TWO-DIMENSIONAL FINITE-VOLUME HYDRODYNAMIC
MODEL FOR RIVER-MARSH SYSTEMS
TUTORIAL, Version 1.0
June 2002

J.D. Lin, W.G. Liao and K.-J. Qiu
Department of Civil and Environmental Engineering
M.W. Lefor
Department of Geology
University of Connecticut

JHR 02-277   Project 93-4

1. Report No.	2. Government Accession No.	3. Recipient's Catalog No.
JHR 02-277		

4. Title and Subtitle	5. Report Date
Two-Dimensional Finite-Volume Hydrodynamic Model for River-Marsh Systems: Tutorial, Version 1.0	June 2002
	6. Performing Organization Code

7. Author(s)	8. Performing Organization Report No.
J.D. Lin, W.G. Liao, K.-J. Qiu, M.W. Lefor	JHR 02-277

9. Performing Organization Name and Address	10. Work Unit No. (TRAIS)
University of Connecticut Connecticut Transportation Institute Storrs, CT 06269-5202	
	11. Contract or Grant No.

12. Sponsoring Agency Name and Address	13. Type of Report and Period Covered
Connecticut Department of Transportation 280 West Street Rocky Hill, CT 06067-0207	Tutorial
	14. Sponsoring Agency Code
	SPR-2214

15. Supplementary Notes

Conducted in cooperation with the U.S. Department of Transportation, Federal Highway Administration

16. Abstract

The Tutorial is a companion document of Report JHR 02-275, entitled *Two-Dimensional Finite–Volume Hydrodynamic Model for River-Marsh Systems: User's Manual*, which provides in detail the description of the model and its operational procedures. An Intel Pentium at 200MHz or above, running under Windows NT 4.0 or Windows 95 or 98 is required to run the model. This document is prepared for users who would like to have a hands-on experience of the model's operation before applying the model to simulate shallow water flow in a river-marsh system. Three examples with analytical solutions and a real site project are provided for practice. The step-by-step instruction will guide a user from creating a project for simulation to presenting the results. The model and all necessary data are provided in the disk included.

17. Key Words	18. Distribution Statement
Hydrodynamics, Model, Tidal Flow, Marsh	No restrictions. This document is available to the public through the National Technical Information Service Springfield, Virginia 22161

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of Pages	22. Price
Unclassified	Unclassified	37	N/A

Form DOT F 1700.7 (8-72)   Reproduction of completed page authorized

SI* (MODERN METRIC) CONVERSION FACTORS

APPROXIMATE CONVERSIONS TO SI UNITS

Symbol	When You Know	Multiply By	To Find	Symbol
LENGTH				
in	inches	25.4	millimetres	mm
ft	feet	0.305	metres	m
yd	yards	0.914	metres	m
mi	miles	1.61	kilometres	km
AREA				
in^2	square inches	645.2	millimetres squared	mm^2
ft^2	square feet	0.093	metres squared	m^2
yd^2	square yards	0.836	metres squared	m^2
ac	acres	0.405	hectares	ha
mi^2	square miles	2.59	kilometres squared	km^2
VOLUME				
fl oz	fluid ounces	29.57	millilitres	mL
gal	gallons	3.785	Litres	L
ft^3	cubic feet	0.028	metres cubed	m^3
yd^3	cubic yards	0.765	metres cubed	m^3

NOTE: Volumes greater than 1000 L shall be shown in m^3

Symbol	When You Know	Multiply By	To Find	Symbol
MASS				
oz	ounces	28.35	grams	g
lb	pounds	0.454	kilograms	kg
T	short tons (2000 lb)	0.907	megagrams	Mg
TEMPERATURE (exact)				
°F	Fahrenheit temperature	5(F-32)/9	Celcius temperature	°C

APPROXIMATE CONVERSIONS TO SI UNITS

Symbol	When You Know	Multiply By	To Find	Symbol
LENGTH				
mm	millimetres	0.039	inches	in
m	metres	3.28	feet	ft
m	metres	1.09	yards	yd
km	kilometres	0.621	miles	mi
AREA				
mm^2	millimetres squared	0.0016	square inches	in^2
m^2	metres squared	10.764	square feet	ft^2
ha	hectares	2.47	acres	ac
km^2	kilometres squared	0.386	square miles	mi^2
VOLUME				
mL	millilitres	0.034	fluid ounces	fl oz
L	litres	0.264	gallons	gal
m^3	metres cubed	35.315	cubic feet	ft^3
m^3	metres cubed	1.308	cubic yards	yd^3
MASS				
g	grams	0.035	ounces	oz
kg	kilograms	2.205	pounds	lb
Mg	megagrams	1.102	short tons (2000 lb)	T
TEMPERATURE (exact)				
°C	Celcius temperature	1.8C+32	Fahrenheit temperature	°F

Temperature scale:

°F -40   0   32   98.6   212
°C -40 -20   0   37   100

* SI is the symbol for the International System of Measurement

TABLE OF CONTENTS

LIST OF FIGURES

1. Examples with Analytical Solutions

Two theoretical sample applications are explained here in detail in order to familiarize the user with the process of using this two-dimensional finite-volume hydrodynamic model (FVHM) for simulating shallow water flows over a river-marsh system. First, the FVHM will be used to simulate a one-dimensional Riemann problem, and the result will be compared with that of an analytical solution. The second application is an open gate example which has been used in many other similar studies. The result of the open gate simulation will be compared with results from other studies in the literature.

A. 1-D Riemann Problem

A one-dimensional open channel flow formulated with shallow water approximations is generally known as a one-dimensional Riemann Problem. This example deals with opening of a gate or breaking of a dam in the middle of a straight, rectangular, open channel. Initially, the water surface forms a step at this location of the gate. No-flow or a vertical wall is considered as the boundary condition at both ends of the channel.

From the folder where you saved the files from the FVHM system diskettes, you may start the model by double-clicking the FVHM icon. This will bring up the **Main Window**, initially with an empty **Workspace** as shown in Figure 1.1.

To solve the 1-D Riemann problem, the channel can be represented by a grid of a single row of computational cells shown below:

Follow the procedure below to create a new tutorial project:

1. Create a new project by using **File|New**. The new project is named "*Untitled*" by default. You may proceed with the default. If you want to name the project now, click on the **Workspace** and then click **File|New** again. The **Estu** dialog appears; click **Yes** to bring up the **Save as** dialog to save the project with a name you choose. If there is a project already loaded and it has been modified, you will be prompted to save the changes.

2. Specify project options using the **Project Options** dialog brought up by clicking **Project|Options** or pressing **F8**. The **Project Options** for a 1-D Riemann problem should appear, and you will enter the items in the boxes as shown in Figure 1.2. Note

that the display device size might be different, depending on the computer you are using to run the model. Click **OK** to close the dialog.
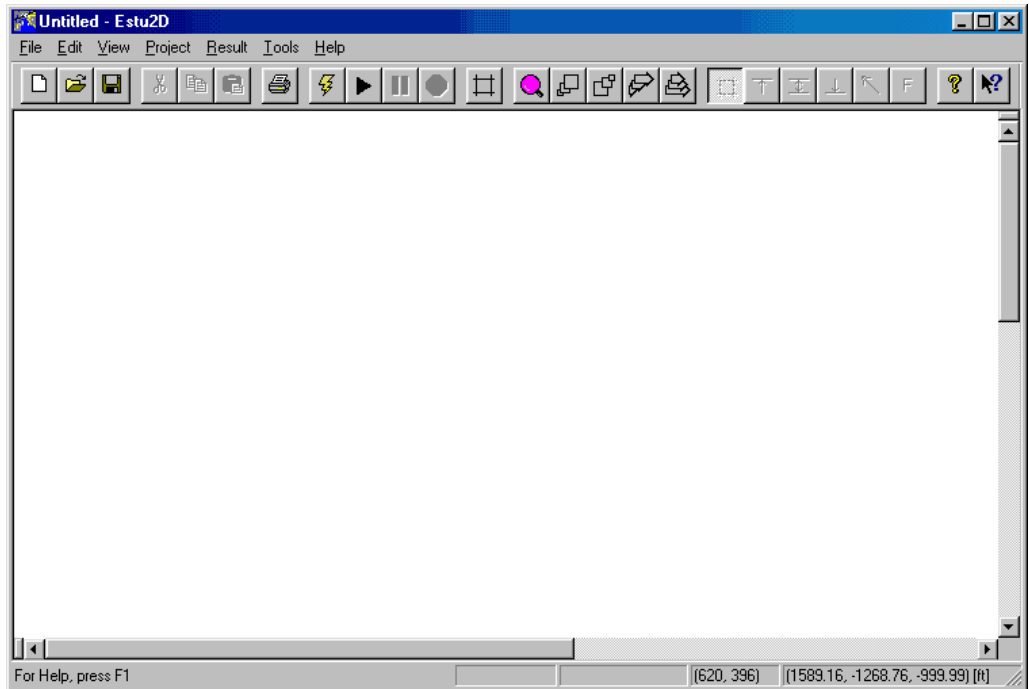

Figure 1.1. The FVHM Main Window
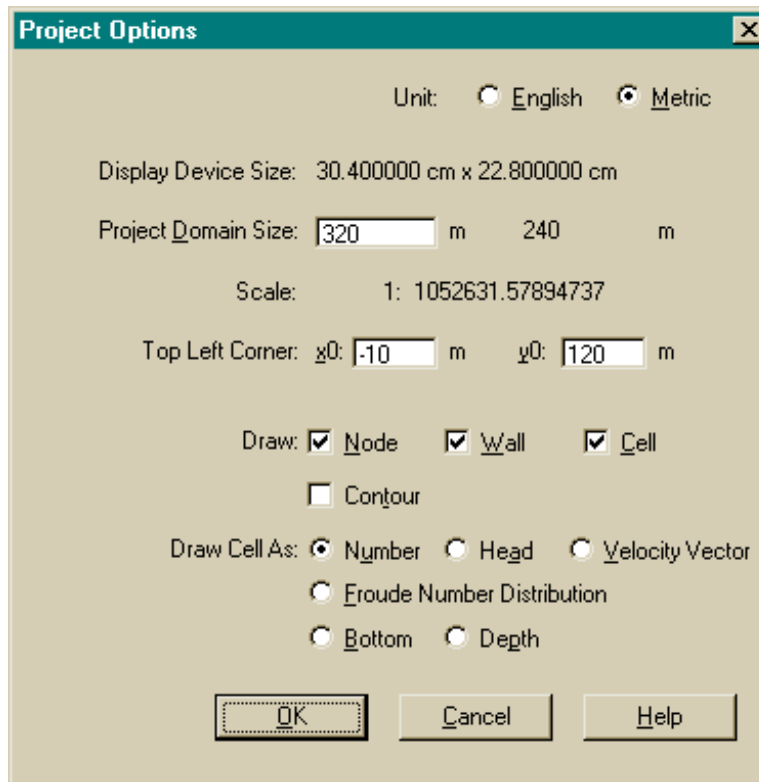
Figure 1.2. Project Options for 1-D Riemann Problem

3. Right click the **Workspace** and bring up the **Context Menu** and select **New Cell** menu item. You are now in the create new cell mode.

4. Create a new cell by clicking the **Workspace** five times to form a closed loop. This will define the corners (nodes) of the computational cell you are creating. Make sure that you position the first and last clicks at the same location. A new cell will be created at the last (fifth) click, when you close the loop. At this time it does not matter what the size and shape of the cell are because you will be able to edit it later using the **Editing** dialogs. Now, you have a cell in the **Workspace**. The number shown in the cell is the cell serial number. (See Figure 1.4.)

5. Right click inside the first node (serial no. 1) of the newly created cell to bring up the **Node** editing dialog as shown in Figure 1.3.
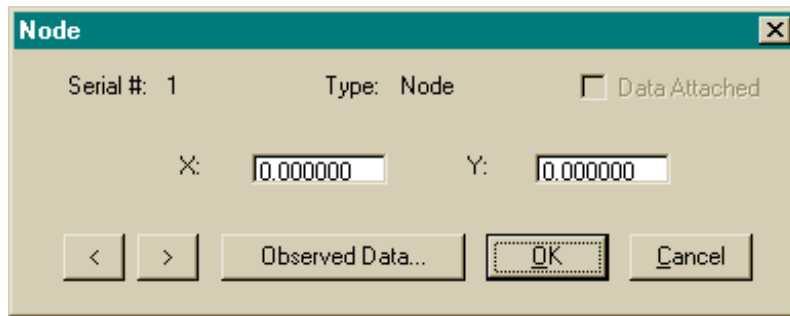
Figure 1.3. Node Editing Dialog

6. Enter 0 for X and 0 for Y, and then click the > button. You should see that the first node moves to a new location, (0, 0). For the next node (serial no. 2), input 5 (meters) for X, 0 for Y, and then click the > button. For the third node (serial no. 4), input 5 for X, 5 for Y, and then click the > button. For the fourth node (serial no. 6), input 0 for X, 5 for Y, and then click the > button. This should bring you back to the first node (serial no. 1) and the cell should become a polygon, in this case a perfect square, displayed near the left lower corner of the **Workspace**. Click **OK** to close the dialog.

Figure 1.4. First Cell

7. Starting with this first cell, you can now use **Copy** and **Paste** to create the rest of the cells for the 1-D Riemann problem. Right click the **Workspace** and bring up the **Context Menu**. Choose the **Select** menu item. Left-click inside the new cell to select it. Bring up the **Context Menu** again and select **Copy**, which will mark the selected cell for copying. Bring up the **Context Menu** once more and select **Paste**. Now when you move the mouse, you should see an outline of the selected cell following the cursor. Move this outline to the right side of the first cell and line the two up. Click the left mouse button and a new cell will be created to the right side of the first cell.

8. Repeat the above **Copy** and **Paste** operation until 60 cells are lined up across the **Workspace**. (To do this more quickly, select five cells after you create five, then copy and paste five cells at a time.) In this case the total length of the 60 cells should be 300 meters because you specified the metric system in the **Project Options** dialog and the side of each cell is 5 meters long. The grid now should look like the grid shown in Figure 1.7 in the **Workspace**.

9. Now, save your work by clicking the **Save** button, saving the grid in the file, **tutorial1.est**. This grid which can be used to solve the 1-D Riemann problem.

10. Next, you must specify the initial condition of the water surface across the 300-meter channel. The bottom elevation of all cells is set at 0 meters by default, as is the initial water surface elevation. The gate open or dam break situation requires that an initial water surface drop be created as the initial condition in the middle of the channel. Select the leftmost 30 cells and choose the **Edit Grid** menu item from the **Context Menu**. The **Cell Property Edit Dialog** will be brought up. (See Figure 1.5) Assign head a value of 5 (meters) and check the checkbox to its right. Click **OK** to close the dialog. This procedure will assign the left 30 cells a water surface elevation of 5 meters above the bottom. Select the rightmost 30 cells and give them a head of 0.25 meters following the same procedure.

11. To see a depiction of the water elevation on your screen, click the **Toolbar** button ⊤.
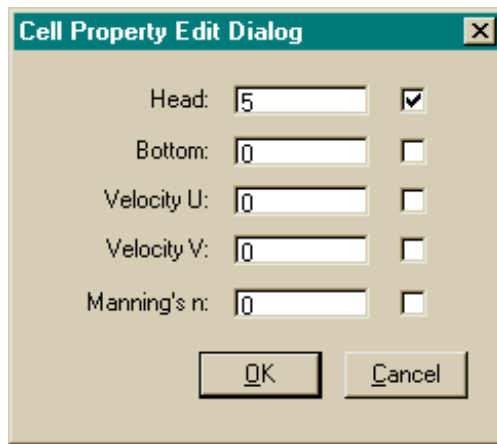
Figure 1.5. Cell Property Edit Dialog

12. For the 1-D Riemann problem you do not have to specify any boundary conditions. The default boundary condition is set for no-flow on all sides, which is just what is needed for this problem. Please remember to save again.

13. To start running the calculation, click the **Toolbar** button ⚡. The **Modeling Options** dialog will be brought up as shown in Figure 1.6. The dialog will show the current system time in your computer, which will be used as the starting time. Therefore, the time values in the dialog shown on your computer will be different from the values shown in Figure 1.6. Add 10 seconds to the ending time to have the program run 10 seconds from the time the water starts to flow. Also type in the other entries as shown in the dialog. Click **OK** and let the program run.
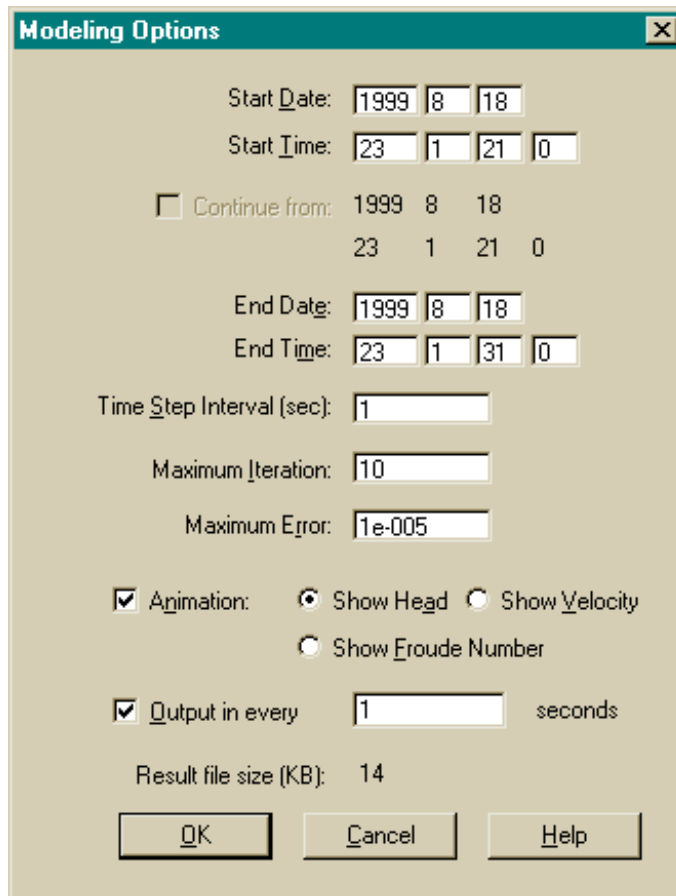
Figure 1.6. Modeling Options Dialog

14. The calculated result of the velocity field should look like that shown below in Figure 1.7 (**Show Velocity** enabled):

Figure 1.7. Velocity Field 10 Seconds After Flow Starts

The bar radiating out from the center of each cell represents the magnitude (bar length) and direction of the velocity.

Using the **Toolbar** buttons, other solutions, such as velocity vectors, Froude number, etc. may also be displayed in the grid. If you now check the folder where you saved **tutorial1.est**, you will find that the calculation results of the 10-minute run have been saved as **tutorial1.r00**. This file can not be viewed via the model, because it is a binary file. However, after the simulation finishes, you may export the result time history of

the water surface elevations and velocity components at selected cells to a text file for post-processing. To do this, first select the cells you want to export and then select **Result|Time History|Export**. You will be prompted to name this exported text file and the folder to save it in. This file will have the extension **.tmh.** You may open this file for review now or later. It consists of three sets of data: Head, U-, and V-components of velocity in the selected cells over the run time. The size of the file depends on the duration and the frequency of output that you have specified in the **Modeling Options** as shown in Figure 1.6. It is usually large. In each set, the heading of Head, U or V is given in the position of first row and first column. Following the heading, serial numbers of the selected cells, separated by spaces, are listed in the first row. From the second row on, the first column is date and time, and the rest are values for Head, U or V in columns corresponding to cell serial numbers. A small sample of a result history file in ASCII format is shown below:

```
Head                    9               159             225
2000 01 19 20 37 37     5               5               0.25
2000 01 19 20 37 38     5               4.9737          0.250067
2000 01 19 20 37 39     5               4.82315         0.26218
2000 01 19 20 37 40     5               4.57209         0.458918
2000 01 19 20 37 41     5               4.30033         0.868783
2000 01 19 20 37 42     5               4.05146         1.18513
2000 01 19 20 37 43     5               3.83808         1.33994
2000 01 19 20 37 44     5               3.6603          1.40224
2000 01 19 20 37 45     4.99996         3.51277         1.43056
2000 01 19 20 37 46     4.99975         3.38912         1.4501
2000 01 19 20 37 47     4.99895         3.28477         1.46904


U                       9               159             225
2000 01 19 20 37 37     0               0               0
2000 01 19 20 37 38     4.16599e-011    0.0367039       0.000421756
2000 01 19 20 37 39     6.44543e-011    0.246589        0.0891948
2000 01 19 20 37 40     7.55947e-011    0.601752        1.77004
2000 01 19 20 37 41     4.57204e-010    0.997408        4.24334
2000 01 19 20 37 42     1.55706e-008    1.37218         5.51405
2000 01 19 20 37 43     2.97451e-007    1.70425         6.04482
2000 01 19 20 37 44     3.90669e-006    1.9887          6.24917
2000 01 19 20 37 45     3.28045e-005    2.23055         6.3188
2000 01 19 20 37 46     0.000174817     2.43801         6.32911
2000 01 19 20 37 47     0.000674845     2.61668         6.30906


V                       9               159             225
2000 01 19 20 37 37     0               0               0
2000 01 19 20 37 38     -4.8511e-011    -4.83992e-011   -2.42994e-012
2000 01 19 20 37 39     -9.7022e-011    -9.58086e-011   -5.85129e-012
2000 01 19 20 37 40     -1.45533e-010   -1.41306e-010   -3.23205e-011
2000 01 19 20 37 41     -1.94044e-010   -1.84817e-010   -8.03447e-011
2000 01 19 20 37 42     -2.42555e-010   -2.26706e-010   -1.21225e-010
2000 01 19 20 37 43     -2.91066e-010   -2.67368e-010   -1.56611e-010
2000 01 19 20 37 44     -3.39577e-010   -3.06982e-010   -1.90189e-010
2000 01 19 20 37 45     -3.88088e-010   -3.45807e-010   -2.23672e-010
2000 01 19 20 37 46     -4.36597e-010   -3.84142e-010   -2.57545e-010
2000 01 19 20 37 47     -4.851e-010     -4.22095e-010   -2.91841e-010
```

To validate the results, the water surface elevation has been compared with the analytical solution of the 1-D Riemann problem (Tan, 1992)*. It indicates good agreement, as shown in the Figure 1.8.
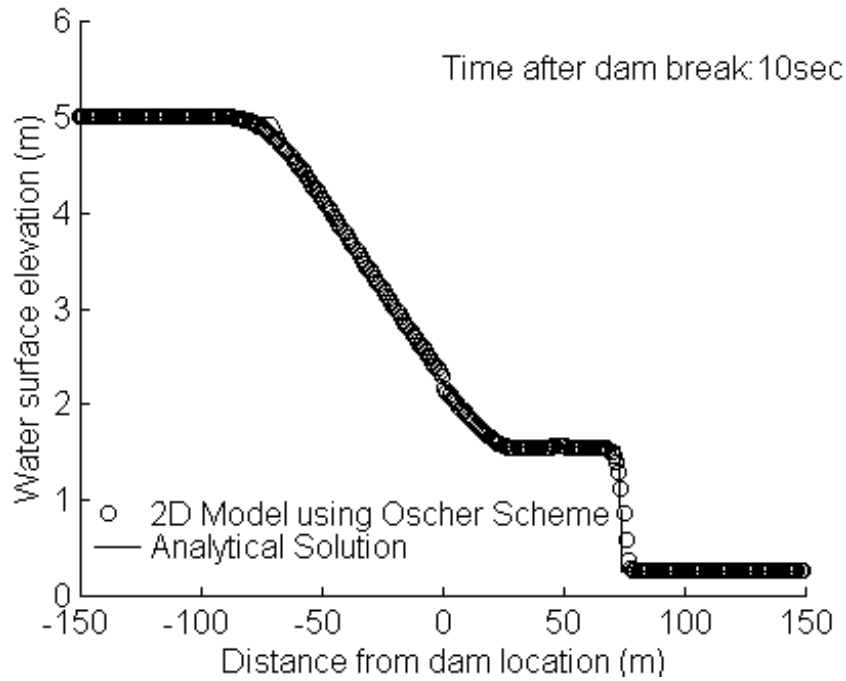

Figure 1.8. Comparison Between Simulated Results and the Analytical Solution

* Tan, W. Y., 1992. Shallow Water Hydrodynamics, Elsevier Oceanography Series Vol. 55, Elsevier, Amsterdam.

B. Involuntary Movement Problem

Bottom slope inside the simulation region has an important effect on the flow pattern throughout the computational domain. One problem associated with numerical models when dealing with bottom slope, however, is the involuntary movement problem. In the case of a confined area flooded with water, the water surface remains still even with bottom slope variations. But because a numerical model introduces round-off errors, the simulated result might show an involuntary movement of water over time if the bottom slope has not been treated properly.

This example deals with a rectangular basin that has a level bottom with an inclined trough running from one end to the other along the centerline of the basin. The basin is filled with water. The calculation starts with a level water surface and proceeds forward in time. If involuntary movement is absent in the FVHM, the water surface should remain level at all time. The grid used for testing the bottom slope effect in the present model is shown in Figure 1.9. The entire computational domain represents 800x600 square meters and each cell represents an area of 50x50 square meters.
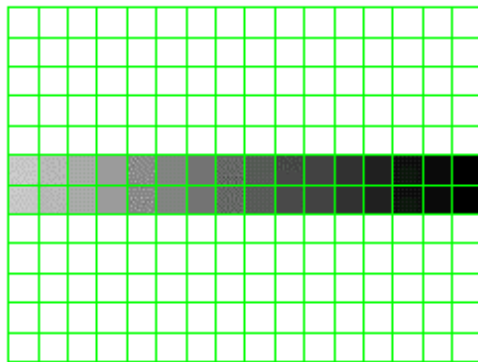

Figure 1.9. Involuntary Movement Problem

To construct this grid, follow the same procedure outlined earlier for the 1-D Riemann problem example. For this problem, your **Project Options** should look like those shown below in Figure 1.10.
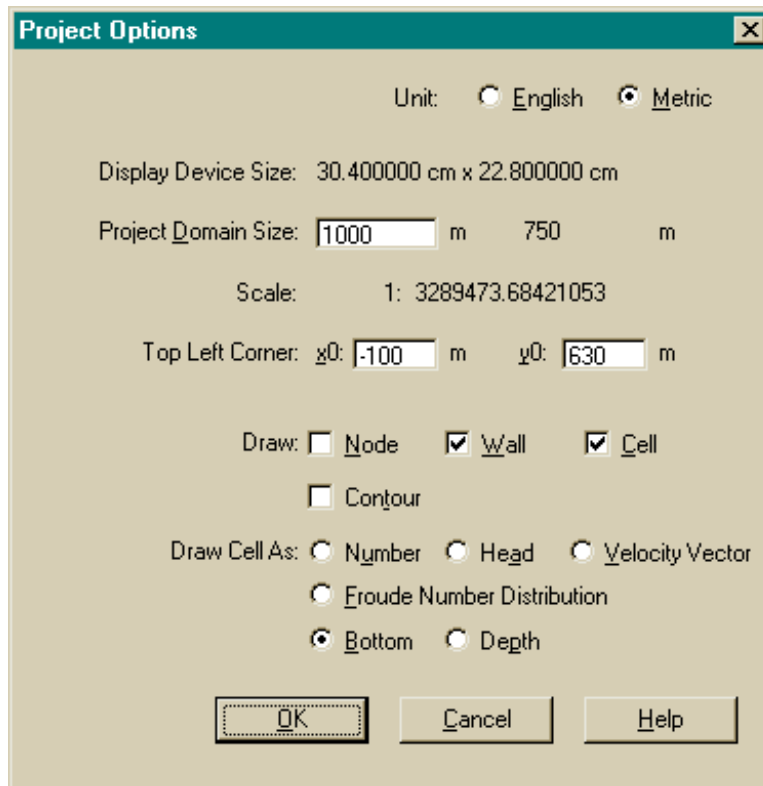
Figure 1.10. Project Options for Involuntary Movement Problem

The procedure is as follows:

1. Construct the first cell with its four nodes at (0, 0), (50, 0), (50, 50) and (0, 50).

2. Use **Copy** and **Paste** to complete the rest of cells to cover the whole domain as shown in Figure 1.9.

3. Right-click the leftmost cell of row 6 and select **Edit Grid** from the **Context Menu**. In the **Cell** property edit dialog, specify a value of -0.4 for its bottom elevation and check.

4. Right-click the rightmost cell in row 6 and specify a value of -1.9 for its bottom elevation as above.

5. Right-click and choose **Select** from the **Context Menu**, and then select row 6 in by dragging the mouse from left to right over the cells on row 6.

6. Select **Tools|Linear Interpolate.** In the **Select Property** dialog check the **Bottom** box (see Figure 1.11), and click **OK**. You should now see that the bottom elevation has been linearly interpolated over the entire row.
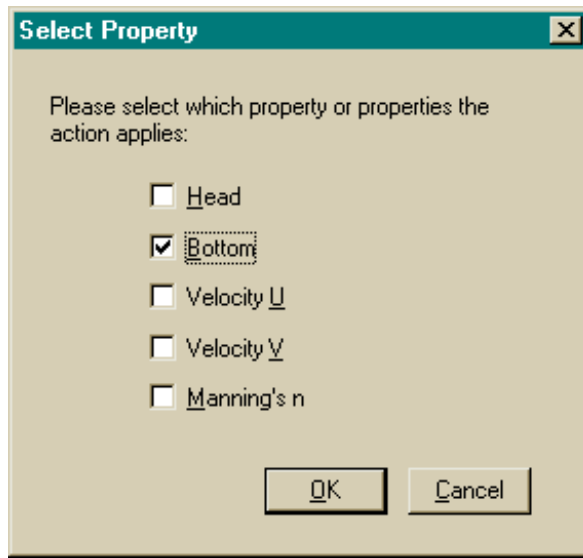
Figure 1.11. Linearly Interpolating the Values in End Cells

7. Repeat the above procedure for row 7. Next, you need to specify a uniform water elevation of 5 meters for the entire computational domain as follows.

8. First, right-click the mouse to select the entire computational domain.

9. Right-click again and select **Edit Grid** to bring up the **Cell Property Edit Dialog.** (See Figure 1.12)

10. Specify a head of 5.0 meters and check the check box for **Head.** Click **OK** to close the dialog.

Figure 1.12. Assigning a Head Value of 5 to All Selected Cells

11. Now save the project as **tutorial2.est** and the user is ready to run the simulation.

Run the simulation for as long as you like; there should be no involuntary water movement.

C. Gate Open Problem

The computational domain of the Gate Open Example is a square basin of 200m x 200m with a level bottom. A solid wall (the white space) with 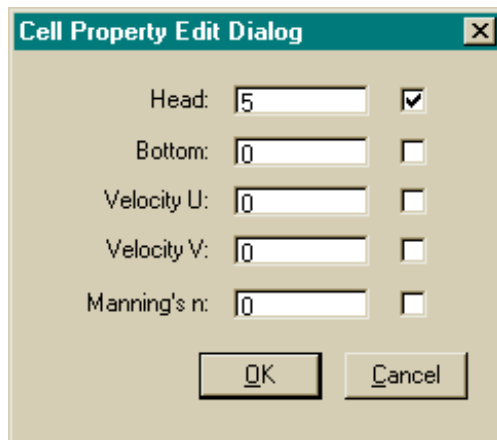an opening (the colored portion) is placed across the middle of the basin as shown in Figure 1.13. The opening is placed off-center and extended to the bottom of the basin, which will introduce asymmetry in flow. Initially, the gate in the opening is closed, and a differential head exists between the upstream (left) and downstream (right) water levels. At time zero, the gate opens instantaneously, and water in the upstream basin gushes through the opening creating a supercritical flow downstream of the gate in the lower basin. (See Figure 1.13)
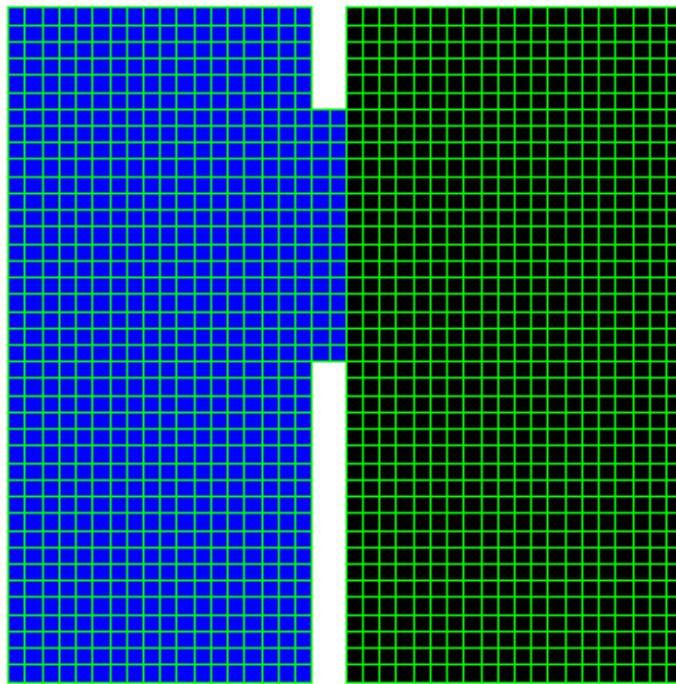
Figure 1.13. Gate Open Example

This exercise starts with the construction of the computational cells, following the procedure outlined in the previous two examples.

1. Open a new project.

2. Specify the entries in the **Project Options** as shown in Figure 1.14.

3. Create the first cell and specify the locations of its four nodes at (0, 0), (5, 0), (5, 5) and (0, 5).

4. Using the cell just created, use **Copy** and **Paste** to make up a 40 x 40 grid.

5. The cell coordinates at the lower left corner are row 1 and column 1 of the grid. Select the cells at column 199, rows 1 to 19 and 35 to 40; column 20, rows 1 to 19 and 35 to 40, and delete them. To select cells, right-click and choose **Select** from the **Context Menu**. Click and drag over the cells you want to select. Clicking a cell again will deselect it. To delete cells, right click and choose **Delete** from the **Context Menu**.
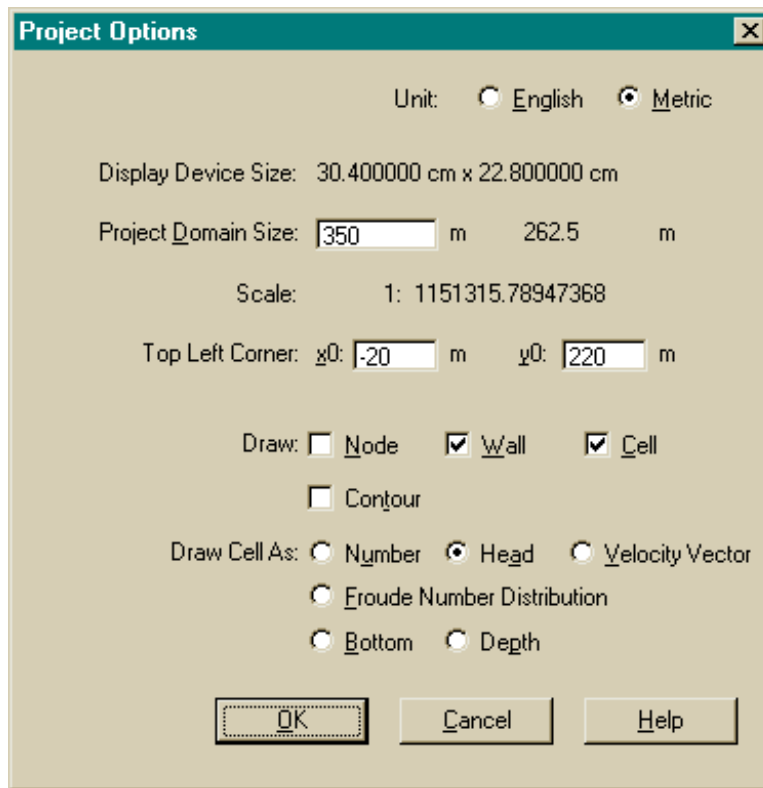
Figure 1.14. Project Options for Gate Open Problem

6. Now save the project as **tutorial3.est**.

7. Right-click and choose **Select** from the **Context Menu**. Left-click and hold the mouse button just outside the cell in row 1 and column 1; then, drag the mouse cursor and enclose all the cells in column 1 through 20 in the select box (vignette box). Release the left mouse button, and you will have the selected the cells. Right-click and choose **Edit Grid** from the **Context Menu**. Assign the selected cells a head of 10 meters and check the **Head** check box, and then click **OK**. This will assign a head value of 10 meters to all the cells upstream of the gate. Follow the same procedure to select all the downstream cells, and specify a head of 5 meters to those cells.

8. Save the **Project Options** and the project is ready to run.

9. Run the project using the **Modeling Options** shown in Figure 1.15. For later validation, let the calculation run for 7.1 seconds. Again, the results will be saved in **tutorial3.r00**.

Figure 1.15. Modeling Options for Gate Open Problem

Figure 1.16 shows the distribution of water surface elevation along a line (row 27) from upstream to downstream through the center of the gate at 7.1 seconds after the gate is opened. In the figure, a comparison is also made with the analytical solution for the same problem.

D. Summary

The above three examples, although simple, have demonstrated almost all of the basic features of FVHM. If you have completed the exercises successfully, you will have acquired the necessary rudimentary skill to use the FVHM. For a complete and detailed explanation of all aspects of the model, please refer to the **User's Manual,** and continue on to next example, one from a real-world site.
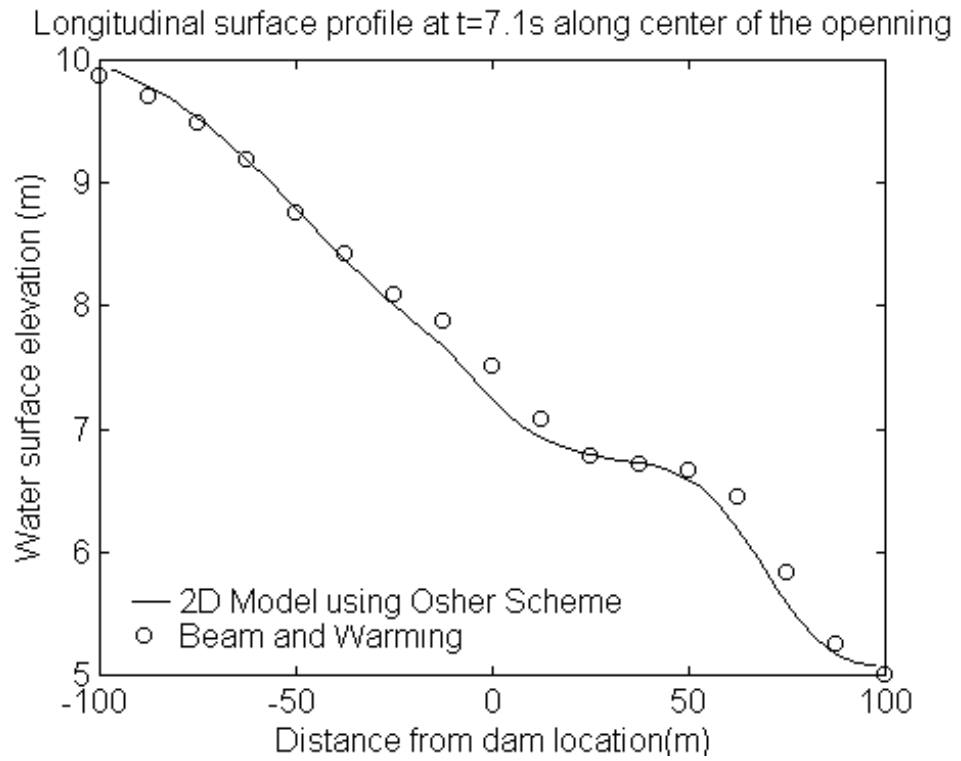
Figure 1.16. Comparison of Analytical and Calculated Results

2. Real Site Simulation

Building a modeling project for a real site simulation can be much more complicated than the examples given above. The following example is based on a coastal river and marsh system consisting of a portion of a river, tributary channels, mosquito ditches, and salt marsh in Westbrook, Connecticut. The site was first surveyed to the Connecticut Coordinate Grid System, which included the (x, y, z) coordinates of a section of the Menunketesuck River, boundaries of the marsh, locations and cross-sections of channels, surface of the marsh, tide staffs, etc. Such a task can be simplified somewhat by the use of high-definition GPS equipment and(or) a laser theodolite. In all cases careful precautions must be taken when working on unstable substrates like marsh peat to avoid gross inaccuracies in the field data. (See the Final Report for JHRAC Project 93-04).

The site was also chosen for calibration and validation of FVHM; therefore, field observed data from this site are available. Outline and boundary condition data are included in the system diskette for you to use in this tutorial. These data were tabulated as ASCII files output from AutoCAD.

The first type of data FVHM can use in initially defining a project is outline data. Outline data are sets of point coordinates (x, y, z), which provide the topography for the interior and the boundaries of the computational domain. In this case, the data consist of surveyed points on the marsh, in the channels, and its boundaries of the site; in other cases, the data may be derived from topographic maps or a combination of both. Once imported into the model, the outline data can be used to define the boundaries of the computational domain and in generating the grid for the site. The second type of data is boundary condition data, which provide the boundary conditions for driving the model through the calculation period.

It is also recommended that you try to compare your calculated results for tidal stage and(or) current with observed data as a way to learn how to generate an effective computational grid for further applications.

Project Options

A new project is always created by first bringing up the **Main Window,** and assigning a file name; then, saving it as a file, here, **tutorial4.est**. Then, bring up the **Project Options** dialog to type in your project entries and to check your options before you do anything else. Figure 2.1 shows the entries in the **Project Options** for the Westbrook site.
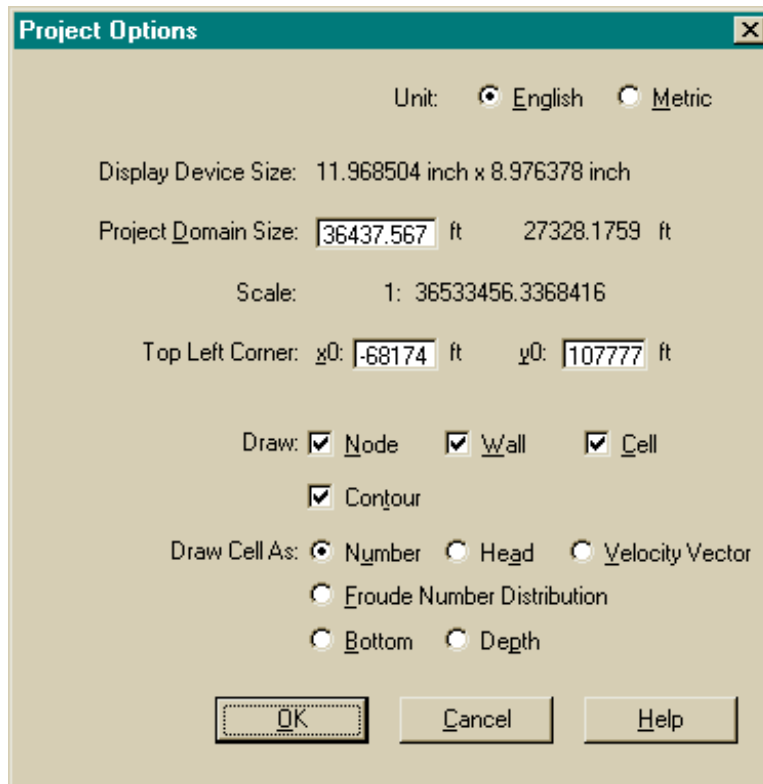
Figure 2.1. Project Options for Westbrook Site

B. Import Outline Data

The outline data file used in this model has the **.oln** extension. A sample section of an outline file for import to the project is given below:

661904.154 1075664.550 2.160

662304.019 1073868.730 1.540

662757.129 1075789.082 1.660

663208.396 1075862.093 2.990

662528.066 1074158.698 2.930

in which the values for (x , y, z) are separated by spaces.

After specifying the entries in **Project Options**, choose **File|Import|Outline** and select the outline file to be imported into the project. After importing, the outline points should be visible in the **Workspace** as shown in Figure 2.2. The check box in

the **Project Options** dialog in Figure 2.1 labeled **Contour** must be checked in order to view the outline.

Figure 2.2. Outline Import for Westbrook Site

You can use **View|Toggle Link** to adjust links if necessary. This will either draw or not draw lines between outline points.

C. Coarse Grid

Based on the outline points now in the **Workspace**, you can start constructing a coarse grid by dividing the domain into large sections as cells, as shown in   Figure 2.3 (the outline data are hidden). Note that in thi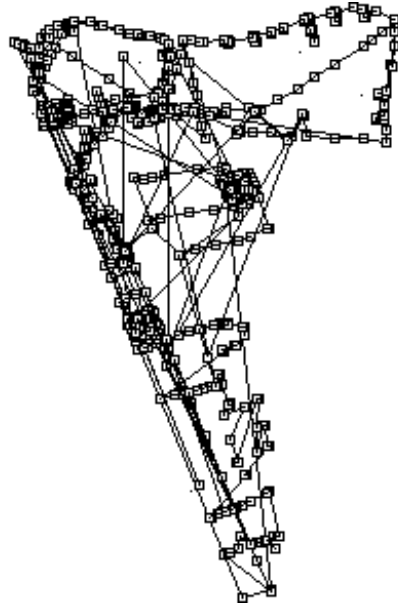s grid, large cells are formed as many-sided polygons, which are not suitable for use in the model, but this is an initial step in constructing a grid for a complex computational domain. This coarse grid will be edited into a fine grid as shown in Figure 2.4.

The procedure here is first to get into the create **New Cell** mode by right click in the **Workspace** to bring up the **Context Menu** and selecting **New Cell**. Then, click continuously to form each cells each of which outlines a section as shown in Figure 2.3. It is important to carefully select where you click to position node points (which may or may not be outline points) and also not to create cells that overlap each other. Note also that the imported outline points, in some cases, may not be sufficient to define the topographic features of the site; therefore, the selection of cells should be guided by a topographic map.
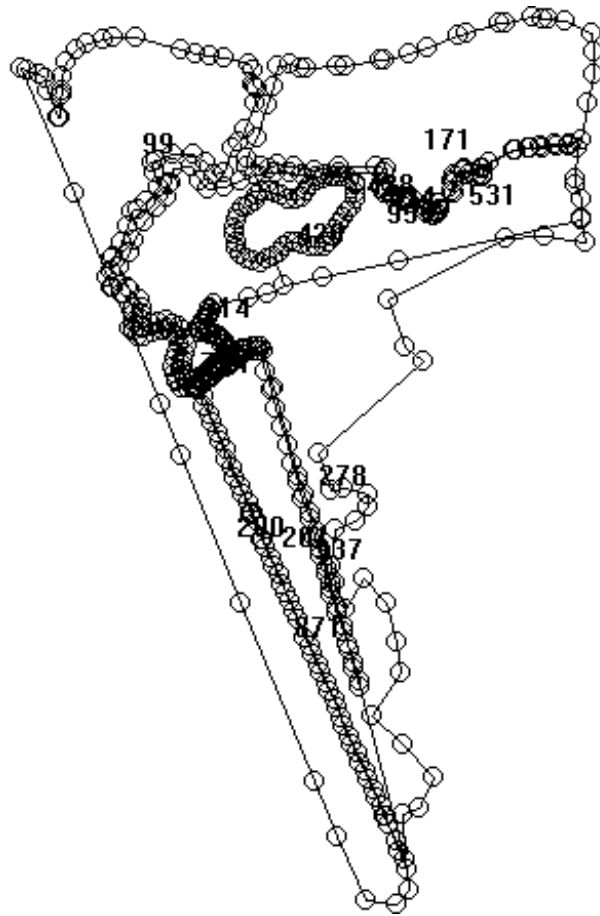
Figure 2.3. Coarse Grid for Westbrook Site

D. Fine Grid

By splitting the walls and the cells, you can arrive at a fine grid as shown in Figure 2.4. In this step, bottom elevations and other cell attributes are assigned to each cell. The attributes of a cell include center coordinates (x, y), head, bottom elevation, and Manning's n. To split a cell, select the **Edit|Split** cell menu and click the node of a cell where you want the split to start. Then, you may click inside each cell to make connecting new nodes and walls. When you click another node of the same cell, the cell will be split. In other words, you create new nodes and walls connecting two nodes of the existing cell. Selected cells may be removed or combined by simply clicking **Delete** or **Merge Cells** in the **Context Menu** or in the **Edit** menu on the **Menu Bar**. Selected walls can be similarly edited. The following tips may help you in these procedures:

    1.   Assign cell attributes before you split a cell. This will result in new cells having the same attribute values as the original cell.

2. Select a group of cells and assign attributes to all of them at the same time.

3. Use **Tools|Linear Interpolate** to automatically assign interpolated attribute values to selected cells.

4. Move the mouse cursor to a cell, and its attribute values will be displayed on the status bar. The value of the attribute being displayed is determined by the selection of the current view icons on the toolbar, such as water surface elevation, velocity, etc.
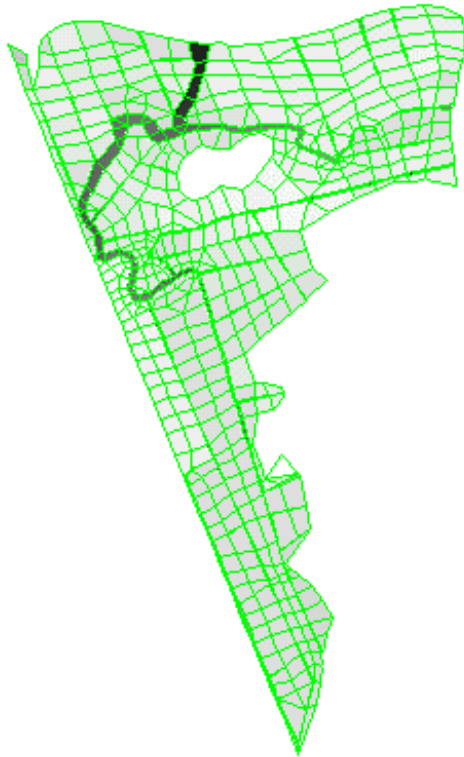
Figure 2.4. Fine Grid for Westbrook Site with Bottom Elevations Shown

E. Boundary Conditions

Boundary conditions of the boundaries enclosing the computational domain can be either fixed, such as the no-flow (Type 2) condition for the upper, landward banks of a marsh, or variable, such as tidal stages (Type 1) or flow (Type 3) at open boundary sections. To apply boundary conditions, you must first specify boundary sections. The first three examples in this Tutorial used the default boundary condition (no-flow), and it was not necessary to specify boundary sections and boundary conditions.

Boundary Sections

A boundary section is a continuous boundary line that has the same boundary condition type. One project grid may have many boundary sections and the overall grid boundaries consists of all boundary sections combined.

The boundary sections for the fine grid in Figure 2.4 are shown in Figure 2.5. The Boundary Walls box on the left lists all of the boundary wall serial numbers in the first column, boundary section number in the second, and boundary condition type in the third. A boundary wall attaches to only one cell, whereas a non-boundary wall attaches to two cells.

Use **Edit|Boundary Section** to bring up the **Boundary Sections** dialog for specifying boundary sections. The boundary wall serial numbers in the box are created when the grid is created. Initially, no boundary sections are specified. The second column will be all 0's and the third column all 2's (no-flow condition) by default. To edit the list, select several walls from the left box and the corresponding walls on the boundary of the grid will be highlighted in the **Workspace**. Check the consistency of the section before clicking the >> button in the middle to transfer them to the Section Walls box on the right. Supply a Section number and check a boundary condition **BC Type**, and click the **Set** button. A new boundary section will be specified in the boundary **Walls** box, together with **BC Type**. Repeat this procedure for all the boundary sections to complete the list in the box as shown in Figure 2.5, and click **OK** to save it.
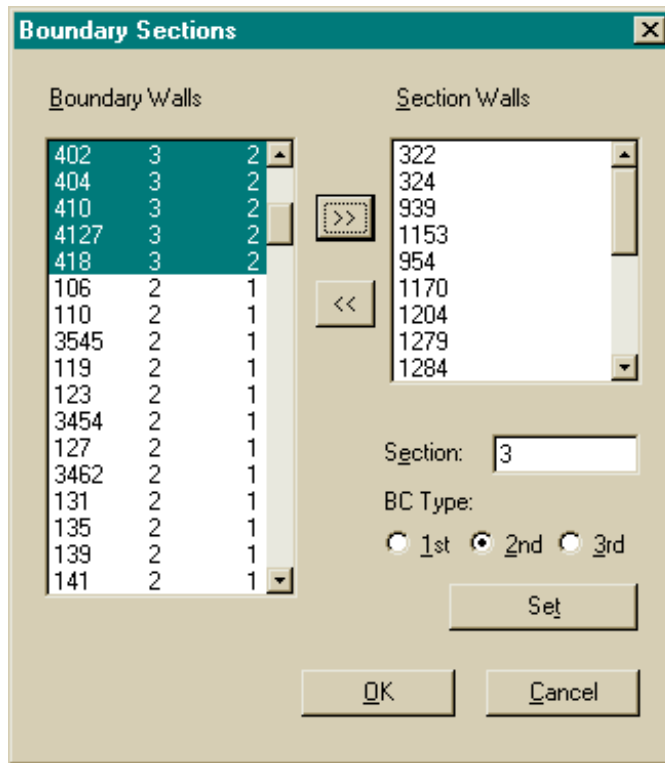
Figure 2.5. Boundary Sections Edit Dialog

The three boundary selections in this Tutorial problem should be highlighted in the **Workspace** as shown in Figure 2.6 through Figure 2.8. When you edit for a section, it is highlighted. Then, when you edit the next section, the new one is highlighted and the previous highlight is removed. Several conditions must be met to successfully specify a boundary section:

1. the section must consist of a single, continuous line segment;

2. sections must not overlap;

3. each section must have its own section number; and,

4. each section may only have one boundary condition type.

The **Boundary Sections** edit dialog checks these conditions before accepting a boundary section into the model.

Section 1 in Figure 2.6 consists of all of the boundary walls in the lower half of the computational domain. The boundary condition is Type 2, the no-flow condition. Section 2 , in Figure 2.7, consists of all of the boundary lines in the upper half of the

computational domain. The boundary condition is Type 1, observed water surface elevation vs. time. These values have to be specified at the selected locations (see next section). Section 3, in Figure 2.8, is the boundary of the central island in the marsh. It has Type 2, the no-flow boundary condition.

Figure 2.6. Boundary Section 1 (Type 2) at Westbrook Site

Boundary Conditions

Each boundary section may have zero (default), one or more boundary conditions specified for it, but the type of boundary condition must match that of the boundary section. FVHM uses boundary condition files to associate boundary conditions with boundary sections. A typical boundary condition file for a boundary section may look like the following :

```
1 10 0
1996 10 26  5 37 59 -0.39
1996 10 26  6  6 23 -0.33
1996 10 26  6 34 47 -0.15
1996 10 26  7  3 11 0.15
1996 10 26  7 31 35 0.53
```

```
1996 10 26   7 59 59 0.99
1996 10 26   8 28 23 1.48
1996 10 26   8 56 47 1.98
1996 10 26   9 25 11 2.46
1996 10 26   9 53 35 2.88
```

In the first row, the first value (1) indicates the type of boundary condition, in this case, Type 1 or water surface elevation, prescribed as a function of time. The second value (10) indicates the number of entries. The third value indicates the system units used (0 = English; 1 = metric). From the second row on, entries in the columns from left to right are: year, month, day of the month, hour, minute, second, and water surface elevation. If there is more than one set of observed boundary conditions in a boundary section, you will need more than one file, i.e. one file for each set of data.

To associate a boundary condition with its associated node in the boundary section, follow the procedure below:

1. First save the boundary condition entries to a text file with the **.bdc** extension using the format above. In this example, there are two files, **tutorial4_s11.bdc** and **tutorial4_s13**.

2. Right-click the associated node in the boundary section, where you want to assign a given boundary condition, to bring up the **Context Menu,** and choose **Edit Grid**.

3. This will bring up the **Node** edit dialog for that node. If the node has no boundary condition already assigned to it, it should appear as shown in Figure 2.9. If there is a boundary condition already attached, it should look like Figure 2.10. Click the button labeled **Observed Data** to bring up a file **Open** dialog.

4. Browse to the boundary condition file and click **Open**. The model will then retrieve the boundary condition from the file and attach it to the node. If there is only one node in a boundary section with attached boundary condition, the whole section will use the same boundary condition on all of its nodes.

5. If two or more nodes have attached boundary conditions, the above procedure has to be repeated for each node. Then the program will interpolate the boundary condition over the section nodes based on the locations of the nodes along the section line.

6. If no boundary conditions have been attached to any of the nodes in a section, a value of zero will be assigned to it. The 0 signifies no flow, the second type boundary condition. For the first type boundary condition, head equals zero. Thus, you must give a value to sections having the first type boundary condition to have meaningful results.
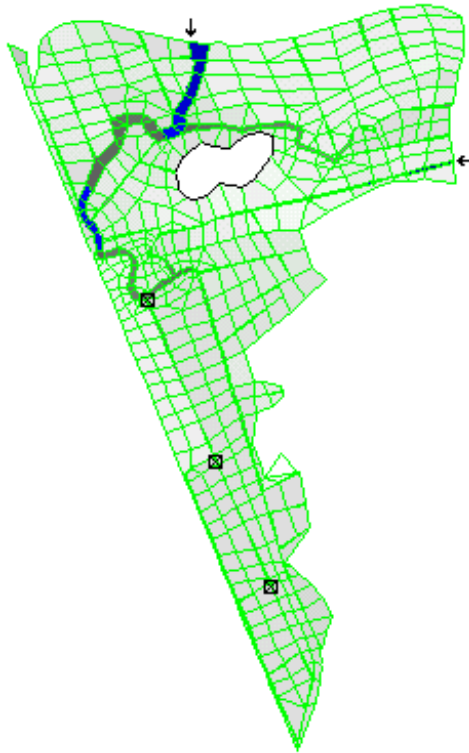
Figure 2.7. Boundary Section 2 (Type 1) at Westbrook Site

You must also consider the starting time differences among the boundary conditions. The model uses the latest starting time found among all of the boundary conditions attached to the grid (shown in the **Modeling Options** dialog) as the starting time for a run. The model also uses the earliest ending time found among all of the boundary conditions attached to the grid (also shown in **Modeling Option** dialog) as the simulation ending time.

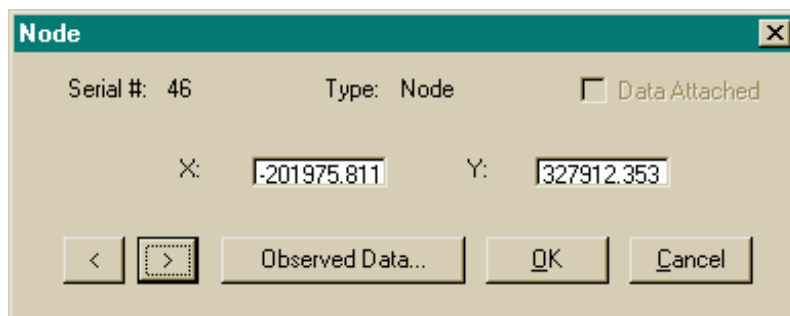Figure 2.8. Boundary Section 3 (Type 2) at Westbrook Site

Figure 2.9. Node Edit Dialog without Boundary Condition Attached

After you have finished attaching a boundary condition, the **Node** Edit Dialog will look like that shown in Figure 2.10. If you click the button labeled **Remove**

Observation, you can remove the attached boundary condition from the node and its section.
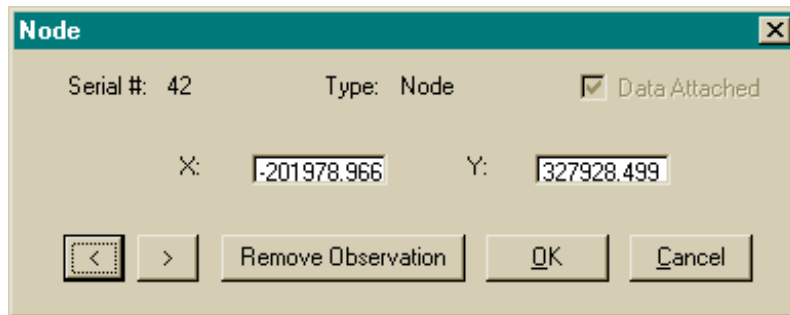

Figure 2.10. Node Edit Dialog with Boundary Condition Attached

In this example, two boundary conditions, **tutorial4_s11.bdc** and **tutorial4_s13.bdc** have been specified at two locations indicated by the two arrows in Figure 2.8. These data were collected in the field on October 26 and 27, 1996. The plot of observed data is shown in Figure 2.11.

As you can see in Figure 2.11, two sinusoid curves were interpolated to fit the data. The boundary conditions used in the calculation were taken from the fitted curves instead of from the actual data. By sampling 100 points from each curve over time, we derived two boundary condition files that contain four full tidal circles lasting for about two days. At the upstream location (to the right in Figure 2.8 at Staff 11), data collection began on October 26, 1996 at 5:20:59 am, and ended on October 28 at 5:45:02 am. At the downstream location (at the top in Figure 2.8 at Staff 13), data collection began on October 26, 1996 at 5:37:59 am and ended on October 28, 1996 at 4:58:05 am. The data collected at the two locations show a time lag in the tidal peak from upstream to downstream. Also notice that four full tidal circles (from the fitted curve) last for about 47 hours 40 minutes, instead of 48 hours, as expected.
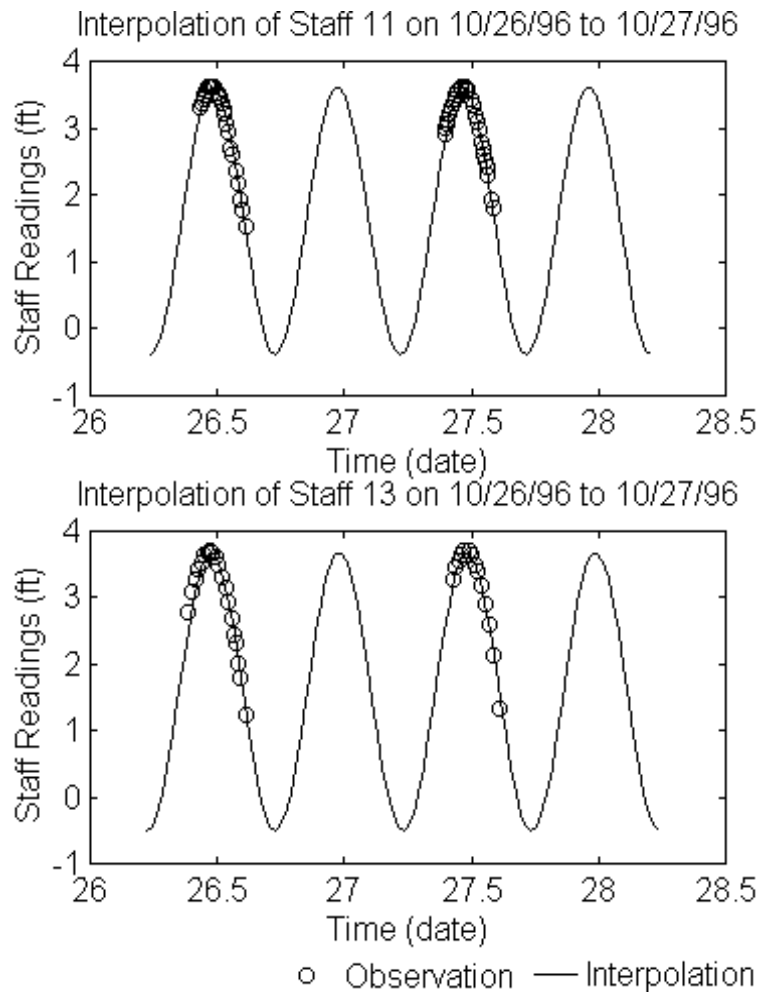
Figure 2.11. Boundary Conditions for Westbrook Site

F. Initial Condition

The initial condition can be specified at each cell while you are generating the grid. Another way to do this by giving an initial uniform value to all of the cells, and let the simulation run through several tidal cycles and use the results as the initial condition.

In this example, we gave an initial uniform value of –3.5 ft, an average water elevation at 5:37 am on October 26, 1996, to all the cells. Figure 2.6 shows the overall picture after the initial value was assigned. You can see that almost all of the region is dry, except for a few wet (blue) spots in the channel, indicating stagnant water.

Because we wanted an initial condition at the later of the two boundary conditions: October 26, 1996 at 5:37:59 am, we started the calculation with a uniform initial value

and let it run from October 26, 1996 at 5:37:59 am through four full tidal circles. We know the water elevation at the downstream location should begin and end at -3.5 ft. (This is not realistic, but is considered acceptable.) The final state of the region was saved into the project as the initial condition. To save the result as the initial condition, you can either choose **File|Save** and overwrite the given uniform initial value or **File|Save As** a new project.
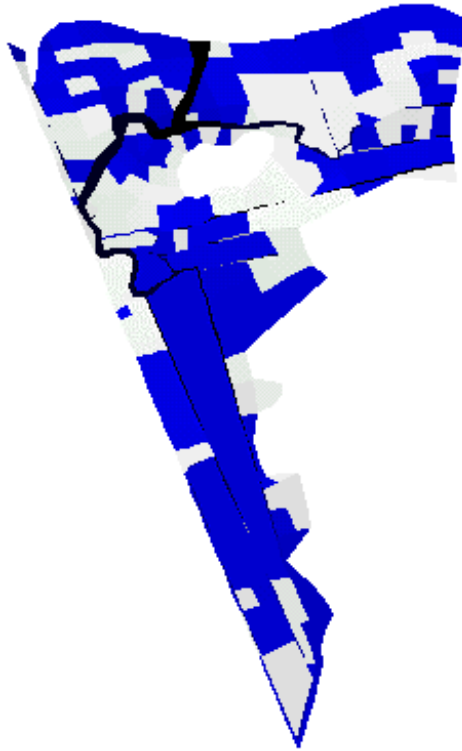

Figure 2.12. Initial Condition of Water Surface Elevation for Westbrook Site

G. Grid Adjustment

One important step in achieving a high quality grid is to adjust the cells so that neighboring cell sizes are compatible. By running a calculation in the **Velocity Vector** mode with **Animation On**, you can very easily spot places in the grid where adjustments need to be made, i.e. where the velocity vectors are pointing in all different directions. The most likely place where this will occur is at channel junctions, where one might create a cell like cell A in Figure 2.13.
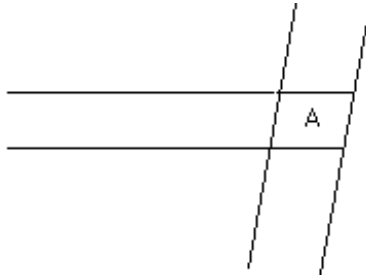
Figure 2.13. Cell Incompatible with Its Neighbors

Here cell A is not a compatibly-sized cell. Instead, you should adjust the cell as shown in Figure 2.14.
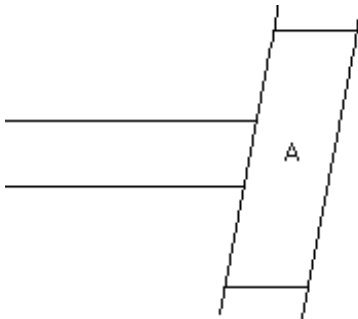

Figure 2.14. Cell Compatible with Its Neighbors

H. Simulation and Results

After all the above preparation, you now can start the real simulation and acquire some results. For a project with a large number of cells, it is better not to have **Animation On** while running the simulation. In this way the model will run much faster.

The calculated results water surface elevation for this example is compared with the tidal staff readings, taken in the field on October 26, 1996 at three locations marked by $\otimes$ in Figure 2.8, appears to be satisfactory as shown in Figure 2.15.
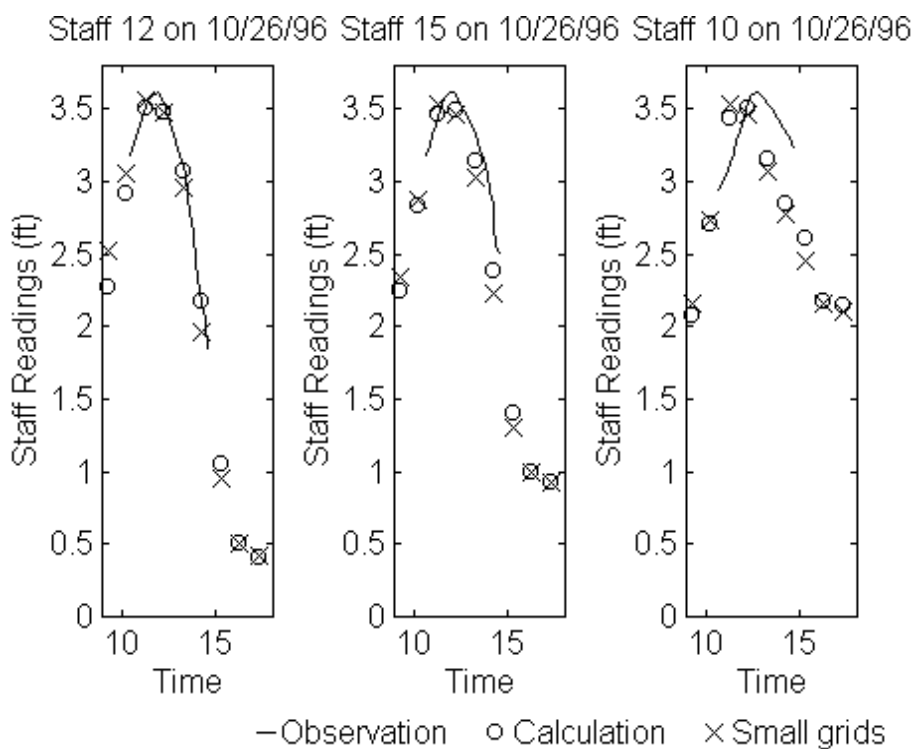
Figure 2.15. Comparison of Calculated and Observed Results at Westbrook Site

I. Conclusions

In this example, it has been shown that the real site simulation is much more complicated. You can rarely use copy-and-paste in constructing the computational grid, because there are not many similar cells that you can copy from. When constructing the computational grid, you more likely will use the split cell, split wall, move node and interpolation tools. You almost certainly will need a lot of grid adjustment. There is also an important issue regarding grid generation, i.e., detail vs. efficiency. An increase in the number of cells will result in more detailed results, but also an increase in computation time. A trade-off has to be made, in most cases, bearing in mind the intent of the work and the accuracy required for the application of the final results. You may run the model with a coarser grid when you are only interested in the general pattern of the flow field; on the other hand, an extremely detailed grid is needed when you are interested in the spatial distribution of tidal ranges and their effects on salt marsh vegetation, for example.

In conjunction with other related documentation for FVHM, we hope that this tutorial will provide you an overview and hands-on experience by working through all the examples.