

**TWO-DIMENSIONAL FINITE-VOLUME HYDRODYNAMIC
MODEL FOR COASTAL RIVER-MARSH SYSTEMS
USER'S MANUAL, 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 02-275

Project 93-4

This research was sponsored by the Federal Highway Administration, the Joint Highway Research Advisory Council (JHRAC) of the University of Connecticut and the Connecticut Department of Transportation and was carried out at the Connecticut Transportation Institute of the University of Connecticut.

This document is disseminated under the sponsorship of the Federal Highway Administration in the interest of information exchange. The United States Government assumes no liability for its contents or use thereof.

The contents of this report reflect the views of the authors who are responsible for the facts and accuracy of the data presented herein. The contents do not necessarily reflect the official views or policies of the Federal Highway Administration, the University of Connecticut or the Connecticut Department of Transportation. This report does not constitute a standard, specification, or regulation.

Technical Report Documentation Page

1. Report No. JHR 02-275		2. Government Accession No.		3. Recipient's Catalog No.	
4. Title and Subtitle Two-Dimensional Finite-Volume Hydrodynamic Model for Coastal River-Marsh Systems: User's Manual, Version 1.0				5. Report Date June 2002	
				6. Performing Organization Code	
7. Author(s) J.D. Lin, W.G. Liao, K.-J. Qiu, M.W. Lefor				8. Performing Organization Report No. JHR 02-275	
9. Performing Organization Name and Address University of Connecticut Connecticut Transportation Institute Storrs, CT 06269-5202				10. Work Unit No. (TRAIS)	
				11. Contract or Grant No.	
12. Sponsoring Agency Name and Address Connecticut Department of Transportation 280 West Street Rocky Hill, CT 06067-0207				13. Type of Report and Period Covered User's Manual	
				14. Sponsoring Agency Code SPR-2214	
15. Supplementary Notes Conducted in cooperation with the U.S. Department of Transportation, Federal Highway Administration					
16. Abstract Two-Dimensional Finite-Volume Hydrodynamic Model was developed as a comprehensive system for modeling the hydrodynamics of tidal flows in salt marshes and other estuarine systems, with particular emphasis on the ability to assess the effects of natural or man-made changes in channel configurations and tidal marshes. The modeling approach is based on applying the finite-volume method to the shallow-water equations, generally used for estuarine flows. The model runs under current MS-DOS operating systems such as Windows 98 and Windows NT, and uses multi-tasking. The core of computer program was first written in Fortran. The graphical interface was first written in Borland C++ and then ported to Microsoft Visual C++. This manual describes each element of the model and its operational procedures in detail, which will guide a user through software installation, generation of computational grids, preparation of data, running simulation and acquiring results.					
17. Key Words Hydrodynamics, Model, Tidal Flow, Marsh			18. Distribution Statement No restrictions. This document is available to the public through the National Technical Information Service Springfield, Virginia 22161		
19. Security Classif. (of this report) Unclassified		20. Security Classif. (of this page) Unclassified		21. No. of Pages 60	22. Price N/A

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 m³

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

Symbol	When You Know	Multiply By	To Find	Symbol
TEMPERATURE (exact)				
°F	Fahrenheit temperature	5(F-32)/9	Celsius 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

Symbol	When You Know	Multiply By	To Find	Symbol
g	grams	0.035	ounces	oz
kg	kilograms	2.205	pounds	lb
Mg	megagrams	1.102	short tons (2000 lb)	T

TEMPERATURE (exact)

Symbol	When You Know	Multiply By	To Find	Symbol
°C	Celsius temperature	1.8C+32	Fahrenheit temperature	°F

°F	°C
32	0
98.6	37
212	100

* SI is the symbol for the International System of Measurement

TABLE OF CONTENTS

I. INTRODUCTION	1
A. OVERVIEW	1
B. FVHM APPLICATIONS	2
C. SOFTWARE INSTALLATION	2
D. SYSTEM REQUIREMENTS	2
E. GETTING STARTED	3
II. MODELING PROCEDURE	4
A. IMPORTING THE OUTLINE OF THE MODEL DOMAIN	4
B. CONSTRUCTING THE GRID	4
C. ASSIGNING MANNING'S COEFFICIENTS AND APPLYING BOUNDARY CONDITIONS ..	5
D. RUNNING FVHM	5
E. POST-PROCESSING THE SOLUTION DATA	5
F. THE FVHM WINDOW	5
III. FILE INPUTS AND OUTPUTS	26
A. THE PROJECT FILE	26
B. IMPORT AND EXPORT FILES	26
C. OUTLINE FILES	29
D. BOUNDARY CONDITION FILES (OBSERVATION DATA FILES)	29
E. RESULT FILE	30
F. EXPORTED TIME HISTORY FILES	31
IV. CONSTRUCTING THE MODEL GRID	33
A. CREATING A NEW PROJECT	33
B. PROJECT OPTIONS	34
C. IMPORTING THE OUTLINE	34
D. CREATING A NEW CELL	35
E. SELECT CELL(S)	37
F. COPY AND PASTE CELL(S)	38
G. DELETE CELL(S)	38
H. MERGE CELL(S)	38
I. MOVE NODE	39
J. REMOVE NODE	41
K. SAVING THE DATA	40
V. ASSIGNMENT OF BASIC PARAMETERS AND BOUNDARY CONDITIONS	41
A. ASSIGNING BASIC PARAMETERS	41
B. ASSIGNING BOUNDARY CONDITIONS	43
C. OBSERVED DATA	46
D. SAVING THE PROJECT	48

VI. RUNNING THE MODEL 49

VII. POST-PROCESSING 51

 A. RESULT PLAY 51

 B. RESULT | TIME HISTORY | EXPORT 53

VIII. FINAL REMARKS 54

* * *

LIST OF FIGURES

1. 1. FVHM Main Window	3
2. 1. The FVHM Main Window, showing a computational domain, &c.	6
2. 2. The File Menu	7
2. 3. File Open Dialog Box	8
2. 4. Save As Dialog Box	9
2. 5. Print Setup Dialog Box	10
2. 6. The Edit Menu	11
2. 7. Cell Property Dialog Box	13
2. 8. Cell Property Edit Dialog Box	14
2. 9. Boundary Section Edit Dialog Box.	14
2. 10. View Menu	16
2. 11. Project Menu Commands	19
2. 12. Modeling Options Dialog Box	20
2. 13. Project Options Dialog Box.	21
2. 14. Result Menu Commands.	22
2. 15. Tools Menu Command.	23
2. 16. Save As Dialog Box.	23
4. 1. Save As Dialog Box	33
4. 2. Project Options Dialog Box.	34
4. 3. Open Dialog Box	35
4. 4. Context Menu.	36
4. 5. Creating New Cells.	37
4. 6. Save Project File	40
5. 1. Cell Property Dialog Box.	42
5. 2. Cell Property Edit Dialog Box.	43
5. 3. Project Options Dialog Box.	44
5. 4. Boundary Sections Edit Dialog Box	45
5. 5. Wall Property Dialog	47
5. 6. Node Property Dialog Box.	47
5. 7. Open Observed Data File.	48
6. 1. Modeling Options Dialog Box.	49
7. 1. Paint Screen.	52
7. 2. Save As Dialog Box.	53

* * *

I. INTRODUCTION

A. OVERVIEW

This Two-Dimensional Finite-Volume Hydrodynamic Model (FVHM) was developed by J. D. Lin, M. W. Lefor, W. G. Liao, and K-J. Qiu of the Civil & Environmental Engineering Department of The University of Connecticut, Storrs. It was designed as a comprehensive system for modeling the hydrodynamics of tidal flows in salt marshes and other estuarine systems, with particular emphasis on the ability to assess the effects of natural or man-made changes in channel configurations and tidal marshes. It is easy to use, and the user will recognize many functions common to Windows programs. This system runs under current MS-DOS operating systems such as Windows 98 and Windows NT, and uses multi-tasking to get the most out of available computer resources.

This is a User's Manual for FVHM only. Further details on the Finite-Volume Hydrodynamic Model are included in the final technical report, part of the series issued by the Joint Highway Research Advisory Council of the Transportation Institute, The University of Connecticut, Box U-5202, 179 Middle Turnpike, Storrs, CT 06269-5202, (860) 486-5400. A companion Tutorial is also available for those unfamiliar with running models of this type.

At its core, FVHM includes the finite-volume hydrodynamic model, along with a pre- and post-processor. FVHM was especially designed to deal with complicated two-dimensional (2-D) shallow-water problems met with in estuaries, such as salt marshes with complex boundaries, hydraulic structures, mosquito drainage ditches, and frequent wetting and drying cycles. Running FVHM will give hydrodynamic solutions including water surface elevations, flow velocities, and wetting and drying cycles under diurnal tidal forcing and storm surge conditions.

The pre- and post- processors provide a user-friendly interface for FVHM. They allow the user to create realistic finite-volume meshes and to view the solutions in a simple and intuitive environment.

B. FVHM APPLICATIONS

FVHM was designed to provide a way to analyze and study 2-D hydrodynamic data from environmental problems in complicated waters, such as rivers, salt marshes, harbors, and estuaries.

The model has the following capabilities:

1. handling complex topography, such as complex boundaries, multiple slough inflows, and ditches;
2. simulating both subcritical and supercritical flows;
3. calculating steady or unsteady flows, such as tidal flows and storm surges;
4. simulating smooth and discontinuous flows;
5. handling wetting and drying cycles; and,
6. handling some simple hydraulic structures, such as bridges, culverts, and tide gates.

C. SOFTWARE INSTALLATION

The model installation sequence consists of the installation of a number of compressed files, *estu.exe*, *model.dll*, *dialogs.dll*, *oscher.dll*, *utility.dll*, *estu.cnt*, and *estu.hlp*. These files are found on the model system diskette. A number of data files are also included there, to be used by those taking the Tutorial. Correct installation requires that all these files be unzipped into the same directory.

D. SYSTEM REQUIREMENTS

An Intel Pentium running at 200MHz or above (or an equivalent processor), running under Windows NT 4.0 or Windows 95 or 98, is required to run the model. A Super VGA monitor with maximum color setting is strongly recommended.

E. GETTING STARTED

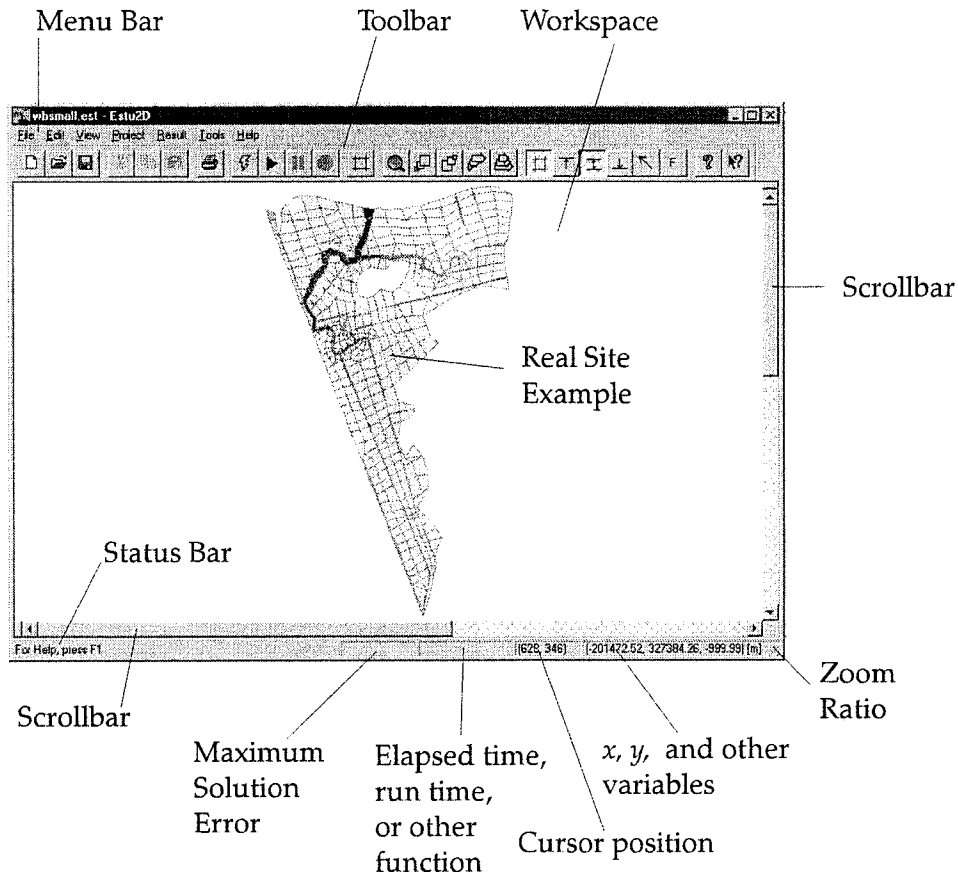


Figure 1.1. FVHM Main Window.

Double-clicking the ESTU.EXE icon will bring up the **Main Window**, as shown in Figure 1.1. Please notify us if there is a problem with installation or with running the model. The **Workspace** (the FVHM Main Window) is empty initially. You may either create a new project, or open an existing project via the drop-down **Menus** or **Toolbar**.

II. MODELING PROCEDURE

Follow the steps given below to use FVHM for solving a 2-D, shallow-water hydrodynamic problem.

A. IMPORTING THE OUTLINE OF THE MODEL DOMAIN

In a typical study, an outline data file for the site to be modeled is needed to define the topography and the boundaries of the computational grid, and defines the model domain.. This outline file consists of a set of x , y , z coordinates, in ASCII format, and has the extension `.oln`. Once the file is imported into FVHM, it may be displayed in the workspace in the **Main Window** (see Fig. 1. 1) as an image of the model domain. You construct a computational grid within this window. The coarseness or fineness of the grid you construct depends on the amount of outline (survey) data you have, the precision of the desired results, and the time it will take for the computer to run the model.

The outline data file can be generated by scanning a map of contour lines on a desktop scanner. Frequently, however, a map of bathymetric contours showing channel sections and geometry, etc., is not available. In that case, a field survey must be conducted to obtain the required data for the outline files. The rule "GIGO" applies: the more complete and accurate the input data, the more accurate the results.

B. CONSTRUCTING THE GRID

Next, you must construct a finite-volume grid. This grid discretizes the model domain into cells, and describes the bathymetric contours (bottom surface geometry) of the body of water to be modeled. Grid construction is done on-screen, intuitively, by using the FVHM program. Topographic data in the form of x , y , z coordinates for each cell of the grid are saved as an FVHM input file. Details of grid construction are described below in Section IV.

C. ASSIGNING MANNING'S COEFFICIENTS AND APPLYING BOUNDARY CONDITIONS

Once the grid has been constructed, the user assigns Manning's n coefficients to each cell, and assigns boundary conditions to the grid, as directed by the FVHM program. Boundary conditions are typically assigned as incoming flow rates at one end of the grid and exit water elevations at the opposite end of the grid. Boundary point coordinates are saved in a data file. There may be many boundary points in a complex study site like a salt marsh.

D. RUNNING THE FVHM

All hydrodynamic solutions are calculated by the FVHM program. The start, progress, and termination of the calculation are managed in the **Main Window**. The results are output to a Results file in binary format.

E. POST-PROCESSING THE SOLUTION DATA

After you have run the FVHM program, the results are post-processed by FVHM. The program can display time-step animations of velocity vector plots, color-coded contour lines of water elevations, and Froude number distributions. The animations may be replayed. Results from selected cells in the Result file may also be copied to an ASCII file for display or other uses.

F. THE FVHM WINDOW

Read this section carefully before you begin to work with the FVHM program, for it is necessary that you have an overall understanding of the basic processes, functions, and tools provided. **IN OTHER WORDS, READ THIS USER'S MANUAL FIRST!**

Go to the directory in which you installed the FVHM package from the Systems Diskette. Double-click the **ESTU.EXE** program icon to open the **FVHM Window** as shown in Fig. 2.1.

The FVHM window is a graphical user interface. It gives you a simple and intuitive environment for creating complicated finite-volume grids, running the program, and for post-processing. The screen is divided into four sections, as shown in Figure 2. 1. You can access On-line Help by clicking the tool bar or **Help** menu.

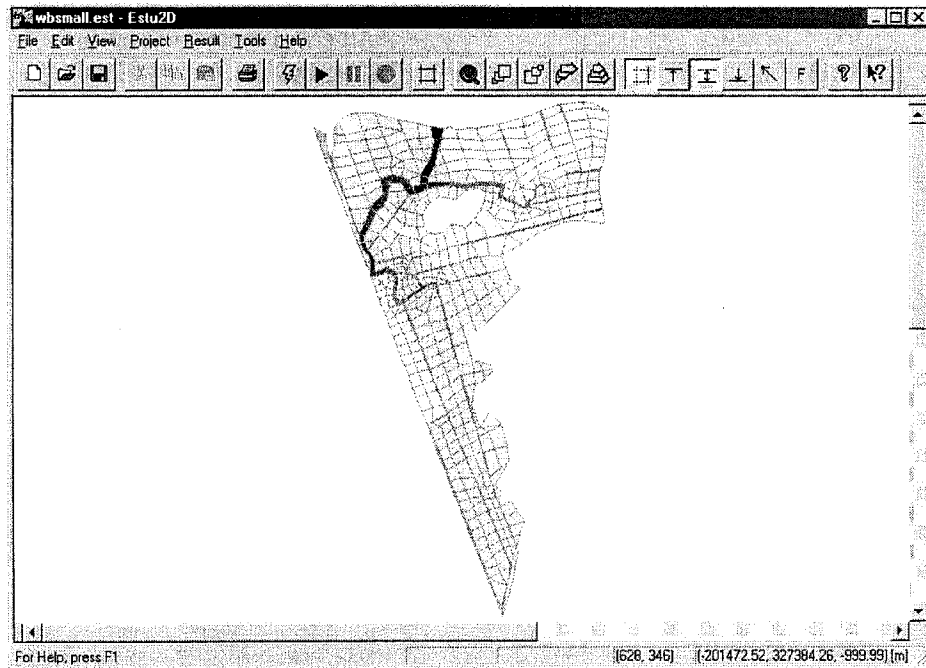


Fig. 2.1. The FVHM Screen, showing a computational domain and the grid cells, density-coded for water depth during part of a tidal cycle (see also Fig. 1. 1).

At the top of the **Main Window**, the **Menu Bar** is used to access FVHM commands; the **Toolbar** allows you to access commands quickly; the **Workspace** displays the current grid; and the **Status Bar** displays help messages and other information. Each of these will be discussed in detail below.

a. **Model Main Window.** The **Model Main Window** is the most important part of your interface. It displays all of the actions you take as you work with FVHM.

b. **The Menu Bar.** The menu bar consists of eight menus: **File**, **Edit**, **Search**, **View**, **Project**, **Result**, **Tool**, and **Help**, used to access all of the FVHM commands. Commands are accessed through pull-down menus located in the menu bar. Each menu can be accessed using the mouse or by holding down the **ALT** key and pressing the respective highlighted letter shown in the **Menu Bar**. Once a menu is visible, the individual command can be chosen with the mouse or by holding down the **ALT** key and pressing the highlighted command letter in the Menu.

THE FILE MENU

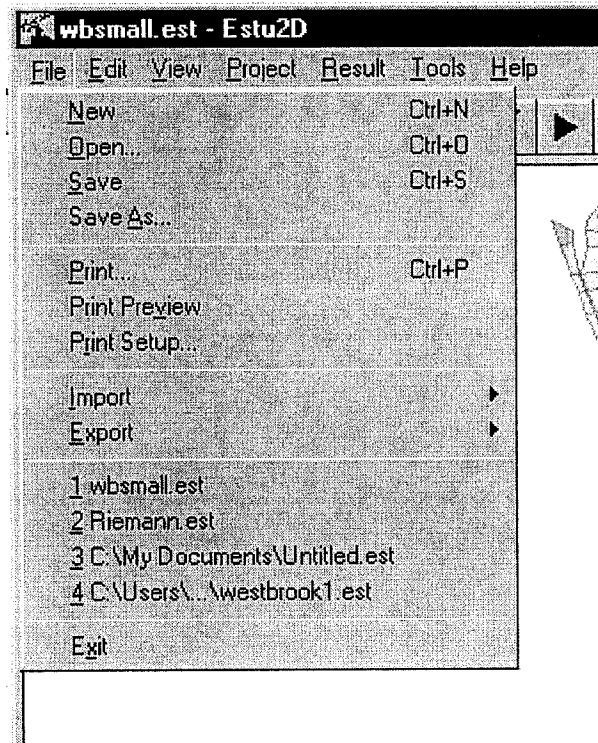


Fig. 2. 2. The File Menu.

The **File** menu provides commands for creating a new project, opening an existing project, saving a project, printing, and exiting the application. See Fig. 2.2.

File | New Command.

The **File | New** command is used to create a new project and to activate the graphics window. After this command is selected, a dialog box will be brought up asking for a file name for the new project (see Fig. 2. 3). Project file names in this model have the extension **.est**. This is an ASCII file, designed to contain all the input data (grid information, boundary conditions, etc.), for FVHM. This command must be selected each time a new modeling project is started. The initial FVHM main window is empty; but, you may start to create a new grid right away, once the new project has been created.

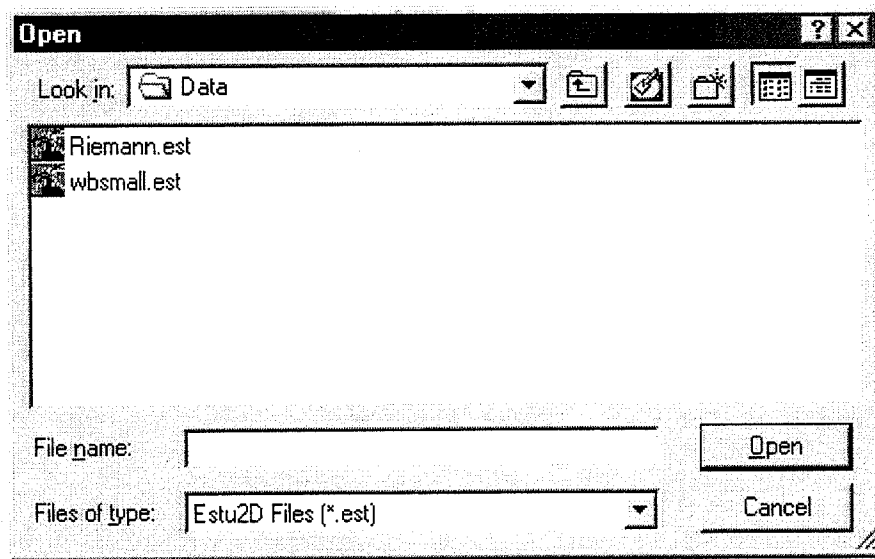


Fig. 2. 3. File Open Dialog Box.

File | Open Command.

The **File | Open** command opens a file dialog box like that shown in Fig. 2. 3, so that the user may browse the directory and select an existing project to load into the project work space. This command should be selected before the user begins editing a project grid or boundary conditions, or running the model.

File | Save Command.

The **File | Save** command is used to save the current active project to the same file from which it was loaded or created in the **Save As** dialog shown in Fig. 2. 4. The project file thus saved will have the file extension **.est**.

File | Save As Command.

The **File | Save As** command allows the user to save the current project under a new name, or in a new disk location. The command displays the **Save File As** dialog box (see Fig. 2. 4). You may enter the new file name if you wish, including the drive and directory. If you choose an existing project name, you will be asked if you want to overwrite the

existing project.

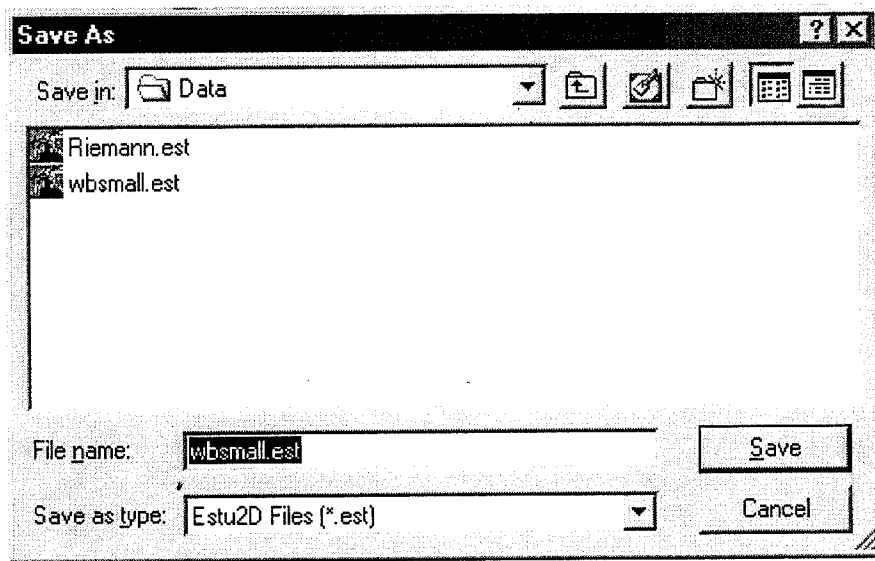


Fig. 2. 4. Save As Dialog Box.

File | Print Command.

The **File | Print** command prints the project grid under the current project settings. Use **File | Print Preview** to see how the grid will be laid out in the printer output. Use **File | Print Setup** to select a printer and to set printer options.

File Print Preview Command.

The **File | Print Preview** command opens a special window showing how the project grid will look when it is printed.

File | Print Setup Command.

The **File | Print Setup** command displays the Printer Setup dialog box (see Fig. 2. 5), which allows you to select and configure the printer in the usual manner.

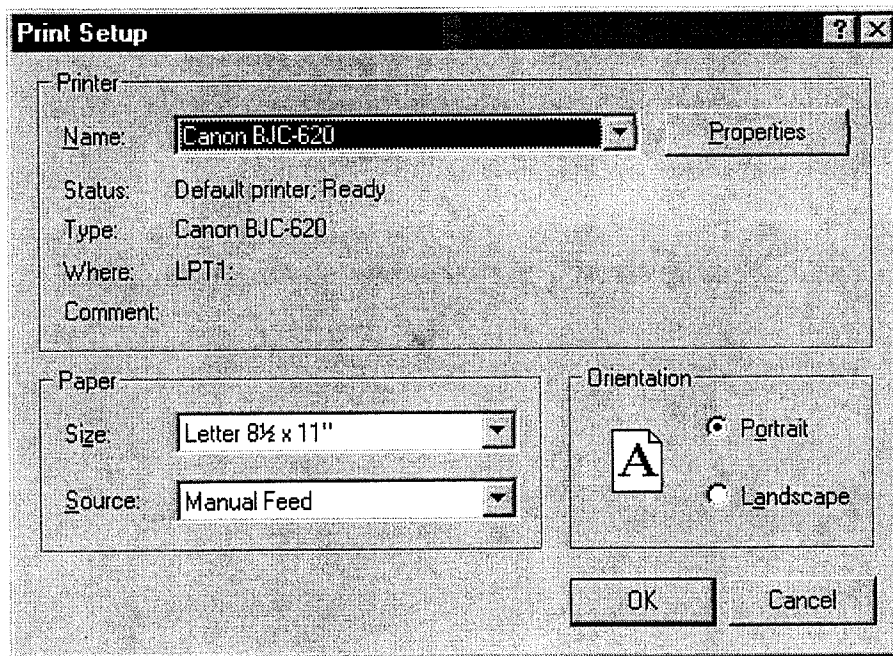


Fig. 2. 5. Print Setup Dialog Box.

File | Import Command.

The **File | Import** command allows you to import from ASCII files. There are two types of ASCII import files currently supported. First, outline data files contain sets of x , y , z coordinate data—the locations and bottom elevations of surveyed points in your study site. Second, previously exported project files in ASCII are also supported, such as result files, which are needed to restart the calculation. Outline data files have the extension **.oln**, and ASCII project files have the extension **.prj**.

File | Export Command.

The **File | Export** command allows you to export data from a binary file into an ASCII file. The exported ASCII project file can be edited directly, and imported back into the model.

File | Exit Command.

The **File | Exit** command exits the program. You will be prompted to save your project

before exiting. This command may also be accessed by pressing ALT + F4.

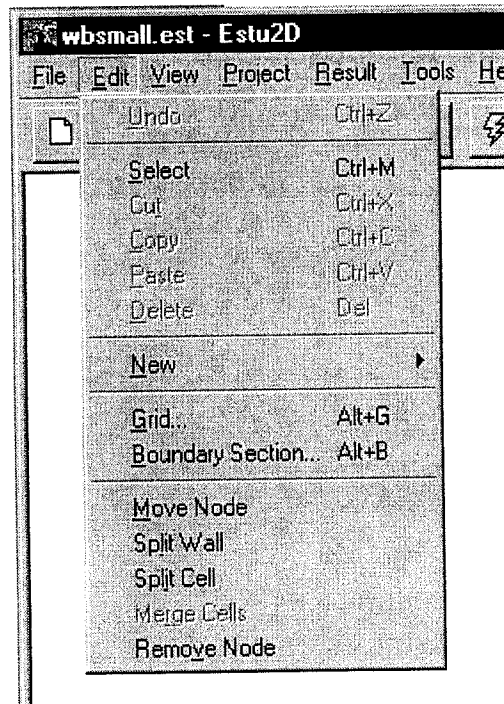


Fig. 2. 6. The Edit Menu.

EDIT MENU

There are fifteen commands under the **Edit** menu (see Fig. 2. 6). These commands are used to edit the grid of the active project.

Edit | Undo Command (not implemented).

The **Edit | Undo** command restores the project to the way it was before your last edit operation. This command is not implemented at present.

Edit | Select Command.

The **Edit | Select** command allows you to select a group of cells. The numbers of the cells selected will also be highlighted (the **Display Cell Number** setting must be selected before you can see the highlighting). Once the cells have been selected they are ready for

commands such as **Copy**, **Delete**, and **Edit Grid**, etc.

Edit | Cut Command (not implemented).

The **Edit | Cut** command removes the selected cells from their current locations and places them in the memory buffer. You may later paste them to some other location within the project grid. This command is not implemented at present.

Edit | Copy Command.

The **Edit | Copy** command leaves the selected cells intact and places an exact copy of them in the memory buffer. You may later paste these cells to some other location within the project grid as necessary.

Edit | Paste Command.

The **Edit | Paste** command will take the cells in the memory buffer (put there by either the **Edit Cut** command or the **Edit Copy** command) and paste them into the project grid. After selecting the cells to paste, an outline of the cells in the buffer will be displayed under the mouse cursor. The next time you click the left mouse button inside the project work space, the cells will be pasted at the location of the mouse pointer.

Edit | Delete Command.

The **Edit | Delete** command deletes the currently selected cells from the project. This command is equivalent to hitting the **Delete** key.

Edit | New Cell Command.

The **Edit | New Cell** command is used to create new cells one at a time. After this command is selected, left-click the cursor wherever you desire to create new nodes and new walls. Complete the new cell by closing its perimeter: click back on the first node, and a new cell will be created. During the process, nodes and walls are generated automatically by FVHM. The **Edit | New Cell** command may also be accessed by clicking on the **Tool** icon or by pressing **ALT + C**.

Edit | Grid Command.

The **Edit | Grid** command will bring up the Cell Property dialog (see Fig. 2. 7). In this dialog you may choose a cell with the < or > key, and you may modify the properties of that cell, *e. g.*, its initial water elevation, its bottom elevation, its velocity components in the *x* and *y* directions, and its Manning's coefficient. This command can also be accessed by pressing **ALT + G**.

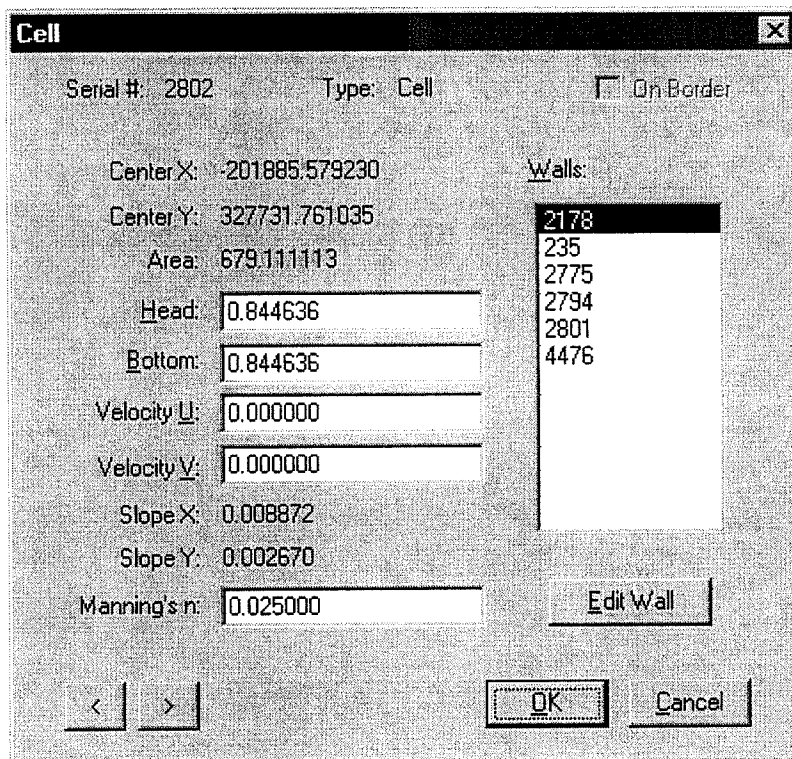


Fig. 2. 7. Cell Property Dialog Box.

If you use **Edit | Select** to select one cell or a group of cells first, and then use the **Edit | Grid** command, the dialog box will look like that shown in Fig. 2. 8, below:

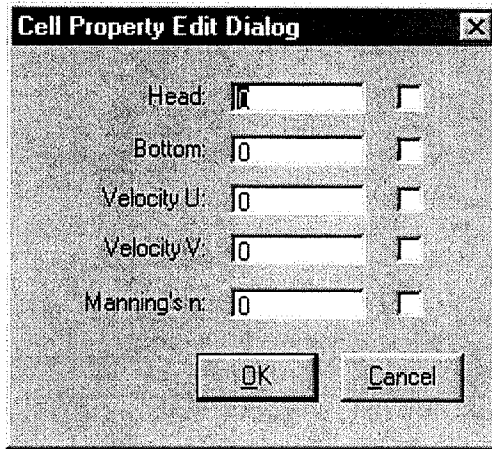


Fig. 2.8 Cell Property Edit Dialog Box.

Edit | Boundary Section Command.

The **Edit | Boundary Section** command brings up the **Boundary Section Edit** dialog (see Fig. 2. 9).

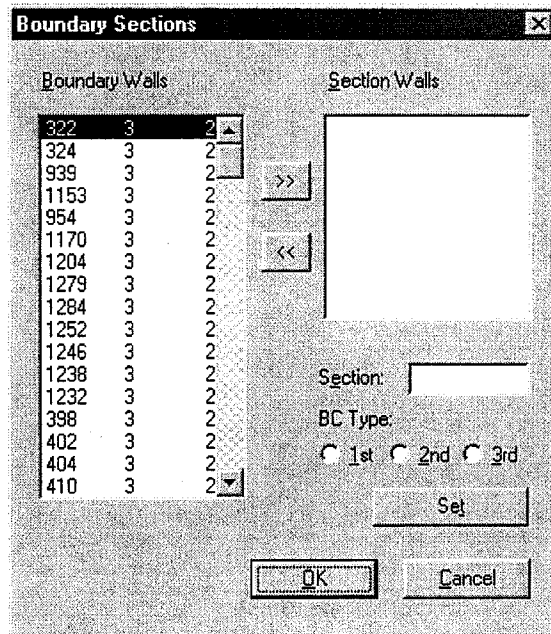


Fig. 2. 9. Boundary Section Edit Dialog Box.

The **Boundary Section Edit Dialog** allows you to group all of the boundary walls into sections, and to specify the type of boundary condition at each section. Details of assigning boundary conditions are given below in Section 5. This command can also be accessed by pressing **ALT + B**.

Edit | Move Node Command.

The **Edit | Move Node** command sets the model into the **Move Node** mode, and allows you to move an existing node from one location to another. By this means, you may adjust the shapes of those cells and walls that share a node. Selecting **Edit | Move Node** again will exit the **Move Node** mode.

Edit | Split Wall Command.

The **Edit | Split Wall** command sets the model into **Split Wall** mode, allowing you to insert nodes in a wall. Click **Edit | Split Wall** again to exit the **Split Wall** mode.

Edit | Split Cell Command.

The **Edit | Split Cell** command allows you to split a cell by connecting two nodes off a cell wall. After choosing the command, you may click on the node where you want to split the cell, and then click another node of the cell to split it. You may also click inside the cell somewhere between the first click that starts the split and the last click that ends the split, to form a more complex split path.

Edit | Merge Cells Command.

The **Edit | Merge Cells** command allows you to merge selected cells.

Edit | Remove Node Command.

The **Edit | Remove Node** command is used to remove an unnecessary node from the grid so a user may adjust the shape of a cell.

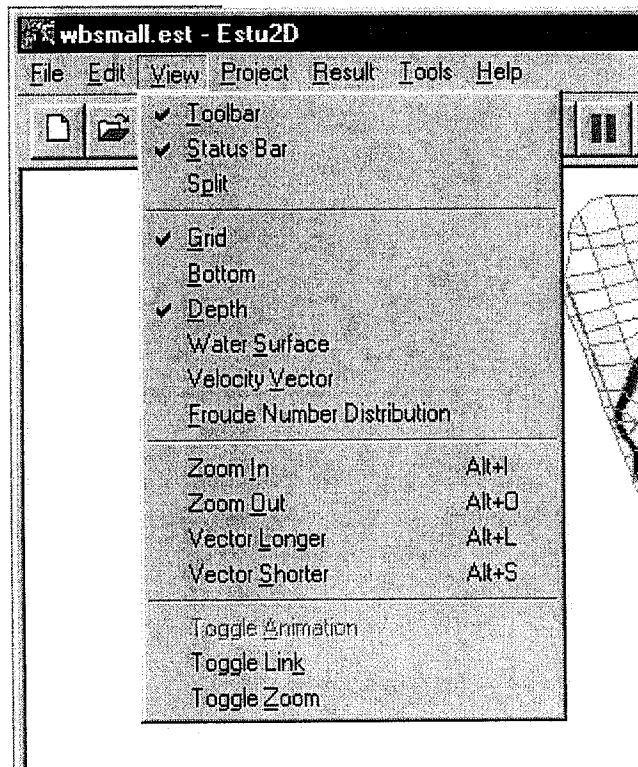


Fig. 2. 10. View Menu.

VIEW MENU

The **View** menu provides commands for what will be shown in the project work space, and how those features will be displayed (see Fig. 2. 10).

View | Toolbar Command.

The **View | Toolbar** command is used to display or hide the toolbar menu.

View | Status Bar Command.

The **View | Status Bar** command is used to display or hide the Status Bar.

View | Split Command.

The **View | Split** command is used to split the application workspace into multiple view ports.

View | Grid Command.

The **View | Grid** Command is used to display the project grid system. This command can also be accessed quickly from the **Toolbar** icon. All of the icons and their respective functions are described under the **Toolbar** section below).

View | Bottom Command.

The **View | Bottom** command is used to display bottom elevations in gray-scale levels within the modeling domain. This command can quickly accessed from the **Toolbar**.

View | Depth Command.

The **View | Depth** command is used to display water depth distributions in blue-scale levels within the modeling domain. This command may be accessed quickly from the **Toolbar**.

View | Water Surface Command.

The **View | Water Surface** command is used to display water surface elevations in blue-scale levels within the modeling domain. It may also be accessed from the **Toolbar**.

View | Velocity Vector Command.

The **View | Velocity Vector** command is used to display velocity distributions with arrows in the modeling domain. It is also accessible from the **Toolbar**.

View | Froude Number Distribution Command.

The **View | Froude Number Distribution** command is used to show regions of supercritical

flow in red, and regions of subcritical flow in blue. It is also accessible from the **Toolbar** .

View | Zoom In Command.

The **View | Zoom In** command is used to magnify the viewing area. This command is also accessible from the **Toolbar** icon.

View | Zoom Out Command.

The **View | Zoom Out** command is used to shrink the viewing area. This command is also accessible from the **Toolbar**.

View | Vector Longer Command.

The **View | Vector Longer** command is used to magnify the velocity vectors. This command is also accessible from the **Toolbar**.

View | Vector Shorter Command.

The **View | Vector Shorter** command is used to decrease the size of the velocity vectors. This command is also accessed from the **Toolbar**.

View | Toggle Animation Command.

Choosing the **View | Toggle Animation** menu command will stop the model from redrawing the workspace at every time-step during the simulation run. This will speed up the simulation process. The command is only available while a simulation is running.

View | Toggle Link Command.

Choosing the **View | Toggle Link** menu command places the project workspace in a special mode under which the lines and square dots representing the imported outlines can be toggled on and off by clicking on them with the left mouse button. This command is useful when you are importing a model outline.

View | Toggle Zoom Link Command.

Choosing the **View | Toggle Zoom Link** menu command places the model in Quick Zoom mode. In this mode, left-clicking the mouse while the cursor is inside the workspace will zoom in, and right-clicking will zoom out.

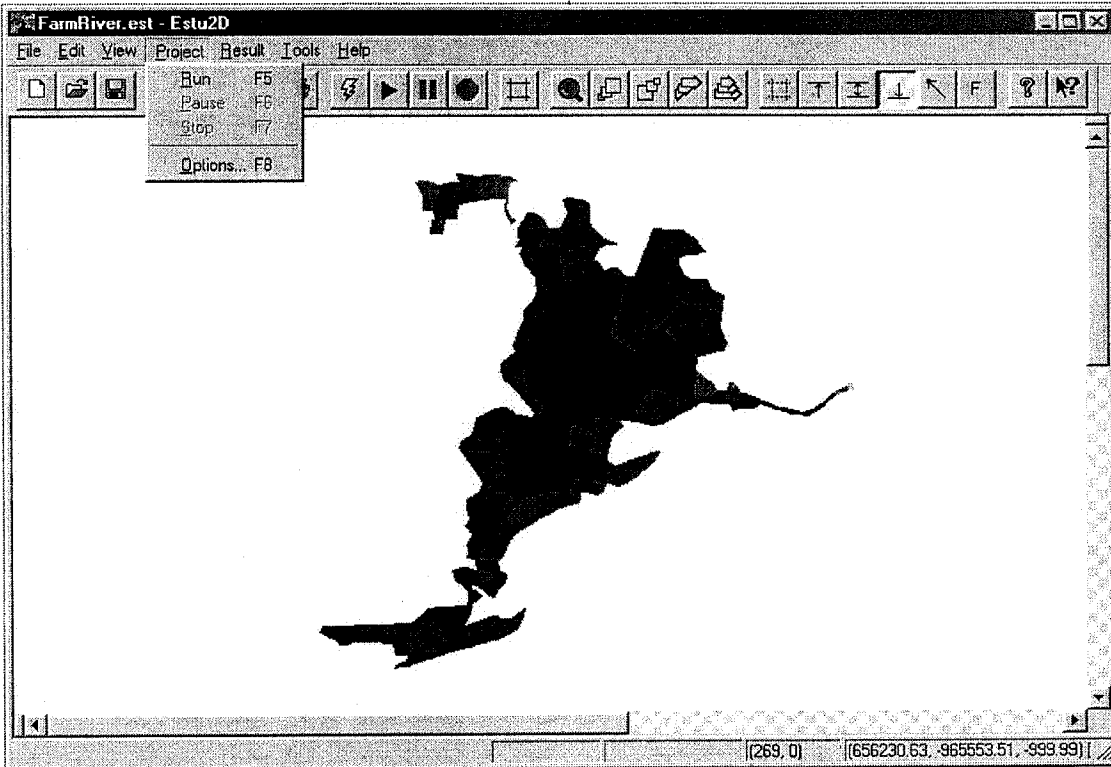


Figure 2. 11. Project Menu Commands.

PROJECT MENU

The **Project** menu includes the commands for operating FVHM.

Project | Run Command.

The **Project | Run** command is used to run the FVHM program. Once the grid has been constructed (including its Manning's n coefficients and initial conditions) and boundary

conditions have been assigned, one may use this command to start the simulation of the hydrodynamic process.

When the **Project | Run** command is selected, a **Modeling Options** dialog box appears (see Fig. 2. 11) to allow you to set the beginning date and time, ending date and time, animation, view content (either water elevation, water depth, velocity, or Froude number), and output time intervals. The **Project | Run** command can also be accessed by pressing **F5**.

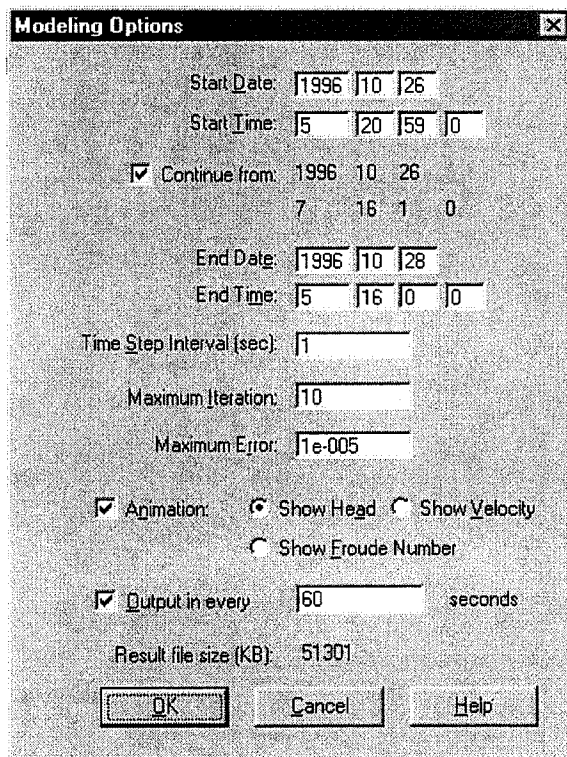


Fig. 2. 12. Modeling Options Dialog Box.

Project | Pause Command.

The **Project | Pause** command is used to pause the simulation in progress. This command can also be accessed by pressing **F6**.

Project | Stop Command.

The **Project | Stop** command is used to stop running the simulation. This command can also be accessed by pressing **F7**.

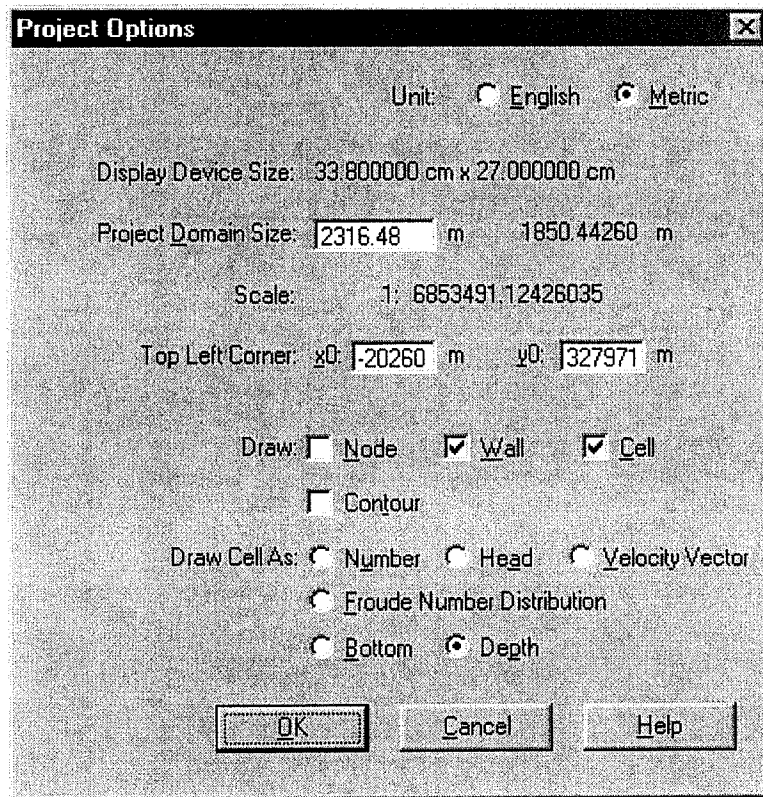


Fig. 2. 13. Project Option Dialog Box.

Project | Options Command.

The **Project | Options** command is used to set the basic parameters of the modeling project. Once you selects this command, a **Project Options** dialog box will pop up (see Fig. 2. 12). You may choose the engineering units, project domain size, screen coordinates, drawing content, and the way cells are drawn.

This command should be used to determine the domain size to be modeled before importing the project outline; it is often used during both editing and post-processing procedures. This command can also be accessed by pressing **F8**.

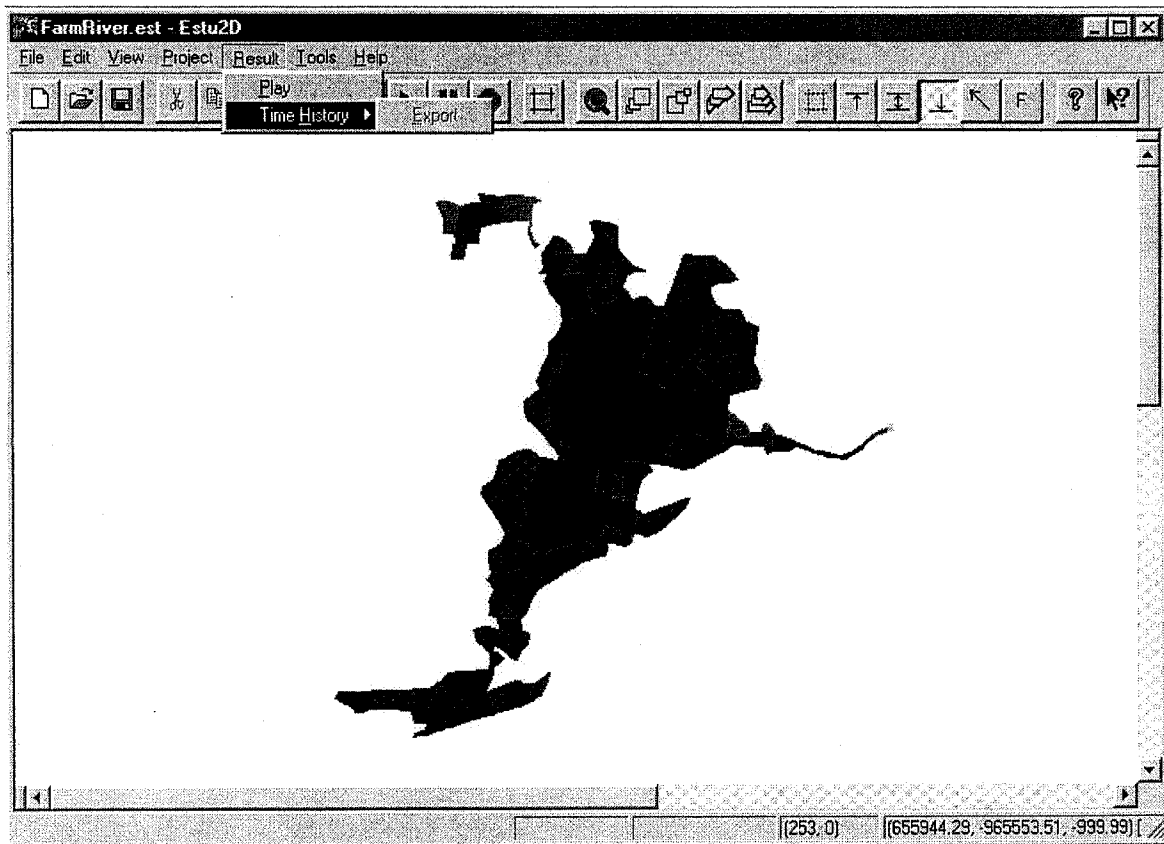


Figure 2.14. Result Menu Commands.

RESULT MENU

The **Result** menu provides two ways for you to handle calculation results.

Result | Play Command.

The **Result | Play** command is used to display the animated hydrodynamic process calculated by a previous simulation.

Result | Time History | Export Command.

The **Result | Time History | Export** command is used to output calculated results for a set of selected cells to an ASCII file (with extension **.tmh**) so that you can process the data for

other applications. Before selecting this command, you should first select the cells of interest by using the **Edit | Select** menu. Then, go to the **Result** menu and select **Time History | Export**. After **Result | Time History | Export** has been selected, a **Save As** dialog box will pop up (see Fig. 2. 13), asking for the filename and address of the file to be saved.

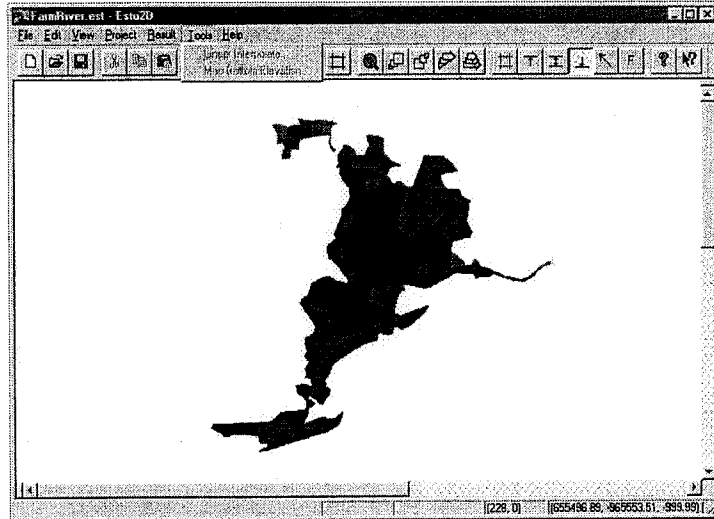


Fig. 2. 15. Tools Menu command.

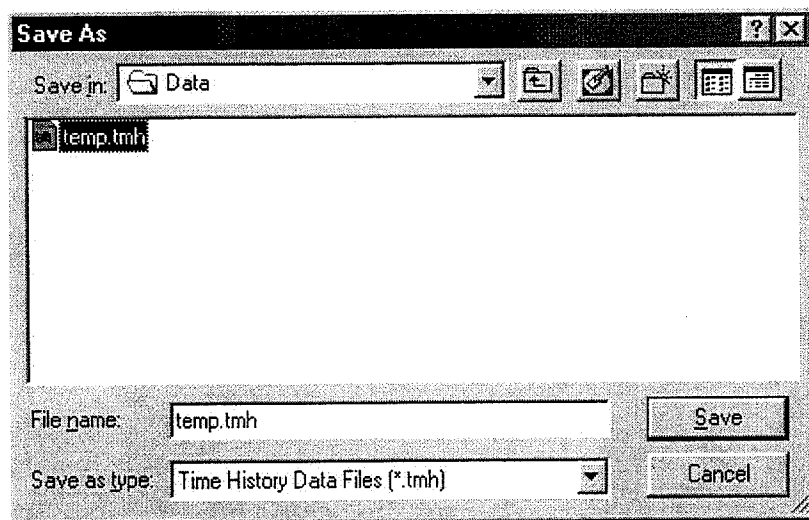


Fig. 2.16. Save As dialog box.










TOOLS MENU
















Tools | Linear Interpolate Command.

The **Tools | Linear Interpolate** command is used to interpolate specific cells state values among two or more selected cells. You must first select a set of cells such that the first and last cells selected contain the values to be interpolated. After selecting those cells and choosing a command, a dialog will pop up, asking which state values of the cells are to be interpolated. When you click on **OK** to close the dialog, the selected state values of the first and last cells you chose will be linearly interpolated onto the other cells you selected, based on the order in which you chose them and the distance between the first and last cells.

HELP MENU (UNDER CONSTRUCTION).

a. Toolbar. The **Toolbar** is a row of buttons at the top of the main window. Each of these buttons controls an application command. Clicking one of the buttons is a quicker alternative to choosing a command from the menu. Depending on the application or command selected, the buttons toggle the commands on and off. The buttons, with their actions and menu equivalents, are listed and shown below.

<i>Button</i>	<i>Action</i>	<i>Menu</i>
	Create a new project	File New
	Locate and open a project	File Open
	Save the file in the active window	File Save
	Cut selected cells to Clipboard	Edit Cut
	Copy selected cells to Clipboard	Edit Copy
	Paste from Clipboard	Edit Paste
	Print the active window	File Print
	Initiate model simulation	Project Run
	Replay simulation result	Result Play

	Pause model simulation or replay	Project Pause
	Stop model simulation or replay	Project Stop
	Edit model grid	Edit Grid
	Enable or disable zoom mode	View Toggle Zoom
	Zoom in for detailed grid structure	View Zoom In
	Zoom out to view whole grid system	View Zoom Out
	Show longer vectors of velocity field	View Vector Longer
	Show shorter vectors of velocity field	View Vector Shorter
	Show Grid Walls	Help Contents
	Show water surface elevation	Help Contents
	Show water depth	Help Contents
	Show bottom elevation	Help Contents
	Show velocity vector	Help Contents
	Show Froude number	Help Contents
	Display help file contents	Help Contents

b. Status Bar. The **Status Bar** at the bottom of the screen displays helpful messages or information regarding the current operation on the left. On the left of the Status Bar is a message area. After the mouse cursor rests on a tool icon for a short time, a description of the icon will appear in this window during calculation. The first box to the left of this one displays the maximum iteration error, and the next box indicates the present time. When the cursor is moved within the **Workspace**, the x, y coordinates of the cursor are displayed in a third box. Its physical coordinates (x, y, z) in the model domain and the selected variable (*e. g.* water surface elevation) will be shown in the fourth box. The box at the extreme right shows the zoom ratio (see also Fig. 1. 1).

III. FILE INPUTS AND OUTPUTS

A. THE PROJECT FILE

The **Project** file is an internal file which contains data that has been input to the model, and (or) that has been generated by the model. It is in binary format, and has the extension **.est**. The data could be interdependent; therefore you should not attempt to edit the file. For example, if the coordinates x, y , of a point in the model are changed or eliminated, the sides of the cell connected to this point (node) and the physical properties of the cell of which the node is a part will be changed as calculated by the model.

B. IMPORT AND EXPORT FILES

Import and Export files are in ASCII format, and have the extension **.prj**. All of the project-related data, such as grid information, initial conditions, boundary conditions, etc. in the **Project** file can be exported as an ASCII file. This file can be imported later if needed. Although **Import and Export** files can be edited with the Text Editor, we do not recommend that beginners directly edit the Project file with it. On the other hand, experienced users may find that editing the Import and Export files directly can save time and energy. But before you start editing, *always remember to back up your data, and to back up the program files!*

The following is an example of a very simple import and export project file:

Example: entry nos. in Line 1.

	1	2	3	4	5	6 - 10
Line No.	1	5	70.2814539930551	-13.3416015624998	639.0400000000001	479.2800000000001 1 1 1 1 0
2	1	1	0	355.0347222	-183.1597222	
3	1	2	1	422.7430556	-183.1597222	
4	1	5	1			
5	1996	10	26	5	37	59 -0.118872
6	1996	10	26	6	6	23 -0.100584
7	1996	10	26	6	34	47 -0.04572
8	1996	10	26	7	3	11 0.04572
9	1996	10	26	7	31	35 0.161544

10 1 4 0 422.743055555555 -246.527777777778
 11 1 6 0 353.298611111111 -246.527777777778
 12 2 3 1 1.57079632678995 67.7083334 1 2 9 0
 13 2 5 1 6.28318530646337 63.3680555777777 2 4 9 0
 14 2 7 1 4.71238898036984 69.4444444444444 4 6 9 0
 15 2 8 1 3.11420224547 63.3918334600072 6 1 9 0
 16 3 9 1 10.1 388.454861116667 -214.843749988889 4345.55242454668 5.2 -0.13 0.05 0 0 0.025 4 3 5 7 8
 17 4 2 2
 18 7
 19 8
 20 4 1 2
 21 3
 22 5
 23 6 2
 24 381.94 -198.78 5 1
 25 401.91 -197.05 4.9 1

Line 1

The first number is a definition number. In this case, "5" means "this line is about model settings."

The second and third numbers are the x and y coordinates of the upper left corner of the application work space.

The fourth and fifth numbers are the x and y dimensions of the application workspace.

The sixth number codes for engineering units (0 = English; 1 = Metric).

The seventh number indicates whether nodes will show on the screen (1 = shown; 0 = not shown).

The eighth number indicates whether walls will show on the screen (1 = shown; 0 = not shown).

The ninth number indicates whether cells will show on the screen (1 = shown; 0 = not shown).

The tenth number indicates how the cell is to be shown on the screen (0 = cell number; 1 = head distribution; 2 = velocity vector; 3 = Froude number; 4 = bottom elevation; 5 = water depth).

Line 2 to End

Beginning with line 2, the first number of each line defines the content of that line, as follows:

Line 2. Node data. The second number is the node number; the third number indicates whether there is attached observation data for this node (0 = no data is attached for this node; 1 = data attached for this node). The fourth and fifth numbers are the x and y coordinates of the node. If there is attached observation data for the node, then the next line will begin an observation data section (for details on the format of observation data sections, see the section on the Observation Data File).

Line 3. Wall data. The second number is the wall number; the third number indicates whether this wall is a boundary wall (0 = boundary wall; 1 = not a boundary wall); the fourth number is the wall angle with respect to the axis; the fifth number is the wall length; the sixth number is the beginning node number; the seventh number is the ending node number; the eighth number is the left side cell number; and the ninth number is the right side cell number. If the wall is a boundary wall, there may not be a left or right side to the cell, and in that case the eighth number or ninth number should be set to zero.

Line 3. Cell data. The second number indicates whether the cell is a structure or non-structure cell (0 = non-structure; 1 = structure); the third number is the cell number; the fourth number indicates whether the cells is a boundary cell (0 = non-boundary cell; 1 = boundary cell); the fifth number is the water elevation; the sixth and seventh numbers are the x and y coordinates of the center; the eighth number is the cell area; the ninth number is the cell bottom elevation; the tenth and eleventh numbers are the initial velocities in the x and y directions respectively; the twelfth and thirteenth numbers are the bottom surface slopes in the x and y directions respectively; the fourteenth number is Manning's coefficient, n ; and the fifteenth number is the number of walls surrounding this cell; the following numbers are the wall serial numbers.

Line 5. Boundary section description line. The second number indicates the boundary type condition for the section; the third number (say, n) indicates how many walls this boundary section contains. The next n lines are the serial numbers of the walls making up this boundary section.

Line 6. Outline section description line. The second number indicates how many points are contained in the outline data. The next n lines (here, 4) will be the x , y , z coordinates for each outline point plus a flag digit indicates whether or not the point is linked with other outline points.

C. OUTLINE FILES

An **Outline** file records a set of locations inside the modeling domain, where the bottom elevations are known. The elevations are normally obtained by field survey. The outline file has a **.oln** file extension, and should be prepared by a text editor before it is imported into the Outline file. The Outline file format is of three columns x , y , z , separated by blank spaces. The following is an example:

```
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
663118.208 1075617.905 3.130
```

Units acceptable for x , y , z are either decimal feet or meters. Before each import, you should make sure that the system of units used in the project file is the same as that in the outline file.

D. BOUNDARY CONDITION FILES (OBSERVATION DATA FILES)

An observation data file (**Boundary Condition File**) contains data normally observed in the field as tide staff readings and current measurements. These observation data are associated with nodes within the model grid. When a node is located on the border, the data associated with that node normally becomes a boundary condition. The **Boundary Condition File** normally has a **.bdc** file extension, and has the following format:

```
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
```

Line 1: The first number is the boundary condition type. In the example shown, 1 means the first type of boundary condition (water elevation boundary); a 2 in this position means the second type of boundary (flow boundary). The second number is the number of entries in the file (how many sets of observation data; here, 10); and the third number represents the engineering unit system (0 = English, decimal, 1 = metric).

Line 2 to end: Each subsequent line represents a set of observation data. The first number is the year, the second number is the month, the third number is the day of the month, the fourth number is the hour (in 24 hr. format), the fifth number is minutes, the sixth number is seconds, and the seventh number is the observed value at that time. (It may seem awkward to have the year, month, etc. in every line, but it is more easily read and managed by the program).

E. RESULT FILE

The **Result** file records the modeling results at fixed time intervals throughout the entire simulation time span. At each cell, center location, water elevation, and current velocity in the x and y directions are recorded.

The **Result** file is a binary file and it has to match the **Project** file. Therefore if the **Project** file is changed, the **Result** file must also be changed to match it. The **Result** file has the file extension **.rnn**, where **nn** (00, 01, etc.) may represent increments in the simulations at different times.

F. EXPORTED TIME HISTORY FILES

Calculation results for the cells under study in the **Result** files can be output to an ASCII file so that you may edit them in a number of ways.

The **Exported Time History** file has the file extension **.tmh**. The following is an example of one of these files:

Head	3862
0	0.878803
300	0.871205
600	0.860556
900	0.856184
1200	0.860379
1500	0.868827
	0.876505
U	3862
0	-0.0316636
300	-0.0757137
600	-0.118039
900	-0.108686
1200	-0.0642724
1500	-0.00458682
1800	0.0662282
V	3862
0	0.0626617
300	0.115999
600	0.175231
900	0.163989
1200	0.0986731
1500	0.00539711
1800	-0.1029

In this example, the number 3862 is the serial number of the cell selected. In the first block of numbers above, columns one through six contain the time (in seconds), and the following columns are the calculated water elevation at the centers of the cells. The second block is the calculated results for the velocity component in the x direction, and the third block contains the calculated results for the velocity component in the y direction, on the common boundary of these two cells.

IV. CONSTRUCTING THE MODEL GRID

A computational grid is here defined as a network of cells constructed from connected nodes and assigned walls. The grid should be so constructed that it represents the essential features of the domain to be modeled. FVHM provides you with a number of tools for grid construction and editing.

A. CREATING A NEW PROJECT

1. The first thing you do is to create a **New Project** for the site to be modeled.
2. To open the **FVHM Main Window**, double click the **ESTU.EXE** program icon from the directory where you installed the FVHM software package. The initial **workspace** screen is empty, and your project has the default name **Untitled**.
3. Go to **File Menu**, select the **File | Save As** command, and a **File Save As** dialog box like that shown in Fig. 4. 1 will appear.

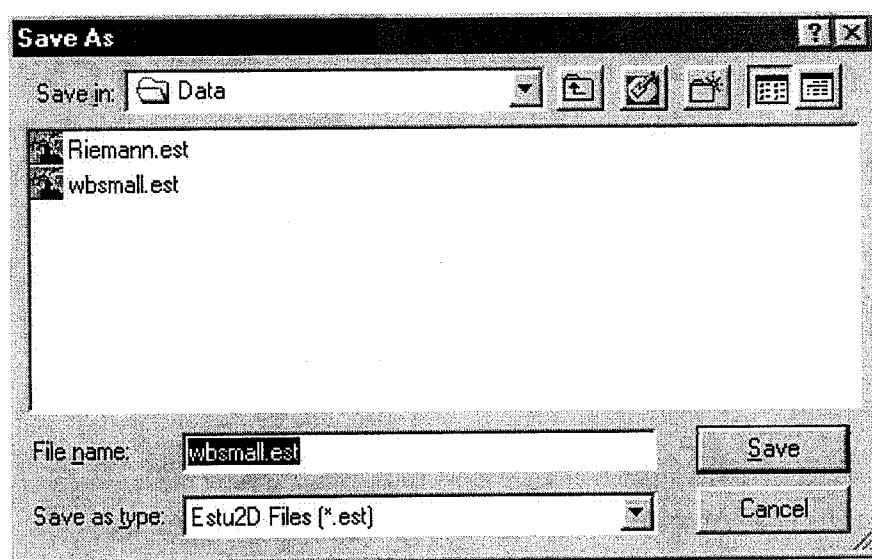


Fig. 4. 1. Save As Dialog Box.

4. Choose the directory and file name for the project (for example, **new.est**). Remember that the project file name must use **.est** as its extension. Then click the **OK** button, and a box will

pop up to confirm that you wish to create the new file.

B. PROJECT OPTIONS

1. Click the **OK** button in the confirmation box that appeared under step 4 above, and a **Project | Options** dialog box pops up (see Fig. 4. 2.). From this box you can define the engineering units to be used, the project domain size, what to draw on the screen, and how to draw the cells.

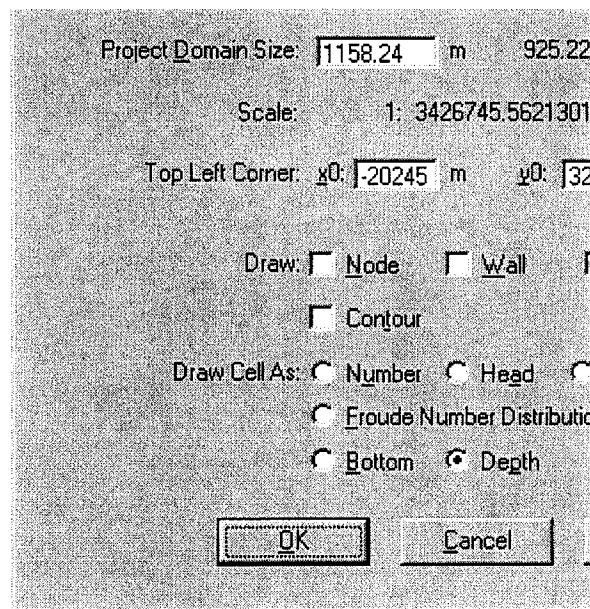


Fig. 4. 2. Project Options Dialog Box.

2. Click the **OK** button after selecting your options. Now you are ready to construct the grid. (Already existing projects may be opened now by using the **File | Open** command.)

C. IMPORTING THE OUTLINE

Before creating or editing the grid, the outline of the site to be modeled (for example, from survey data, existing maps, aerial photographs, or orthophotographs) may be imported to

the main graphics window. This outline file has the extension **.oln**, and may consist of a set of points with *x*, *y*, *z* coordinates, and can be displayed in the background as a guide for creating or editing cells, nodes, and walls. To import an outline:

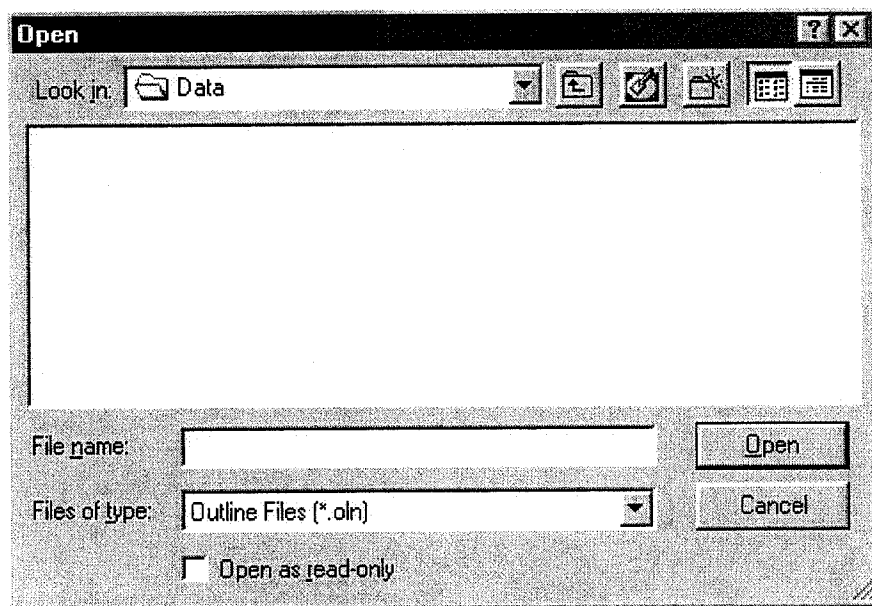


Fig. 4. 3. Open Dialog Box.

1. Select **File | Import | Outline**, and an **Open Dialog** box will be brought up (see Fig. 4. 3).
2. In the **Drives** box, choose the drive in which the project directory was created.
3. In the **Folders** box, locate and open the project directory.
4. In the **File Name** box, find and highlight the outline file, and click the **OK** button. This will import the outline data points to the FVHM main window.

D. CREATING A NEW CELL

A new cell can be created by the following procedure:

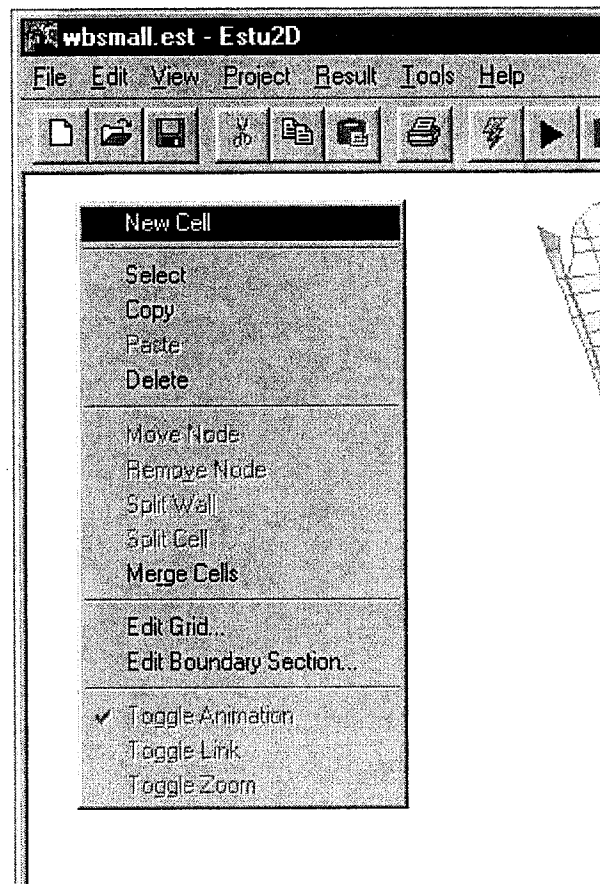


Fig. 4. 4. Context Menu.

1. Select **Edit | New Cell**, or right-click inside the project work space. A **Context Menu** will pop up under the mouse cursor (see Fig. 4. 4). Select **New Cell**.
2. Move the cursor to a starting point in this **workspace** and left-click the mouse. A new node will be created. This is the first node.
3. Next, move the mouse to another location inside the project work space, and left-click again to create the second node. The program will automatically create a new wall.
4. Repeating this process will create a series of walls. Once you close a path by clicking back on its first node, a new cell will be created and the process ends (see Fig. 4. 5).

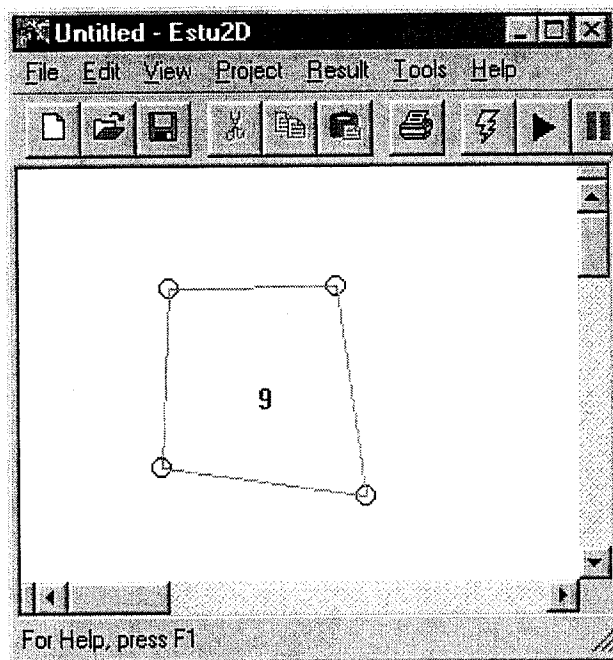


Fig. 4. 5. Creating New Cells.

5. Repeat the above four steps to create additional cells as desired.

E. SELECT CELL(S)

The **Select** command is used to select cells that user wants to **copy**, **delete**, **edit**, or **merge**.

To select cells:

1. Click the **Edit | Select** command, or press **Ctrl + M**. FVHM is now in **Select** mode.
2. Click on any cells you wish to select. The numbers of the cells will be highlighted.
3. To select a contiguous group of cells, click and *hold* the left mouse button, then drag a rectangle around the group of cells.
4. Click again on an already highlighted cell to deselect it.

After the selected cells have been **edited, deleted, copied, or merged**, FVHM will exit the **Select** mode automatically.

F. COPY AND PASTE CELL(S)

The Copy and Paste commands can save you a great deal of time in constructing cells. To copy a cell or a group of cells,

1. Click the **Edit | Select** command.
2. Click on the cell(s) you want to copy.
3. Click the **Edit | Copy** command. The cell(s) has now been copied into a memory buffer.
4. Click the **Edit | Paste** command. The copied cell(s) appear highlighted at the cursor.
5. Move the cursor to where you want to paste the cells, and left-click the mouse. The **Paste** procedure is now completed.

G. DELETE CELL(S)

To **delete** a cell or a group of cells,

1. Click the **Edit | Select** command.
2. Click on the cell(s) you want to delete.
3. Click the **Edit | Delete** command. The cell(s) have now been deleted.

H. MERGE CELL(S)

1. Click on the **Edit | Select** command.
2. Click on the cell(s) you want to merge.

3. Click on the **Edit | Merge Cells** command. The cells will be merged if the model permits it. Not all cells can be merged, because, for example, a cell representing a man-made structure (“structure cell”) cannot be merged with normal cells.

I. MOVE NODE

If a cell is poorly shaped, you can adjust it by moving its nodes. To move a node:

1. Click the **Edit | Move Node** command. FVHM is now in **Move Node** mode, and only the **Move Node** command shows on the **Edit** menu.
2. Move the cursor to the node you want to adjust.
3. Click and hold the left mouse button, and move the cursor to a new location.
4. Release the left mouse button. The node has now been repositioned.
5. Click the **Edit | Move Node** command again and FVHM will exit the **Move Node** mode.

J. REMOVE NODE

You can also adjust a the shape of a cell shape by removing one or more nodes, as follows.

1. Click the **Edit | Remove Node** command. FVHM is in **Remove Node** mode, and only the **Remove Node** command shows in the **Edit** menu.
2. Move the cursor to the node you want to remove.
3. Click the left mouse button, and the node will be removed.
4. Click the **Edit | Remove Node** command again and FVHM will exit the **Remove Node** mode.

Note: Once a cell becomes a triangle none of its three nodes may be removed.

K. SAVING THE DATA

Once the grid has been constructed and edited, it should be saved to the **Project File**. To do this simply click the **File | Save** command. A box will be brought up indicating that the grid has been saved successfully (see Fig. 4. 6). Click **OK** to return to the main window.

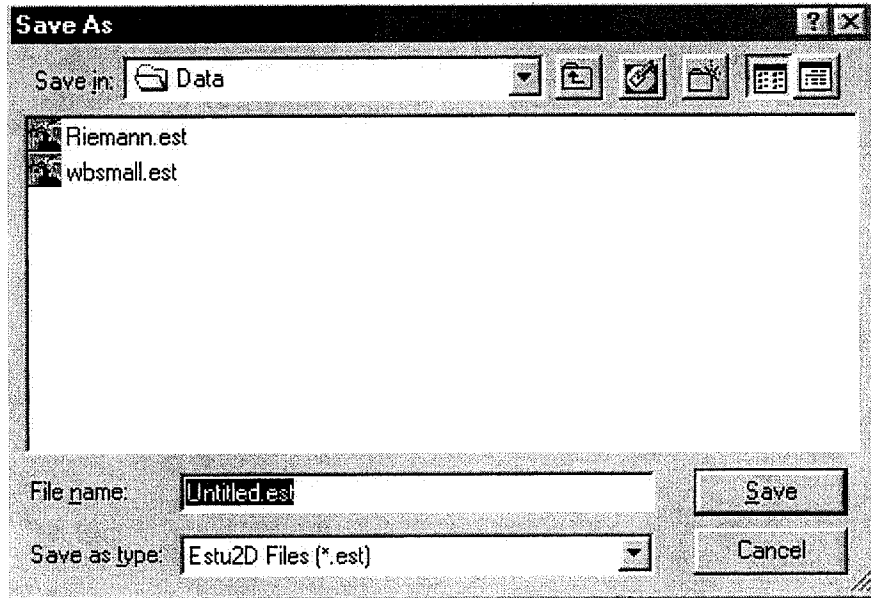


Fig. 4. 6. Save Project File.

We recommend that you save the grid once in a while during the process of grid construction and editing, especially if the grid is large.


V. ASSIGNMENT OF BASIC PARAMETERS AND BOUNDARY CONDITIONS

Once a grid has been constructed and saved, FVHM automatically defines and numbers both boundary and interior nodes, walls, and cells. What you need to do next is:

1. Assign initial conditions, bottom elevations, and Manning's n coefficients to each of the cells; and,
2. Assign boundary conditions.

A. ASSIGNING BASIC PARAMETERS

The basic parameters of a grid are **initial conditions, bottom elevations, and Manning's roughness coefficients**. To assign these basic parameters to a cell,

1. move the cursor to the cell first, then right-click the mouse and choose the **Edit Grid** command, or click **Edit | Grid** in the menu, or click the Toolbar icon . A **Cell Property** dialog box will be brought up (see Fig. 5. 1).
2. The **Cell Property Dialog** allows you to assign or modify the properties of each individual cell (in Fig. 5. 1, the individual cell is cell no. 23), such as the initial water elevation of the cell, the bottom elevation of the cell, the velocity in x and y directions and the Manning's coefficient, n .

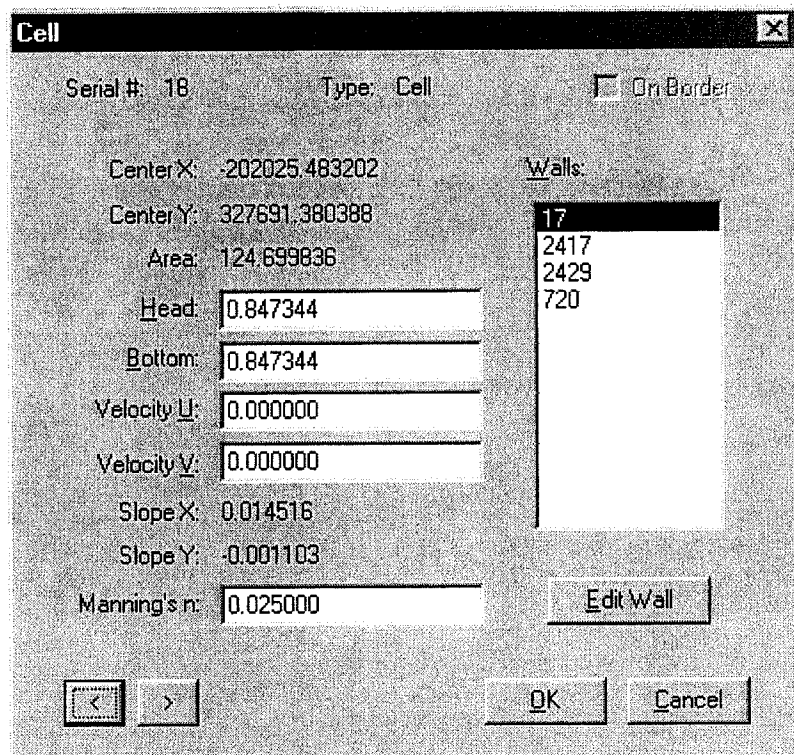


Fig. 5. 1. Cell Property Dialog Box.

In the dialog box, the center coordinates are the averages of all the nodal coordinates of the cell. Slope of a cell bottom is calculated by comparing it with the bottoms of neighboring cells.

3. The **Walls** panel on the right lists all the walls belonging to this cell. By selecting one of the walls and clicking the **Edit Wall** button, the **Wall Property** dialog for that wall will be brought up, allowing you to examine the properties of that wall. The panel tells you if the cell is a boundary cell. You may use the < and > keys to browse through all the cells in the grid.

It may be convenient (and it decreases run time) if you assign the same parameters to a group of cells using the **Group Edit** dialog:

1. Click the **Edit | Select** command.
2. **Select** a group of cells with the same basic features.

3. Click the right mouse button and choose the **Edit Grid** command to bring up the **Cell Property Edit** dialog box (see Fig. 5. 2).

4. Choose and input the parameters, then click **OK**.

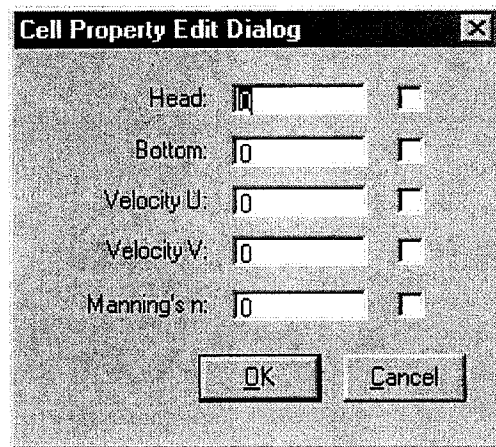


Fig. 5. 2. Cell Property Edit Dialog Box.

B. ASSIGNING BOUNDARY CONDITIONS

Normally, the model boundaries consist of the land boundary, the water boundary, and the flow—no flow boundary. Specifying boundary sections and boundary types is a critical step in the use of this model. You must specify boundary sections, and associate boundary condition types with each section unless the project has no flow boundary conditions that apply uniformly to all the of boundaries. The no-flow condition is the default if boundary sections are not specified. In FVHM, there are three types of boundary conditions: Type 1. water surface elevation; Type 2. no flow; and, Type 3. flow or velocity. Assigning boundary conditions consists of two operations:

- a. Specifying boundary sections and boundary types;
- b. Assigning observed data to boundary nodes.

You may specify boundary sections and boundary types via the following steps:

1. Click the **Project | Options** command to bring up the **Project Options** dialog box (see Fig. 5. 3).

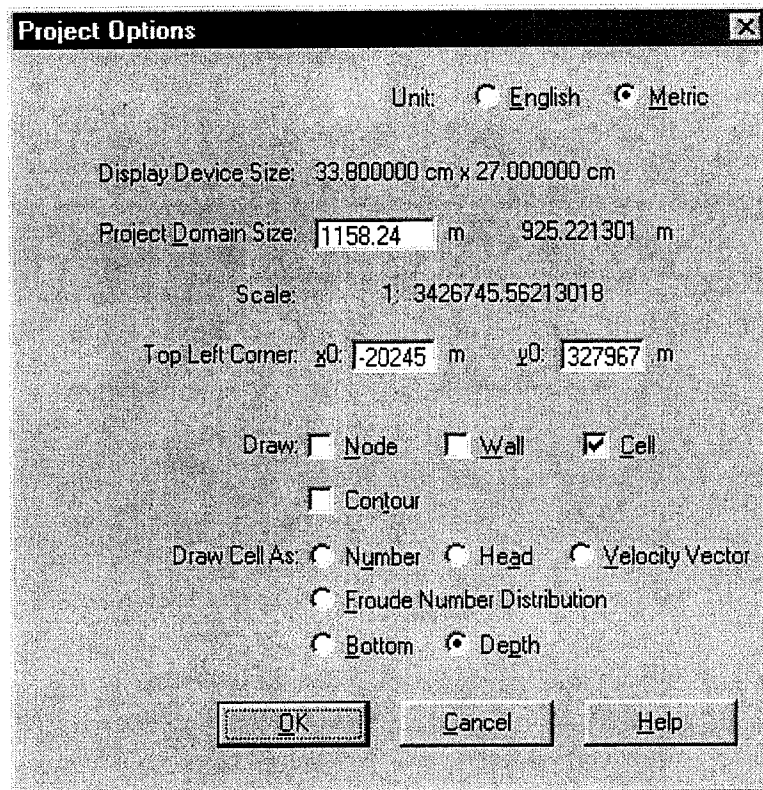


Fig. 5. 3. Project Options Dialog Box

2. From **Draw** options, select **Wall**, then click **OK**. This shows the project as it will appear within the walls. The walls will be highlighted when they are selected in the **Boundary Section Edit** dialog.

3. Click the **Edit | Boundary Sections** command to bring up the **Boundary Sections Edit** dialog box (see Fig. 5. 4). The box on the left lists all the walls at the border. In the list-box, the first column is boundary wall number, the second column is boundary section number, and the third column is the boundary type. The **Boundary Section Edit** dialog allows you to group all of the boundary walls into sections, and to specify the type of boundary condition at each section.

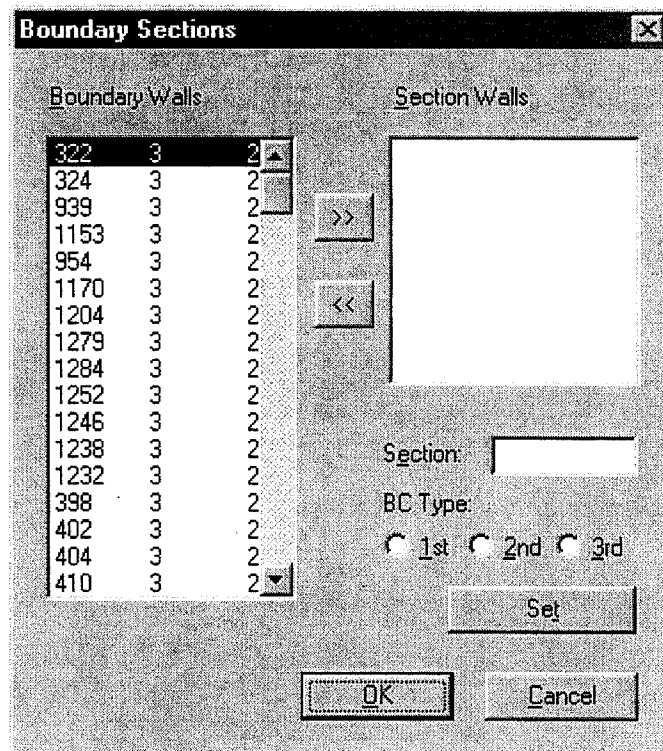


Fig. 5. 4. Boundary Sections Edit Dialog Box.

4. Select a group of walls from the **Boundary Walls** list-box either by **Shift+click** or **Ctrl+Left-click**. The walls will be highlighted in the project workspace as they are selected.

5. Click the **>>** button in the middle of the **Boundary Section Edit** dialog box. The selected walls will be listed in the list-box on the right. You may select more walls from the left and add them to the right or select them from the right and use the **<<** button to remove them. Keep in mind that you should keep the walls selected in the right list-box within a consecutive boundary section, and highlighted sections in the workspace should be one contiguous unit.

6. **Assign a section number** in the **Section** box, and specify the boundary condition type for this section. Remember, Type 1 is the water surface elevation boundary, Type 2 is the land boundary (the no-flow boundary), and Type 3 is the flow boundary, through which the flow rate is controlled by a given formula.

7. Click the **Set** button, and the right **Section Walls** list-box will be cleared. The section number and boundary condition type number will be listed along with the wall number in the left **Boundary Walls** list-box.

8. By repeating steps 4 through 7, you can specify multiple boundary sections, and assign different boundary condition types to those sections.

9. After specifying the boundary sections and boundary types, click the **OK** button. The program will then perform a check on the sections. The following errors may occur if the boundaries are not correctly specified:

- a. walls in a section may not be continuous; and(or),
- b. the boundary condition type may not match the observed data already attached to one or more nodes in a section.

If no error is found, the program will keep the the section information, and save it when you save the project. The section information will be cleared if any boundary walls were removed as a result of **Cell Delete** and **Cell Merge**. If that happens you has to go through the **Boundary Section Edit Dialog** again.

C. OBSERVED DATA

After specifying the boundary sections and boundary types, you should assign the observed data to the boundary nodes. Commonly, observed data are available for only a few nodes or for just a single node, for a boundary section. In such cases, fro example, where data could be the diurnal variation of tidal stage, or a flow hydrograph at an upstream boundary, FVHM automatically interpolates between nodes.

Here are the steps you follow to assign observed data to boundary nodes:

1. Prepare the **observed data file (boundary condition file)** in the format described in Section III, and save the file into the project directory. The observed data file must have the extension **.bdc**.

2. Move the cursor to the boundary cell with the observed data, and then click the **right**

mouse button and select the **Edit Grid** command. A **Cell Property Dialog** is brought up (see Fig. 5. 1).

3. Click the **Edit Wall** button. A **Wall Property dialog** box will be brought up (see Fig. 5. 5).

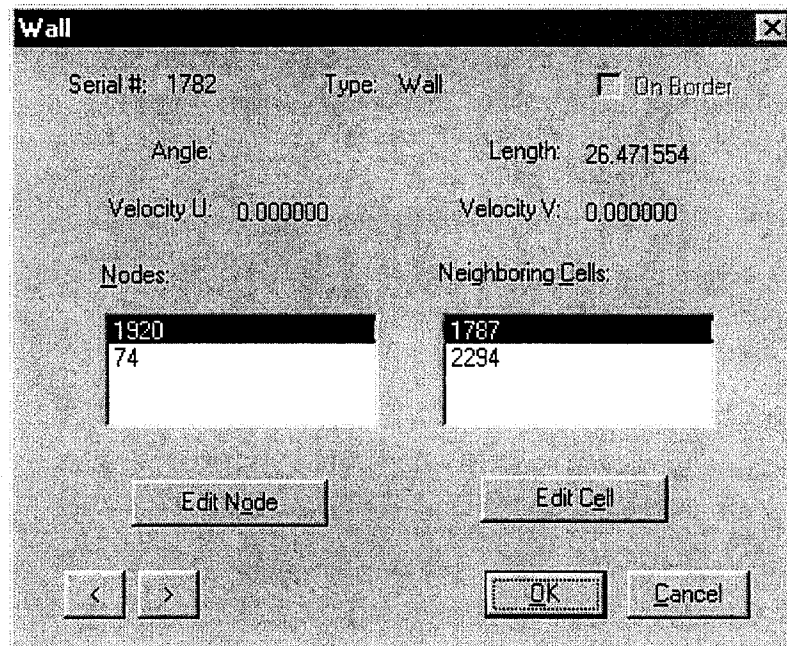


Fig. 5. 5. Wall Property Dialog.

4. Click the **Edit Node** button to bring up the **Node Property dialog** (see Fig. 5. 6).

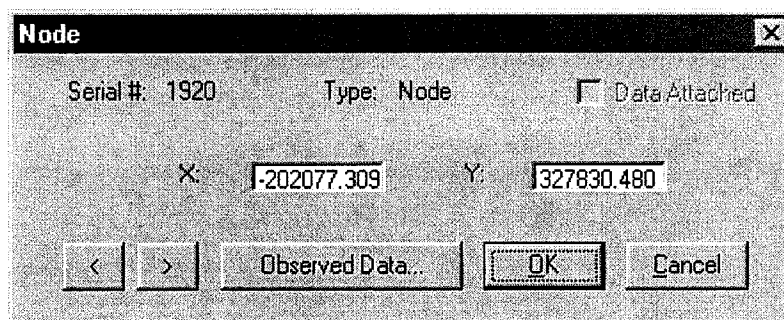


Fig. 5. 6. Node Property Dialog Box.

5. From the **Node Property** dialog, you may specify a new x and y location for the node. The node will be moved to a new location, the length of the wall the node is attached to will be recalculated, and the area of the cell the node belongs to will also be adjusted.

6. The < and > buttons will allow you to browse through all the nodes in the project. Use these to browse the boundary nodes. Selected nodes will be highlighted. Find the node with the observed data, and click the **Observed Data** button. This will bring up a **File | Open** dialog box, which allows you to attach a string of observed data to the node (see Fig. 5. 7).

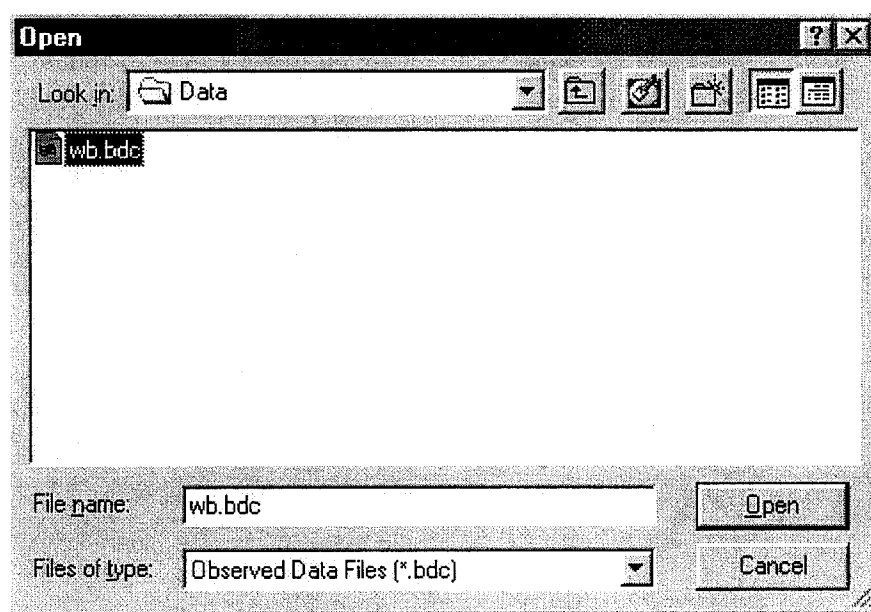


Fig. 5. 7. Open Observed Data File.

7. Select the observed data (that were input earlier to a **Boundary Condition File**, described in Section III), and click the **OK** button. The observed data will be assigned to the node automatically.

8. Repeat steps 1 through 4 for all other boundary nodes for which observed data are available.

D. SAVING THE PROJECT

After assigning basic parameters and boundary conditions, don't forget to **Save** the project.

VI. RUNNING THE MODEL

Once you have finished building the grid and assigning basic parameters and boundary conditions to it, the project file—which includes all of the data necessary to run the model—has been generated. Now you are ready to have FVHM simulate the dynamic process. To run FVHM:

1. Click the **Project | Run** command to bring up the **Modeling Options** dialog box (see Fig. 6. 1). This dialog allows you to specify the parameters of how the simulation will run.

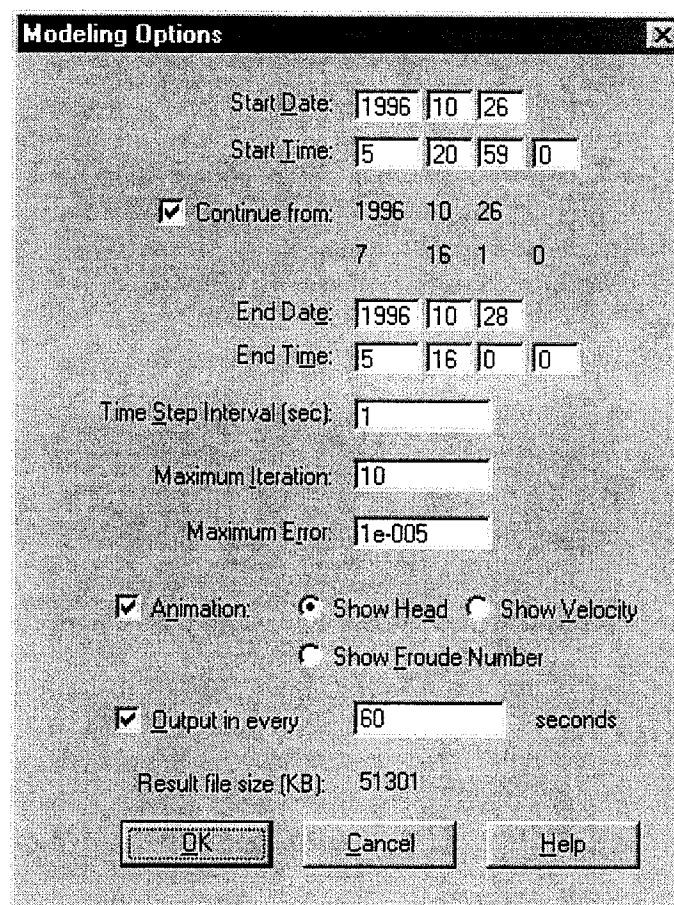


Fig. 6. 1. Modeling Options Dialog Box.

2. Select **Start Date** and **Start Time** for the simulation. You may start the simulation at any time after the last boundary condition starting time.

3. Select the **End Date** and **End Time** for the calculation. You may end the calculation anywhere inside the boundary (the end date and time must not be earlier than the start date and time).
4. Select the **First Time Step**, **Maximum Iteration**, and **Maximum Error Allowance**.
5. Select **Animation** if required. If animation is required, you can further choose the **View Content (Head, Velocity, or Froude Number.)** Animation allows you to check the computation dynamically, but it slows down the simulation.
6. Select the **Output Time Interval**.
7. Click **OK** to run the model. You can stop or pause the simulation anytime in between the calculation by clicking the **Project | Stop** or **Project | Pause** command. The simulation can be continued from the time you stopped or paused so long as you **Save** the previous calculation results.

VII. POST-PROCESSING

The results calculated by FVHM are saved in a single (binary) results file with the extension .R00. FVHM provides two ways you to view this result file, described in this section.

A. RESULT PLAY

Just like showing a set of slides, **Result Play** shows the whole dynamic process of the simulation in a series of plots. The time interval between plots is fixed, and was set in **Modeling Options** (the Output Time Interval in Fig. 6. 1.) when you ran the model. You can play the results for **Water Depth**, **Water Elevation**, **Velocity**, or **Froude Number** by clicking the corresponding toolbar. The following steps are taken to view the simulation process:

1. Click **File | Open** to open the **Project File** (extension .est).
2. Click the **View | Depth**, **View | Water Surface**, **View | Velocity Vector**, or **View | Froude Number Distribution** command to set up the view content. You may also do this by clicking the corresponding toolbar.
3. Click **Result | Play** to play the result.

Note: At present the print function under FVHM is still under construction, so you cannot print result plots directly. To output a plot however, your can use following procedure:

1. Click **File | Open** to open the **Project File** (extension .est).
2. Click the **View | Depth**, **View | Water Surface**, **View | Velocity Vector**, or **View | Froude Number Distribution** command to set up the view content. You can also do this by clicking the corresponding toolbar.
3. Click the **Result | Play** command to play the result.
4. Press the **Pause/Break** button as soon as he required plot appears in the FVHM main

window. This stops **Result Play** temporarily, and the plot you want shows in the main window.

5. Press **Alt + Print Screen** to copy the plot to your memory buffer.

6. Open the **Windows Paint** program from your computer's **Accessories** (see Fig. 7. 1).

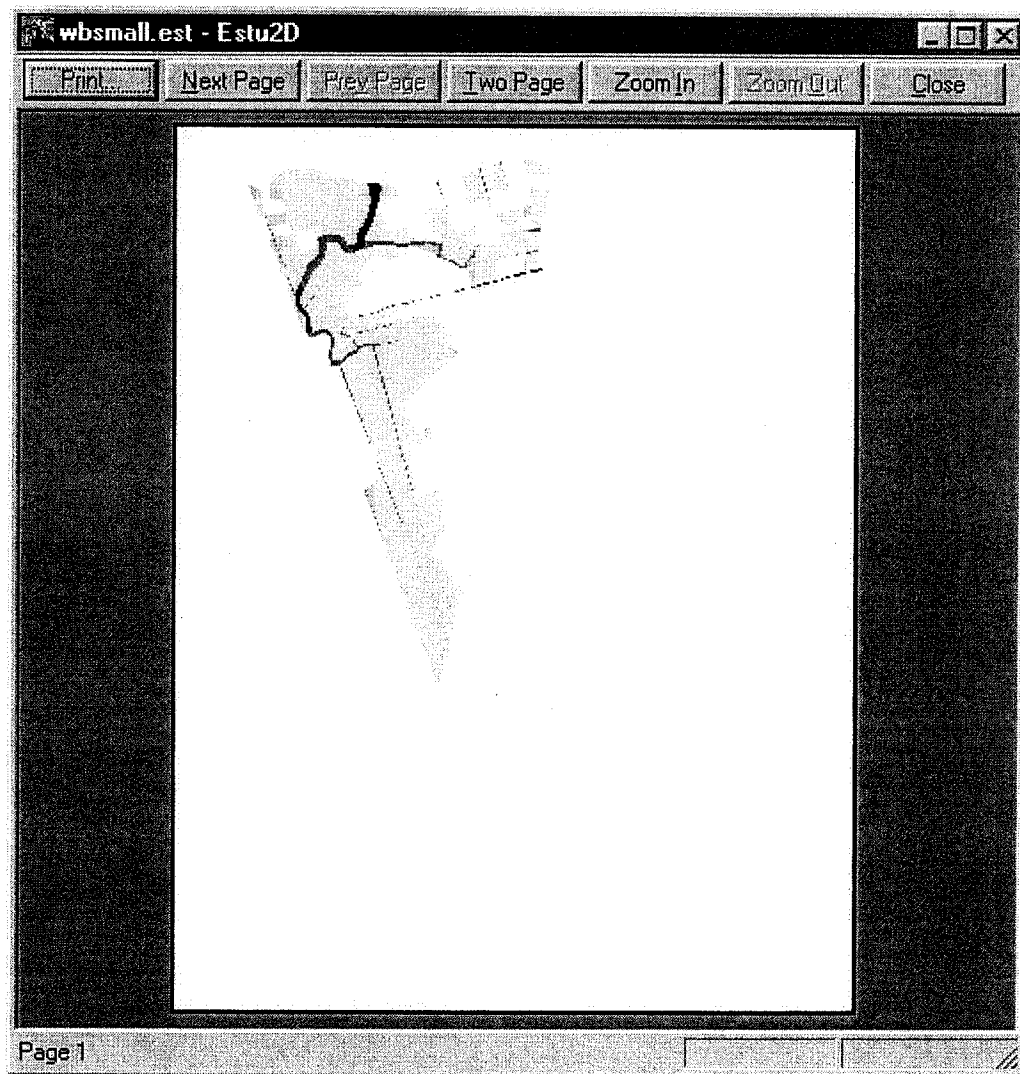


Fig. 7. 1. Paint Screen.

7. Click **Paint's Edit | Paste** command, and the plot you saved in the memory buffer will be moved into the **Paint** window.

8. Click **Paint's File | Print** command to output the plot to your printer.
9. Repeat Step 2 through Step 8 to print all of the other plots you may want.

B. RESULT | TIME HISTORY | EXPORT

The **Result** file contains all of the solutions for each node in binary format, and is therefore not conveniently handled by you. FVHM provides a way to output those results to an ASCII file, so you can handle the Result data with other tools, such as Microsoft Excel. This ASCII file is called the **Time History File**, and has the extension **.tmh**. The format of this file was introduced in Section III. To output the results at certain nodes to an ASCII file, take the following steps:

1. Click **File | Open** to open the **Project File** (extension **.est**).
2. Click the **Edit | Select** command and select the cell(s) for which you want to output the calculation results. The **Result | Time History | Expert** mode is enabled only after the cell(s) has been selected.
3. Click the **Result | Time History | Expert** command to output the results for the selected cell (s). A **Save As** dialog will pop up, asking for a **File Name** and **Folder** (see Fig. 7. 2.).

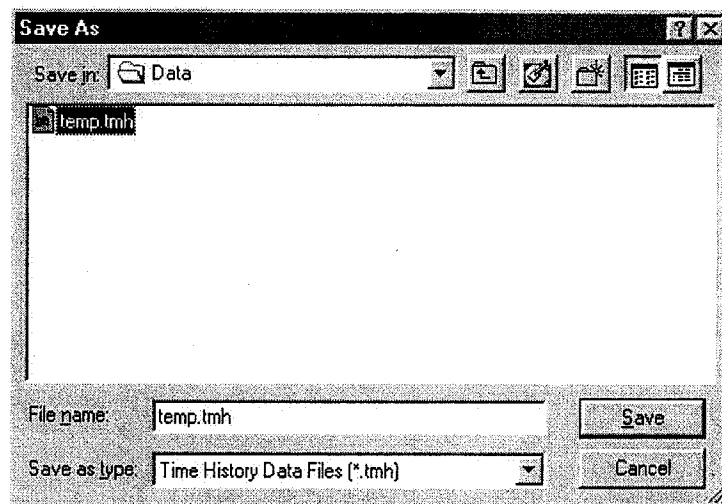


Fig. 7. 2. Save As Dialog Box.

4. Select the folder and input a file name; then click the **OK** button.

VIII. FINAL REMARKS

In this version of FVHM, the finite-volume method for the solution of the shallow-water hydrodynamics in a river—marsh system has been completed. The graphical user interface in the Windows environment, including pre- and post-processing of input and output data, graphical presentation of the results, and animation, is not considered fully completed, however. We stress that FVHM will only be as useful as the input data is complete: that is, if the regulatory process requires demonstration of final water surface elevations to a particular tolerance, then the precision of field surveys and other data observations must reflect that. We are continuing to implement, refine, and update various elements of this model, and would appreciate any comments or feedback from users.

* * *