- **1D Control**

This command will bring up the *Control 1D* dialog where various model parameters are set. See *Model Parameters* page.

- **Model Check**

This command will cause SMS to perform a check for invalid settings that could prevent TUFLOW from running properly. If errors are found, they will be reported in the *Model Check* dialog with information on how to correct them.

Related Topics

- TUFLOW

TUFLOW Model Parameters

Run control parameters are divided into two sections for TUFLOW simulations. The main model control dialog and the model control parameters that apply specifically to the 1D portions of the TUFLOW run. Right-click on the TUFLOW simulation and choose the appropriate menu command to edit either set of model parameters.

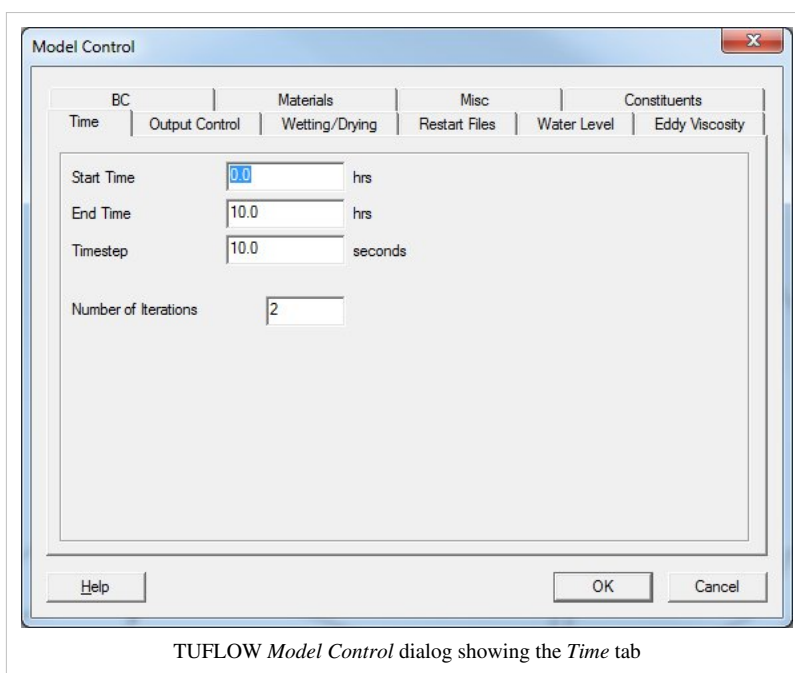
This document will try to highlight frequently used model control parameters. Please see the TUFLOW documentation for a description of those items left out or for more information on specific items.

Model Control

The main *Model Control* dialog sets parameters for the run in general and some 2D specific controls.

Time

This page is used to set the run times for the simulation. The optimum time step size is determined by the courant number. The rule of thumb is that the time step should be about half as many seconds as the grid cell size in meters. So a grid with 10 meter cells, should run with a 5 second time step. The time step may be reduced to increase stability, but generally problems indicate another type of problem.



TUFLOW *Model Control* dialog showing the *Time* tab

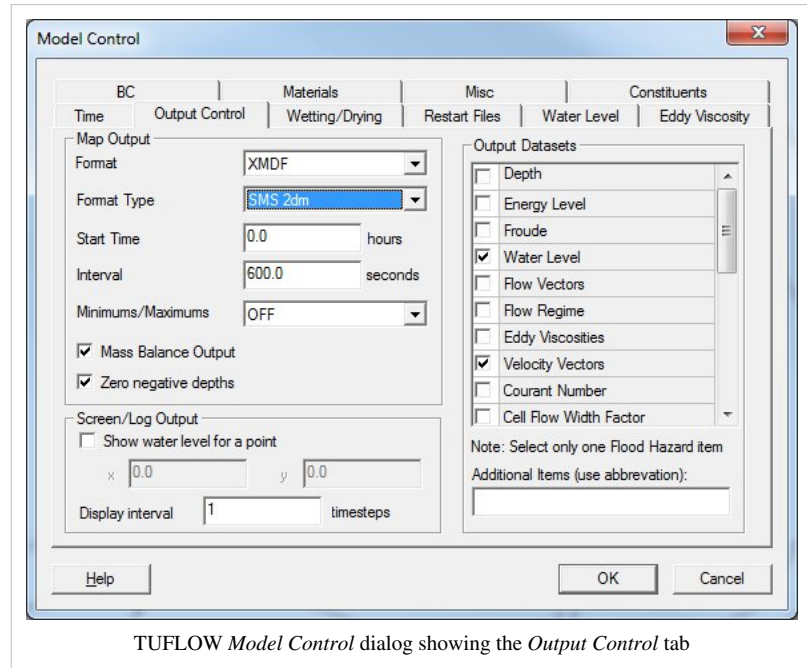
Output Control

This *Output Control* tab allows the user to specify what two dimensional (spatially varied) data should be saved in the TUFLOW run. The tab is broken down into three sections. These include the *Map Output*, *Output Datasets* and *Screen/Log Output* sections.

"Map Output"

This section allows the user to specify the format and frequency of output created by TUFLOW. The user specifies:

- **Format** – Choose between XMDF and binary data (*.dat) file format or both. (Map Output Format line in the *.tcf file)
- **Format Type** – Choose between normal grid resolution (SMS 2dm), double normal grid resolution (SMS high resolution), or reaveraged grid resolution (SMS low resolution fancy). (Map Output Format line in the *.tcf file)
- **Start Time** – Choose the simulation time in hours at which spatially variable dataset output starts. (Start Map Output line in the *.tcf file).
- **Interval**: Choose the time interval in seconds between times steps in spatial datasets saved by TUFLOW (Map Output Interval line in the *.tcf file).
- **Minimums/Maximums** – TUFLOW can compute minimum and maximum values or only the maximum values for the simulation. This can be very handy in visualizing flood extents. (Store Maximums and Minimums line in the *.tcf file).
- **Mass Balance Output toggle** – turn this toggle on to have TUFLOW save the mass error (%) or cumulative mass error (%). The specific option must also be turned on in the "Output Datasets" section. (Mass Balance Output line in the *.tcf file).
- **Zero negative depths** – By default, TUFLOW zeros depths that are computed as negative values at cell locations. If you turn this off, and setup the display options in SMS to only show positive depths you can see partially wet cells. (Zero Negative Depths in SMS line in the *.tcf file).



TUFLOW Model Control dialog showing the *Output Control* tab

"Output Datasets"

This section controls which datasets should be created by TUFLOW. The list is long and getting longer with each revision of TUFLOW. Select the toggle next to the desired items. (Map Output Data Types line in the *.tcf file).

"Screen/Log Output"

By default TUFLOW outputs to the screen information for each time step. This information is also written to a log file (*.tlf). TUFLOW supports various options for this type of run time feedback that are controlled in this section of the output control tab.

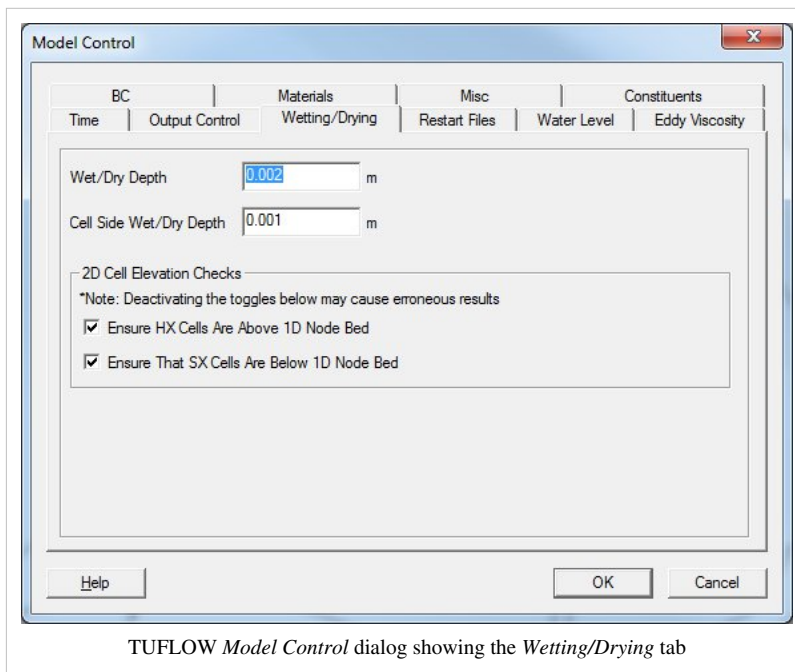
- **Show water level for a point toggle and location** – This option allows the user to instruct TUFLOW to display the water level on the screen for the cell located at the specified location. (Display Water Level line in the *.tcf file).

- Display interval – this control allows the user to decrease the frequency of the output information during a TUFLOW run. A value of 0 or 1 outputs at each time step. A larger positive value decreases the frequency to one in the specified number of time steps. A value of -2 suppresses all output except negative depth warnings and a value of -3 suppresses all time step display (Screen/Log Display Interval line in the *.tcf file).
- The user or developer can now specify additional text in the TUFLOW model control output tab which will be treated as custom output types. This will be used by developers and users to specify output types that are not yet supported directly in the interface.

Wetting/Drying

See the FESWMS manual for information on these options. Generally the default options are fine.

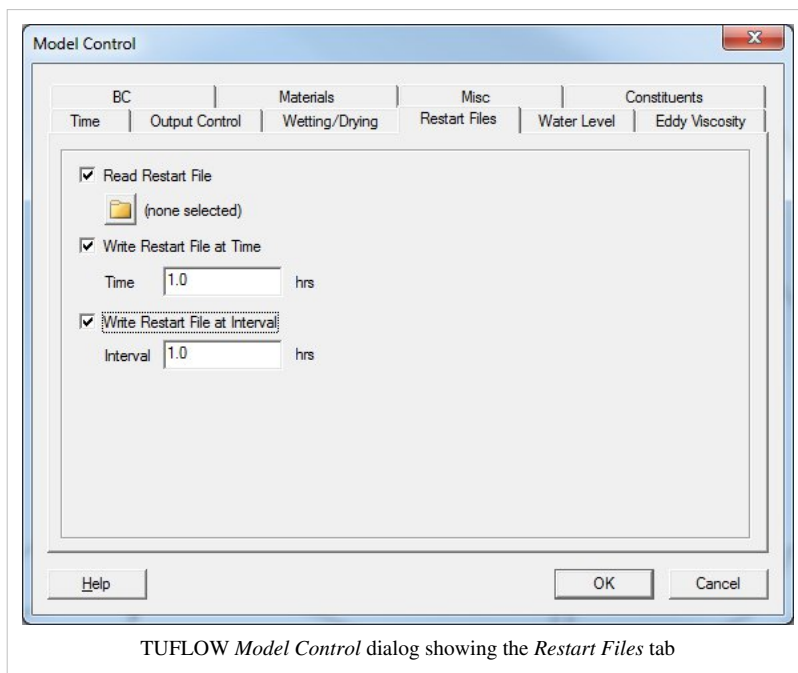
- Wet/Dry Depth
- Cell Side Wet/Dry Depth
- 2D Cell Elevation Checks
 - Ensure HX Cells Are Above 1D Node Bed
 - Ensure That SX Cells Are Below 1D Node Bed



TUFLOW Model Control dialog showing the Wetting/Drying tab

Restart Files

Restart files can be used to start a simulation part way through a run. They are generally used to generate an initial condition for several simulations that start with the same boundary conditions but are later changed for different events. For example to evaluate 10, 20, and 100 year events in a tidal area it might be useful to run a couple tidal cycles of a normal event and save these to a restart file. The restart file could be used with the 10, 20, and 100 year events without rerunning the first couple tidal cycles.

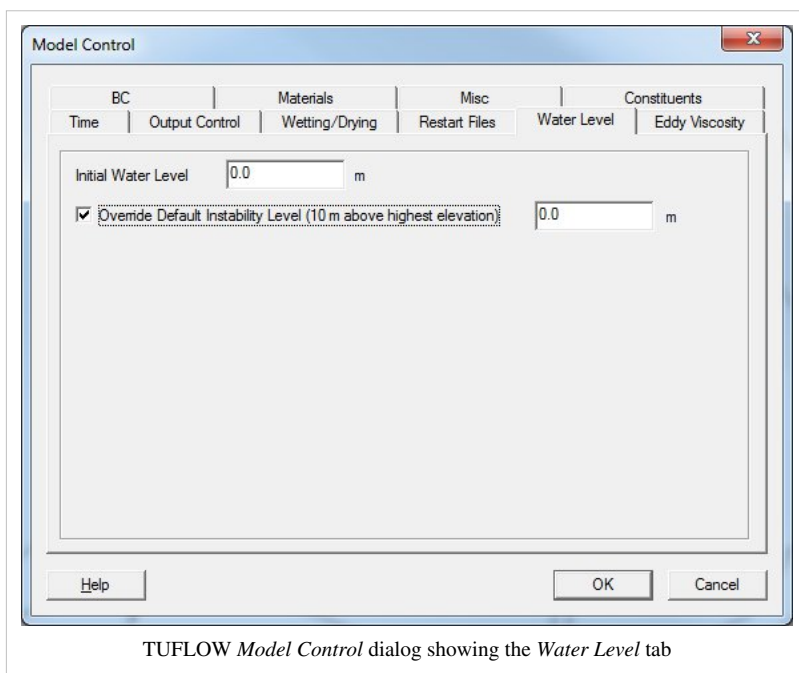


TUFLOW Model Control dialog showing the *Restart Files* tab

- Initial Water Level – All models

need to have an initial water level. Any cells with a lower elevation than the initial water level will start out wet and the rest will start dry. A TUFLOW model does not require a fully wet domain when starting like FESWMS or RMA2. It is generally a good idea for the initial water level to match the downstream boundary condition at the start of a run. The global initial water level can be overridden on a local level using a 2D Spatial Attributes Coverage.

- Override default instability level – By default TUFLOW will assume that if the water rises 10 m above the highest elevation in the grid then the run is unstable. This value may be overridden to allow the water level to get higher than this without causing TUFLOW to abort.



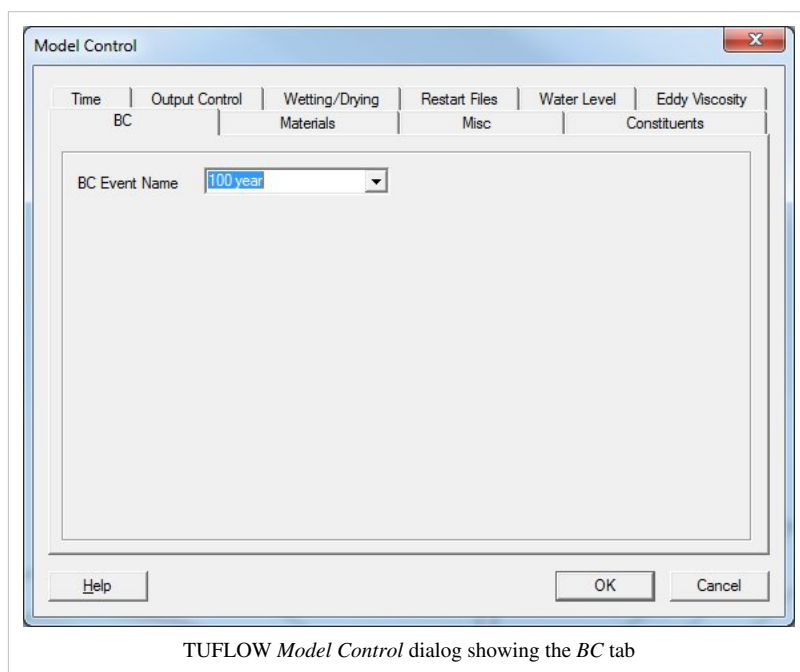
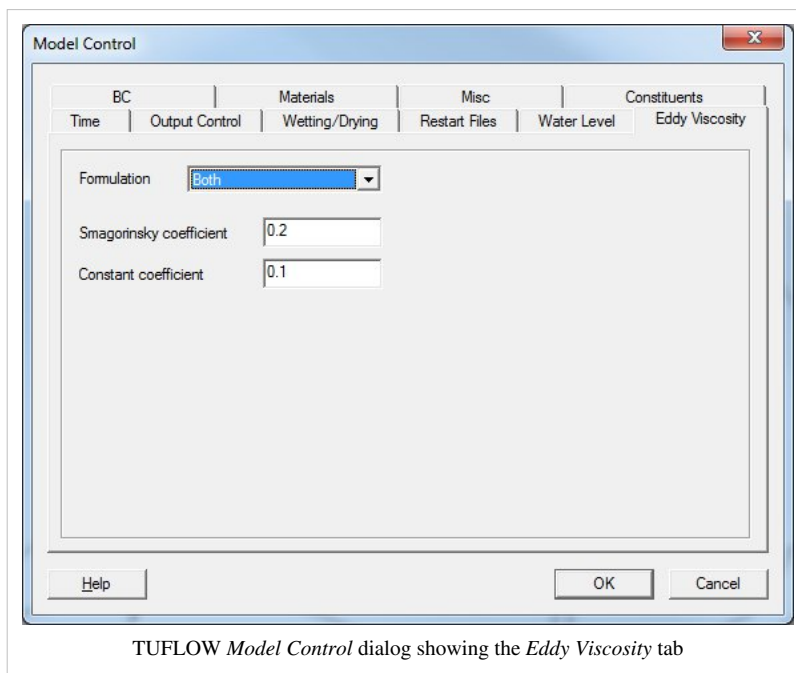
TUFLOW Model Control dialog showing the *Water Level* tab

Eddy Viscosity

Eddy viscosity is used to compute energy losses due to turbulence not modeled at the scale of the simulation. TUFLOW supports specification of a constant eddy viscosity or using Smagorinsky specification. When using the Smagorinsky specification, the coefficient is generally between 0.06 and 1.0. It is recommended to use the default eddy viscosity of 1.0 when using constant eddy viscosities.

BC

- BC Event Name – The boundary condition event to use is specified.
- SA Minimum Depth – This value sets the minimum depth a wet cell must have to apply an SA inflow.
- SA Proportion to Depth – This allows the user to set whether or not to proportion SA inflows according to the depth of water.

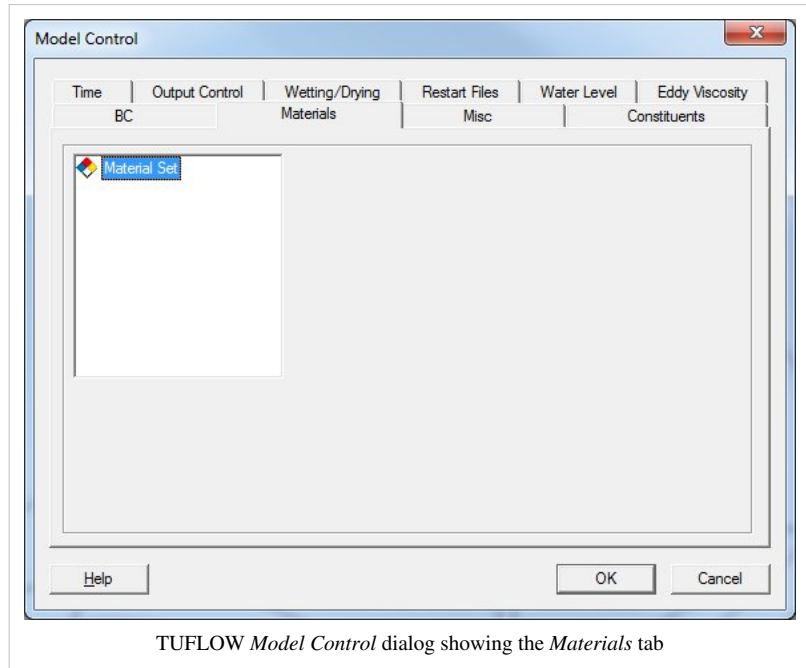


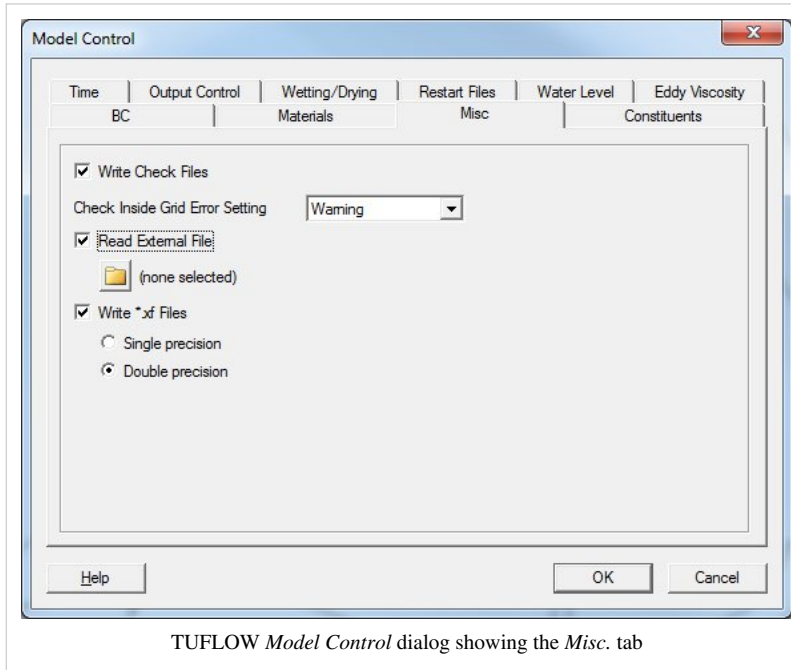
Materials

The material set to use is specified in this tab.

Misc.

- **Write Check Files – TUFLOW** creates a number of check files that can be used to verify model inputs. TUFLOW will run faster if not writing check files.
- **Check inside grid error setting –** By default TUFLOW will generate an error of entities exist outside the grid domain. If you want to use data that extends beyond your grid, you can change this to warning or off.
- **Read External File –** The SMS interface does not support every option available in TUFLOW. Unsupported options can be used by creating a TUFLOW command file (see the TUFLOW documentation) and specifying the file name here.
- **Write Z pts as Binary Files (*.xf)**
- **TUFLOW Executable –** This option determines which executable to use when running the simulation. The options are double and single precision for both 32 and 64bit. By default, it is the option chosen in the startup preferences.
- **Use Mass Balance Corrector –** This option sets whether or not TUFLOW should use the mass balance corrector, which carries out an additional iteration of the mass balance equation every half time step. This can result in significant reductions in mass error for problematic models, particularly those with steep and/or very shallow flow.





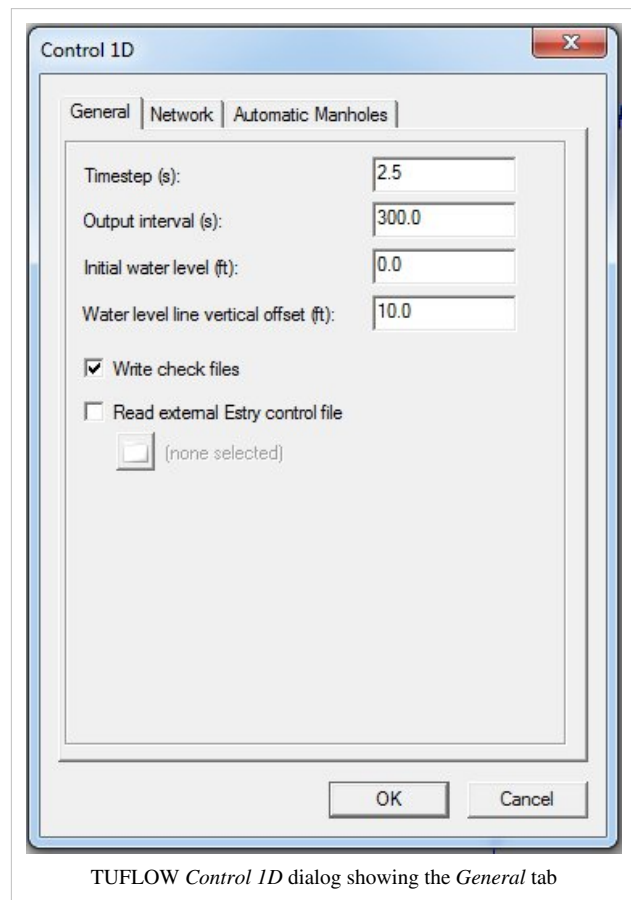
1D Control

General

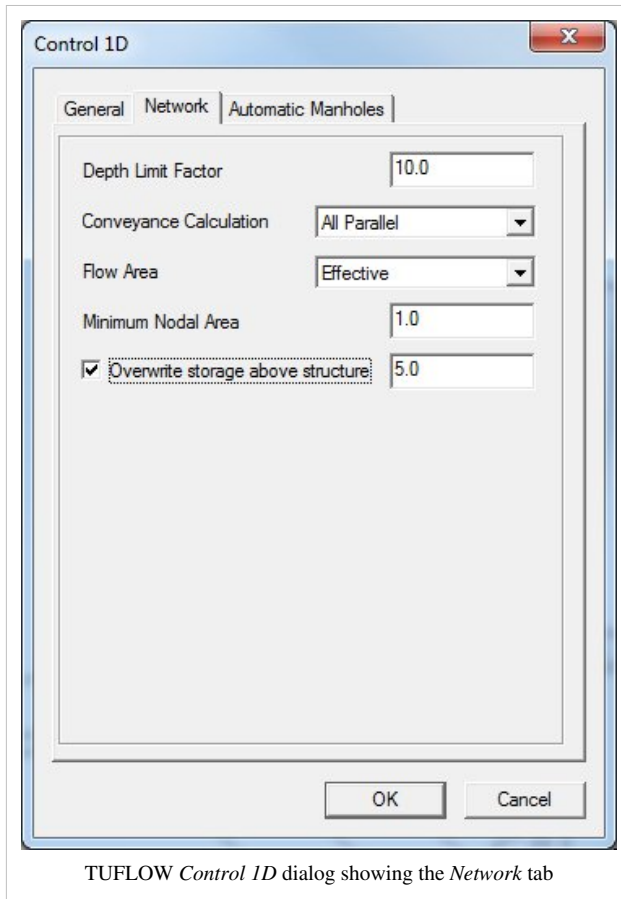
- **Timestep** – The optimum 1D timestep is based upon the courant number. The 1D timestep can generally be made quite small without affecting run times.
- **Output Interval** – How often to write 1D solution data to the csv solution files.
- **Initial Water Level** – The initial water level should generally be the same as the 2D water level.
- **Water level lines vertical offset** – This value sets the vertical adjustment of the water level line elevations in the SMS *.2dm file. This command only affects the *.2dm file read in by SMS for visualization and does affect the hydraulic calculations.
- **Write Check Files** – If this is off TUFLOW will not write check files for the 1D data.
- **Read external estry control file** – The SMS interface does not support all of the options available for 1D controls. You can build an estry control file (see the TUFLOW documentation for instructions) and have it read at the end of the SMS generated files.

Network

- **Depth Limit Factor** – This determines the depth that TUFLOW will consider unstable and abort the run. The default value is 10, meaning that if the channel depth is 10 times larger than the depth of the channel the simulation will be considered unstable.



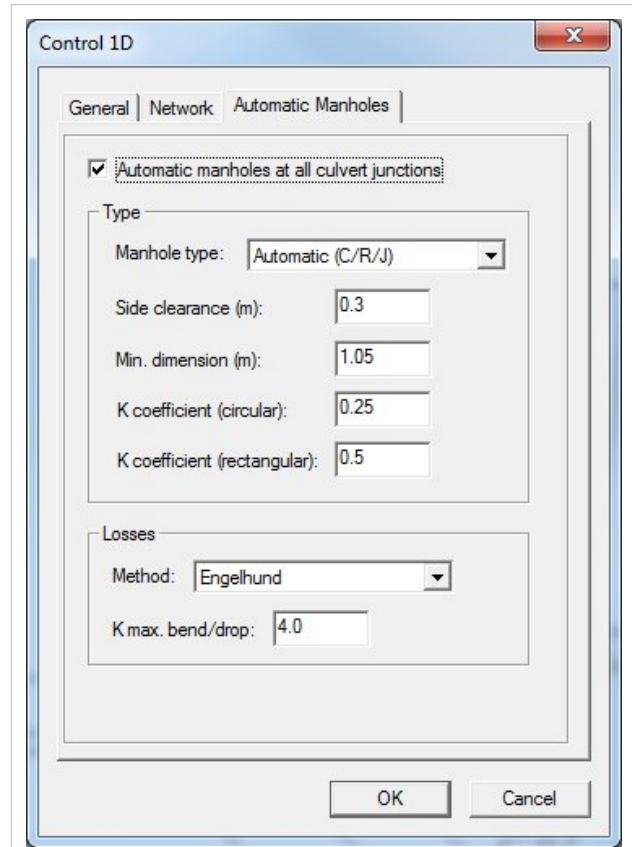
- Conveyance calculation – For 1D cross-section based channels, the conveyance calculations are divided into subsections. The subsections can be divided anywhere there is a change in resistance or a channel can be formed around each channel point(All parallel option). If the all parallel option is not used it is possible for the conveyance in a section to decrease when the depth increases which gives TUFLOW an error.
- Minimum nodal area – Nodes with a small storage area can have stability problems. Adding a minimum nodal area for all nodes has the potential to make the run more stable but may attenuate the model solution.



TUFLOW *Control 1D* dialog showing the *Network* tab

Automatic Manholes

- Automatic manholes at all culvert junctions – whether or not to have TUFLOW automatically create manholes.
- Type
 - Manhole type – circular, rectangular, junction, or automatically determined.
 - Side clearance (m) – for circular manholes, the default side clearance from the side of the largest culvert to the side of the manhole chamber. For rectangular manholes, the side clearance from the side of the culvert to the side of the manhole chamber.
 - Min. dimension (m) – minimum diameter for circular manholes and minimum width and length for rectangular manholes.
 - K coefficient (circular) – K coefficient for flow out of circular manholes.
 - K coefficient (rectangular) – K coefficient for flow out of rectangular manholes.
- Losses
 - Method – loss approach to be used (none, Engelhund, Fixed).
 - K max bend/drop – maximum K energy loss coefficient that can occur for the sum of the loss coefficients for bends and drops at a manhole when using the Engelhund approach.



TUFLOW Model Control 1D showing the Automatic Manhole tab

See the article TUFLOW Manholes for more information.

Related Topics

- TULFOW

TUFLOW Numeric Engine

TUFLOW is a computational engine that provides two-dimensional (2D) and one-dimensional (1D) solutions of the free-surface flow equations to simulate flood and tidal wave propagation. It is specifically beneficial where the hydrodynamic behaviour in coastal waters, estuaries, rivers, floodplains and urban drainage environments have complex 2D flow patterns that would be awkward to represent using traditional 1D network models.

A powerful feature of TUFLOW is its 2D/1D dynamic linking, first pioneered in 1990, and subsequently enhanced to the point where it offers unparalleled flexibility and robustness.

TUFLOW continues to develop and evolve to meet the challenges of hydrodynamic modelling. Its strengths include:

- rapid and stable wetting and drying;
- 1D and 2D linking;
- multiple 2D domains (optional);
- both 1D and 2D representation of hydraulic structures;
- automatic upstream/downstream controlled flow regime switching;
- 1D and 2D supercritical flow;
- highly flexible and efficient data handling;
- GIS based; and
- extensive quality control outputs.

It is suited to modelling flooding in major rivers through to complex overland and piped urban flows, along with estuarine and coastal hydraulics. [3]

TUFLOW was written and is maintained by Bill Syme at WBM TUFLOW Webpage ^[2].

Related Topics

- TUFLOW

TUFLOW Simulation

A simulation includes the domain, boundary conditions, model parameters, event definition, and material set used for a single TUFLOW run. Multiple simulations can exist within the same project in order to compare alternatives such as before/after a levee or to look at multiple events such as 10 year, 20 year, and 50 year.

Working with Simulations

A Simulation can be created by right-clicking on a blank portion of the project explorer and choosing **New → TUFLOW Simulation**. Once a simulation exists future simulations are often similar to one already created. Often it is easier to make a copy of the simulation and make changes where appropriate. A simulation can be copied by right-clicking on the simulation and choosing **Duplicate**.

The right-click menu for a simulation includes commands to change model parameters or 1D model parameters as well as launching TUFLOW to run the simulation.

Simulation Components

Much of the data for a simulation is contained in Geometry components and TUFLOW coverages. These items are included in a TUFLOW simulation using project explorer links. The model parameters are stored with the simulation item and can be accessed from the simulation's right-click menu.

TUFLOW ZShape

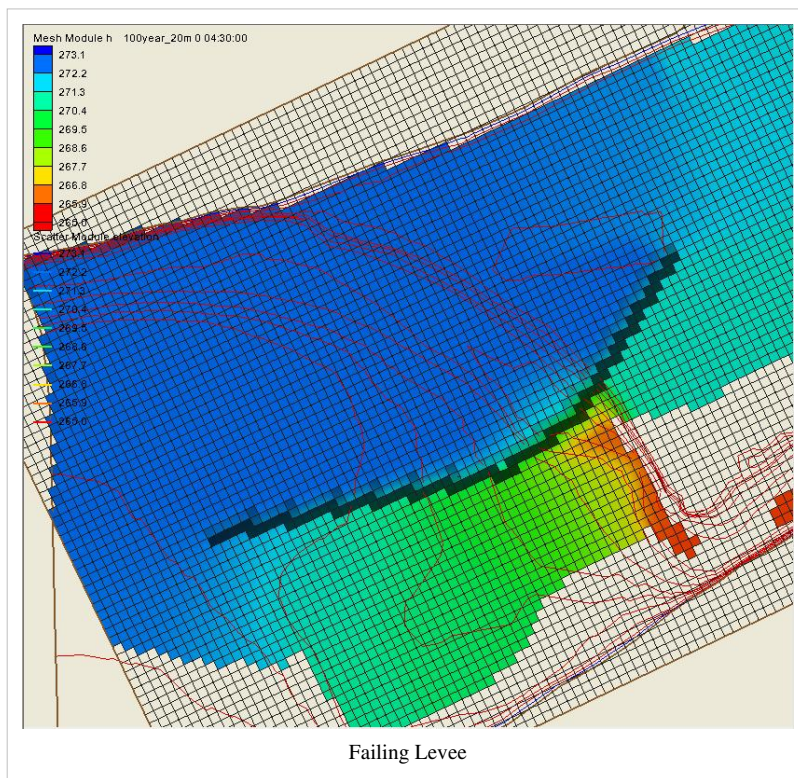
The TUFLOW ZShape coverage allows you to modify terrain with arcs and set up triggers to change those modifications, such as at a specific time during the simulation or when water depth exceeds a certain amount. An example application is a levee collapsing when flood water overtops it.

TUFLOW also supports polygon ZShapes. However, SMS does not support polygon ZShapes at this time. If you need to create static modifications to terrain using polygons you can use the Geometry Modification Coverage.

The ZShape Coverage

The ZShape coverage is available under the models/TUFLOW folder. ZShape data is stored in a feature arc's attributes. The Z values of the feature arc's points determine its Z, not the Z of the arc itself (exceptions to this are explained in the Arc Properties Dialog section below).

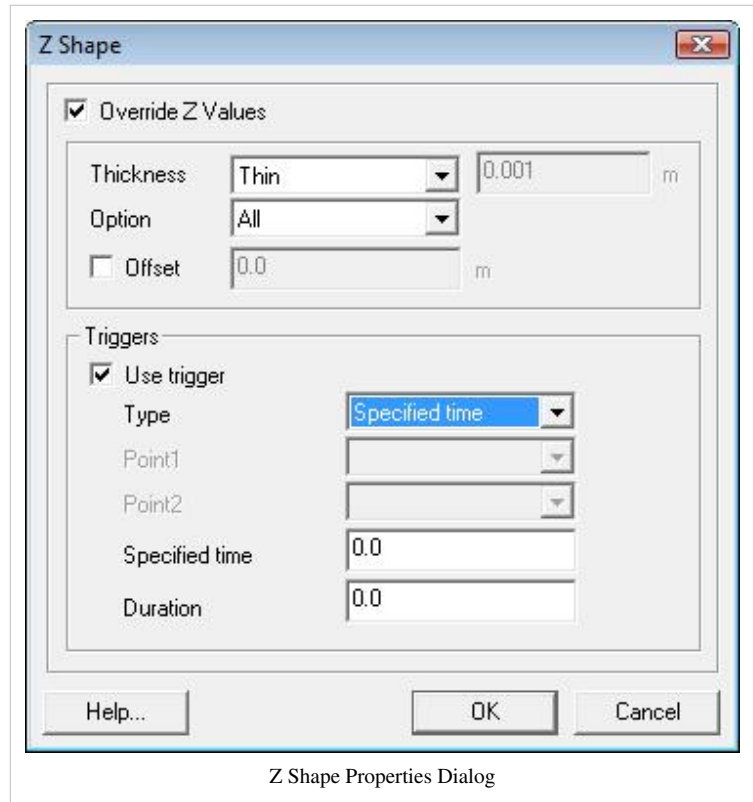
ZShapes are split into two main categories, Static ZShapes and Variable ZShapes. Static ZShapes are simple terrain modifications that do not change over time. They can be used to create a levee, pit, sand bar, dam, etc. Variable ZShapes have a trigger that causes them to raise or lower the terrain during the simulation. Often a variable ZShape is combined with a static one to provide raised terrain for the trigger to modify. This is necessary because a variable z shape cannot raise the same area it intends to lower when its trigger activates -- it needs the terrain to already be there and cannot create it itself, only modify once its trigger activates.



Arc Properties Dialog

Arcs in a ZShape coverage store ZShape data only if they are told to "override Z values". Otherwise they are not exported for TUFLOW's use. To reach the properties dialog, right-click on an arc and choose **Attributes....** Below is an explanation of the zshape options.

- **Override Z Values:** ZShape data is only stored and exported if this checkbox is enabled. It tells TUFLOW to override the current terrain Z values with the new values from the arc.
- **Thickness:** Arcs can be Thin Lines, Thick Lines, or Wide Lines. Thin lines have a width of 0, thick lines have a thickness less than or equal to 1.5 times the 2D cell size, and wide lines are any width larger than that. Thin lines follow some special rules that affect other options in the dialog, such as Option and Trigger Type.
- **Option:** The ZShape Options are All, Add, Min, Max, and Offset. Offset is only available for variable thin line Z Shapes, which cannot use the Add, Min or Max options. Add is unavailable for any variable Z Shape.
 - Offset is not listed by the same name in the TUFLOW documentation and is an alias in SMS for not specifying "NO MERGE". This causes the points of the arc to not be written, and the Z of the arc is determined by the Offset field.
- **Offset:** Raises the entire arc by this amount. Negative values are appropriate.
 - When offset is specified as the Option for variable thin lines, this field specifies the height to adjust terrain to (instead of adjusting it up or down by this amount) when triggered.
 - For all other variable arcs, this the the amount to adjust the terrain by when the trigger activates.
- **Use Trigger:** Toggles the arc between being a static zshape and a variable zshape. Variable z shapes use triggers to change terrain when a specific event occurs. All variable Z Shapes have the Duration option to span the change over a certain period of time. Other options will change with the Trigger Type.
 - **Trigger Type:** The type of event that will trigger the Z modification.
 - Specified time: Triggers at a specific time during the simulation, measured in hours.
 - Water level at point: Uses a Trigger Point to measure water level, and triggers when the water level reaches or exceeds the value in the *Water level* field.
 - Water level difference: Uses two trigger points. Triggers when the difference between water levels measured at both points exceeds the value in *Water level difference*. A difference in either direction is treated the same in TUFLOW, and there is no need for negative numbers.
 - Thin line water level: Similar to *Water level at point*, but for thin lines. Does not use a trigger point, instead measuring the water levels on either side of the thin line.
 - Thin line water level difference: Triggers when the difference between water levels on either side of the thin line reaches or exceeds the value in *Water level difference*.



- Point1 and Point2: Both point selection boxes list the trigger points available in the coverage. These fields are only enabled for trigger types that use them.
- Specified time, Water level, Water level difference: These fields become available based on the trigger type selected.
- Duration: If set to 0, changes are instantaneous once triggered. Otherwise the change will be interpolated over this many hours.

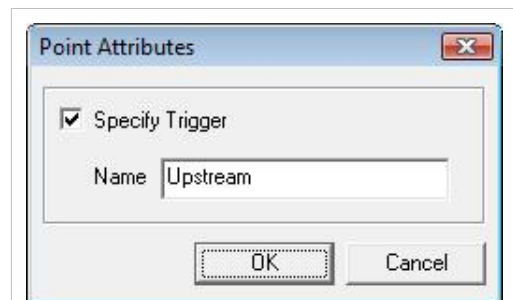
Trigger Points

Trigger points are used by the *Water level at point* and *Water level difference* trigger types. Simply create a feature point and enter its attributes dialog to create a trigger point.

Trigger Point Dialog

The *trigger point* dialog is used to turn a feature point into a ZShape trigger point. Enabling the Specify Trigger checkbox and giving the point a name will cause it to appear in the Point1 and Point2 selection boxes in the arc attributes dialog for use as a trigger point.

If you rename a trigger point that is being used by triggers, or delete it by unchecking *Specify Trigger*, SMS will notify you that there are one or more arcs/polygons being updated with the new change.



Trigger Point Attributes Dialog

Article Sources and Contributors

SMS Models *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64297> *Contributors:* Aanderson, Azundel, Jcreer, Rjones

Generic Mesh Model *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63934> *Contributors:* Cmballard, Jcreer

Generic Model Files *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64590> *Contributors:* Jcreer, Jdhowlett

Generic Model Graphical Interface *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64868> *Contributors:* Jcreer, Sramiah

FVCOM *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63922> *Contributors:* Jcreer, Jdhowlett

PTM *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64916> *Contributors:* Jcreer, Morelandtom

PTM Model Control *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64824> *Contributors:* Jcreer, Jdhowlett

PTM Graphical Interface *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64118> *Contributors:* Jcreer, Jdhowlett

PTM Particle Filters *Source:* <http://www.xmswiki.com/xms/index.php?oldid=48036> *Contributors:* Jcreer

PTM Traps *Source:* <http://www.xmswiki.com/xms/index.php?oldid=62127> *Contributors:* Jcreer, Jdhowlett

PTM Coverages *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64178> *Contributors:* Jcreer

PTM Gage Coverage *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64179> *Contributors:* Jcreer

PTM Sources *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64120> *Contributors:* Jcreer, Jdhowlett

PTM Arc Attributes Dialog *Source:* <http://www.xmswiki.com/xms/index.php?oldid=62143> *Contributors:* Jcreer, Jdhowlett

PTM Feature Point Attributes Dialog *Source:* <http://www.xmswiki.com/xms/index.php?oldid=62145> *Contributors:* Jcreer, Jdhowlett

PTM Files *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64387> *Contributors:* Jcreer, Morelandtom

PTM Control File *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64388> *Contributors:* Jcreer

PTM Sediment File *Source:* <http://www.xmswiki.com/xms/index.php?oldid=62150> *Contributors:* Jcreer, Jdhowlett

PTM Source File *Source:* <http://www.xmswiki.com/xms/index.php?oldid=62152> *Contributors:* Jcreer, Jdhowlett

PTM Wave File *Source:* <http://www.xmswiki.com/xms/index.php?oldid=66070> *Contributors:* Jcreer, Jdhowlett

PTM Trap File *Source:* <http://www.xmswiki.com/xms/index.php?oldid=52930> *Contributors:* Jcreer, Jdhowlett

PTM Trap Output *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64389> *Contributors:* Jcreer

PTM Boundary Condition File *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64862> *Contributors:* Jcreer

Generic Grid Model *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64075> *Contributors:* Jcreer

Creating a Generic Grid *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64082> *Contributors:* Jcreer

ADCIRC *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63965> *Contributors:* Aanderson, Azundel, Jcreer, Jdhowlett, Wood

ADCIRC Database *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63961> *Contributors:* Azundel, Jcreer, Jdye

ADCIRC Files *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64385> *Contributors:* Cballard, Jcreer

ADCIRC Graphical Interface *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63974> *Contributors:* Cballard, Jcreer

ADCIRC Menu *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64875> *Contributors:* Jcreer, Jdhowlett

ADCIRC Mesh *Source:* <http://www.xmswiki.com/xms/index.php?oldid=58159> *Contributors:* Jcreer, Jdhowlett

ADCIRC Model Control *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64826> *Contributors:* Jcreer, Rjones

ADCIRC Spatial Attributes *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63960> *Contributors:* Azundel, Jcreer, Rjones

LTEA *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63964> *Contributors:* Jcreer, Wood

ADCIRC Boundary Conditions *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64860> *Contributors:* Azundel, Jcreer, Jdhowlett

ADCIRC Weirs and Island Barriers *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64861> *Contributors:* Jcreer

BOUSS-2D *Source:* <http://www.xmswiki.com/xms/index.php?oldid=61851> *Contributors:* Jcreer, Jdhowlett

BOUSS-2D Using the Model *Source:* <http://www.xmswiki.com/xms/index.php?oldid=51399> *Contributors:* Jcreer

BOUSS-2D Simulations *Source:* <http://www.xmswiki.com/xms/index.php?oldid=62208> *Contributors:* Jcreer

BOUSS-2D Calculators *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63838> *Contributors:* Jcreer, Jdhowlett

BOUSS-2D Files *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64391> *Contributors:* Azundel, Jcreer

BOUSS-2D Graphical Interface *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64869> *Contributors:* Jcreer, Jdhowlett

BOUSS-2D Model Control *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64827> *Contributors:* Jcreer, Jdhowlett

BOUSS-2D Parameter File *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64393> *Contributors:* Azundel, Jcreer

BOUSS-2D Probes *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63981> *Contributors:* Jcreer

BOUSS-2D Test Cases *Source:* <http://www.xmswiki.com/xms/index.php?oldid=51470> *Contributors:* Dmyers, Jcreer

BOUSS2D Runup / Overtopping *Source:* <http://www.xmswiki.com/xms/index.php?oldid=66071> *Contributors:* Jcreer

BOUSS2D Runup / Overtopping Input Files *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64383> *Contributors:* Jcreer

BOUSS2D Runup / Overtopping Viewing Data *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63979> *Contributors:* Jcreer

CGWAVE *Source:* <http://www.xmswiki.com/xms/index.php?oldid=66072> *Contributors:* Azundel, Jcreer, Morelandtom

CGWAVE Overview *Source:* <http://www.xmswiki.com/xms/index.php?oldid=62862> *Contributors:* Jcreer

CGWAVE Math Details: Boundary Conditions *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64859> *Contributors:* Jcreer, Jdhowlett

CGWAVE Boundary Conditions Dialog *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64864> *Contributors:* Jcreer, Jdhowlett

CGWAVE Files *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64384> *Contributors:* Azundel, Jcreer

CGWAVE Math Details: Governing Equations *Source:* <http://www.xmswiki.com/xms/index.php?oldid=62244> *Contributors:* Jcreer, Jdhowlett

CGWAVE Graphical Interface *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63745> *Contributors:* Azundel, Jcreer

CGWAVE Incident Wave Conditions *Source:* <http://www.xmswiki.com/xms/index.php?oldid=62252> *Contributors:* Azundel, Jcreer, Morelandtom

CGWAVE Model Control *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64828> *Contributors:* Jcreer, Jdhowlett

CGWAVE Math Details: Numerical Solution *Source:* <http://www.xmswiki.com/xms/index.php?oldid=62255> *Contributors:* Jcreer, Jdhowlett

CGWAVE Practical Notes *Source:* <http://www.xmswiki.com/xms/index.php?oldid=62257> *Contributors:* Astrassburg, Jcreer

CGWAVE Test Cases *Source:* <http://www.xmswiki.com/xms/index.php?oldid=62259> *Contributors:* Jcreer, Jdhowlett

Long Wave Input Toolbox *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64909> *Contributors:* Azundel, Jcreer

CMS *Source:* <http://www.xmswiki.com/xms/index.php?oldid=62268> *Contributors:* Jcreer, Mebrown998

CMS-Flow/CMS-Wave Steering *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64878> *Contributors:* Jcreer, Jdhowlett

CMS-Flow/Save Points *Source:* <http://www.xmswiki.com/xms/index.php?oldid=62273> *Contributors:* Jcreer

CMS-Flow *Source:* <http://www.xmswiki.com/xms/index.php?oldid=62275> *Contributors:* Azundel, Jcreer, Twood, Vnarteh, Wood

CMS-Flow Cell Attributes *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63863> *Contributors:* Jcreer, Jdhowlett

CMS-Flow Graphical Interface *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64867> *Contributors:* Azundel, Jcreer

CMS-Flow Grid Modification Flag *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63866> *Contributors:* Jcreer

CMS-Flow Hard Bottom *Source:* <http://www.xmswiki.com/xms/index.php?oldid=52104> *Contributors:* Jcreer, Jdhowlett

CMS-Flow Hot Start *Source:* <http://www.xmswiki.com/xms/index.php?oldid=62279> *Contributors:* Jcreer, Jdy

CMS-Flow Menu *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63865> *Contributors:* Jcreer, Jdhowlett

CMS-Flow Model Control *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64829> *Contributors:* Azundel, Jcreer

CMS-Flow Observation Cells *Source:* <http://www.xmswiki.com/xms/index.php?oldid=49191> *Contributors:* Jcreer, Jdhowlett

CMS-Flow Output Control *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63870> *Contributors:* Azundel, Jcreer

CMS-Flow Spatial Datasets *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63867> *Contributors:* Azundel, Jcreer

CMS-Flow Time Control *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63872> *Contributors:* Azundel, Jcreer

CMS-Flow Transport Control *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63873> *Contributors:* Jcreer, Jschultze

CMS-Flow User-Editable Datasets *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63868> *Contributors:* Jcreer

Lund Cirp and Watanabe Formula *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63877> *Contributors:* Azundel, Jcreer

CMS-Flow Advanced Cards *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63876> *Contributors:* Jcreer, Sramiah

M2D *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63878> *Contributors:* Jcreer, Mebrown998

Telescoping Grids *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63880> *Contributors:* Jcreer

CMS-Wave *Source:* <http://www.xmswiki.com/xms/index.php?oldid=66073> *Contributors:* Azundel, Jcreer

CMS-Wave Cell Attributes Dialog *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63883> *Contributors:* Jcreer, Sramiah

CMS-Wave Graphical Interface *Source:* <http://www.xmswiki.com/xms/index.php?oldid=50184> *Contributors:* Azundel, Jcreer

CMS-Wave Menu *Source:* <http://www.xmswiki.com/xms/index.php?oldid=62855> *Contributors:* Jcreer, Jdhowlett

CMS-Wave Model Control *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64830> *Contributors:* Jcreer, Jdhowlett

CMS-Wave Nesting Options Dialog *Source:* <http://www.xmswiki.com/xms/index.php?oldid=62321> *Contributors:* Jcreer

CMS-Wave Files *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64394> *Contributors:* Azundel, Jcreer

CMS-Wave Control File *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64395> *Contributors:* Azundel, Jcreer

CMS-Wave Depth File *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64396> *Contributors:* Azundel, Jcreer

CMS-Wave Simulation File *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64397> *Contributors:* Azundel, Jcreer

CMS-Wave Spectral Energy File *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64398> *Contributors:* Jcreer, Ladamson

CMS-Wave Spectral Table File *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64399> *Contributors:* Azundel, Jcreer

CMS-Wave STD *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64401> *Contributors:* Jcreer

CMS-Wave Structure File *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64400> *Contributors:* Azundel, Jcreer

CSHORE *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64857> *Contributors:* Jcreer

CSTORM-MS *Source:* <http://www.xmswiki.com/xms/index.php?oldid=62338> *Contributors:* Jcreer

GenCade *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64165> *Contributors:* Jcreer, Wood

GenCade Files *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64402> *Contributors:* Jcreer

GenCade Modeling Process *Source:* <http://www.xmswiki.com/xms/index.php?oldid=48376> *Contributors:* Jcreer

GenCade Graphical Interface *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64910> *Contributors:* Jcreer

GenCade Arc Attributes *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63844> *Contributors:* Jcreer

GenCade Events *Source:* <http://www.xmswiki.com/xms/index.php?oldid=48975> *Contributors:* Jcreer

GenCade Menu *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63845> *Contributors:* Jcreer

GenCade Model Control Dialog *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64870> *Contributors:* Jcreer

GenCade Result Visualization *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63846> *Contributors:* Jcreer

GenCade Structures *Source:* <http://www.xmswiki.com/xms/index.php?oldid=48987> *Contributors:* Jcreer

Wave Gages *Source:* <http://www.xmswiki.com/xms/index.php?oldid=48995> *Contributors:* Jcreer

STWAVE *Source:* <http://www.xmswiki.com/xms/index.php?oldid=66074> *Contributors:* Jcreer, Wood

Grid Nesting *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63899> *Contributors:* Jcreer, Morelandtom

STWAVE Graphical Interface *Source:* <http://www.xmswiki.com/xms/index.php?oldid=62356> *Contributors:* Jcreer, Jhworth

STWAVE Iterations Control Dialog *Source:* <http://www.xmswiki.com/xms/index.php?oldid=48936> *Contributors:* Jcreer

STWAVE Menu *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63895> *Contributors:* Jcreer, Jdhowlett

STWAVE Model Control *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64872> *Contributors:* Jcreer, Jdhowlett

STWAVE Output Control Dialog *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63897> *Contributors:* Jcreer, Jdhowlett

WAM *Source:* <http://www.xmswiki.com/xms/index.php?oldid=66075> *Contributors:* Jcreer

WAM Map to Raster Utility *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63904> *Contributors:* Jcreer

WAM Graphical Interface *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63903> *Contributors:* Jcreer

WAM Grid Options *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63905> *Contributors:* Jcreer, Tnewbold

WAM Simulation Model Control *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64833> *Contributors:* Jcreer

WAM Spectra from STWAVE Grids *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64142> *Contributors:* Jcreer

ADH *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64588> *Contributors:* Jcreer

ADH Bed Layers Assignment *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63815> *Contributors:* Jcreer

ADH Boat Definition File Cards *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64404> *Contributors:* Jcreer, Kwinters

ADH Extract WSE *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63818> *Contributors:* Jcreer

ADH Hot Start File *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64403> *Contributors:* Jcreer

ADH Hot Start Initial Conditions *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63820> *Contributors:* Jcreer, Kwinters

ADH Material Properties *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64884> *Contributors:* Jcreer, Kwinters

ADH Run Model *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63812> *Contributors:* Jcreer, Jdhowlett

Sediment Library Control *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63823> *Contributors:* Jcreer

ADH Sediment Transport and Bed Layers *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63824> *Contributors:* Jcreer

ADH Time Series *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63825> *Contributors:* Jcreer

ADH Time Series Attributes *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63826> *Contributors:* Jcreer

ADH Velocity Series Editor *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63828> *Contributors:* Jcreer

ADH Vessel Coverage *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64173> *Contributors:* Jcreer

ADH Wind Stations *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63822> *Contributors:* Jcreer

ADH Boundary Condition *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64855> *Contributors:* Jcreer, Kwinters

ADH Boundary Condition File Cards *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64856> *Contributors:* Jcreer, Kwinters

ADH Model Control *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64834> *Contributors:* Jcreer, Kwinters

ADH Model Control Model Parameters *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64836> *Contributors:* Jcreer, Kwinters

ADH Model Control Iterations *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64838> *Contributors:* Jcreer

ADH Model Control Time *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64839> *Contributors:* Jcreer, Kwinters

ADH Model Control Output *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64840> *Contributors:* Jcreer, Kwinters

ADH Model Control Global Material Properties *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64885> *Contributors:* Jcreer, Kwinters

ADH Model Control Transport Constituents *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63991> *Contributors:* Jcreer, Kwinters

ADH Model Control Consolidation *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64842> *Contributors:* Jcreer

ADH Model Control Advanced *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64843> *Contributors:* Jcreer

ADH Model Control Solver *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64844> *Contributors:* Jcreer

FESWMS *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64336> *Contributors:* Jcreer, Jdhowlett, Vnarteh

FESWMS Arc Attributes Dialog *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63807> *Contributors:* Jcreer, Jdhowlett

FESWMS BC Nodestrings *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64854> *Contributors:* Jcreer, Jdhowlett

FESWMS Executable Known Issues *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63801> *Contributors:* Jcreer, Jschultze

FESWMS Files *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64406> *Contributors:* Jcreer, Jdhowlett

FESWMS Graphical Interface *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63808> *Contributors:* Jcreer, Jdhowlett

FESWMS Hydraulic Structures *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63809> *Contributors:* Jcreer, Ndye

FESWMS Material Properties *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64886> *Contributors:* Jcreer, Rjones

FESWMS Menu *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64337> *Contributors:* Jcreer, Jdhowlett

FESWMS Model Control Dialog *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64837> *Contributors:* Disleman, Jcreer

FESWMS Point Attributes Dialog *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64876> *Contributors:* Jcreer, Jdhowlett

FESWMS Sediment Control *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64339> *Contributors:* Jcreer, Twood

FESWMS Spindown *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64338> *Contributors:* Jcreer, Jdhowlett

HYDRO AS-2D *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63776> *Contributors:* Jcreer, Wood

RIVERFLO-2D *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63777> *Contributors:* Aanderson, Azundel, Jcreer

SRH-2D *Source:* <http://www.xmswiki.com/xms/index.php?oldid=62954> *Contributors:* Azundel, Jcreer, Jdhowlett

Steering *Source:* <http://www.xmswiki.com/xms/index.php?oldid=65656> *Contributors:* Jcreer, Jdhowlett

TABS *Source:* <http://www.xmswiki.com/xms/index.php?oldid=52595> *Contributors:* Jcreer, Jdhowlett

TABS Attribute Dialog *Source:* <http://www.xmswiki.com/xms/index.php?oldid=62491> *Contributors:* Jcreer

Total Flow Nodestring *Source:* <http://www.xmswiki.com/xms/index.php?oldid=40366> *Contributors:* Jcreer, Jdhowlett

GFGEN *Source:* <http://www.xmswiki.com/xms/index.php?oldid=40367> *Contributors:* Jcreer, Jdhowlett

GFGEN Executable Known Issues *Source:* <http://www.xmswiki.com/xms/index.php?oldid=45685> *Contributors:* Jcreer, Wood

RMA2 *Source:* <http://www.xmswiki.com/xms/index.php?oldid=62493> *Contributors:* Jcreer, Jdhowlett

Nodal Transition (Marsh Porosity) Dialog *Source:* <http://www.xmswiki.com/xms/index.php?oldid=62494> *Contributors:* Jcreer, Jdhowlett

Rainfall Values Dialog *Source:* <http://www.xmswiki.com/xms/index.php?oldid=48685> *Contributors:* Jcreer, Jdhowlett

RMA2 1D Control Structure *Source:* <http://www.xmswiki.com/xms/index.php?oldid=52632> *Contributors:* Jcreer

RMA2 Boundary Conditions *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64853> *Contributors:* Jcreer, Sramiah

RMA2 Files *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64408> *Contributors:* Jcreer, Sramiah

RMA2 Graphical Interface *Source:* <http://www.xmswiki.com/xms/index.php?oldid=47958> *Contributors:* Jcreer, Sramiah

RMA2 Material Properties *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64887> *Contributors:* Jcreer, Jdhowlett

RMA2 Menu *Source:* <http://www.xmswiki.com/xms/index.php?oldid=50243> *Contributors:* Jcreer, Jdhowlett

RMA2 Model Control Dialog *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64848> *Contributors:* Jcreer, Jdhowlett

RMA2 Spindown *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64341> *Contributors:* Jcreer, Jdhowlett

Roughness Options Dialog *Source:* <http://www.xmswiki.com/xms/index.php?oldid=62509> *Contributors:* Jcreer, Jdhowlett

RMA2 GCL Card *Source:* <http://www.xmswiki.com/xms/index.php?oldid=50249> *Contributors:* Jcreer, Jdhowlett

RMA4 *Source:* <http://www.xmswiki.com/xms/index.php?oldid=52737> *Contributors:* Jcreer, Rjones

RMA4 Boundary Conditions *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64852> *Contributors:* Jcreer, Jdhowlett

RMA4 Files *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64350> *Contributors:* Jcreer, Jdye

RMA4 Graphical Interface *Source:* <http://www.xmswiki.com/xms/index.php?oldid=47925> *Contributors:* Jcreer, Jdhowlett

RMA4 Material Properties *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64888> *Contributors:* Jcreer

RMA4 Menu *Source:* <http://www.xmswiki.com/xms/index.php?oldid=50256> *Contributors:* Jcreer, Sramiah

RMA4 Model Control *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64849> *Contributors:* Jcreer

TUFLOW *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64085> *Contributors:* Azundel, Jcreer, Rjones

TUFLOW Overview *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64018> *Contributors:* Jcreer

Batch Runs *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64000> *Contributors:* Jcreer

Building TUFLOW 2D Domains *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64067> *Contributors:* Jcreer, Rjones

Command Objects *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64001> *Contributors:* Jcreer

Define TUFLOW domain *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64079> *Contributors:* Jcreer

Reading a TUFLOW Simulation *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64003> *Contributors:* Jcreer

TUFLOW 2D Geometry Components *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64015> *Contributors:* Jcreer, Rjones

TUFLOW AD *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64004> *Contributors:* Jcreer

TUFLOW Boundary Conditions *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64851> *Contributors:* Jcreer, Rjones

TUFLOW Check Files *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64005> *Contributors:* Jcreer, Rjones

TUFLOW Combining 1D and 2D Domains *Source:* <http://www.xmswiki.com/xms/index.php?oldid=47892> *Contributors:* Jcreer, Rjones

TUFLOW Grid Options *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64008> *Contributors:* Jcreer, Rjones

TUFLOW Inlet Database *Source:* <http://www.xmswiki.com/xms/index.php?oldid=47894> *Contributors:* Jcreer, Jdhowlett

TUFLOW Irregular Culverts *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64028> *Contributors:* Jcreer

TUFLOW Linking 2D Domains *Source:* <http://www.xmswiki.com/xms/index.php?oldid=47896> *Contributors:* Jcreer, Rjones

TUFLOW Links *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64013> *Contributors:* Jcreer

TUFLOW Manholes *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64348> *Contributors:* Jcreer

TUFLOW Material properties *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64889> *Contributors:* Jcreer

TUFLOW Menu *Source:* <http://www.xmswiki.com/xms/index.php?oldid=63758> *Contributors:* Azundel, Jcreer

TUFLOW Model Parameters *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64850> *Contributors:* Jcreer, Jhworth

TUFLOW Numeric Engine *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64029> *Contributors:* Jcreer

TUFLOW Simulation *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64027> *Contributors:* Jcreer

TUFLOW ZShape *Source:* <http://www.xmswiki.com/xms/index.php?oldid=64031> *Contributors:* Jcreer, Rjones

Image Sources, Licenses and Contributors

Image:define model dialog.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:Define_model_dialog.jpg *License:* unknown *Contributors:* Jcreer

Image:Mesh Model Parameters.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:Mesh_Model_Parameters.jpg *License:* unknown *Contributors:* Jcreer

Image:Mesh Global Parameters.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:Mesh_Global_Parameters.jpg *License:* unknown *Contributors:* Jcreer

Image:Mesh BC.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:Mesh_BC.jpg *License:* unknown *Contributors:* Jcreer

Image:Mesh 2D Define Materials.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:Mesh_2D_Define_Materials.jpg *License:* unknown *Contributors:* Jcreer

Image:Mesh MP.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:Mesh_MP.jpg *License:* unknown *Contributors:* Jcreer

Image:Mesh Global Parameter Definition.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:Mesh_Global_Parameter_Definition.jpg *License:* unknown *Contributors:* Jcreer

File:Feature Arc Attributes.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:Feature_Arc_Attributes.jpg *License:* unknown *Contributors:* Jcreer

File:Mesh Nodestring BC.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:Mesh_Nodestring_BC.jpg *License:* unknown *Contributors:* Jcreer

Image:Mesh Nodal BC Display Options.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:Mesh_Nodal_BC_Display_Options.jpg *License:* unknown *Contributors:* Jcreer

Image:PTM MC Time.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:PTM_MC_Time.jpg *License:* unknown *Contributors:* Jcreer

Image:PTM MC Files.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:PTM_MC_Files.jpg *License:* unknown *Contributors:* Jcreer

Image:PTM Create External Input.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:PTM_Create_External_Input.jpg *License:* unknown *Contributors:* Jcreer

Image:PTM MC Computations.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:PTM_MC_Computations.jpg *License:* unknown *Contributors:* Jcreer

Image:PTM MC Output.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:PTM_MC_Output.jpg *License:* unknown *Contributors:* Jcreer

Image:PTM Polygon Att 1.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:PTM_Polygon_Att_1.jpg *License:* unknown *Contributors:* Jcreer

Image:PTM Polygon Att 2.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:PTM_Polygon_Att_2.jpg *License:* unknown *Contributors:* Jcreer

Image:PTM Feature Attribute Arc Source.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:PTM_Feature_Attribute_Arc_Source.jpg *License:* unknown *Contributors:* Jcreer

Image:PTM Feature Attribute Arc Trap.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:PTM_Feature_Attribute_Arc_Trap.jpg *License:* unknown *Contributors:* Jcreer

Image:PTM Feature Attributes Point.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:PTM_Feature_Attributes_Point.jpg *License:* unknown *Contributors:* Jcreer

Image:PTM Feature Attributes Point Vert.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:PTM_Feature_Attributes_Point_Vert.jpg *License:* unknown *Contributors:* Jcreer

Image:ADCIRC MC General.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ADCIRC_MC_General.jpg *License:* unknown *Contributors:* Jcreer

Image:ADCIRC MC Timing.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ADCIRC_MC_Timing.jpg *License:* unknown *Contributors:* Jcreer

Image:ADCIRC MC Files.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ADCIRC_MC_Files.jpg *License:* unknown *Contributors:* Jcreer

Image:ADCIRC MC Tidal.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ADCIRC_MC_Tidal.jpg *License:* unknown *Contributors:* Jcreer

Image:LeProvost_Domain.png *Source:* http://www.xmswiki.com/xms/index.php?title=File:LeProvost_Domain.png *License:* unknown *Contributors:* Mebrown998

Image:ADCIRC MC Wind.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ADCIRC_MC_Wind.jpg *License:* unknown *Contributors:* Jcreer

Image:ADCIRC Fleet Wind File.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ADCIRC_Fleet_Wind_File.jpg *License:* unknown *Contributors:* Jcreer

Image:ADCIRC Binary Wind.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ADCIRC_Binary_Wind.jpg *License:* unknown *Contributors:* Jcreer

Image:ADCIRC MC Sediment.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ADCIRC_MC_Sediment.jpg *License:* unknown *Contributors:* Jcreer

File:SMSSpatialAtt.jpg *Source:* <http://www.xmswiki.com/xms/index.php?title=File:SMSSpatialAtt.jpg> *License:* unknown *Contributors:* Jcreer

File:SpatialAttsInitialValues.PNG *Source:* <http://www.xmswiki.com/xms/index.php?title=File:SpatialAttsInitialValues.PNG> *License:* unknown *Contributors:* Jcreer

File:LTEA1.jpg *Source:* <http://www.xmswiki.com/xms/index.php?title=File:LTEA1.jpg> *License:* unknown *Contributors:* Jcreer

File:LTEA2.jpg *Source:* <http://www.xmswiki.com/xms/index.php?title=File:LTEA2.jpg> *License:* unknown *Contributors:* Jcreer

File:LTEA3.jpg *Source:* <http://www.xmswiki.com/xms/index.php?title=File:LTEA3.jpg> *License:* unknown *Contributors:* Jcreer

File:LTEA4.jpg *Source:* <http://www.xmswiki.com/xms/index.php?title=File:LTEA4.jpg> *License:* unknown *Contributors:* Jcreer

Image:AmericanSamoaPaved.jpg *Source:* <http://www.xmswiki.com/xms/index.php?title=File:AmericanSamoaPaved.jpg> *License:* unknown *Contributors:* Azundel

Image:AmericanSamoaLT24K.jpg *Source:* <http://www.xmswiki.com/xms/index.php?title=File:AmericanSamoaLT24K.jpg> *License:* unknown *Contributors:* Azundel

Image:AmericanSamoaLT12K.jpg *Source:* <http://www.xmswiki.com/xms/index.php?title=File:AmericanSamoaLT12K.jpg> *License:* unknown *Contributors:* Azundel

Image:AmericanSamoaPaved_Zoom.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:AmericanSamoaPaved_Zoom.jpg *License:* unknown *Contributors:* Azundel

Image:AmericanSamoaLT24K_Zoom.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:AmericanSamoaLT24K_Zoom.jpg *License:* unknown *Contributors:* Azundel

Image:AmericanSamoaLT12K_Zoom.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:AmericanSamoaLT12K_Zoom.jpg *License:* unknown *Contributors:* Azundel

Image:GlacierBayImage.jpg *Source:* <http://www.xmswiki.com/xms/index.php?title=File:GlacierBayImage.jpg> *License:* unknown *Contributors:* Azundel

Image:GlacierBayConceptual.jpg *Source:* <http://www.xmswiki.com/xms/index.php?title=File:GlacierBayConceptual.jpg> *License:* unknown *Contributors:* Azundel

Image:GlacierBayLinear.jpg *Source:* <http://www.xmswiki.com/xms/index.php?title=File:GlacierBayLinear.jpg> *License:* unknown *Contributors:* Azundel

Image:LTEA_Harmonics.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:LTEA_Harmonics.jpg *License:* unknown *Contributors:* Azundel

Image:GlacierBay10K.jpg *Source:* <http://www.xmswiki.com/xms/index.php?title=File:GlacierBay10K.jpg> *License:* unknown *Contributors:* Azundel

Image:GlacierBay20K.jpg *Source:* <http://www.xmswiki.com/xms/index.php?title=File:GlacierBay20K.jpg> *License:* unknown *Contributors:* Azundel

Image:GlacierBay30K.jpg *Source:* <http://www.xmswiki.com/xms/index.php?title=File:GlacierBay30K.jpg> *License:* unknown *Contributors:* Azundel

Image:ADCIRC Arc BC.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ADCIRC_Arc_BC.jpg *License:* unknown *Contributors:* Jcreer

Image:ADCIRC Normal BC.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ADCIRC_Normal_BC.jpg *License:* unknown *Contributors:* Jcreer

Image:ADCIRC Mainland BC.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ADCIRC_Mainland_BC.jpg *License:* unknown *Contributors:* Jcreer

Image:ADCIRC Island BC.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ADCIRC_Island_BC.jpg *License:* unknown *Contributors:* Jcreer

Image:ADCIRCWeir5.jpg *Source:* <http://www.xmswiki.com/xms/index.php?title=File:ADCIRCWeir5.jpg> *License:* unknown *Contributors:* Jcreer

Image:ADCIRC recording station.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ADCIRC_recording_station.jpg *License:* unknown *Contributors:* Jcreer

Image:ADCIRCweir1.jpg *Source:* <http://www.xmswiki.com/xms/index.php?title=File:ADCIRCweir1.jpg> *License:* unknown *Contributors:* Jcreer

Image:ADCIRCweir2.jpg *Source:* <http://www.xmswiki.com/xms/index.php?title=File:ADCIRCweir2.jpg> *License:* unknown *Contributors:* Jcreer

Image:ADCIRCweir4.jpg *Source:* <http://www.xmswiki.com/xms/index.php?title=File:ADCIRCweir4.jpg> *License:* unknown *Contributors:* Jcreer

File:ADCIRCweir6.jpg *Source:* <http://www.xmswiki.com/xms/index.php?title=File:ADCIRCweir6.jpg> *License:* unknown *Contributors:* Jcreer

Image:Damping Reflection Relationship-Fig 5- BOUSS-2D Technical Report.png *Source:* http://www.xmswiki.com/xms/index.php?title=File:Damping_Reflection_Relationship-Fig_5-_BOUSS-2D_Technical_Report.png *License:* unknown *Contributors:* Azundel

Image:BOUSS 2D Calc Wave.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:BOUSS_2D_Calc_Wave.jpg *License:* unknown *Contributors:* Jcreer

Image:BOUSS 2D Calc Runup.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:BOUSS_2D_Calc_Runup.jpg *License:* unknown *Contributors:* Jcreer

Image:SMS BOUSS 2D BC.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:SMS_BOUSS_2D_BC.jpg *License:* unknown *Contributors:* Jcreer

Image:BOUSS-2D_Boundary_Image.png *Source:* http://www.xmswiki.com/xms/index.php?title=File:BOUSS-2D_Boundary_Image.png *License:* unknown *Contributors:* Azundel

Image:SMS BOUSS2D MC.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:SMS_BOUSS2D_MC.jpg *License:* unknown *Contributors:* Jcreer

Image:Probe manager.png *Source:* http://www.xmswiki.com/xms/index.php?title=File:Probe_manager.png *License:* unknown *Contributors:* Jcreer

Image:New Runup.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:New_Runup.jpg *License:* unknown *Contributors:* Jcreer

Image:Runup MC.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:Runup_MC.jpg *License:* unknown *Contributors:* Jcreer

File:TransectProfile.jpg Source: <http://www.xmswiki.com/xms/index.php?title=File:TransectProfile.jpg> License: unknown Contributors: Jcreer

File:SolutionProfile.jpg Source: <http://www.xmswiki.com/xms/index.php?title=File:SolutionProfile.jpg> License: unknown Contributors: Jcreer

File:SolutionTimeSeries.jpg Source: <http://www.xmswiki.com/xms/index.php?title=File:SolutionTimeSeries.jpg> License: unknown Contributors: Jcreer

Image:CGWave_figure1.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CGWave_figure1.jpg License: unknown Contributors: Abarney

Image:Dialog1.jpg Source: <http://www.xmswiki.com/xms/index.php?title=File:Dialog1.jpg> License: unknown Contributors: Abarney

Image:Dialog2.jpg Source: <http://www.xmswiki.com/xms/index.php?title=File:Dialog2.jpg> License: unknown Contributors: Abarney

Image:CGWAVE BC.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CGWAVE_BC.jpg License: unknown Contributors: Jcreer

Image:SelectMeshNodestring.JPG Source: <http://www.xmswiki.com/xms/index.php?title=File:SelectMeshNodestring.JPG> License: unknown Contributors: Jdhowlett

Image:CGWAVE Model Control.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CGWAVE_Model_Control.jpg License: unknown Contributors: Jcreer

Image:CMS Flow CMS Wave Steering.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CMS_Flow_CMS_Wave_Steering.jpg License: unknown Contributors: Jcreer

Image:CMS-Flow Cell.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CMS-Flow_Cell.jpg License: unknown Contributors: Jcreer

Image:CMS-Flow Hard Bottom.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CMS-Flow_Hard_Bottom.jpg License: unknown Contributors: Jcreer

Image:CMS-Flow Hard Bottom Symbols.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CMS-Flow_Hard_Bottom_Symbols.jpg License: unknown Contributors: Jcreer

Image:CMS-Flow MC Flow.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CMS-Flow_MC_Flow.jpg License: unknown Contributors: Jcreer

Image:CMS-Flow MC Sediment.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CMS-Flow_MC_Sediment.jpg License: unknown Contributors: Jcreer

Image:CMS-Flow MC Salinity.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CMS-Flow_MC_Salinity.jpg License: unknown Contributors: Jcreer

Image:CMS-Flow Salinity Global Concentration.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CMS-Flow_Salinity_Global_Concentration.jpg License: unknown Contributors: Jcreer

Image:CMS-Flow MC Tidal.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CMS-Flow_MC_Tidal.jpg License: unknown Contributors: Jcreer

Image:CMS-Flow MC Wind.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CMS-Flow_MC_Wind.jpg License: unknown Contributors: Jcreer

Image:CMS-Flow MC Output.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CMS-Flow_MC_Output.jpg License: unknown Contributors: Jcreer

Image:CMS-Flow MC Cells.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CMS-Flow_MC_Cells.jpg License: unknown Contributors: Jcreer

Image:CMS-Flow MC Input.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CMS-Flow_MC_Input.jpg License: unknown Contributors: Jcreer

Image:CMS-Flow MC Advanced.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CMS-Flow_MC_Advanced.jpg License: unknown Contributors: Jcreer

Image:CMS-Flow Observation Symbols.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CMS-Flow_Observation_Symbols.jpg License: unknown Contributors: Jcreer

File:561px-CMSFlowModelControl.jpg Source: <http://www.xmswiki.com/xms/index.php?title=File:561px-CMSFlowModelControl.jpg> License: unknown Contributors: Jcreer

File:CustomDatasets.jpg Source: <http://www.xmswiki.com/xms/index.php?title=File:CustomDatasets.jpg> License: unknown Contributors: Jcreer

File:CMS-Wave Cell Attributes.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CMS-Wave_Cell_Attributes.jpg License: unknown Contributors: Jcreer

File:CMSWaveModelControl.jpg Source: <http://www.xmswiki.com/xms/index.php?title=File:CMSWaveModelControl.jpg> License: unknown Contributors: Jcreer

Image:CMS-Wave Nesting Options.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CMS-Wave_Nesting_Options.jpg License: unknown Contributors: Jcreer

Image:CSHORE Arc Att.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CSHORE_Arc_Att.jpg License: unknown Contributors: Jcreer

Image:CSHORE Friction Factor.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CSHORE_Friction_Factor.jpg License: unknown Contributors: Jcreer

Image:CSHORE Transects.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CSHORE_Transects.jpg License: unknown Contributors: Jcreer

Image:CSHORE MC Comp.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CSHORE_MC_Comp.jpg License: unknown Contributors: Jcreer

Image:CSHORE MC Sediment.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CSHORE_MC_Sediment.jpg License: unknown Contributors: Jcreer

Image:CSHORE MC Wave Water.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CSHORE_MC_Wave_Water.jpg License: unknown Contributors: Jcreer

Image:Create HPC Scripts.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:Create_HPC_Scripts.jpg License: unknown Contributors: Jcreer

Image:CSTORM Model Project Sum Over.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CSTORM_Model_Project_Sum_Over.jpg License: unknown Contributors: Jcreer

Image:CSTORM Set Thread1.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CSTORM_Set_Thread1.jpg License: unknown Contributors: Jcreer

File:CSTORM St Thread2.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CSTORM_St_Thread2.jpg License: unknown Contributors: Jcreer

Image:CSTORM Model Project Sum ESMF.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CSTORM_Model_Project_Sum_ESMF.jpg License: unknown Contributors: Jcreer

Image:CSTORM Model Project Sum Spatial.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CSTORM_Model_Project_Sum_Spatial.jpg License: unknown Contributors: Jcreer

Image:CSTORM Model Project Sum.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:CSTORM_Model_Project_Sum.jpg License: unknown Contributors: Jcreer

Image:GenCade Arc Attributes.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:GenCade_Arc_Attributes.jpg License: unknown Contributors: Jcreer

Image:GenCade Beach Fill.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:GenCade_Beach_Fill.jpg License: unknown Contributors: Jcreer

Image:GenCade Bypassing.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:GenCade_Bypassing.jpg License: unknown Contributors: Jcreer

Image:GenCade Grid Frame.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:GenCade_Grid_Frame.jpg License: unknown Contributors: Jcreer

Image:GenCade Water Level Events.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:GenCade_Water_Level_Events.jpg License: unknown Contributors: Jcreer

Image:GenCade MC Setup.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:GenCade_MC_Setup.jpg License: unknown Contributors: Jcreer

Image:GenCade MC Beach.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:GenCade_MC_Beach.jpg License: unknown Contributors: Jcreer

Image:GenCade MC Seaward.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:GenCade_MC_Seaward.jpg License: unknown Contributors: Jcreer

Image:GenCade MC Lateral.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:GenCade_MC_Lateral.jpg License: unknown Contributors: Jcreer

Image:GenCade Detached Breakwaters.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:GenCade_Detached_Breakwaters.jpg License: unknown Contributors: Jcreer

Image:GenCade Seawalls.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:GenCade_Seawalls.jpg License: unknown Contributors: Jcreer

Image:GenCade Groins.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:GenCade_Groins.jpg License: unknown Contributors: Jcreer

Image:GenCade Inlets.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:GenCade_Inlets.jpg License: unknown Contributors: Jcreer

Image:GenCade Wave Gages.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:GenCade_Wave_Gages.jpg License: unknown Contributors: Jcreer

Image:STWAVE MC Iterations.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:STWAVE_MC_Iterations.jpg License: unknown Contributors: Jcreer

Image:STWAVE Cell Attributes.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:STWAVE_Cell_Attributes.jpg License: unknown Contributors: Jcreer

File:STWAVEModelControl.jpg Source: <http://www.xmswiki.com/xms/index.php?title=File:STWAVEModelControl.jpg> License: unknown Contributors: Jcreer

Image:STWAVE MC Output.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:STWAVE_MC_Output.jpg License: unknown Contributors: Jcreer

Image:WAM Cell Attributes.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:WAM_Cell_Attributes.jpg License: unknown Contributors: Jcreer

Image:WAM MC general.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:WAM_MC_general.jpg License: unknown Contributors: Jcreer

Image:WAM MC Output.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:WAM_MC_Output.jpg License: unknown Contributors: Jcreer

Image:WAM MC Spatial Inputs.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:WAM_MC_Spatial_Inputs.jpg License: unknown Contributors: Jcreer

Image:WAM Sim MC general.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:WAM_Sim_MC_general.jpg License: unknown Contributors: Jcreer

Image:WAM Sim MC Spectra.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:WAM_Sim_MC_Spectra.jpg License: unknown Contributors: Jcreer

Image:ADH Hot Start Initial.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:ADH_Hot_Start_Initial.jpg License: unknown Contributors: Jcreer

File:ADH Material Properties Prop.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:ADH_Material_Properties_Prop.jpg License: unknown Contributors: Jcreer

File:ADH Material Properties Bound.jpg Source: http://www.xmswiki.com/xms/index.php?title=File:ADH_Material_Properties_Bound.jpg License: unknown Contributors: Jcreer

File:ADH Material Properties refine.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ADH_Material_Properties_refine.jpg *License:* unknown *Contributors:* Jcreer

Image:Right arrow.png *Source:* http://www.xmswiki.com/xms/index.php?title=File:Right_arrow.png *License:* unknown *Contributors:* Kwinters

Image:ADH Sedminet Library.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ADH_Sedminet_Library.jpg *License:* unknown *Contributors:* Jcreer

Image:ADH Vessel Node Att.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ADH_Vessel_Node_Att.jpg *License:* unknown *Contributors:* Jcreer

Image:ADH Vessel Att.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ADH_Vessel_Att.jpg *License:* unknown *Contributors:* Jcreer

Image:ADH Model Control Parameters.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ADH_Model_Control_Parameters.jpg *License:* unknown *Contributors:* Jcreer

Image:ADH Model Control Iterations.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ADH_Model_Control_Iterations.jpg *License:* unknown *Contributors:* Jcreer

File:ADH Model Control Time.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ADH_Model_Control_Time.jpg *License:* unknown *Contributors:* Jcreer

Image:ADH Model Control Output.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ADH_Model_Control_Output.jpg *License:* unknown *Contributors:* Jcreer

Image:ADH Model Control Global.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ADH_Model_Control_Global.jpg *License:* unknown *Contributors:* Jcreer

Image:ADH Model Control Transport.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ADH_Model_Control_Transport.jpg *License:* unknown *Contributors:* Jcreer

File:ADH Model Control Consolidation.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ADH_Model_Control_Consolidation.jpg *License:* unknown *Contributors:* Jcreer

Image:ADH Model Control Advanced.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ADH_Model_Control_Advanced.jpg *License:* unknown *Contributors:* Jcreer

Image:FESMWS Arc Attributes.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:FESMWS_Arc_Attributes.jpg *License:* unknown *Contributors:* Jcreer

Image:FESWMS Nodestring BC2.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:FESWMS_Nodestring_BC2.jpg *License:* unknown *Contributors:* Jcreer

Image:culvert.jpg *Source:* <http://www.xmswiki.com/xms/index.php?title=File:Culvert.jpg> *License:* unknown *Contributors:* Cjenkins

Image:select_node.gif *Source:* http://www.xmswiki.com/xms/index.php?title=File:Select_node.gif *License:* unknown *Contributors:* Jdhowlett

File:FESWMS Culvert.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:FESWMS_Culvert.jpg *License:* unknown *Contributors:* Jcreer

Image:wier.jpg *Source:* <http://www.xmswiki.com/xms/index.php?title=File:Wier.jpg> *License:* unknown *Contributors:* Cjenkins

File:FESWMS Weir.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:FESWMS_Weir.jpg *License:* unknown *Contributors:* Jcreer

File:FESWMS Drop Inlet.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:FESWMS_Drop_Inlet.jpg *License:* unknown *Contributors:* Jcreer

File:FESWMS Channel Link.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:FESWMS_Channel_Link.jpg *License:* unknown *Contributors:* Jcreer

File:FESWMS Gate.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:FESWMS_Gate.jpg *License:* unknown *Contributors:* Jcreer

Image:SelectMeshElement.gif *Source:* <http://www.xmswiki.com/xms/index.php?title=File:SelectMeshElement.gif> *License:* unknown *Contributors:* Jdhowlett

File:FESWMS Pier.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:FESWMS_Pier.jpg *License:* unknown *Contributors:* Jcreer

Image:FESWMS MP Roughness.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:FESWMS_MP_Roughness.jpg *License:* unknown *Contributors:* Jcreer

Image:FESWMS MP Turbulence.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:FESWMS_MP_Turbulence.jpg *License:* unknown *Contributors:* Jcreer

Image:FESWMS MP Wind.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:FESWMS_MP_Wind.jpg *License:* unknown *Contributors:* Jcreer

File:FESWMS Nodestring BC.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:FESWMS_Nodestring_BC.jpg *License:* unknown *Contributors:* Jcreer

File:FESWMS Local Parameters.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:FESWMS_Local_Parameters.jpg *License:* unknown *Contributors:* Jcreer

File:FESWMS Initial.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:FESWMS_Initial.jpg *License:* unknown *Contributors:* Jcreer

Image:FESWMS MC General.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:FESWMS_MC_General.jpg *License:* unknown *Contributors:* Jcreer

Image:FESWMS MC Timing.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:FESWMS_MC_Timing.jpg *License:* unknown *Contributors:* Jcreer

Image:FESWMS MC Parameters.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:FESWMS_MC_Parameters.jpg *License:* unknown *Contributors:* Jcreer

Image:FESWMS MC Print.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:FESWMS_MC_Print.jpg *License:* unknown *Contributors:* Jcreer

Image:FESWMS MC Wind.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:FESWMS_MC_Wind.jpg *License:* unknown *Contributors:* Jcreer

Image:FESWMS Point Attr.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:FESWMS_Point_Attr.jpg *License:* unknown *Contributors:* Jcreer

Image:FESWMS MC Sediment.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:FESWMS_MC_Sediment.jpg *License:* unknown *Contributors:* Jcreer

Image:SMS FESWMS Spindown.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:SMS_FESWMS_Spindown.jpg *License:* unknown *Contributors:* Jcreer

Image:SMS Steering Wizard.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:SMS_SteeringWizard.jpg *License:* unknown *Contributors:* Jcreer

Image:TABS Feature Point.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:TABS_Feature_Point.jpg *License:* unknown *Contributors:* Jcreer

Image:Assign 1D Geometry.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:Assign_1D_Geometry.jpg *License:* unknown *Contributors:* Jcreer

File:TABS Arc Attributes.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:TABS_Arc_Attributes.jpg *License:* unknown *Contributors:* Jcreer

Image:TotalFlow1.gif *Source:* <http://www.xmswiki.com/xms/index.php?title=File:TotalFlow1.gif> *License:* unknown *Contributors:* Jdhowlett

Image:TotalFlow2.gif *Source:* <http://www.xmswiki.com/xms/index.php?title=File:TotalFlow2.gif> *License:* unknown *Contributors:* Jdhowlett

File:RMA2 Nodal Transition.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:RMA2_Nodal_Transition.jpg *License:* unknown *Contributors:* Jcreer

File:RMA2 Rainfall Values.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:RMA2_Rainfall_Values.jpg *License:* unknown *Contributors:* Jcreer

Image:RMA2 Nodal BC.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:RMA2_Nodal_BC.jpg *License:* unknown *Contributors:* Jcreer

Image:RMA2 Assign BC.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:RMA2_Assign_BC.jpg *License:* unknown *Contributors:* Jcreer

Image:DisFactors.JPG *Source:* <http://www.xmswiki.com/xms/index.php?title=File:DisFactors.JPG> *License:* unknown *Contributors:* Cballard

Image:RMA2 MP Turbulence.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:RMA2_MP_Turbulence.jpg *License:* unknown *Contributors:* Jcreer

Image:RMA2 MP Roughness.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:RMA2_MP_Roughness.jpg *License:* unknown *Contributors:* Jcreer

Image:RMA2 MP March.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:RMA2_MP_March.jpg *License:* unknown *Contributors:* Jcreer

Image:RMA2 MP Wind.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:RMA2_MP_Wind.jpg *License:* unknown *Contributors:* Jcreer

Image:RMA2 MP Rainfall.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:RMA2_MP_Rainfall.jpg *License:* unknown *Contributors:* Jcreer

Image:RMA2 MP Coriolis.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:RMA2_MP_Coriolis.jpg *License:* unknown *Contributors:* Jcreer

Image:SMS RMA2 Revisions.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:SMS_RMA2_Revisions.jpg *License:* unknown *Contributors:* Jcreer

Image:RMA2 MC General.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:RMA2_MC_General.jpg *License:* unknown *Contributors:* Jcreer

Image:RMA2 MC Timing.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:RMA2_MC_Timing.jpg *License:* unknown *Contributors:* Jcreer

Image:RMA2 MC Files.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:RMA2_MC_Files.jpg *License:* unknown *Contributors:* Jcreer

Image:RMA2 MC Global.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:RMA2_MC_Global.jpg *License:* unknown *Contributors:* Jcreer

Image:RM2 MC Weather.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:RM2_MC_Weather.jpg *License:* unknown *Contributors:* Jcreer

Image:RMA2 Wind Values.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:RMA2_Wind_Values.jpg *License:* unknown *Contributors:* Jcreer

Image:SMS Steering Module RMA2 Spindown.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:SMS_Steering_Module_RMA2_Spindown.jpg *License:* unknown *Contributors:* Jcreer

File:RMA2 Roughness Options.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:RMA2_Roughness_Options.jpg *License:* unknown *Contributors:* Jcreer

File:RMA4_Assign_Element_Mass>Loading.JPG *Source:* http://www.xmswiki.com/xms/index.php?title=File:RMA4_Assign_Element_Mass>Loading.JPG *License:* unknown *Contributors:* Jdye

File:rma4 bc nodestrings.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:Rma4_bc_nodestrings.jpg *License:* unknown *Contributors:* Jcreer

Image:SMS RMA4 Assign Element Mass.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:SMS_RMA4_Assign_Element_Mass.jpg *License:* unknown *Contributors:* Jcreer

Image:RMA 4 MP.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:RMA_4_MP.jpg *License:* unknown *Contributors:* Jcreer

File:rma4 model control dialog 1.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:Rma4_model_control_dialog_1.jpg *License:* unknown *Contributors:* Jcreer

File:RMA4 MC Files.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:RMA4_MC_Files.jpg *License:* unknown *Contributors:* Jcreer

File:RMA4 MC Constituents.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:RMA4_MC_Constituents.jpg *License:* unknown *Contributors:* Jcreer

File:SMS Batch Run.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:SMS_Batch_Run.jpg *License:* unknown *Contributors:* Jcreer

Image:TUFLOW AD.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_AD.jpg *License:* unknown *Contributors:* Jcreer

Image:TUFLOW BC Node Attribute.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_BC_Node_Attribute.jpg *License:* unknown *Contributors:* Jcreer

Image:TUFLOW Arc BC.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_Arc_BC.jpg *License:* unknown *Contributors:* Jcreer

Image:TUFLOW Polygon BC.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_Polygon_BC.jpg *License:* unknown *Contributors:* Jcreer

Image:TUFLOW BC Events.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_BC_Events.jpg *License:* unknown *Contributors:* Jcreer

Image:TUFLOW 1D 2D linking mechanisms.png *Source:* http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_1D_2D_linking_mechanisms.png *License:* unknown *Contributors:* Rjones

Image:TUFLOW transitioning1Dcross 2Ddomain.png *Source:* http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_transitioning1Dcross_2Ddomain.png *License:* unknown *Contributors:* Rjones

Image:TUFLOW Grid Options.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_Grid_Options.jpg *License:* unknown *Contributors:* Jcreer

Image:Irregular Culvert Att.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:Irregular_Culvert_Att.jpg *License:* unknown *Contributors:* Jcreer

Image:2Dto2DLinkage.png *Source:* <http://www.xmswiki.com/xms/index.php?title=File:2Dto2DLinkage.png> *License:* unknown *Contributors:* Rjones

File:tufLOW links.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:TufLOW_links.jpg *License:* unknown *Contributors:* Jcreer

Image:TUFLOW MC 1D Auto.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_MC_1D_Auto.jpg *License:* unknown *Contributors:* Jcreer

Image:TUFLOW MP.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_MP.jpg *License:* unknown *Contributors:* Jcreer

Image:TUFLOW Cell Attributes.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_Cell_Attributes.jpg *License:* unknown *Contributors:* Jcreer

Image:TUFLOW MC Time.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_MC_Time.jpg *License:* unknown *Contributors:* Jcreer

Image:TUFLOW MC output.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_MC_output.jpg *License:* unknown *Contributors:* Jcreer

File:TUFLOW MC Wetting.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_MC_Wetting.jpg *License:* unknown *Contributors:* Jcreer

Image:TUFLOW MC Restart.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_MC_Restart.jpg *License:* unknown *Contributors:* Jcreer

Image:TUFLOW MC Water.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_MC_Water.jpg *License:* unknown *Contributors:* Jcreer

Image:TUFLOW MC Eddy.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_MC_Eddy.jpg *License:* unknown *Contributors:* Jcreer

Image:TUFLOW MC BC.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_MC_BC.jpg *License:* unknown *Contributors:* Jcreer

Image:TUFLOW MC Material.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_MC_Material.jpg *License:* unknown *Contributors:* Jcreer

Image:TUFLOW MC Misc.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_MC_Misc.jpg *License:* unknown *Contributors:* Jcreer

File:TUFLOW 1D MC General.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_1D_MC_General.jpg *License:* unknown *Contributors:* Jcreer

Image:TUFLOW MC 1D network.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_MC_1D_network.jpg *License:* unknown *Contributors:* Jcreer

Image:ZShape_failing_levée.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ZShape_failing_levée.jpg *License:* unknown *Contributors:* Jdhowlett

Image:ZShape_properties_dialog.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ZShape_properties_dialog.jpg *License:* unknown *Contributors:* Jdhowlett

Image:ZShape_trigger_point_dialog.jpg *Source:* http://www.xmswiki.com/xms/index.php?title=File:ZShape_trigger_point_dialog.jpg *License:* unknown *Contributors:* Jdhowlett